Analysis of a reinforced concrete hypar roof

Exercise of course CIEM5301 Shell Structures

P.C.J. Hoogenboom, 20 September 2025

In this exercise we analyse a reinforced concrete shell roof by the finite element method. The roof consists of four hypars and eight edge beams. We will model the roof, perform a linear elastic analysis, perform a linear buckling analysis and design reinforcement. At the end, we conclude that this roof can be built.

In preparation of this classroom exercise, you need to install SCIA Engineer on your computer. You can download the software for free from

https://www.scia.net/nl/forms/gratis-engineering-software-studentenlicentie-aanvraagformulier

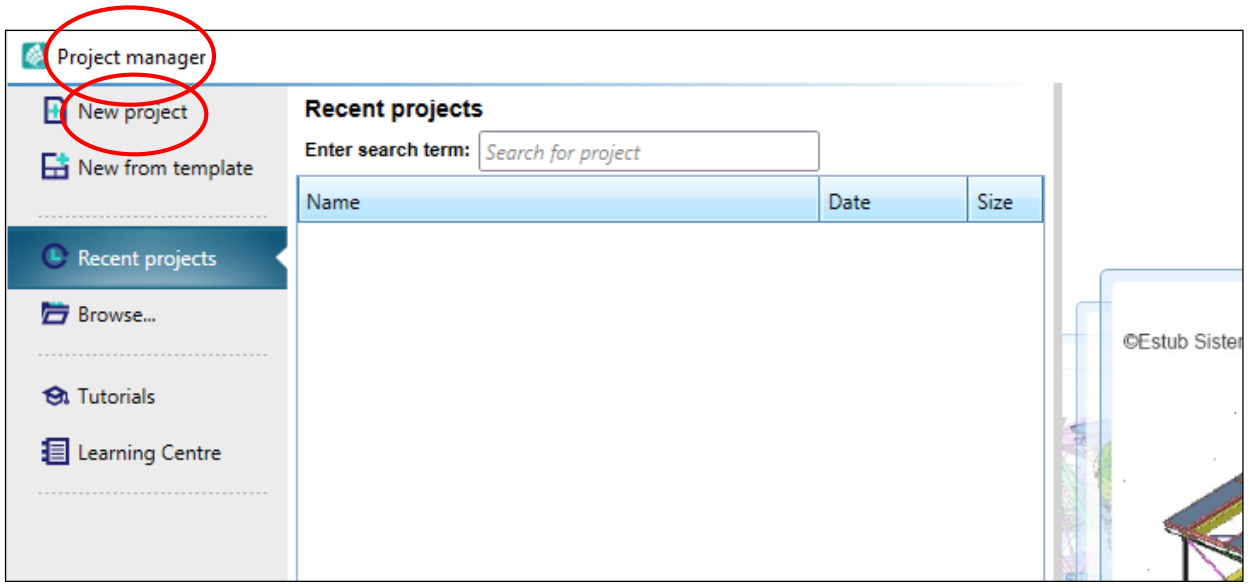
Use your university email address. Installing takes less than an hour. The installation goes wrong on about 10% of computers. In this case, do the exercise together with another student.

This classroom exercise takes about 1.5 hour. You can do it at your own pace, however, do not go as fast as possible. You learn most if you keep an overview. You are invited to discuss things with fellow students and the teacher. If you discover something interesting, show it to others. You do not need to write a report. You will not receive a grade for this exercise. When you are done, you may leave … or help others.

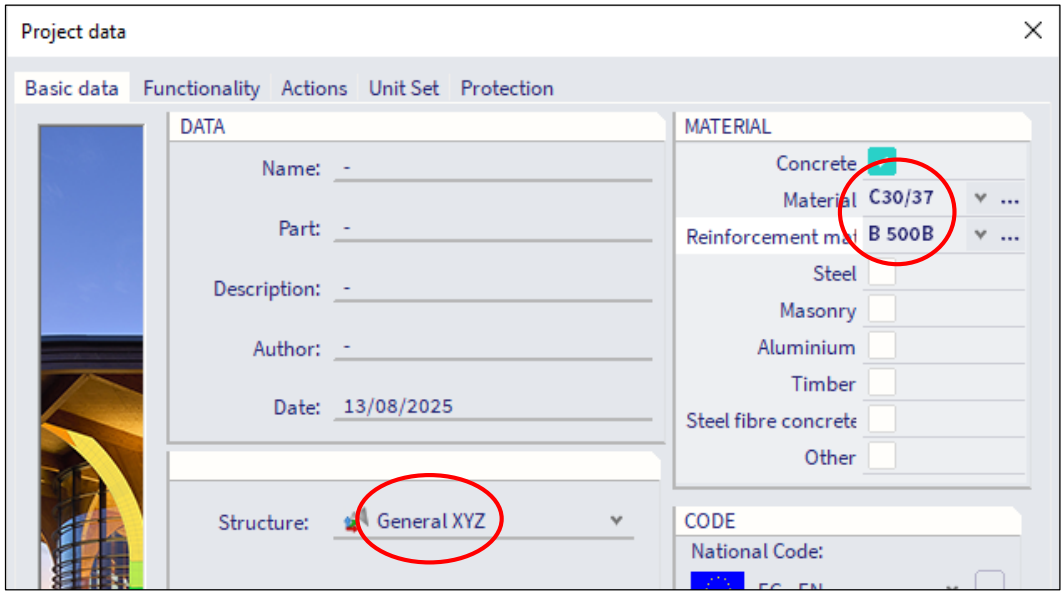
You need to be online when using SCIA Engineer because your license is in the cloud and the software occasionally checks this licence.

The name SCIA is derived from scientific analyser. The “C” in “SCIA” is not pronounced. Note that the first “c” in science is not pronounced either.

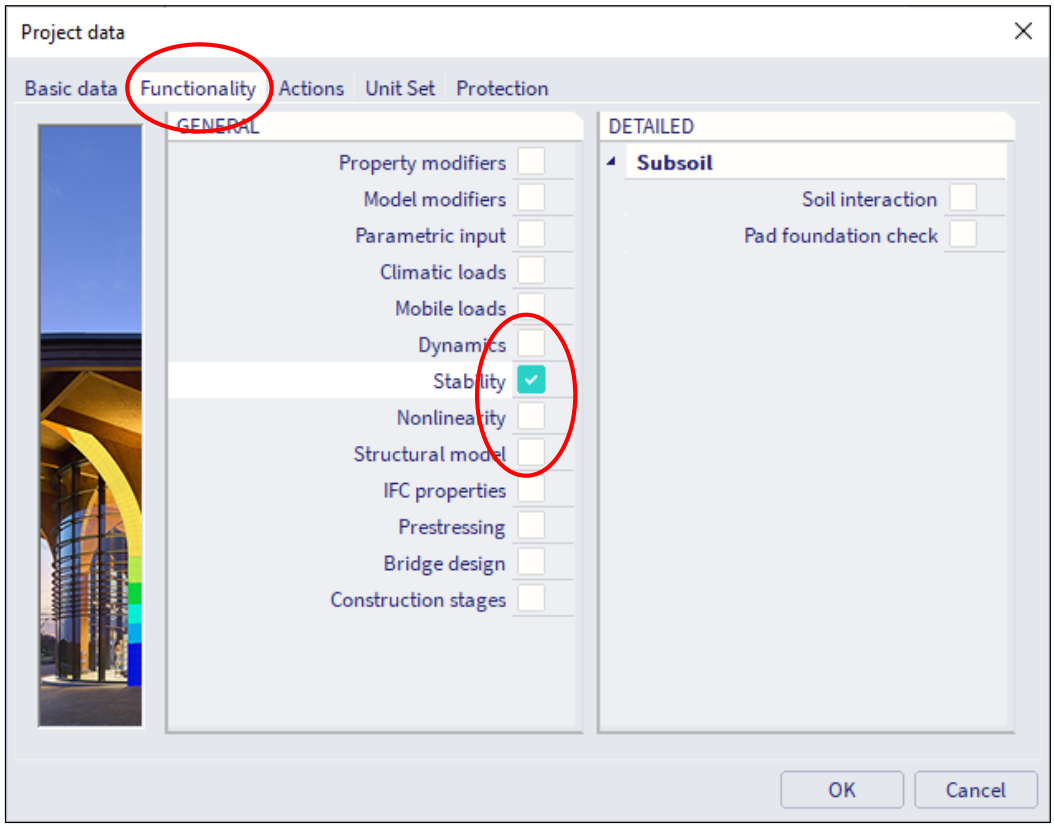
Start SCIA Engineer and start a new project.



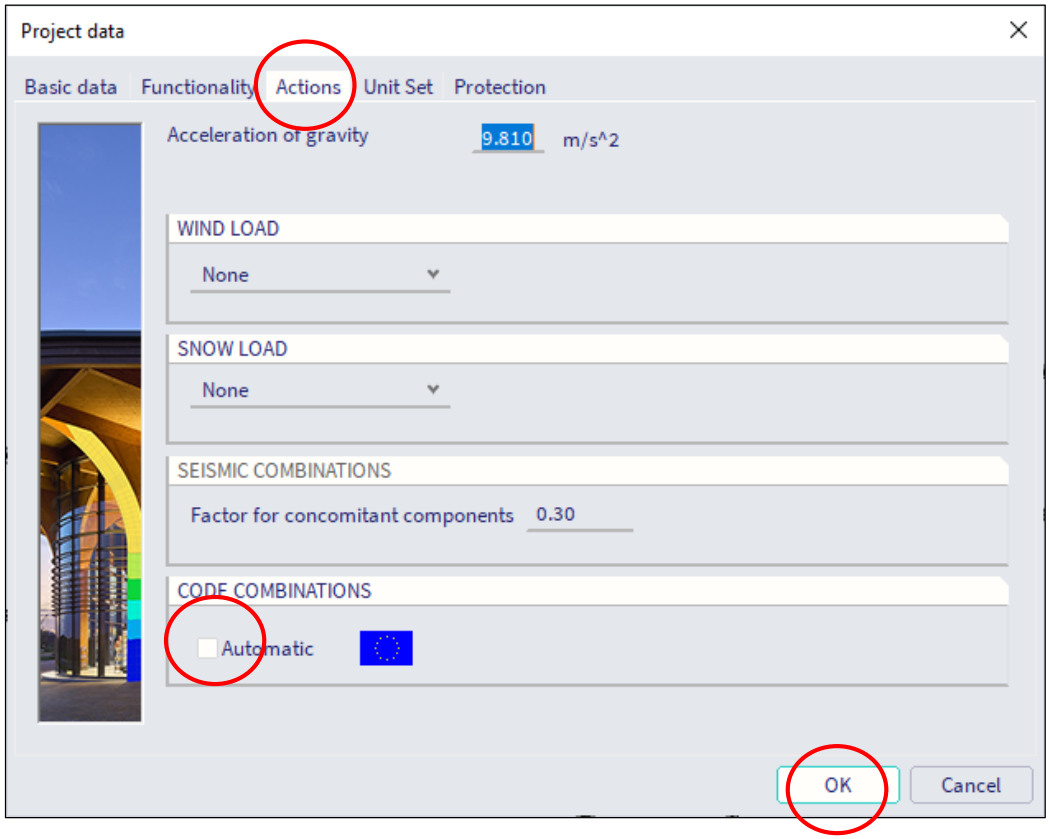
Select General XYZ. Select Concrete C30/37 and Reinforcement B 500 B.



By selecting functionality we add commands to the menu. Commands that we later need. Select Stability. The others we do not need today.



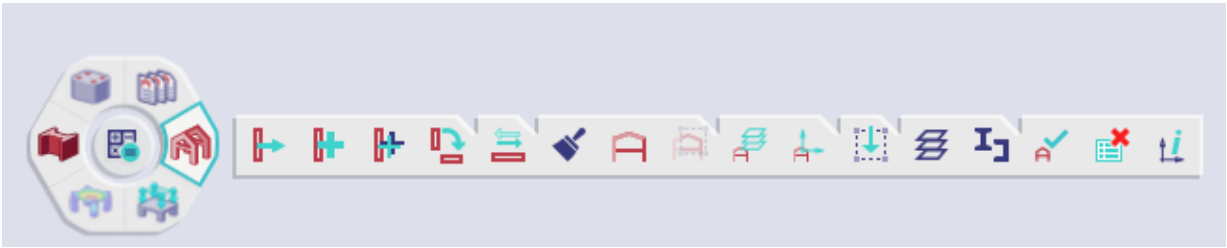
We switch off automatic generation of load combinations. We will enter these ourselves.



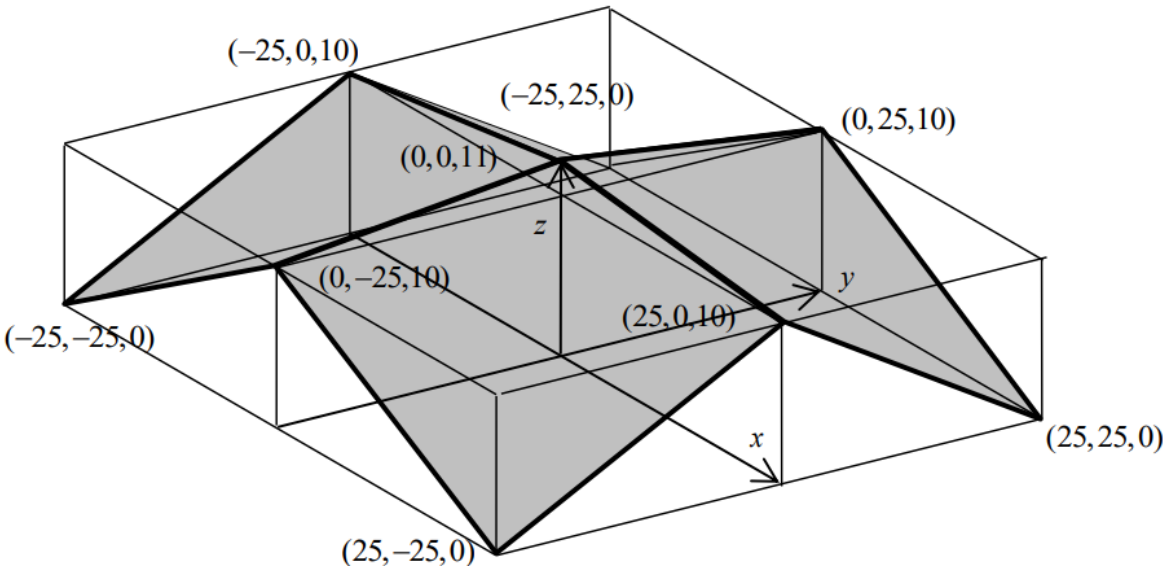
When we continue, more commands will become available on the circle menu. The commands can be organised in three parts.

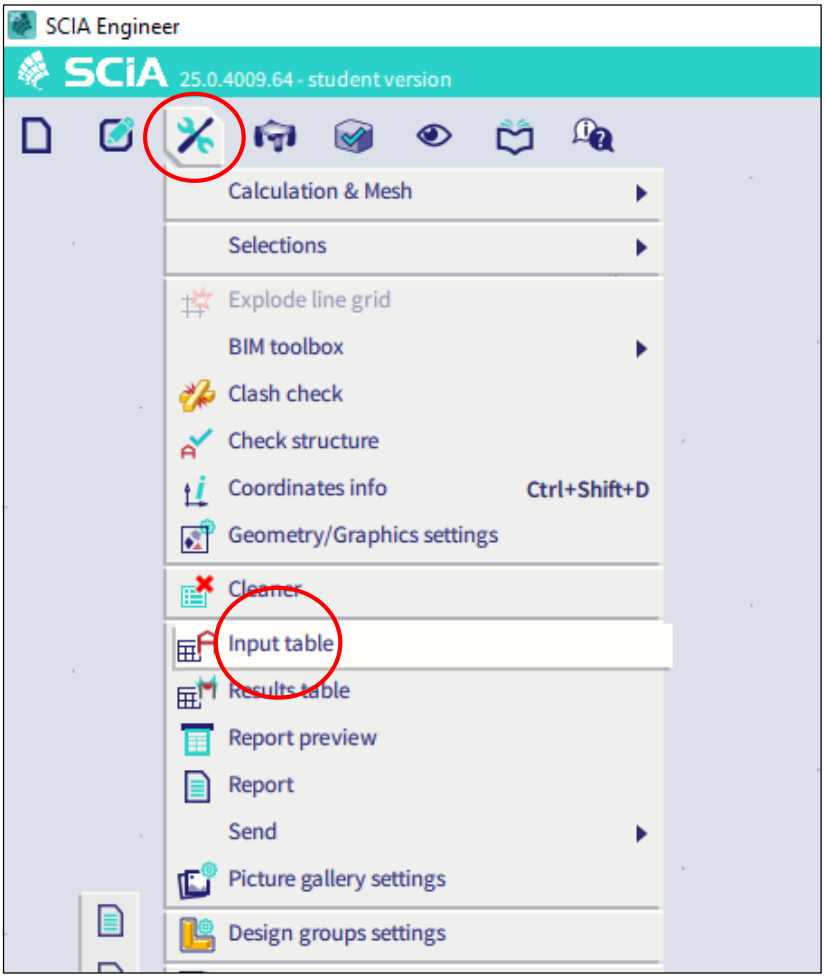
1 Structure, Loads …….. Pre-process
2 Calculate ..... Analyse
3 Results, Concrete, Steel, Reports ..... Post-process

All structural analysis programs and finite element programs are organised this way. In the old days, these three parts were separate programs. Today, the parts are unified by a graphical user interface. When designing, we cycle through the parts 1 – 2 – 3 – 1 – 2 – 3 – et cetera.



We model a four hypar roof with a span of 50 m.

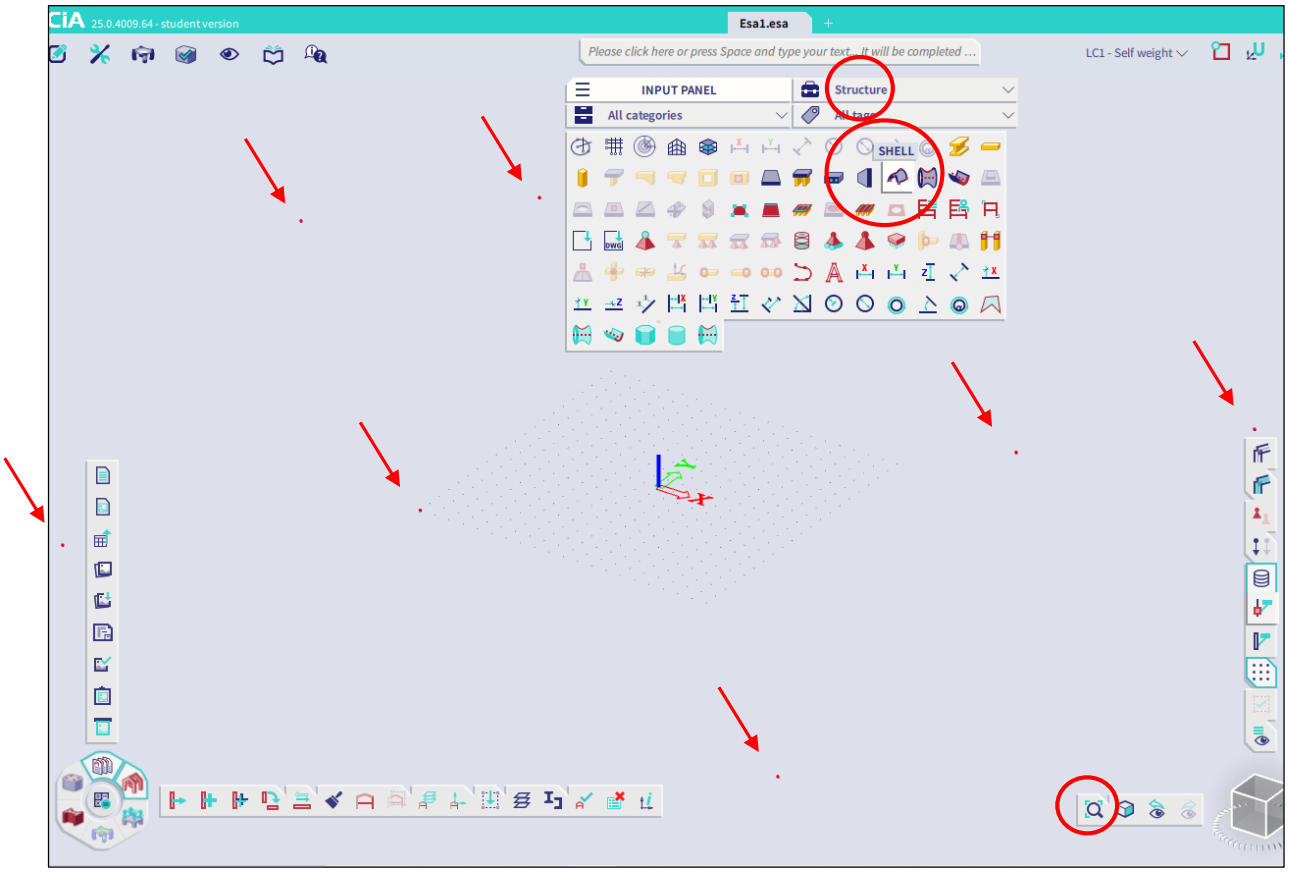




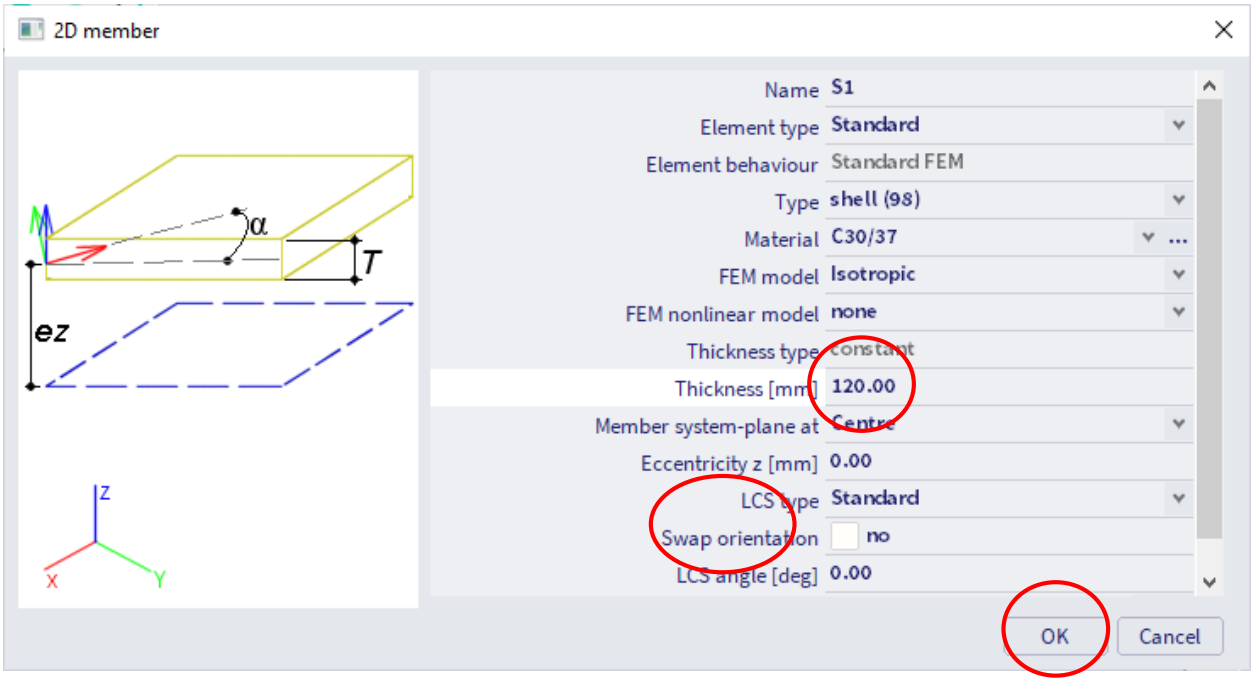
INPUT TABLE | Structure | Type here

	Name	X [m]	Y [m]	Z [m]	Member	2D member
1	N1	-25.000	-25.000	0.000		
2	N2	0.000	-25.000	10.000		
3	N3	25.000	-25.000	0.000		
4	N4	25.000	0.000	10.000		
5	N5	25.000	25.000	0.000		
6	N6	0.000	25.000	10.000		
7	N7	-25.000	25.000	0.000		
8	N8	-25.000	0.000	10.000		
9	N9	0.000	0.000	11.000		
*						

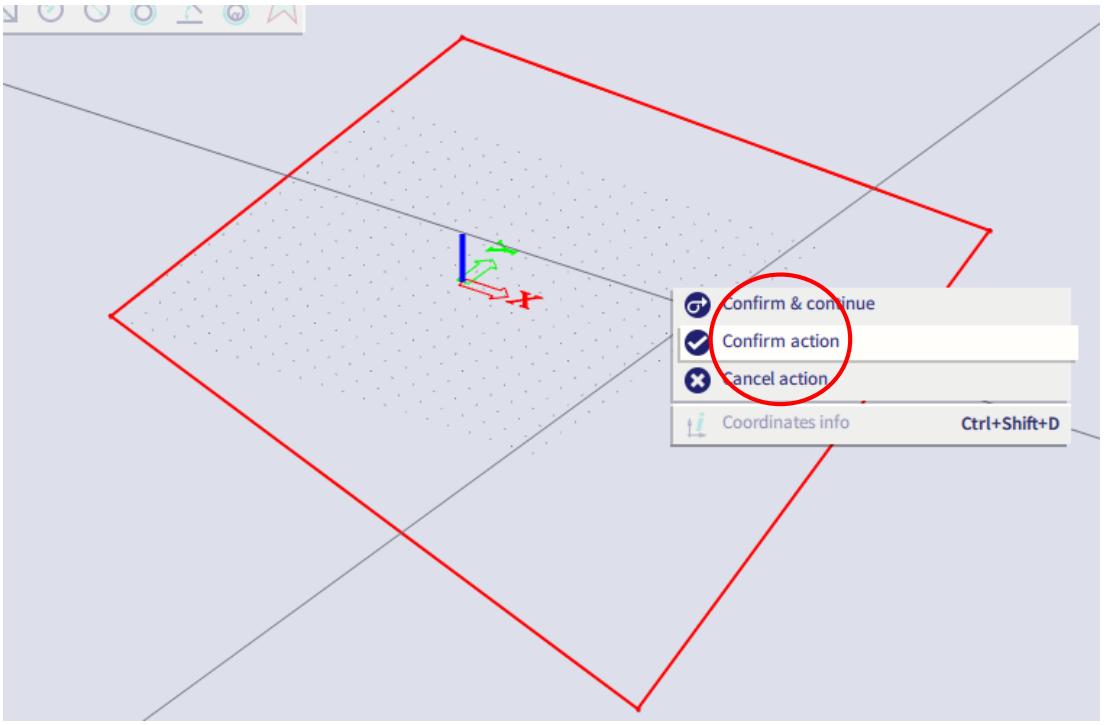
Zoom all. All nine points can be seen. Click on Shell.



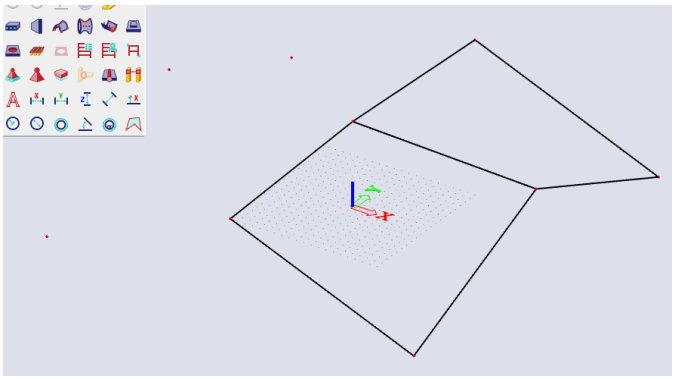
Enter the thickness of 120 mm. Note that in this window, we can also change the z axis of the local coordinate system (LCS) (inwards or outwards). Perhaps we need to do this later.

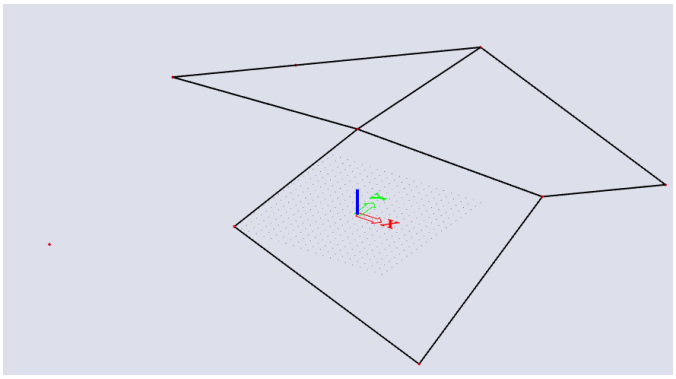


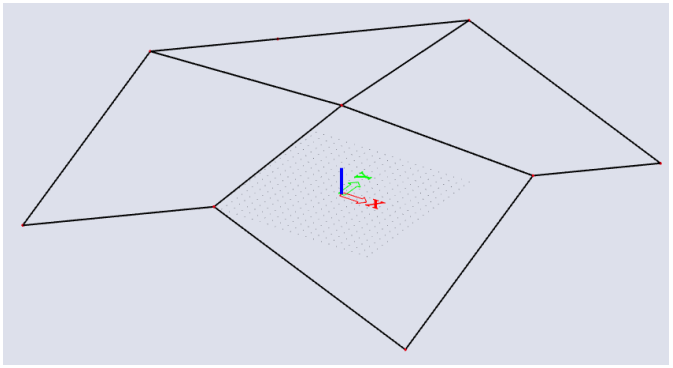
We enter one of the four hypars by clicking on the points. A command can be ended by clicking right and on confirm action. If you want to get out of a command, press “esc” three times.



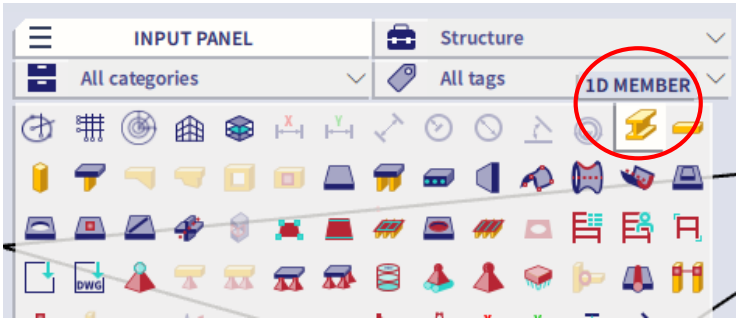
The other three hypars can be entered in the same way.

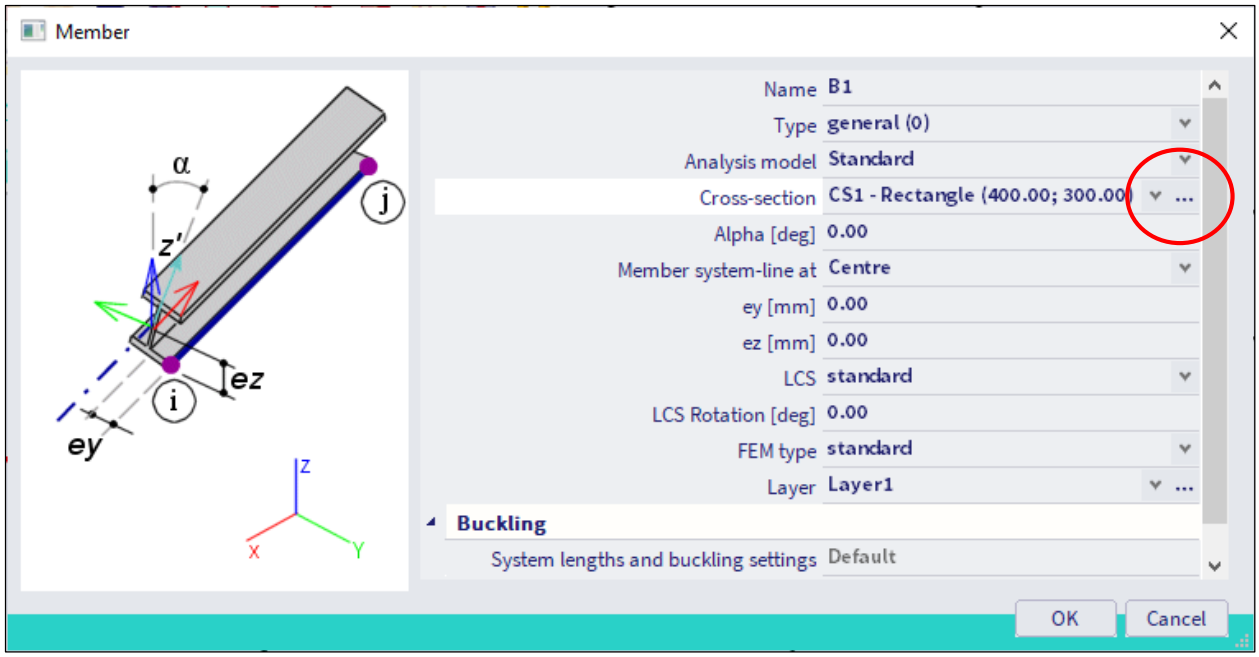


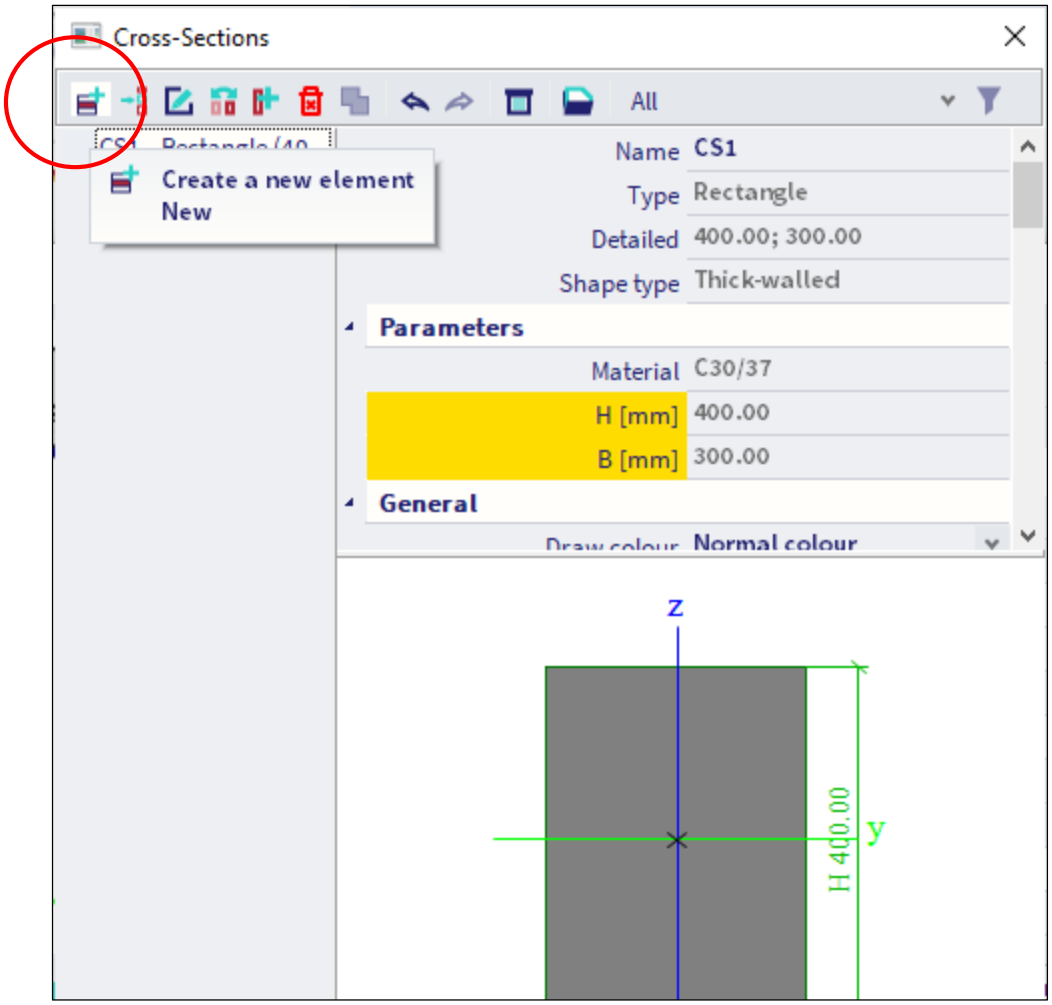


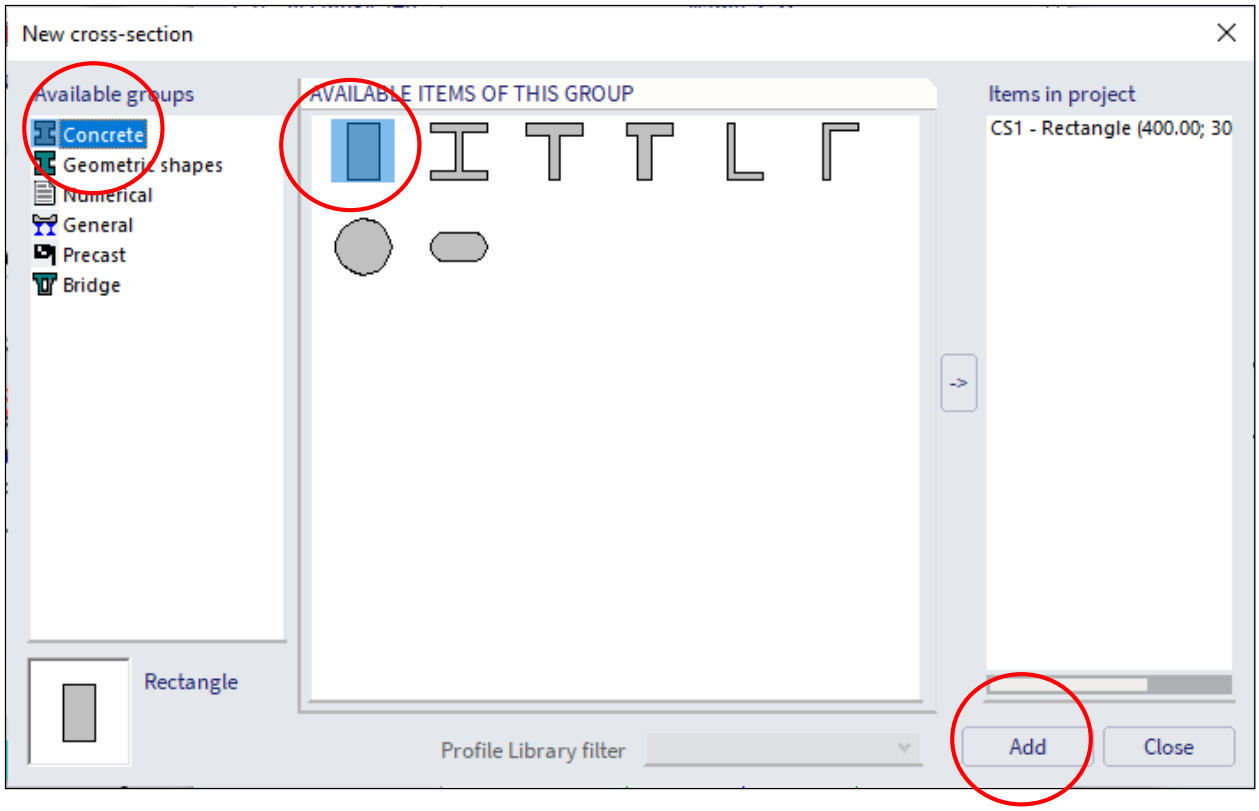


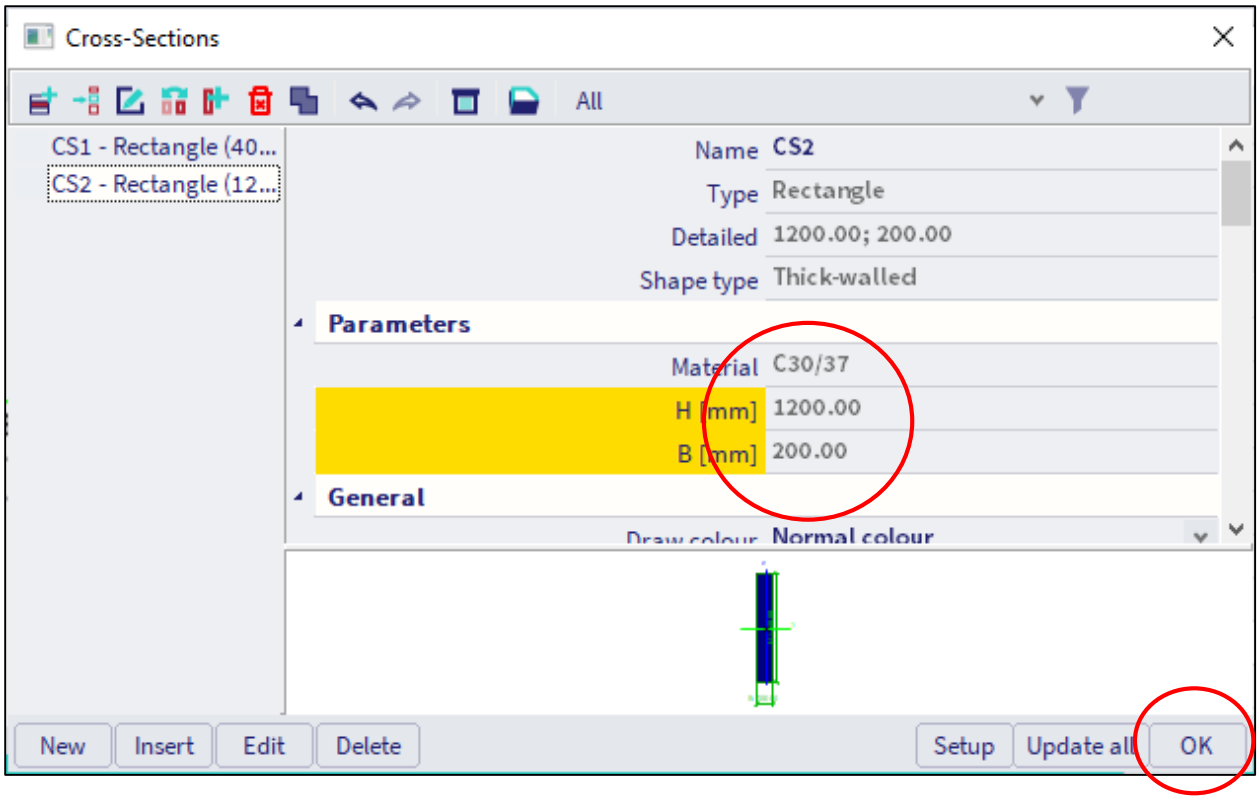
Before entering edge beams, we need to define a cross-section.

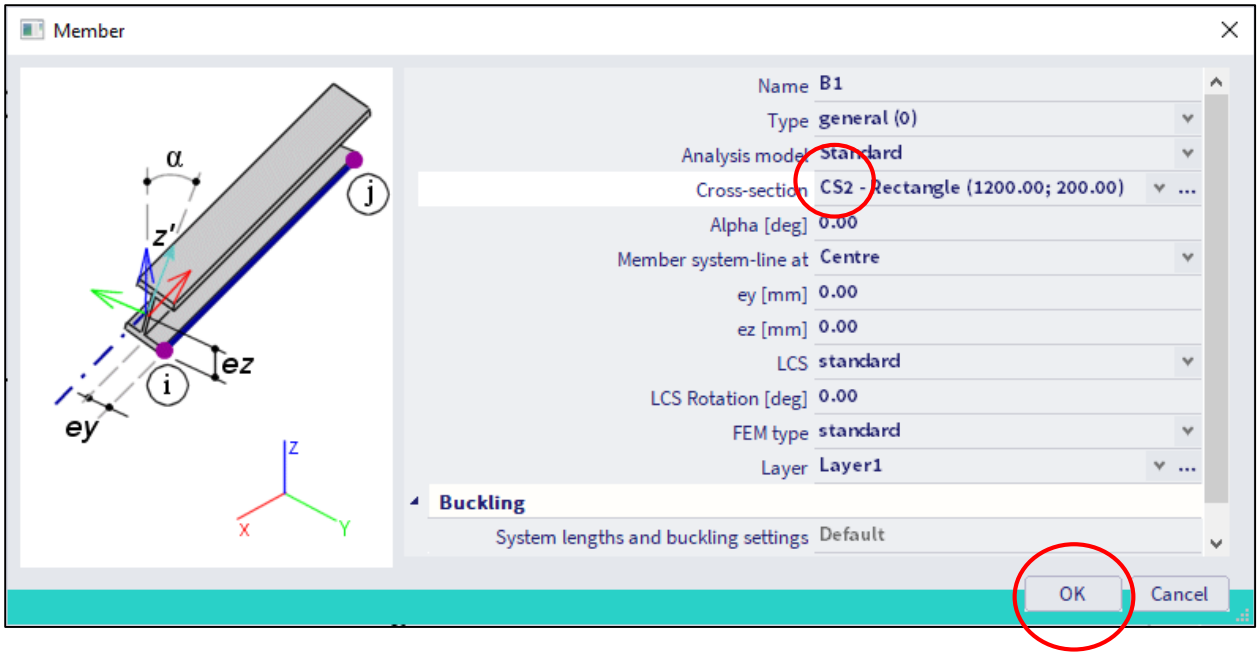



Member
α
z'
j
i
ez
ey
Z
X
Y
Name B1
Type general (0)
Analysis model Standard
Cross-section CS1 - Rectangle (400.00; 300.00)
Alpha [deg] 0.00
Member system-line at Centre
ey [mm] 0.00
ez [mm] 0.00
LCS standard
LCS Rotation [deg] 0.00
FEM type standard
Layer Layer1
Buckling
System lengths and buckling settings Default
OK
Cancel

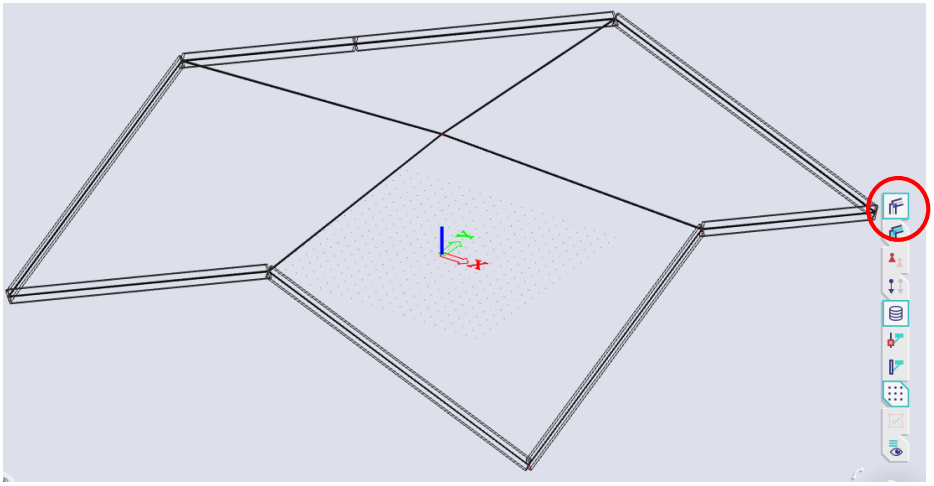

Cross-Sections
All
Create a new element
New
Name CS1
Type Rectangle
Detailed 400.00; 300.00
Shape type Thick-walled
Parameters
Material C30/37
H [mm] 400.00
B [mm] 300.00
General
Normal colour
Z
y
H 400.00


New cross-section
Available groups
Concrete
Geometric shapes
Numerical
General
Precast
Bridge
AVAILABLE ITEMS OF THIS GROUP
Items in project
CS1 - Rectangle (400.00; 30
->
Rectangle
Profile Library filter
Add
Close

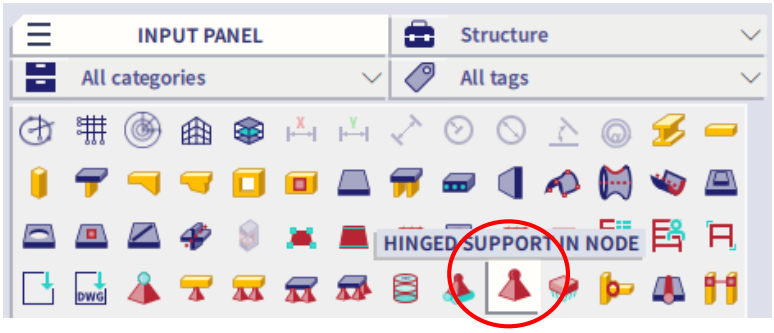

Cross-Sections
All
CS1 - Rectangle (40...
CS2 - Rectangle (12...
Name CS2
Type Rectangle
Detailed 1200.00; 200.00
Shape type Thick-walled
Parameters
Material C30/37
H [mm] 1200.00
B [mm] 200.00
General
Draw colour Normal colour
New
Insert
Edit
Delete
Setup
Update all
OK



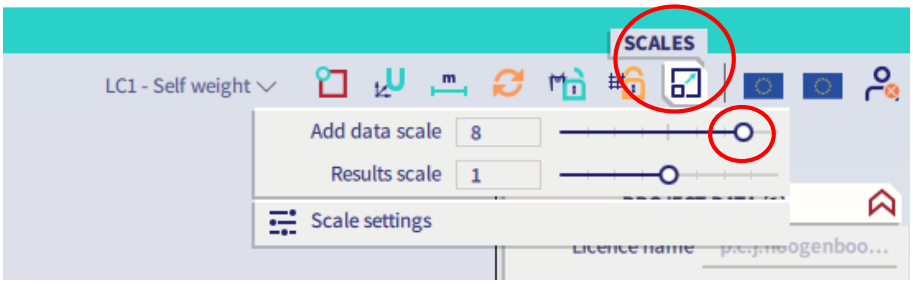
Add an edge beam to the model by clicking on a node and then on another node. Repeat this seven times. Click the Volumes toggle. Now we see the edge beams.

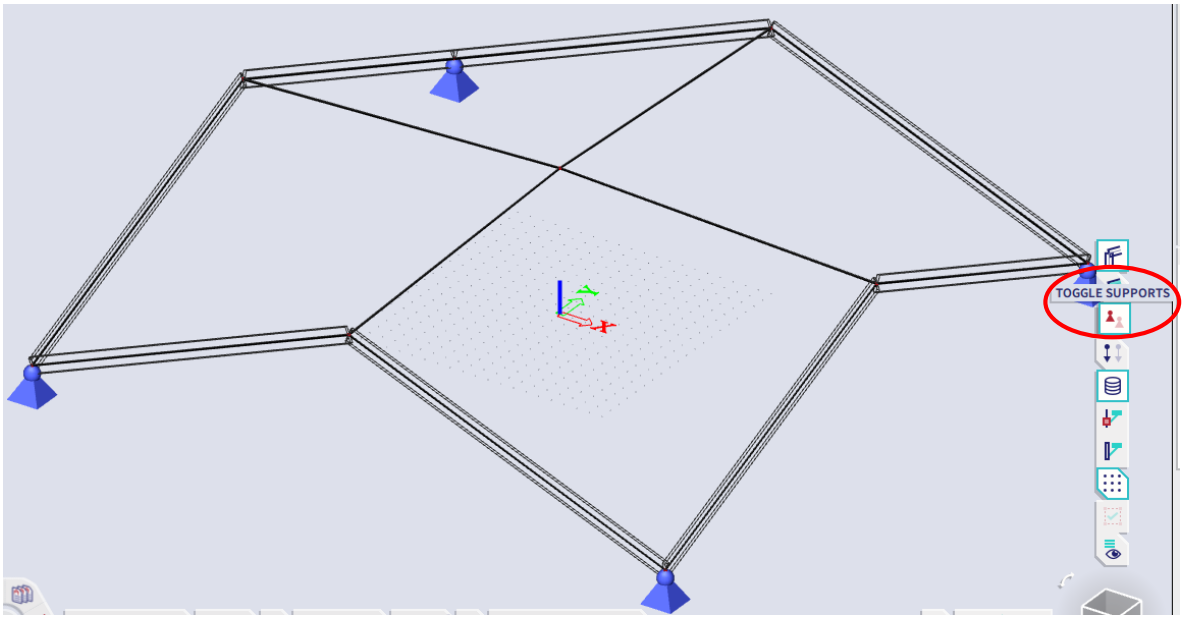


The shell is simply supported in four points.



Click in the drawing on a support point four times. We can see the supports by toggling them on and enlarging them.





Self-weight is 2400 kg/m^3 x 10 N/kg x 0.120 m = 2880 N/m^2 = 2.88 kN/m^2.
The safety factor for self-weight is 1.3.

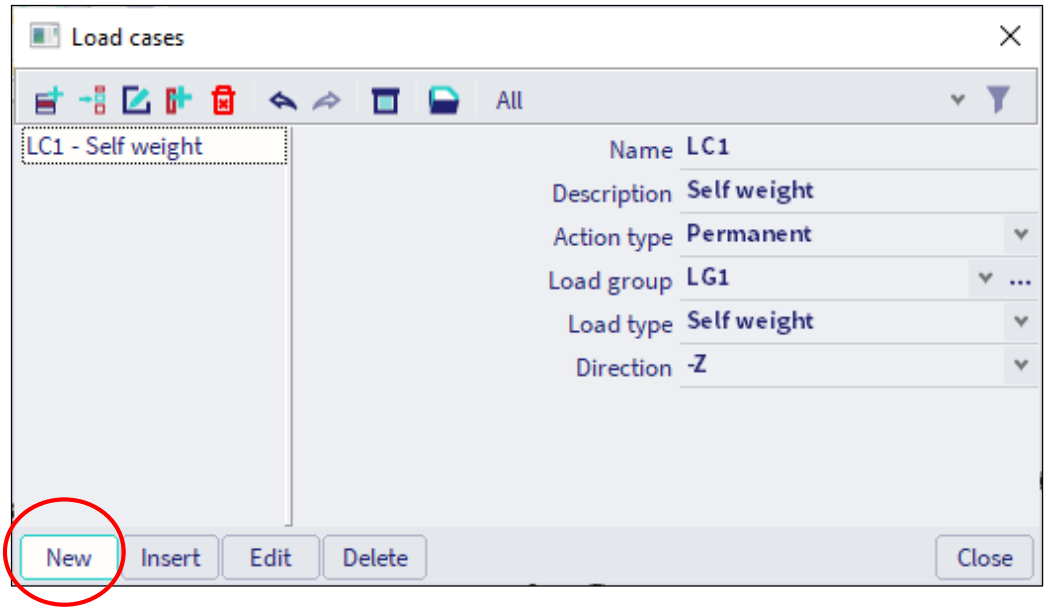
Snow load is 1 kN/m^2. The safety factor for snow is 1.5

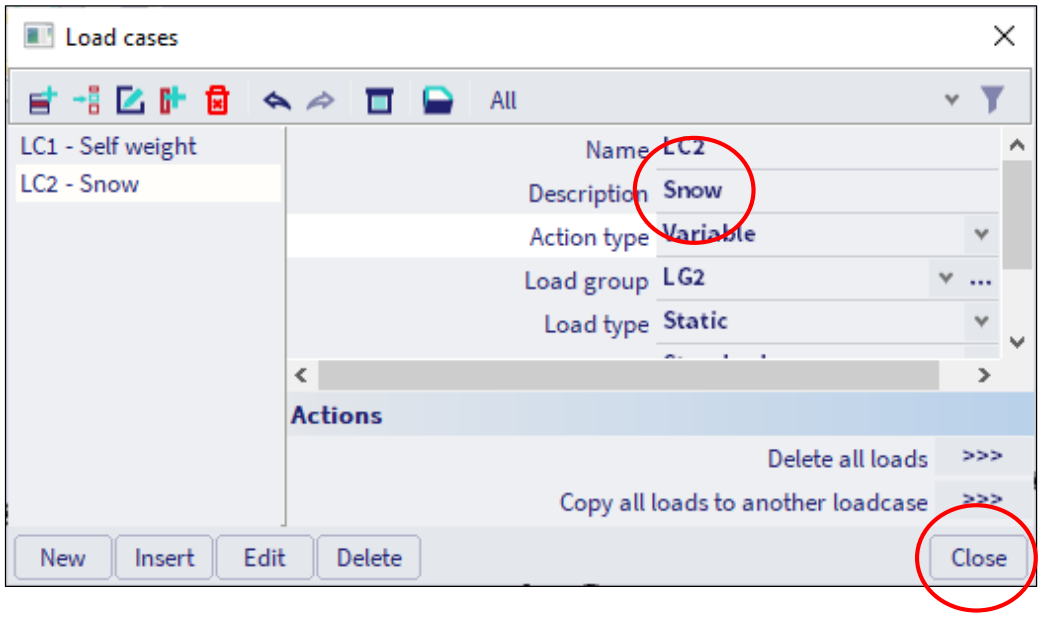
SLS load combination: 2.88 + 1.00 = 3.88 kN/m^2

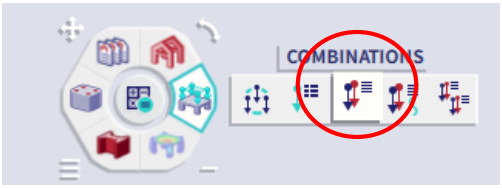
ULS load combination: 2.88 x 1.3 + 1.00 x 1.5 = 5.24 kN/m^2

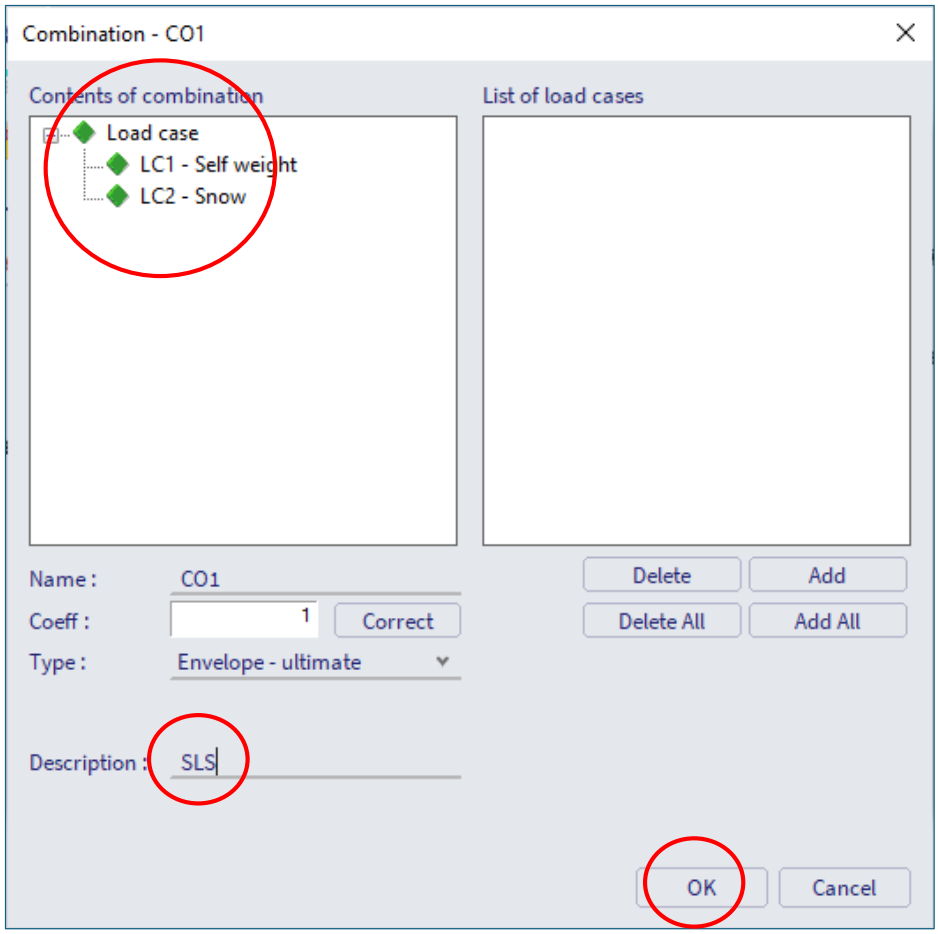
So, there are two load cases and two load combinations. Let's enter this.

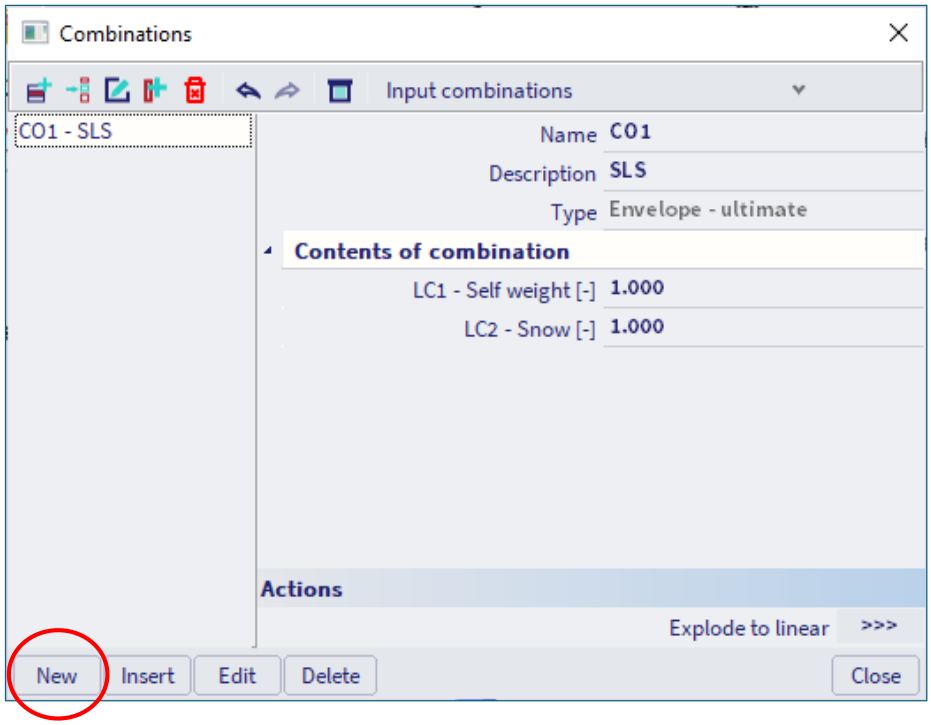


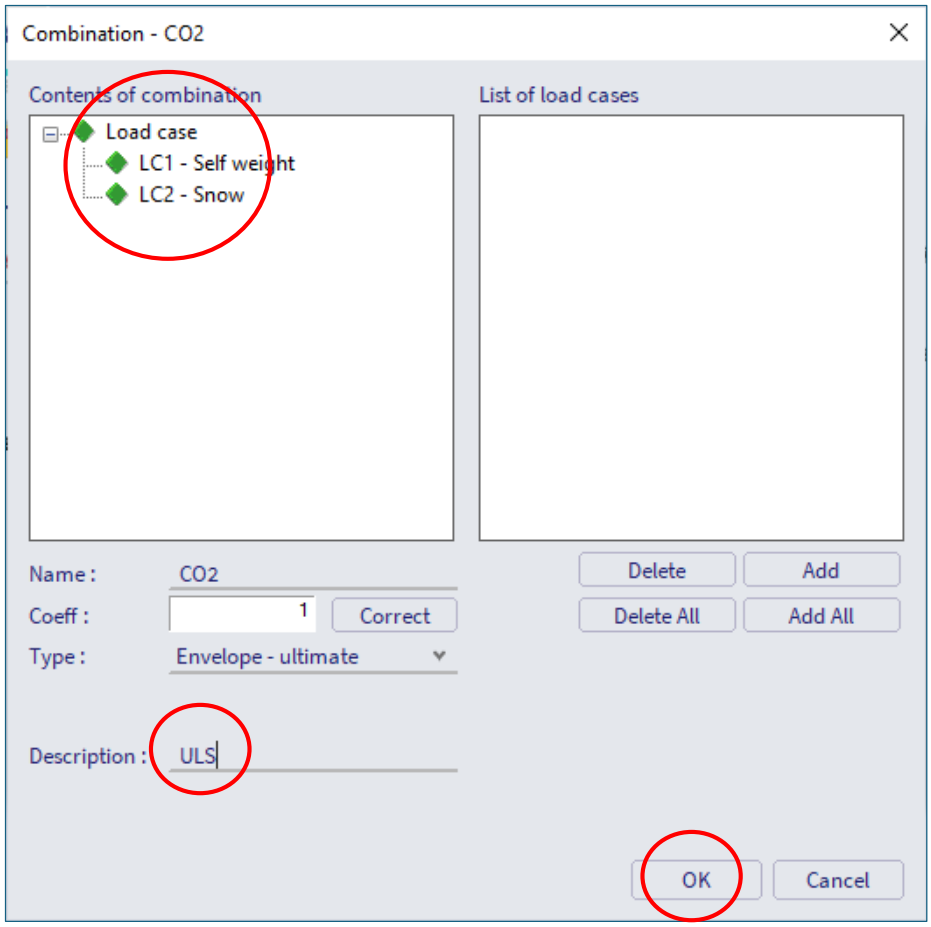

Load cases
All
LC1 - Self weight
Name LC1
Description Self weight
Action type Permanent
Load group LG1
Load type Self weight
Direction -Z
New
Insert
Edit
Delete
Close

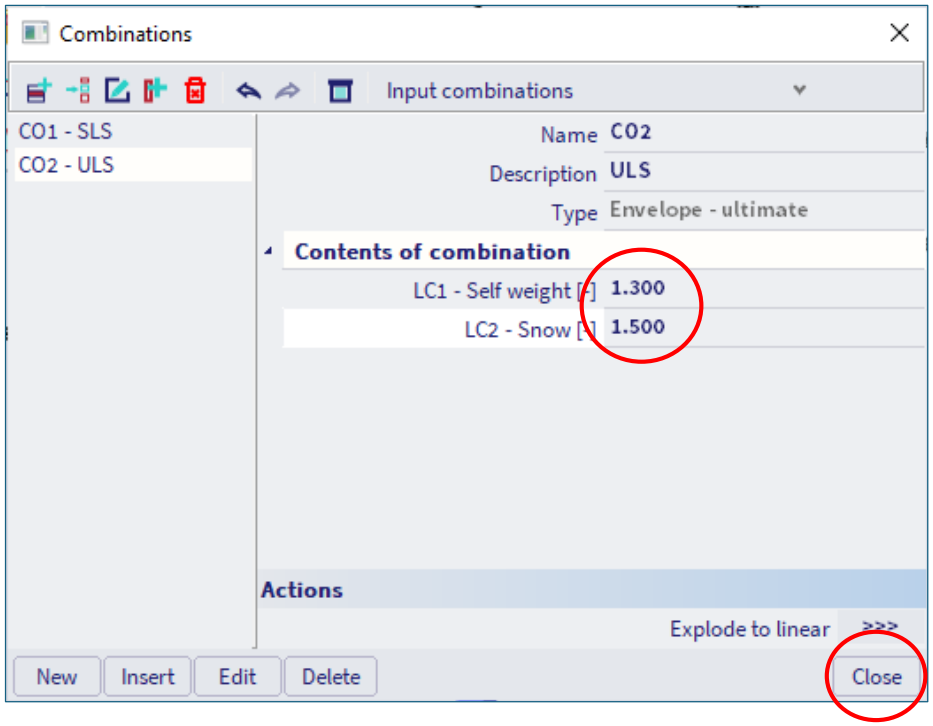

Load cases
All
LC1 - Self weight
LC2 - Snow
Name LC2
Description Snow
Action type Variable
Load group LG2
Load type Static
Actions
Delete all loads >>>
Copy all loads to another loadcase >>>
New
Insert
Edit
Delete
Close

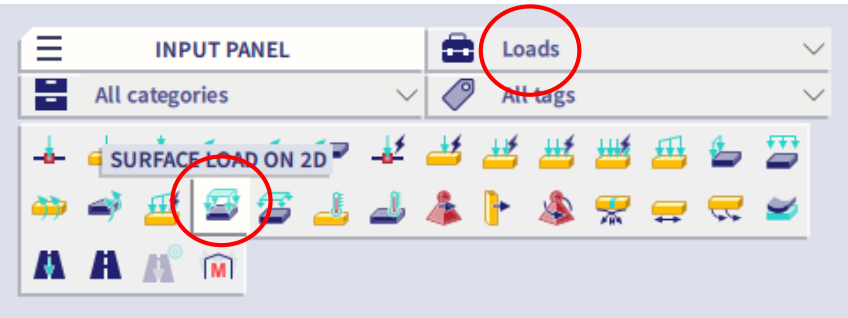

COMBINATIONS


Combination - CO1
Contents of combination
List of load cases
Load case
LC1 - Self weight
LC2 - Snow
Name :
CO1
Coeff :
1
Correct
Type :
Envelope - ultimate
Description :
SLS
Delete
Add
Delete All
Add All
OK
Cancel

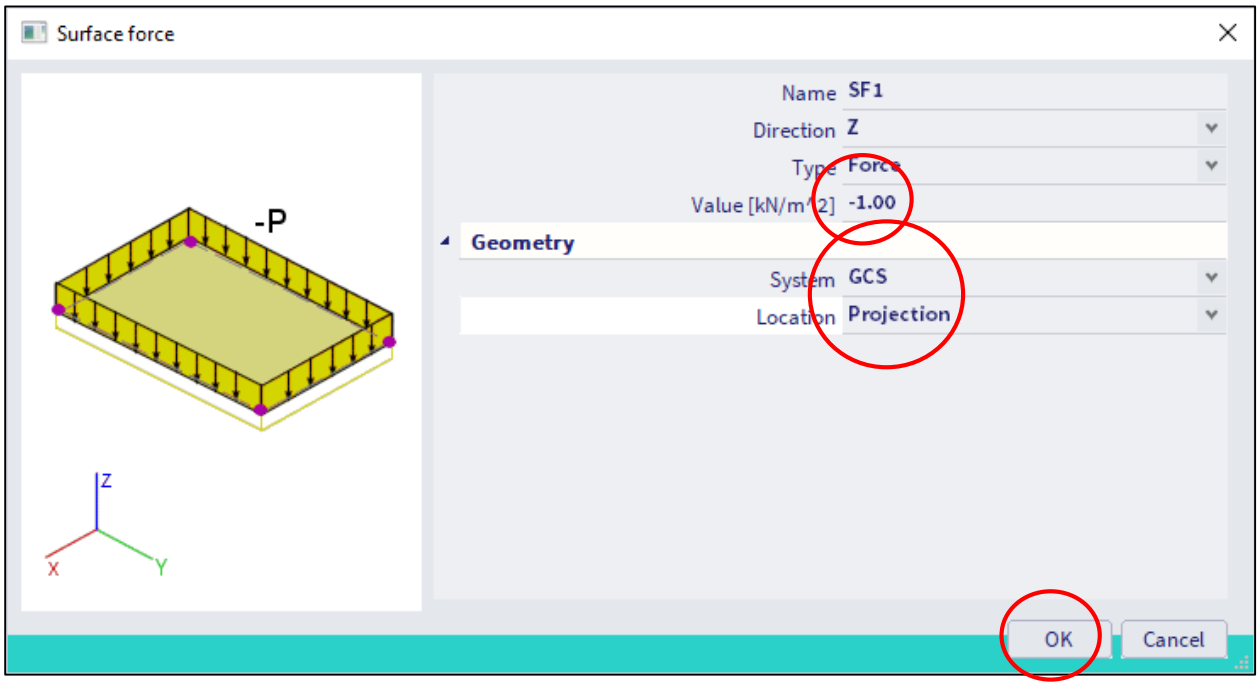

Combinations
Input combinations
CO1 - SLS
Name
CO1
Description
SLS
Type
Envelope - ultimate
Contents of combination
LC1 - Self weight [-]
1.000
LC2 - Snow [-]
1.000
Actions
Explode to linear
>>>
New
Insert
Edit
Delete
Close


Combination - CO2
Contents of combination
List of load cases
Load case
LC1 - Self weight
LC2 - Snow
Name :
CO2
Coeff :
1
Correct
Type :
Envelope - ultimate
Description :
ULS
Delete
Add
Delete All
Add All
OK
Cancel

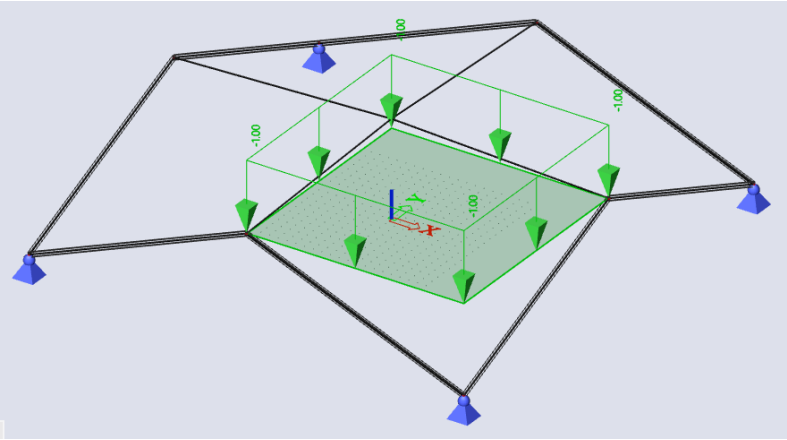

Combinations
Input combinations
CO1 - SLS
CO2 - ULS
Name
CO2
Description
ULS
Type
Envelope - ultimate
Contents of combination
LC1 - Self weight [-]
1.300
LC2 - Snow [-]
1.500
Actions
Explode to linear
>>>
New
Insert
Edit
Delete
Close



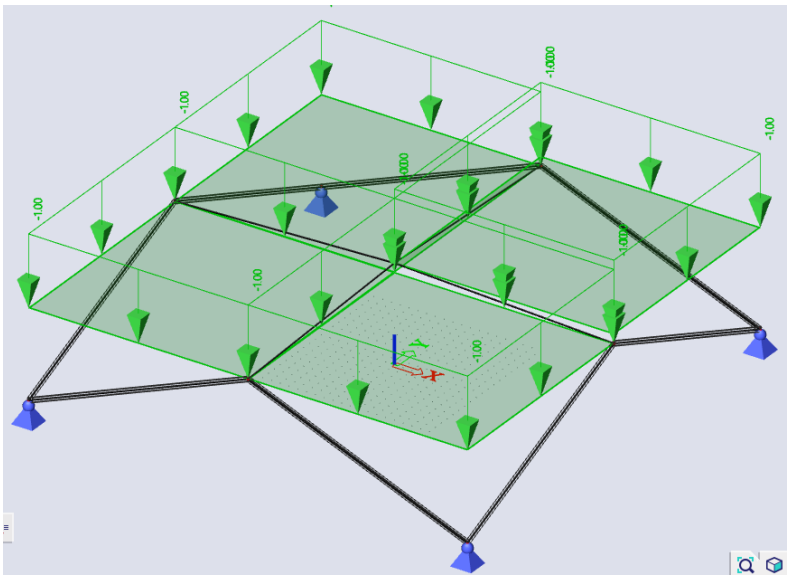
We do not need to enter self-weight. The program already knows. Let's enter snow load. In SCIA, de z axis of the global coordinate system (GCS) points up. Snow load goes down. The snow load is 1 kN per horizontal m^2. This is how meteorologists measured it.



Select a hypar edge. Right-click. Confirm action.



Repeat for the other three hypars.



We have entered the roof, the supports and the load. The model is ready.
Let's generate a mesh and perform a linear elastic finite element analysis.

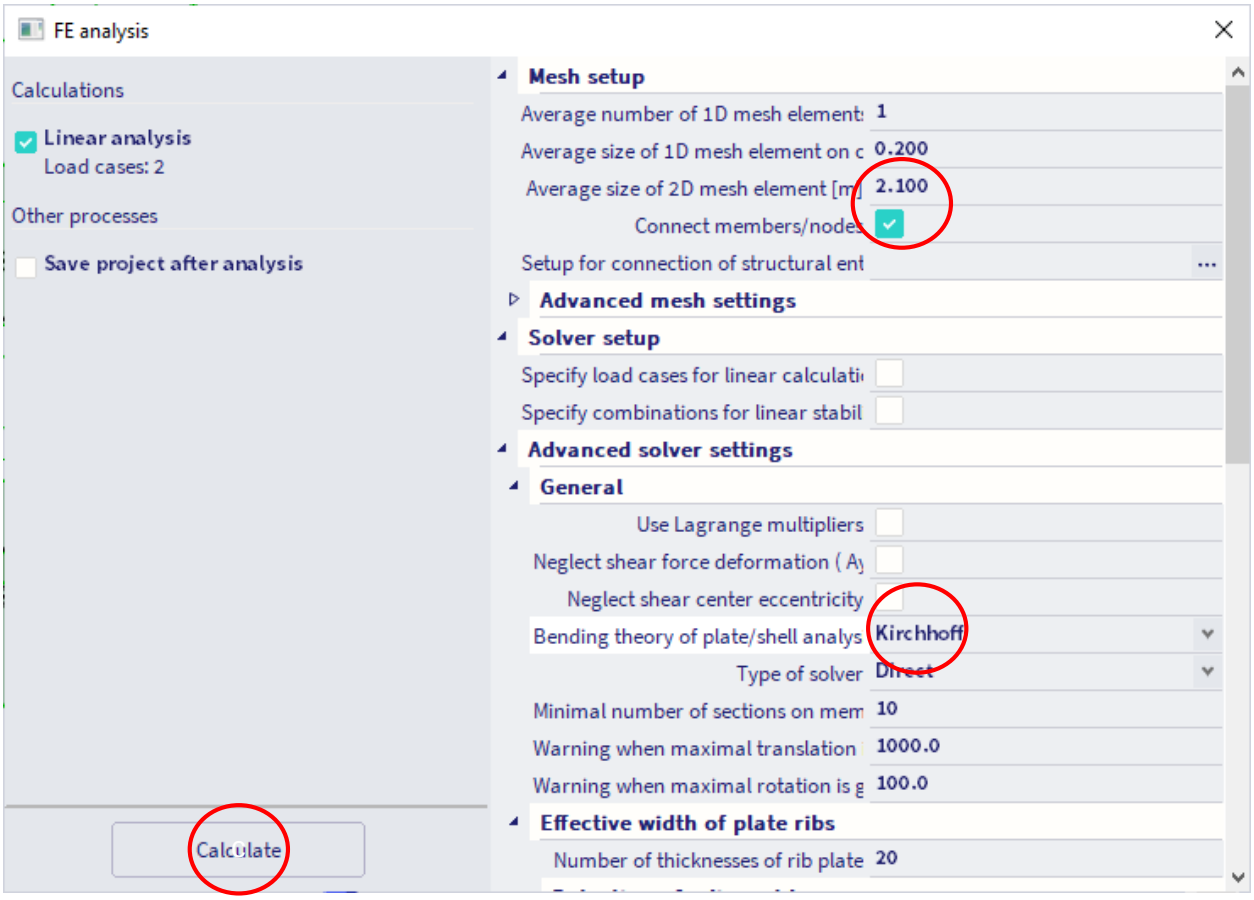
We need about 6 elements in the influence length.

$$a = \frac{bc}{h} = \frac{25 \times 25}{10} = 62.5 \text{ m} \quad \text{..... [book p. 102]}$$

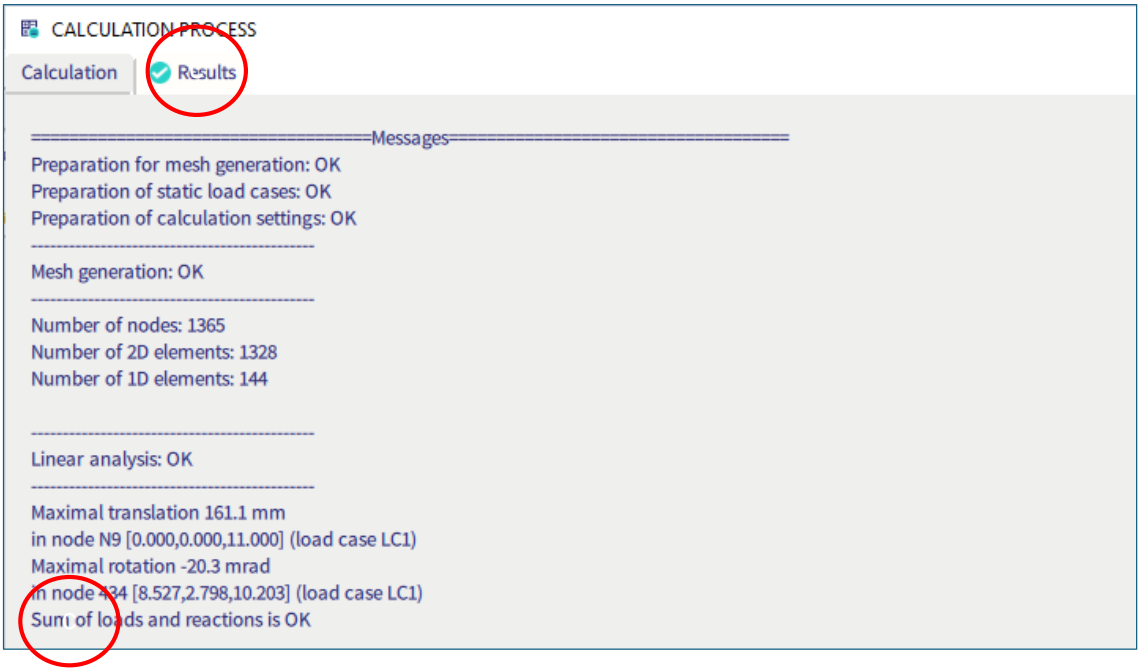
$$l_i = 2.25 \sqrt[3]{atl} = 2.25 \sqrt[3]{26.5 \times 0.120 \times 25} = 12.9 \text{ m} \quad \text{..... [book p. 75]}$$

$$h = \frac{l_i}{6} = \frac{12.9}{6} = 2.1 \text{ m} \quad \text{..... [book p. 75]}$$





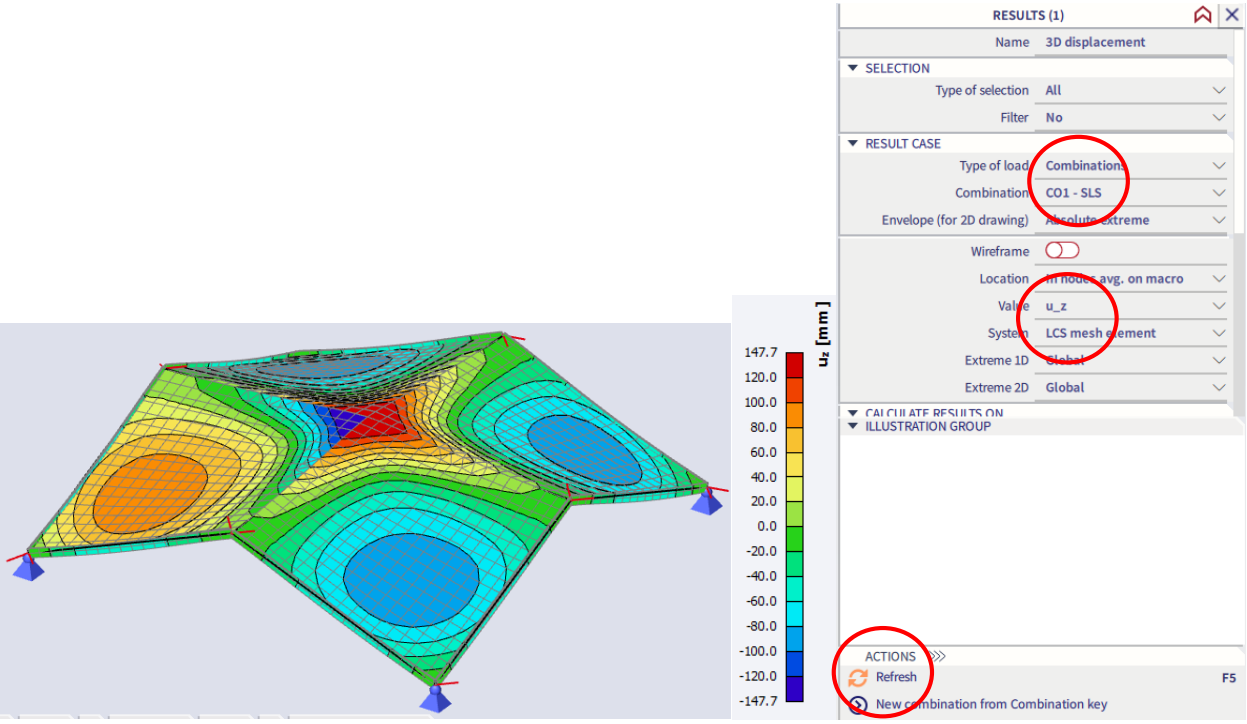
Note that the software has done an equilibrium check for us. The load and the support reactions are in equilibrium, consequently, the equations have been solved accurately.



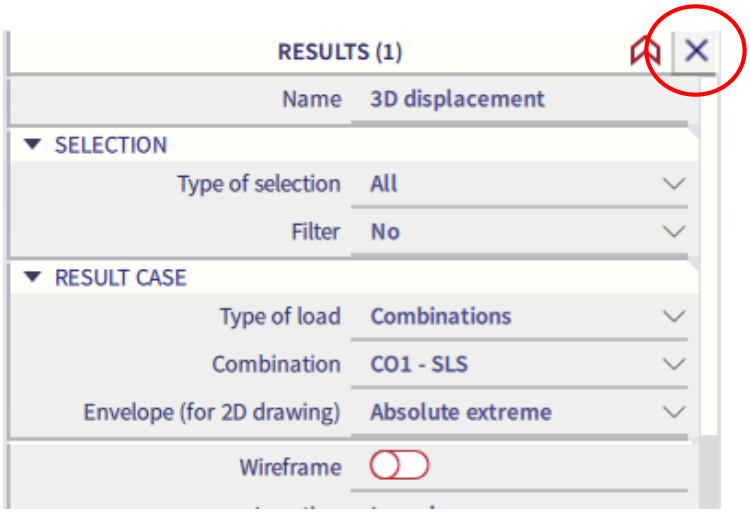
Let’s look at the deformation.



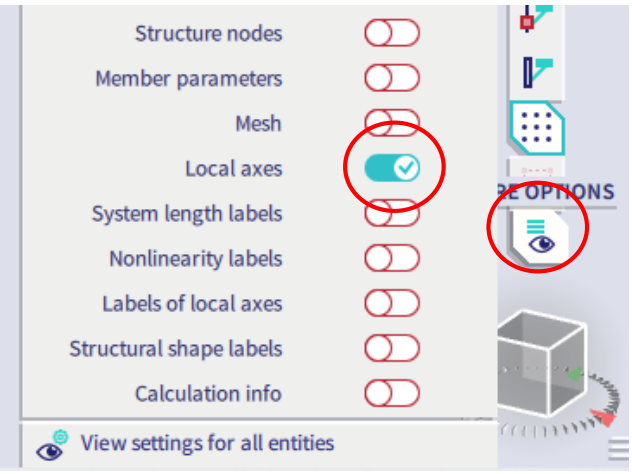
We plot u_z, which is the displacement perpendicular to the surface. We see that one of the hypars is up-side-down.



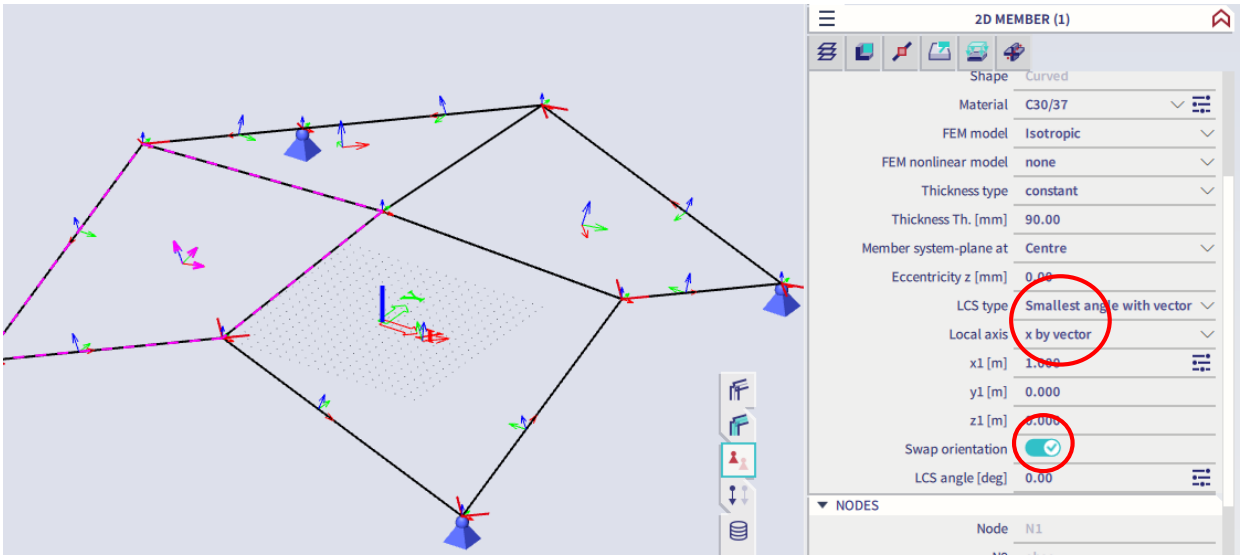
Let’s fix this. We leave post-processing and go back to pre-processing.



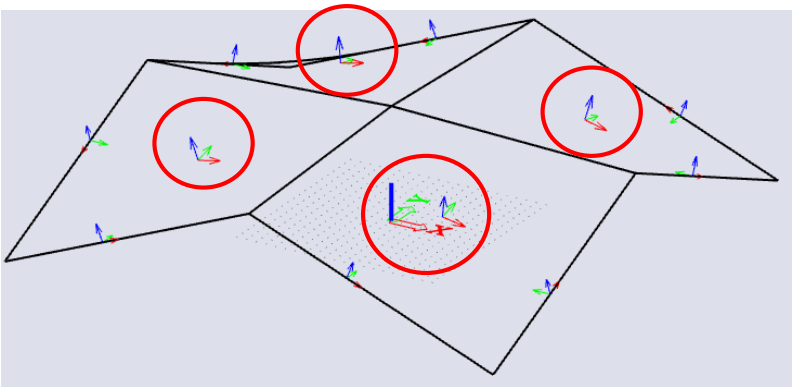
We show the local coordinate systems of the hypars.



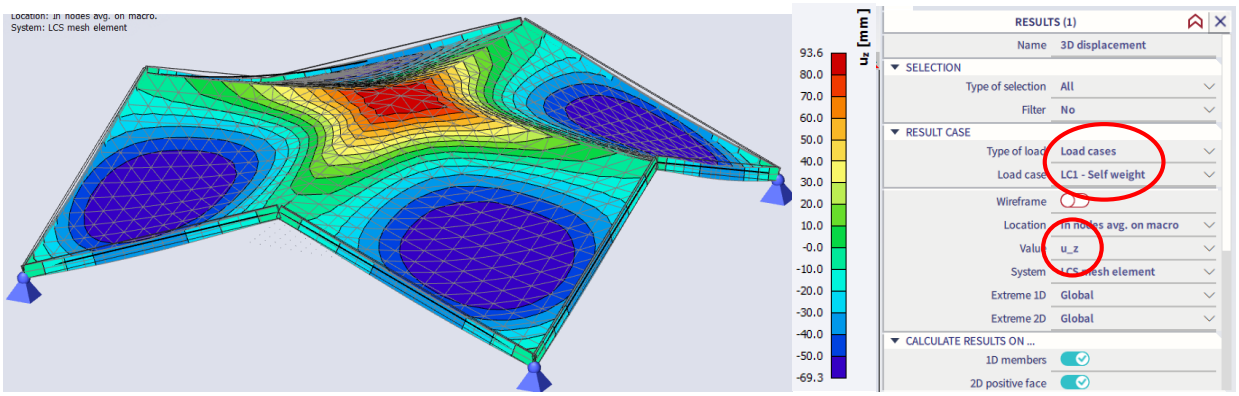
We select a hypar by clicking on its edge.
We align the local x axis with the global x axis.



Repeat for the other three hypars.



We analyse the model again. Plot u_z for just self-weight.

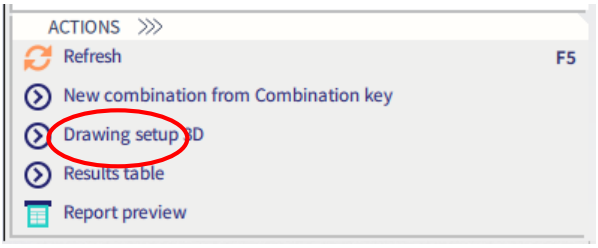


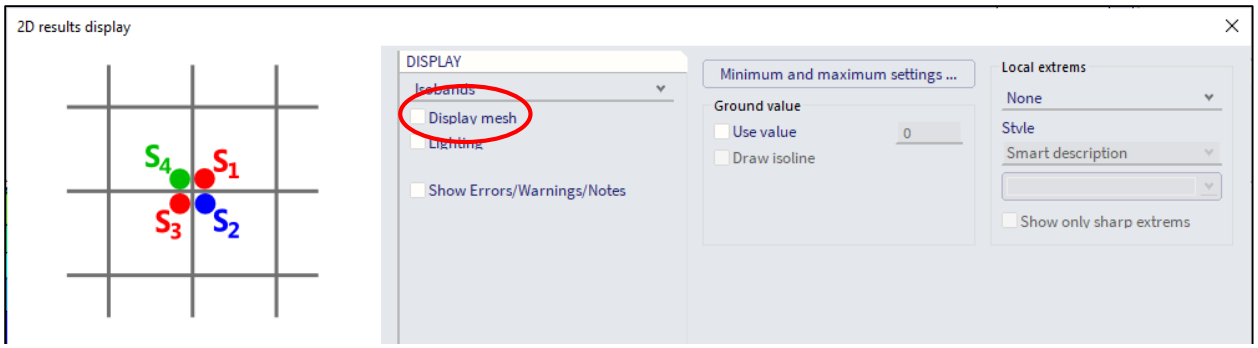
First, we check *ourselves*. For this rotate the model and zoom in.

- Are the hypars and edge beams connected? …… no gaps, no overlaps …….. OK
- Are the shapes of the elements okay? …… triangles ……..... OK
- Do the supports translate? …… no …….. OK
- Is the deflection symmetrical? …….. yes, to the shape and the load ……. OK
- Is the deflection in the direction of the load? ……. yes, but the top moves up …… OK

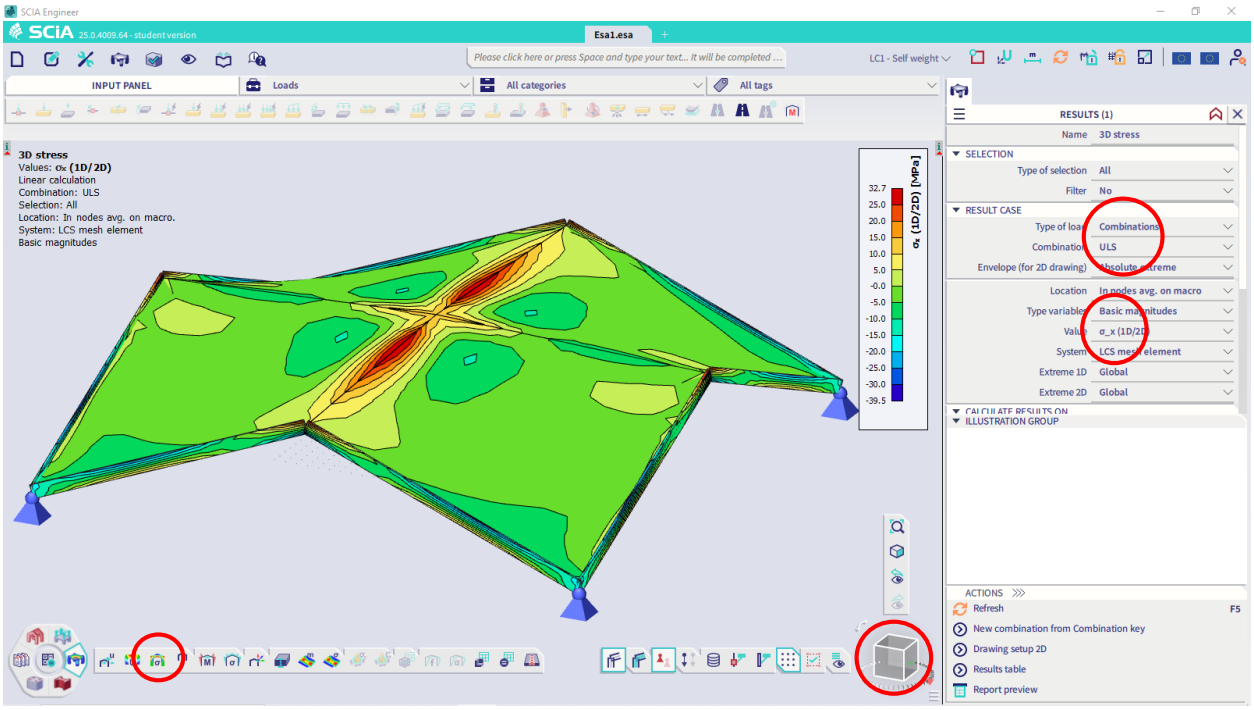
Second, we check the *design*. The top displacement is 94 mm up. There is no practical reason why it is too large. The edge beams deflect 35 mm. The edge beam should not touch the glass façade. Due to cracking the deflection will triple. Due to creep it will double. So, in ten years, we can expect a deflection of 35 x 3 x 2 = 210 mm. As a rule of thumb, deflection should be less than span / 250 = 50000 / 250 = 200 mm, so about okay. The edge beam deflection due to snow is 6 x 3 = 18 mm. We better design a 250 mm gap between the edge beams and the façade.

We switch of displaying the finite element mesh.





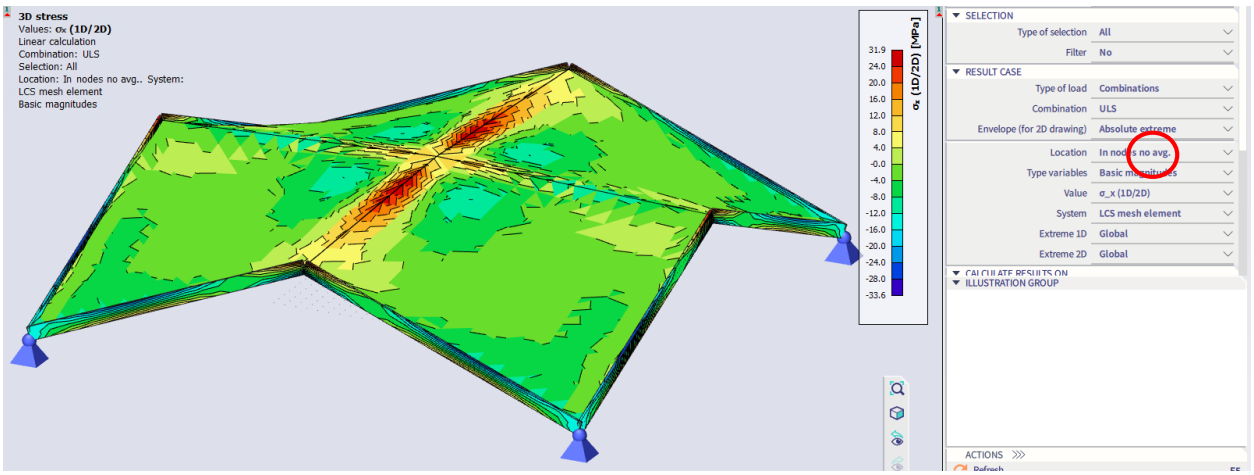
Linear elastic stresses. We can turn the shell over to see the stresses in the other side.



First, we check *ourselves*.
- Are the contour lines smooth? (no jumps) ……..…….. yes ……OK
- Compression in the hypar top and tension in the bottom? ……. yes …… OK
- Is this ultimate limit state (ULS) loading? …… yes …… OK
- Are the stresses in the local coordinate system (LCS) …….. yes …… OK

Second, we check the *design*. The stresses are large compared to the 30 N/mm^2 compressive strength of the concrete. However, these are linear elastic stresses and proper dimensioning of cross-sections will provide sufficient moment capacity.

This is how it looks, if we do not average in nodes. Not necessary now.

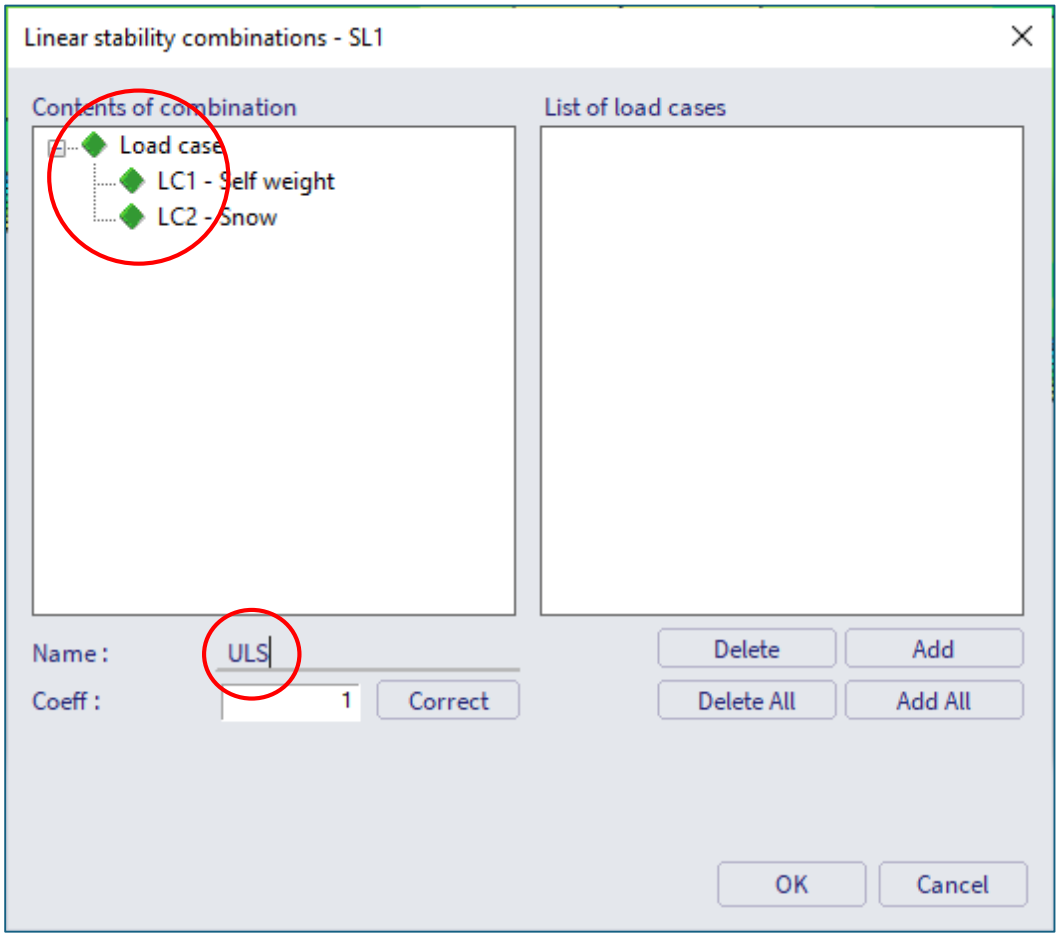


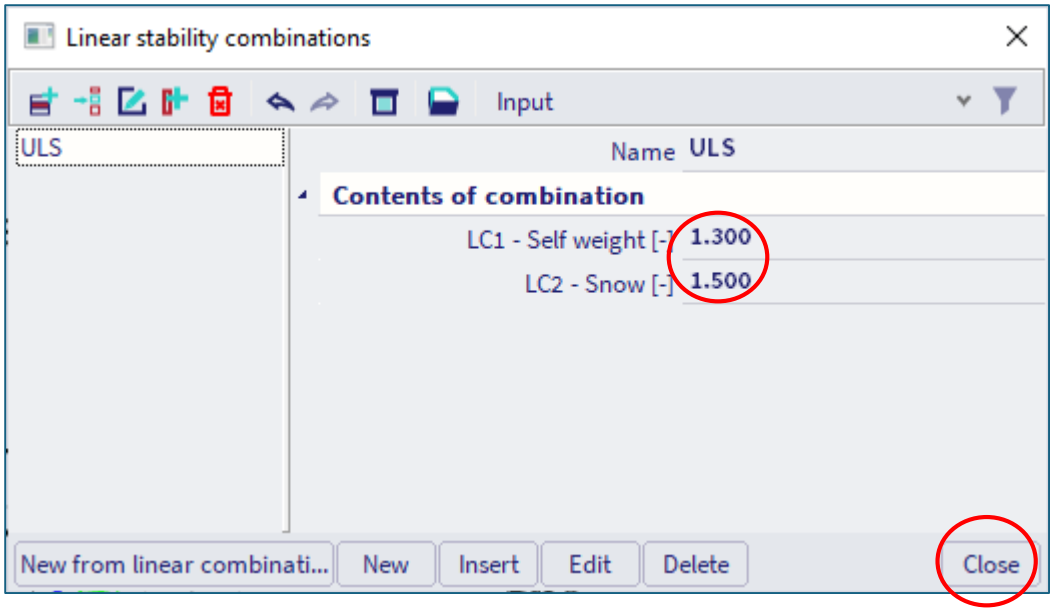
We can change the finite element size in Mesh Setup. Half the element size, perform the analysis again and if the important results do not change much than the element size is sufficiently small.

Buckling check

For stability, we need to enter a new load combination. Normal load combinations are okay for stability too.

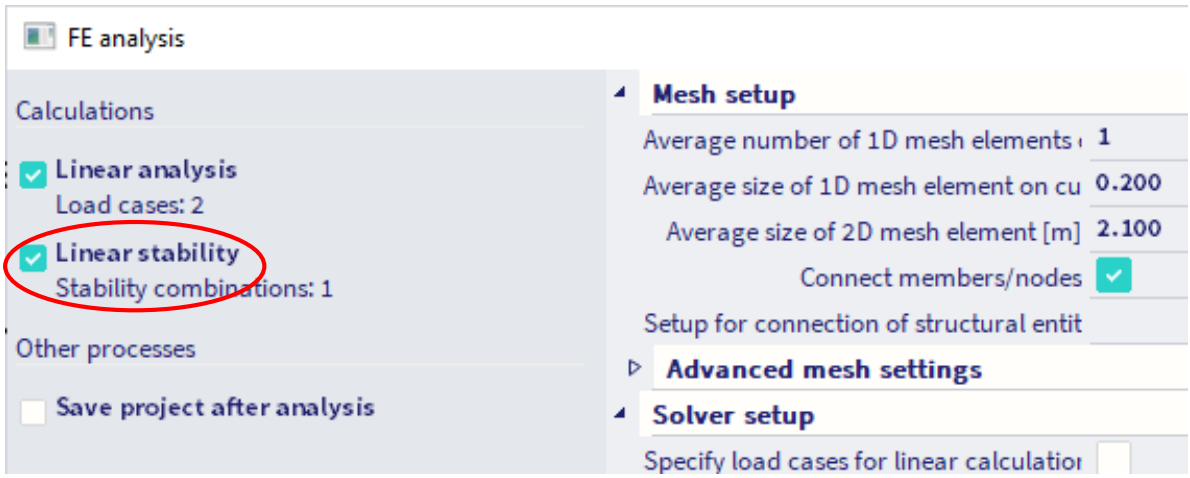






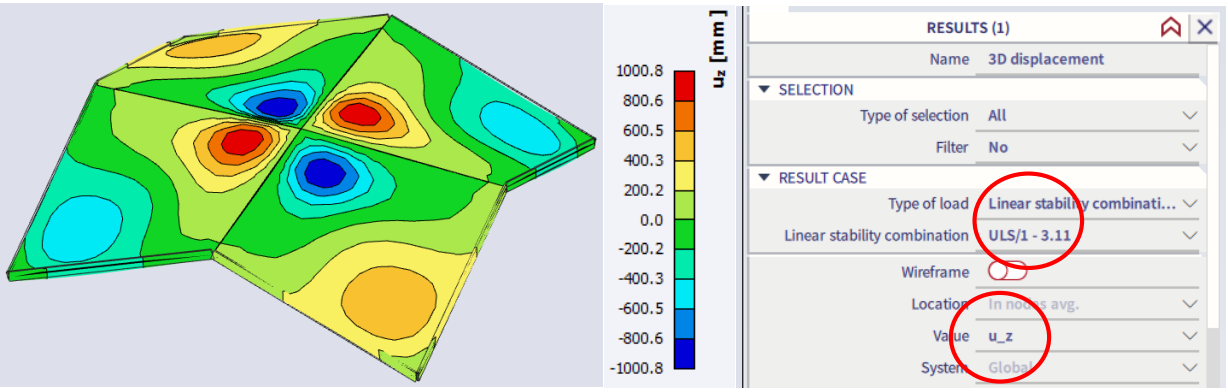
We perform the analyses.





Question: The program does first the linear analyses and then the linear stability analysis. Why?





The first buckling mode has a buckling load factor of 3.11. This means that the shell buckles at a load of 3.11 times its ULS load. However, concrete cracking is not included. Cracking can reduce the buckling load factor to 3.11 / 3 = 1.04. Shape imperfections are not included either but hypars are not very sensitive to imperfections. The design seems okay. The final answer comes from a non-linear analysis (physically and geometrically non-linear analysis with imperfections). This will be practiced in the next practical lecture.

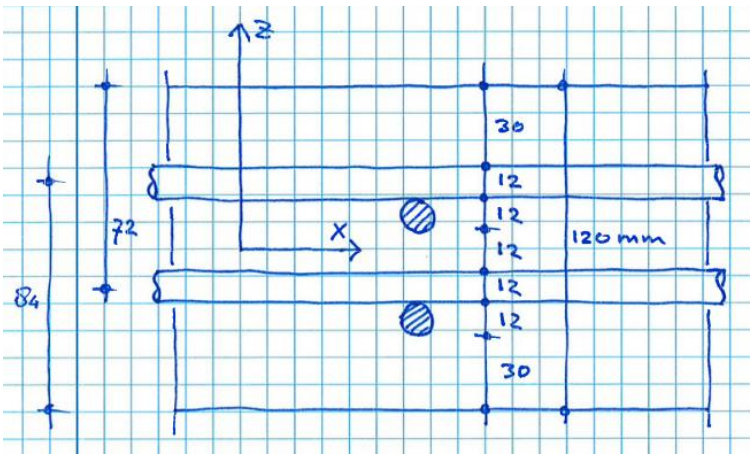
Question: The largest buckling displacement is always 1 m. Why was it programmed like this?

Question: The stresses after buckling cannot be displayed. Why not?

Have a look at the second buckling mode: ULS/2 – 3.93. The number of buckling modes that are computed can be selected in *Advanced solver settings* just before we do the analyses.

Reinforcement design

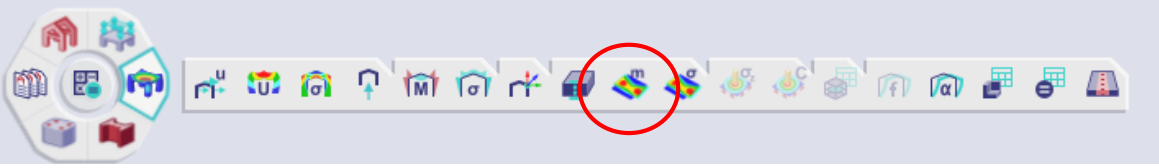
We use 12 mm bars with a spacing of 200 mm. The bars have 30 mm cover. The effective depth is 84 mm or 72 mm.



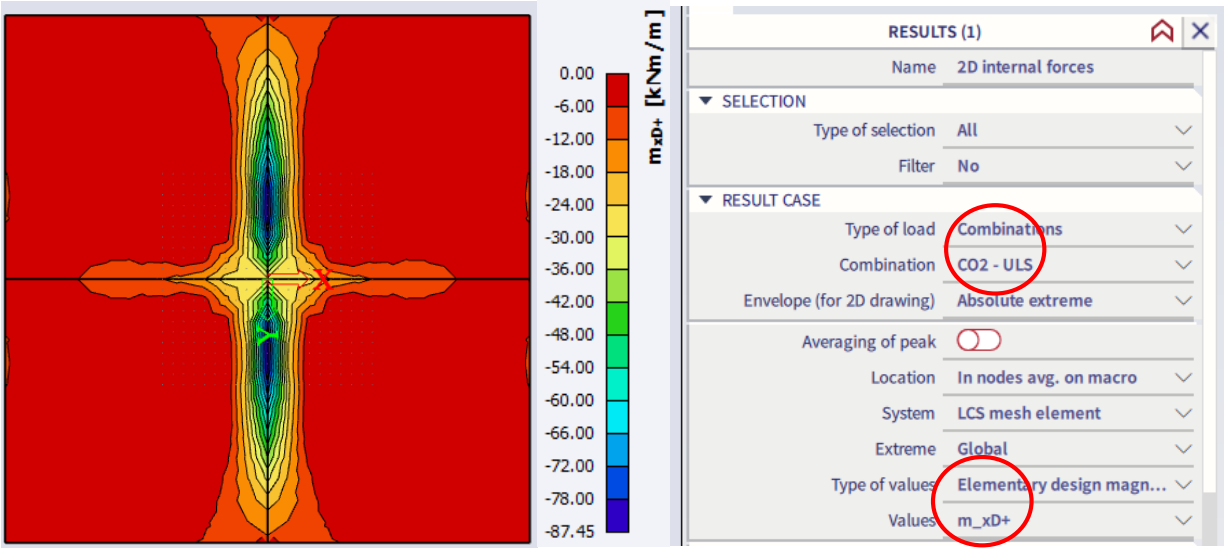
The yield force of a bar is $F = ¼\ \pi\ (12\ mm)^2 \times 500\ N/mm^2 / 1.15 = 49171\ N$
The concrete stress is $\sigma = F / (0.2 \times 84\ mm \times 200\ mm) = 14.6\ N/mm^2 < 30/1.3\ N/mm^2$ … OK
The moment capacity is $m_{xx} = 0.9 \times 84\ mm\ F / (200\ mm) = 18580\ N = 18.6\ kNm/m$

top bars in *x* direction …….…….. $m_{xx}+ = 18.6$ kNm/m
bottom bars in *x* direction ..…….... $m_{xx}– = -15.9$ kNm/m
top bars in *y* direction ……,,…. $m_{yy}+ = 15.9$ kNm/m
bottom bars in *y* direction …… $m_{yy}– = -18.6$ kNm/m

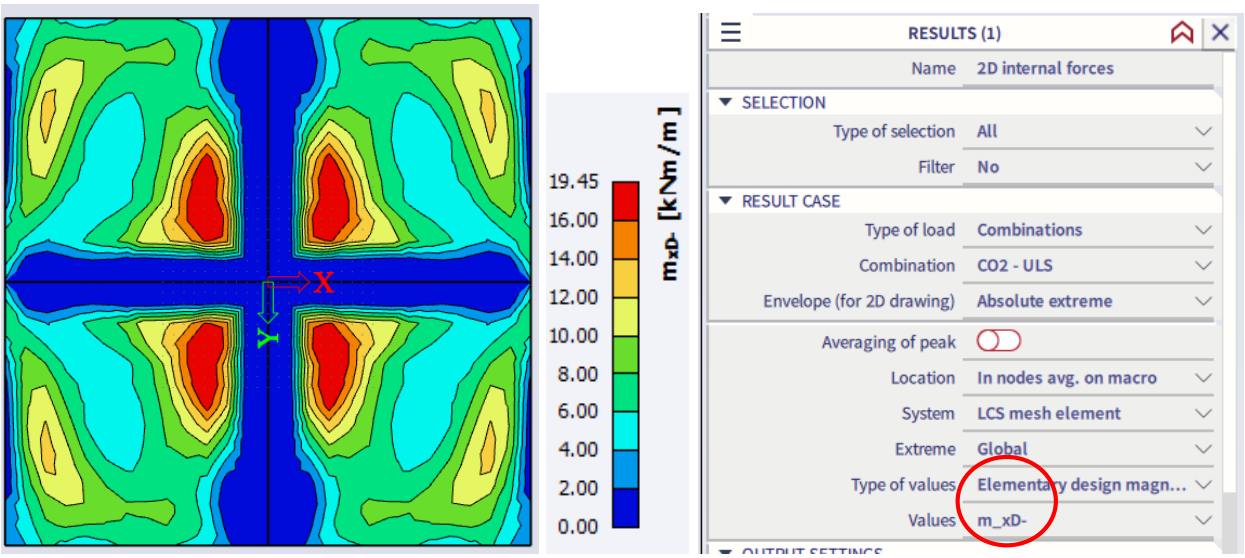
Let's display the required reinforcement moments.



Note that SCIA has different positive moment directions than the course.



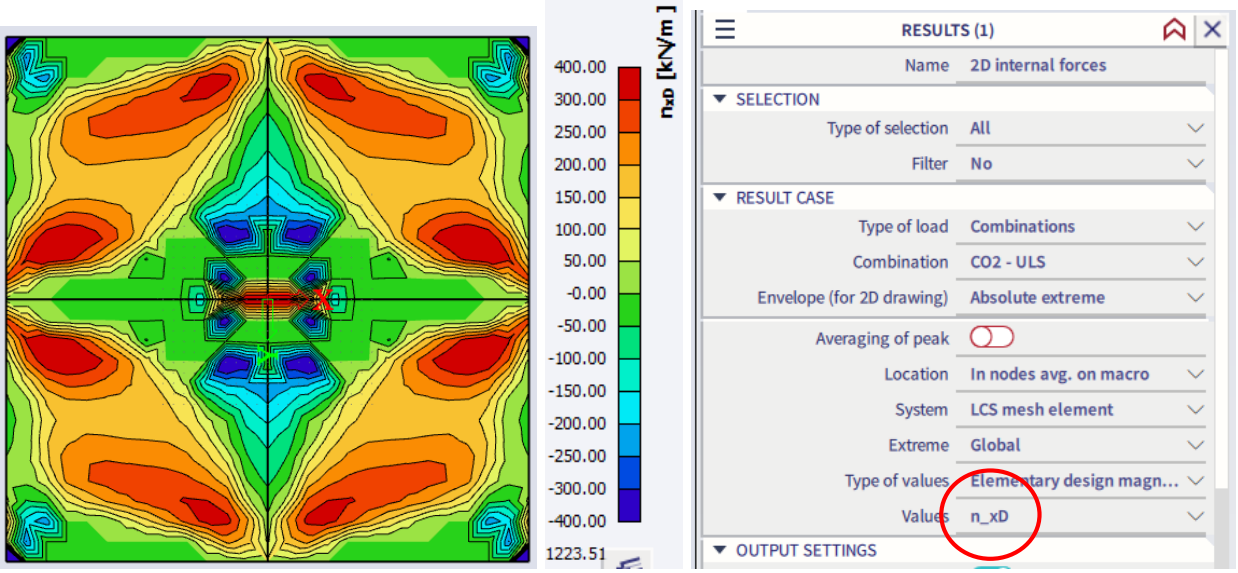
m_xD+ = m_{xx} + | m_{xy} |. The 18.6 kNm/m is sufficient, except for in the top (edge disturbance). In the top we need 4 extra bars every 200 mm. 5 x 18.6 = 93.0 kNm/m. This is a lot but we can make it fit.



m_xD– = -mxx - | mxy |. The 15.9 kNm/m is sufficient, except for in the red areas. In the red areas we put an extra bar every 400 mm. 1.5 x 15.9 = 23.9 kNm/m.

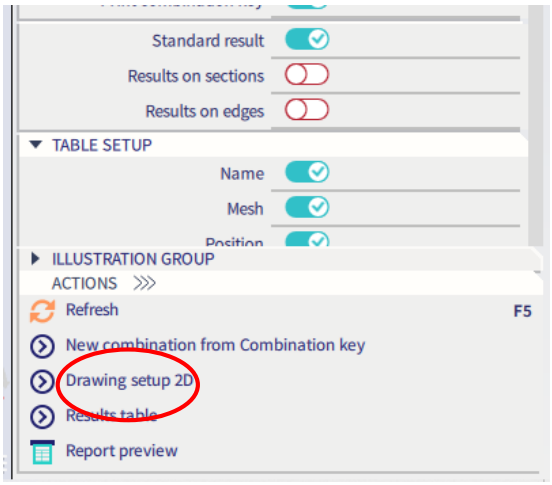
The bars not only need to carry the moments; they also need to carry the membrane forces.

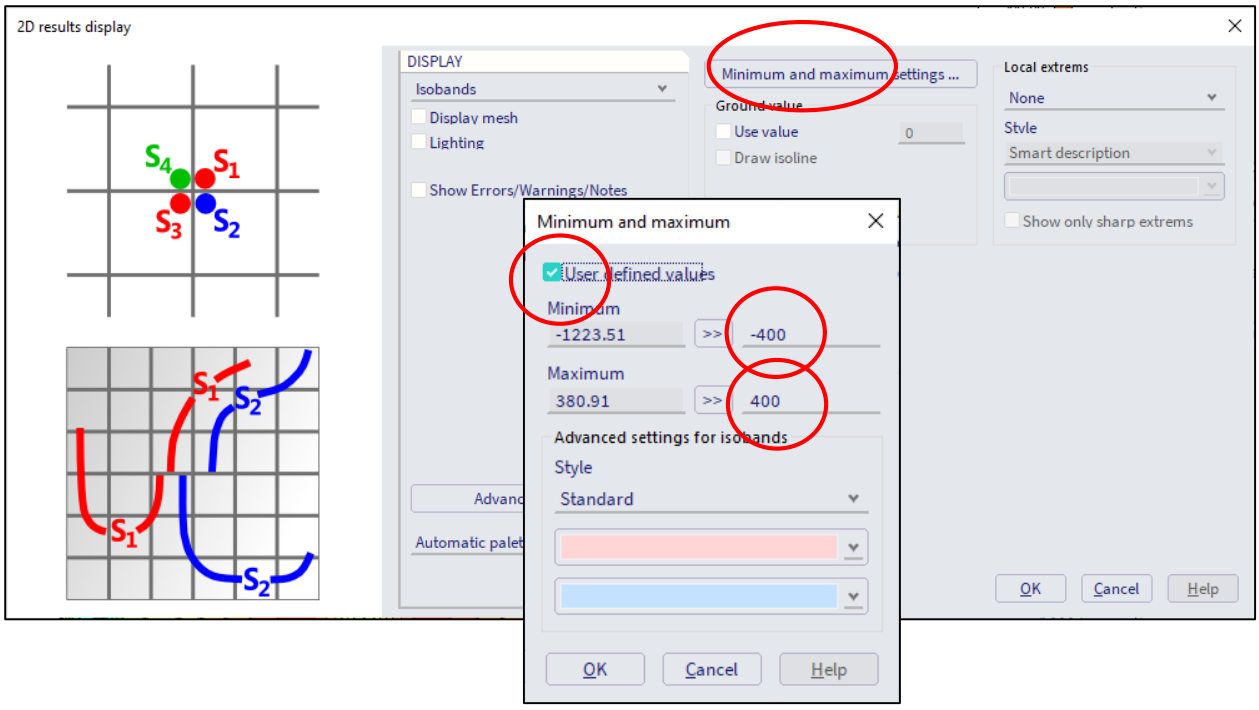
The membrane force that the bars can carry is n_{xx} = 2 F / (200 mm) = 492 N/mm = 492 kN/m.



n_xD = n_{xx} + | n_{xy} |. The 492 kN/m is sufficient.

To obtain this nice plot we need to adjust the colour bar.





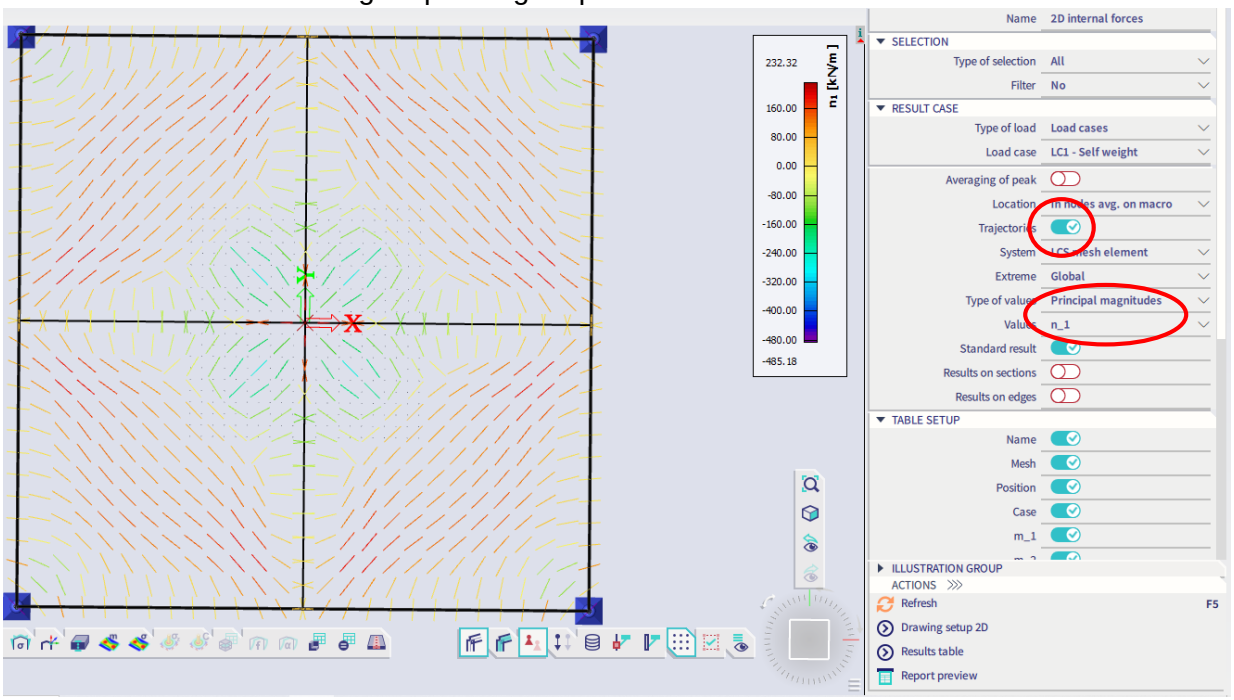
Question: At the supports the membrane forces are very large. What is going on there?

The large membrane forces occurs in different parts of the shell than the large moments. This is fortunate because the bars need to carry both.

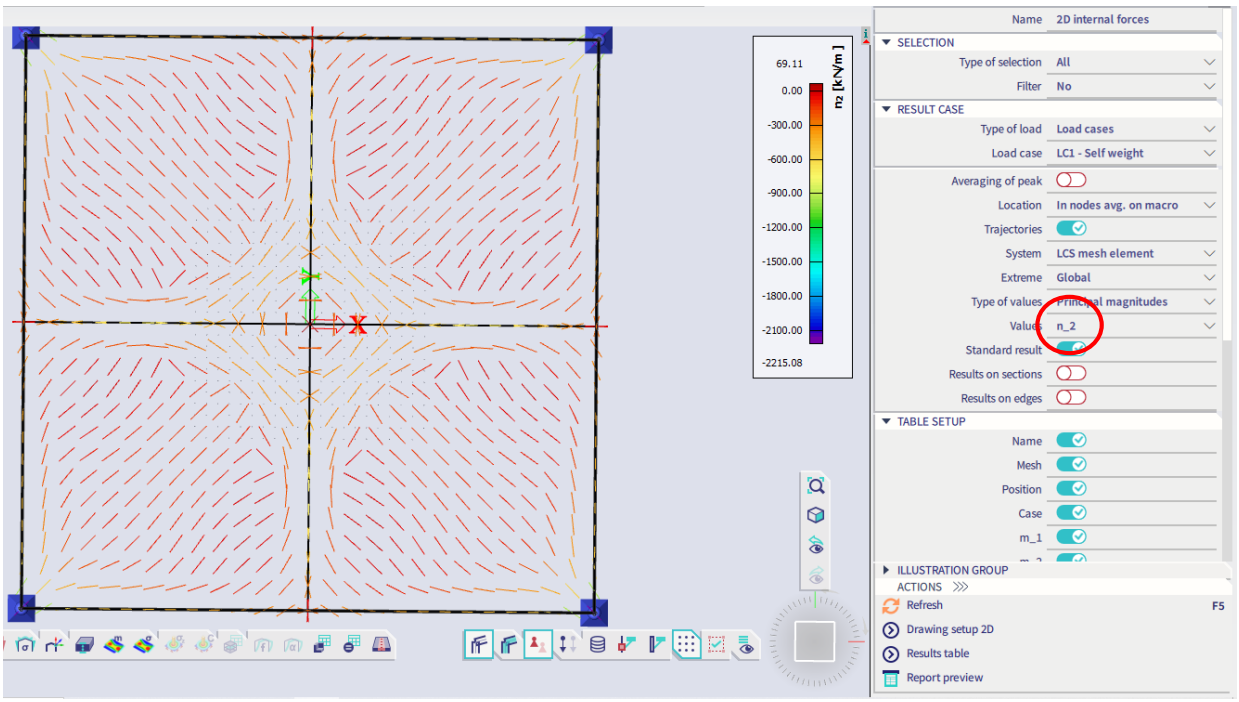
We also want to enter the bars in SCIA and let SCIA do unity checks on strength and crack widths. Unfortunately, this is not possible for shells as yet. Surely, in the future this will be implemented.

Appendix 1. Principal normal forces

Let’s look at the force flow. These are the directions of the first principal membrane force. We see a tension band from edge top to edge top. We can call this cable action.



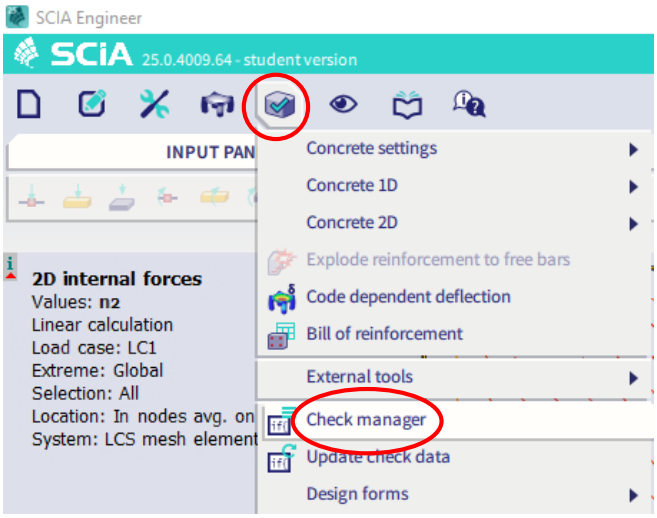
We see compression from shell top to support. We can call this arch action. Around the top there is a compression system that unloads the top. Perhaps we can fit in an oculus.

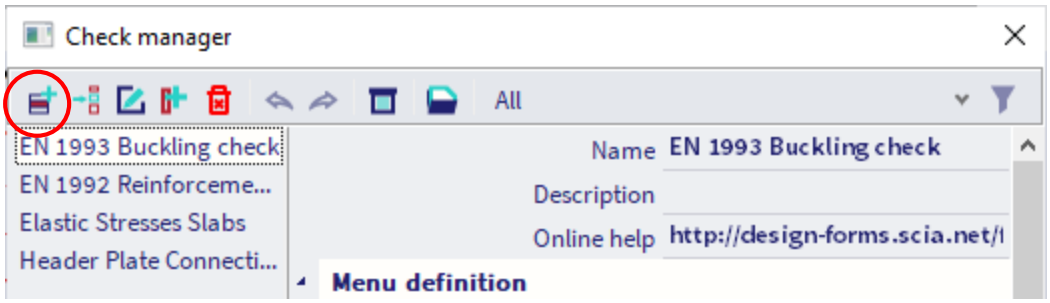


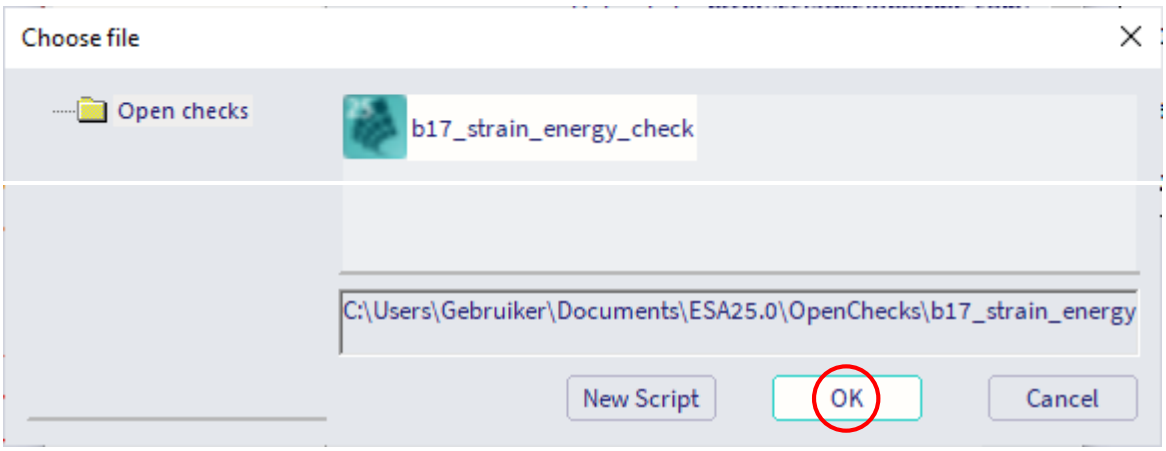
Appendix 2. Membrane forces or bending

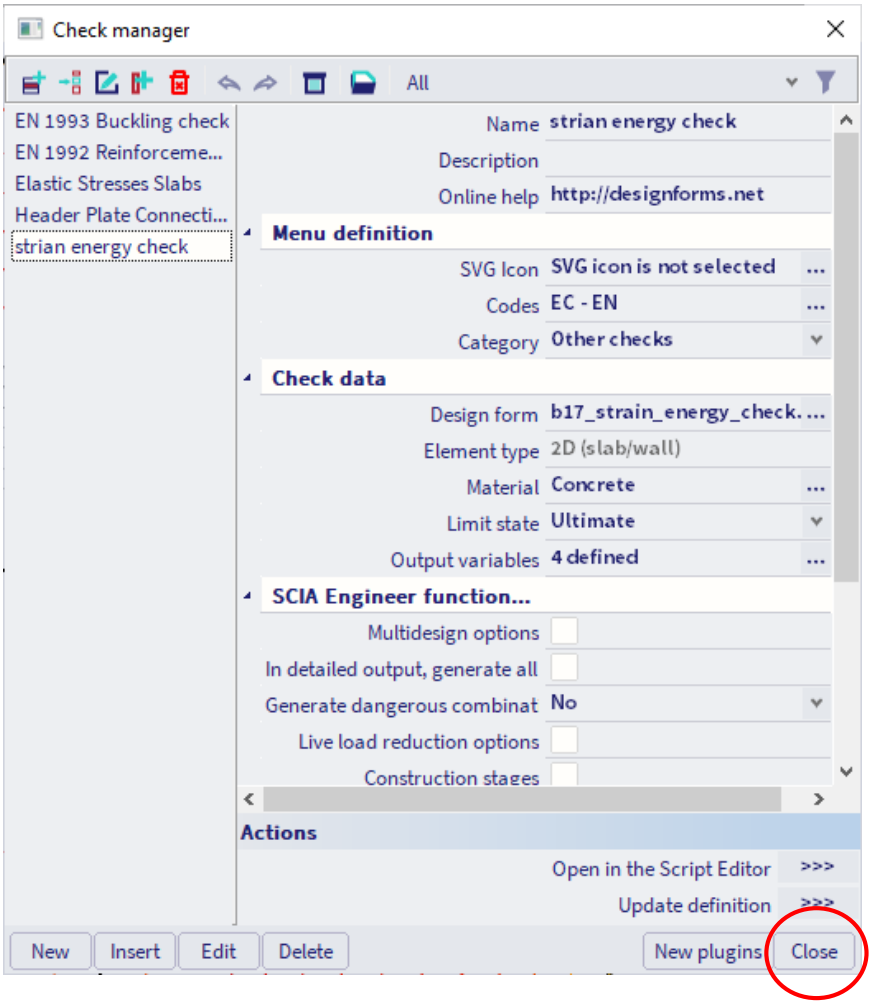
You can write your own programs in SCIA with a tool called “design forms builder”. You can use not only computation results but also model data such as node coordinates. This program (*.CLS) is first compiled (*.CLC) and then loaded into SCIA.

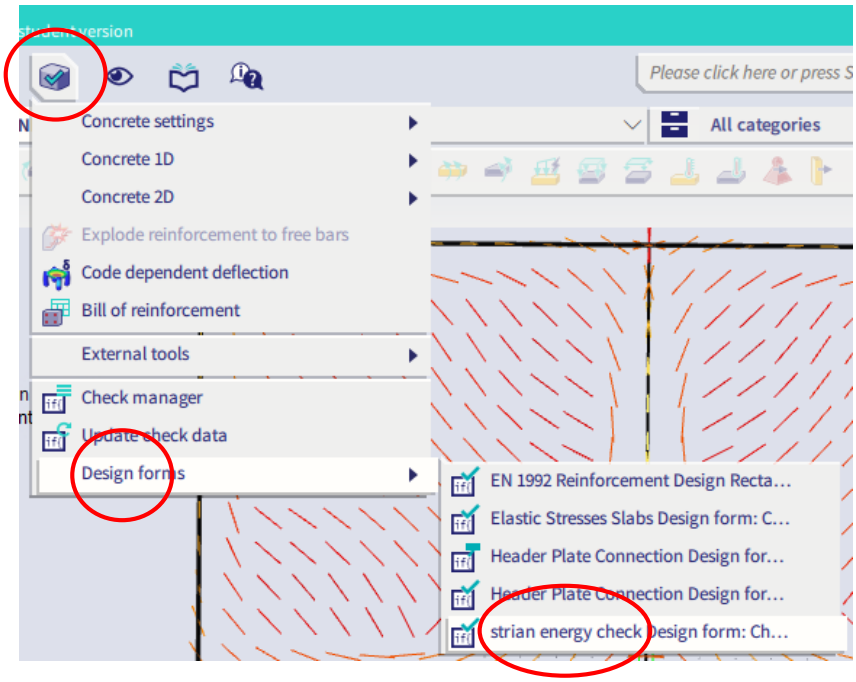
Marieke Vergeer has made a small program in her BEP. The program displays the strain energy in shells. Let’s see if this works. The name of the program is “b17_strain_energy_check.clc”. It can be downloaded from (23 KB) http://phoogenboom.nl/b17_strain_energy_check.zip
The *.clc program needs to be copied to the folder C:\Users\Documents\ESA25\OpenChecks\
We need to load the program in SCIA.



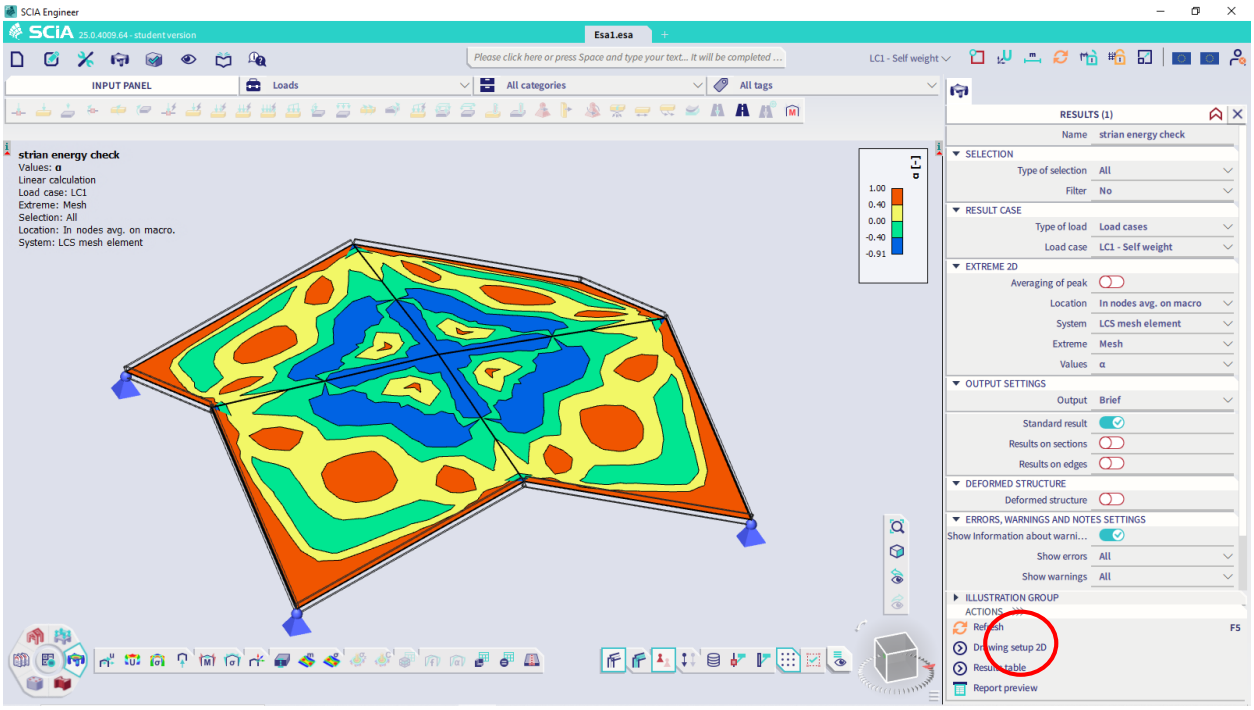


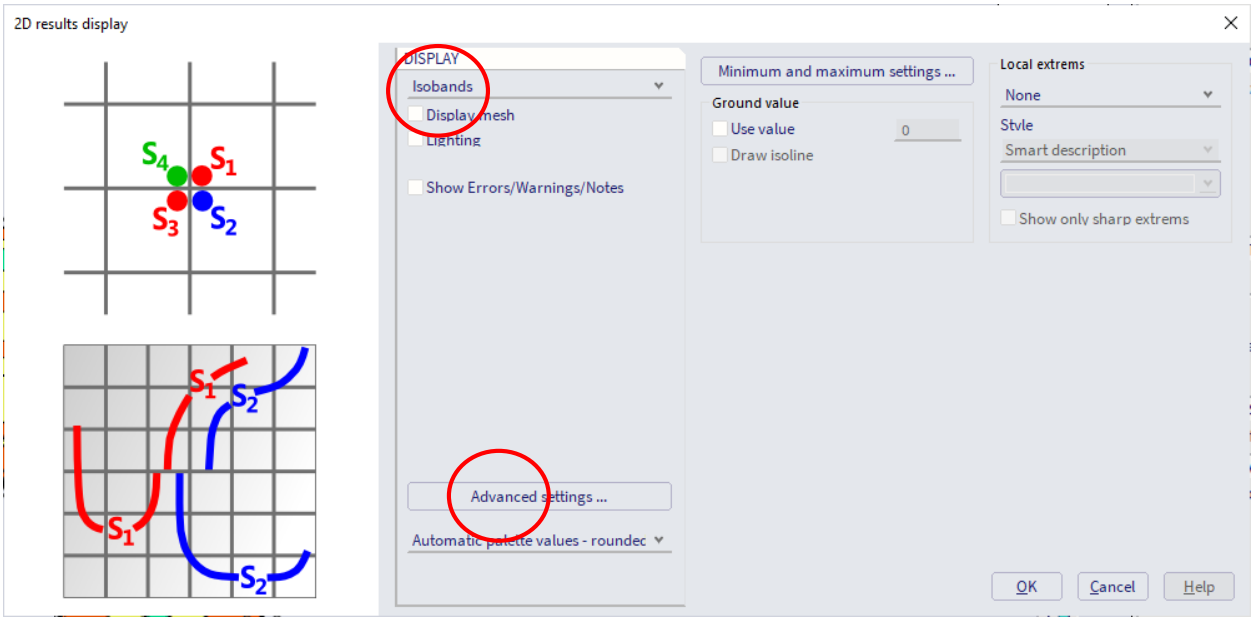


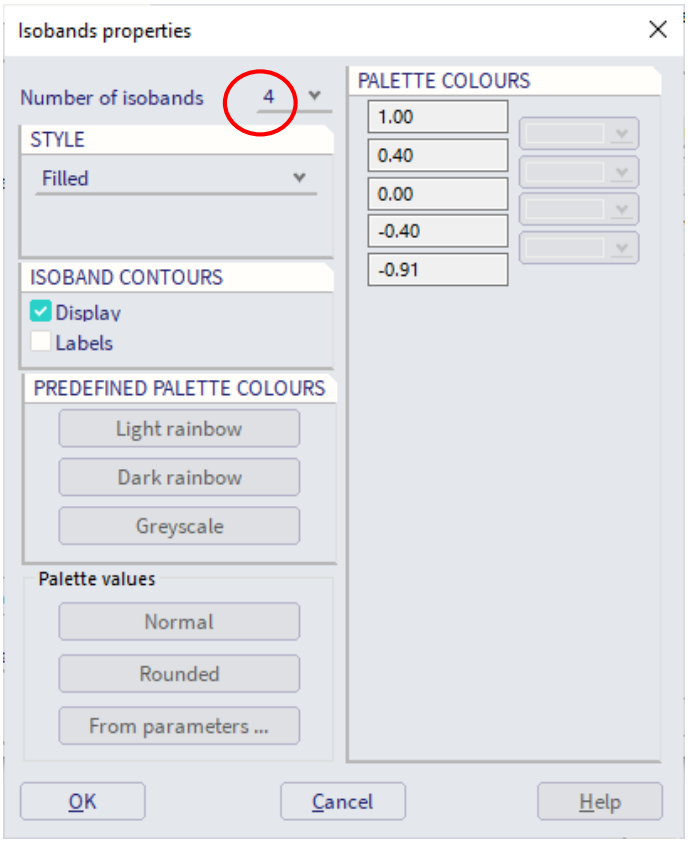

Check manager
All
EN 1993 Buckling check
EN 1992 Reinforceme...
Elastic Stresses Slabs
Header Plate Connecti...
strian energy check
Name strian energy check
Description
Online help http://designforms.net
Menu definition
SVG Icon SVG icon is not selected
Codes EC - EN
Category Other checks
Check data
Design form b17_strain_energy_check....
Element type 2D (slab/wall)
Material Concrete
Limit state Ultimate
Output variables 4 defined
SCIA Engineer function...
Multidesign options
In detailed output, generate all
Generate dangerous combinat No
Live load reduction options
Construction stages
Actions
Open in the Script Editor >>>
Update definition >>>
New
Insert
Edit
Delete
New plugins
Close


Please click here or press S
All categories
Concrete settings
Concrete 1D
Concrete 2D
Explode reinforcement to free bars
Code dependent deflection
Bill of reinforcement
External tools
Check manager
Update check data
Design forms
EN 1992 Reinforcement Design Recta...
Elastic Stresses Slabs Design form: C...
Header Plate Connection Design for...
Header Plate Connection Design for...
strian energy check Design form: Ch...

The quantity α is the ratio of the membrane strain energy and the bending strain energy. Where $\alpha = 1$ the load is carried in by membrane forces. Where $\alpha = -1$ the load is carried by moments. It gives in one contour plot an overview of the shell behaviour. There is a lot of moment in the hypars. We cannot do much about these because they are due to edge disturbances.






Isobands properties
Number of isobands
4
STYLE
Filled
ISOBAND CONTOURS
Display
Labels
PREDEFINED PALETTE COLOURS
Light rainbow
Dark rainbow
Greyscale
Palette values
Normal
Rounded
From parameters ...
PALETTE COLOURS
1.00
0.40
0.00
-0.40
-0.91
OK
Cancel
Help