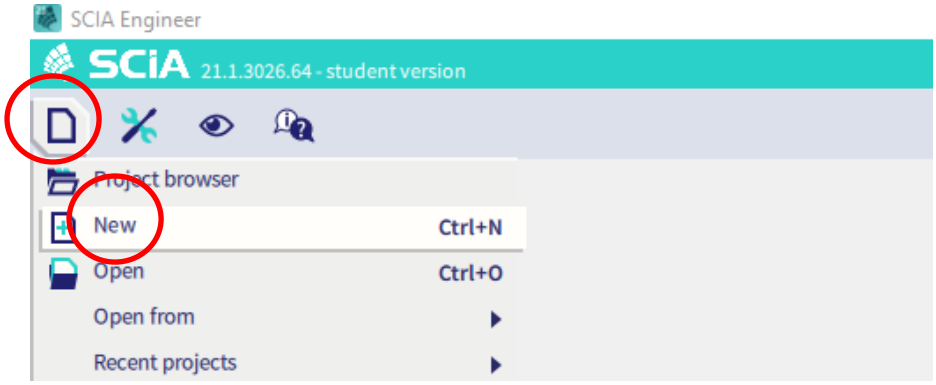
Finite element analysis of a steel conoid roof

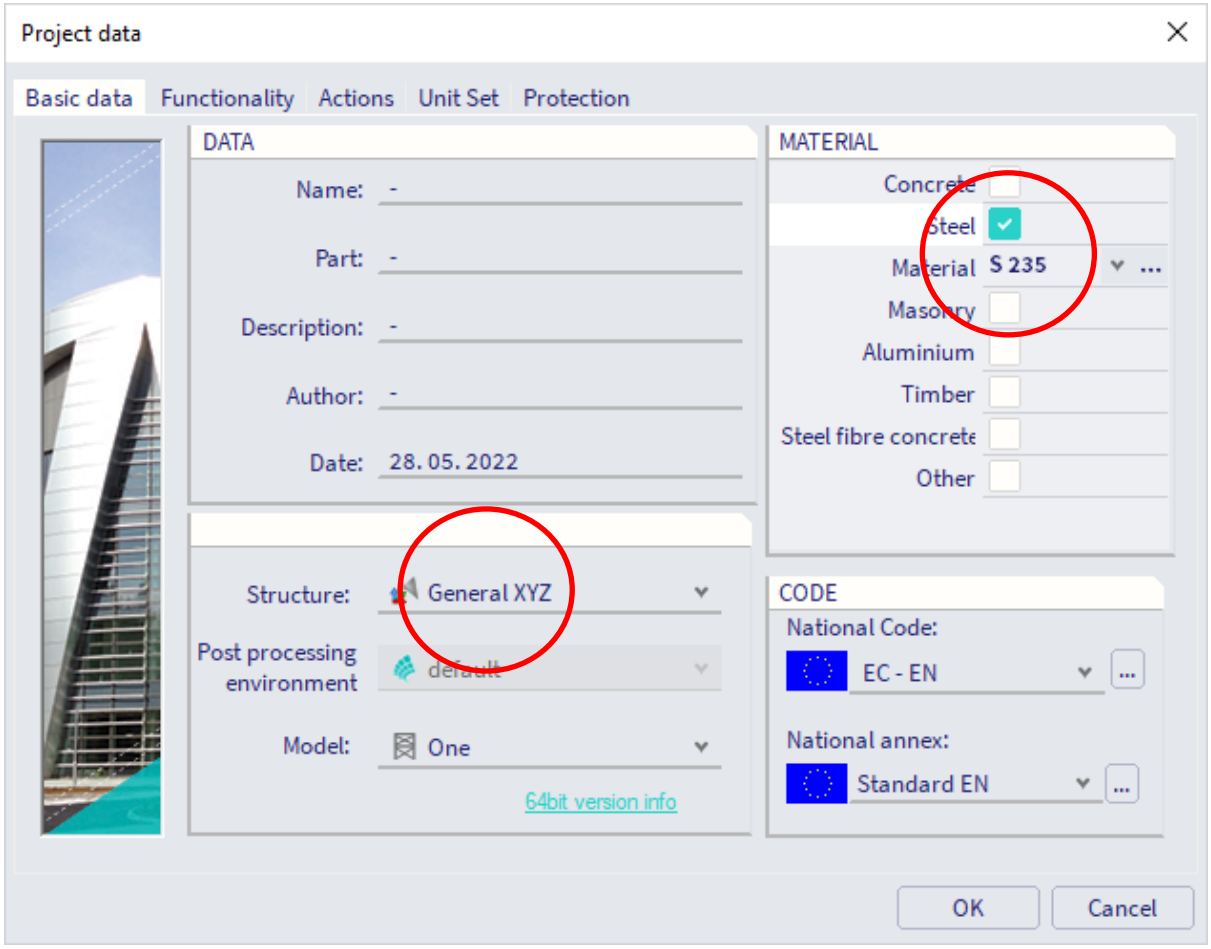
Exercise of course CIEM5301 Shell Structures

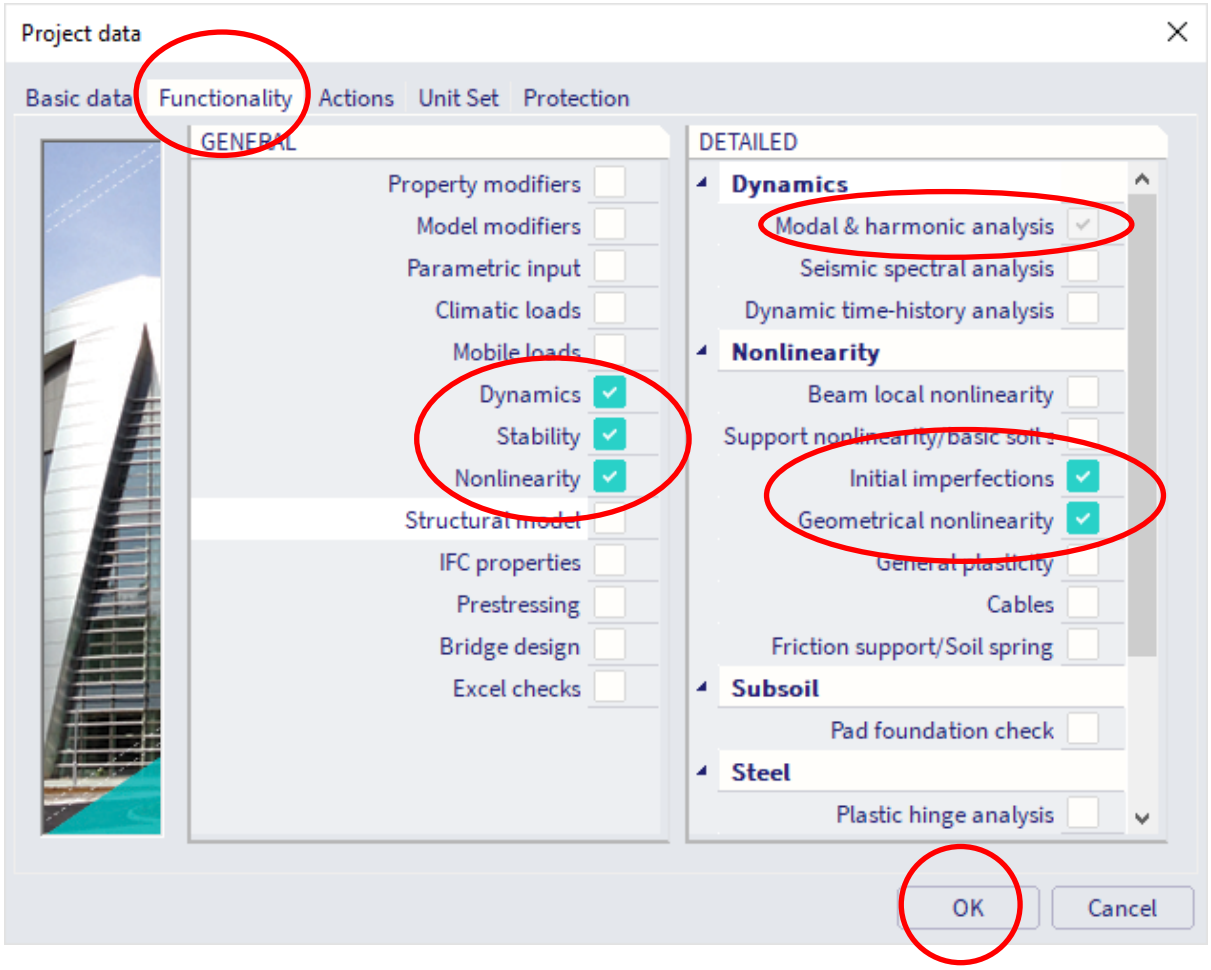
P.C.J. Hoogenboom, 4 October 2026

In this exercise we analyse a steel conoid shell roof by the finite element method. We will model the roof, perform a linear elastic analysis, calculate the natural frequencies perform a linear buckling analysis and a geometrically nonlinear analysis. At the end, we conclude that this roof can be built. This classroom exercise takes about 1.5 hour.

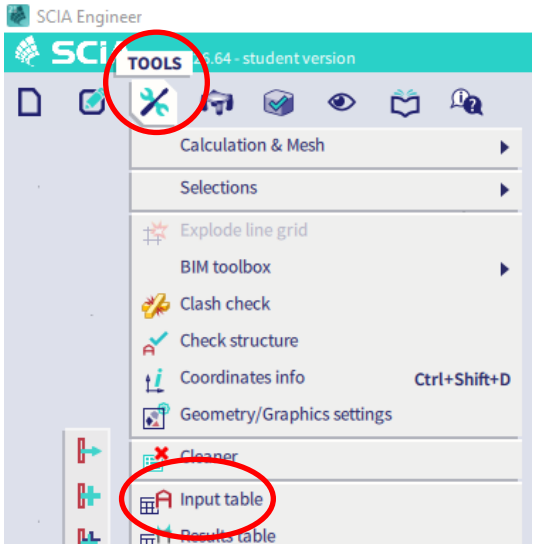
Start SCIA Engineer and start a new project.

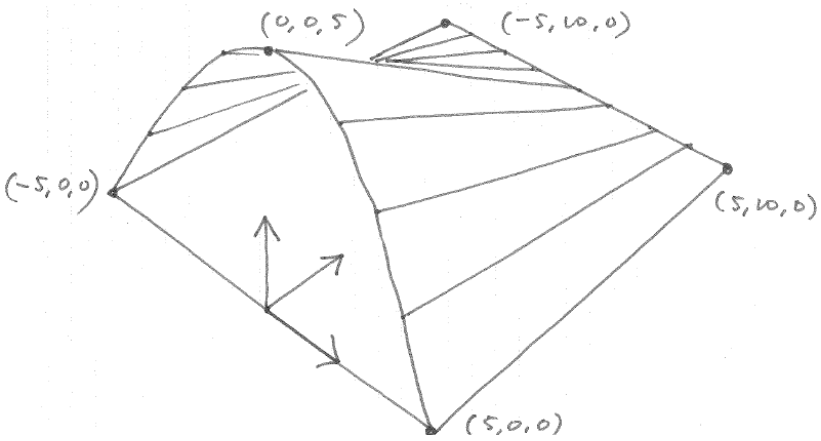


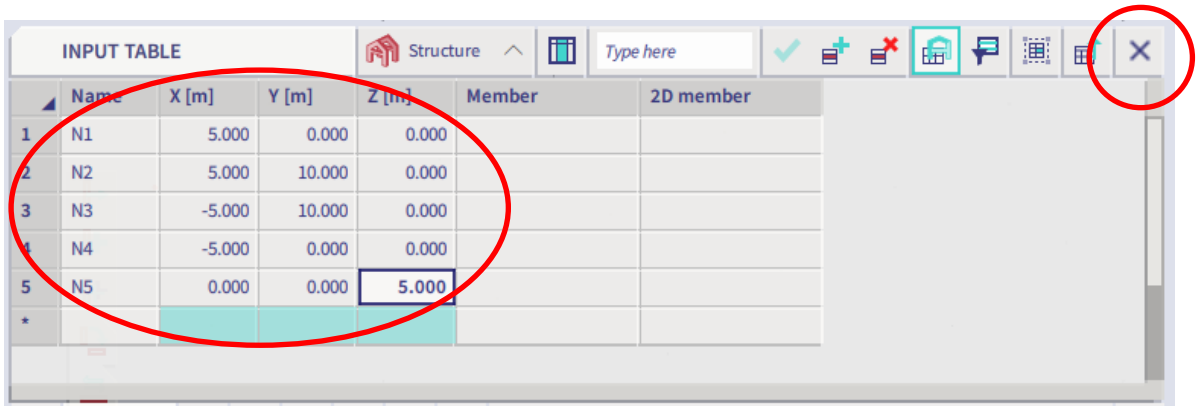


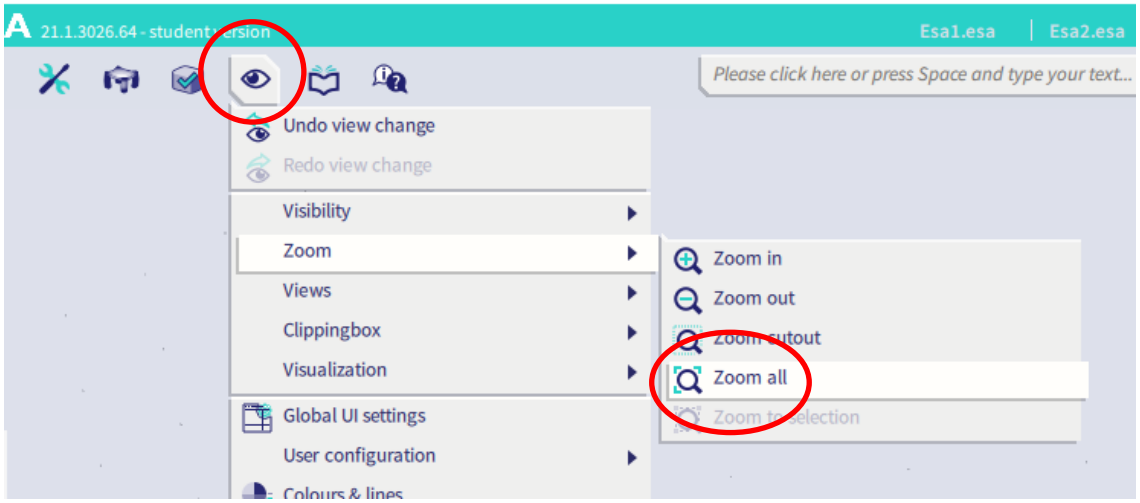


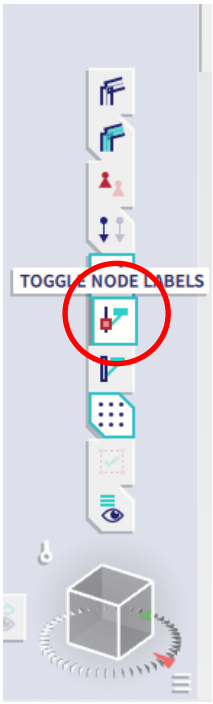
We first enter 5 points for drawing a conoid shell.

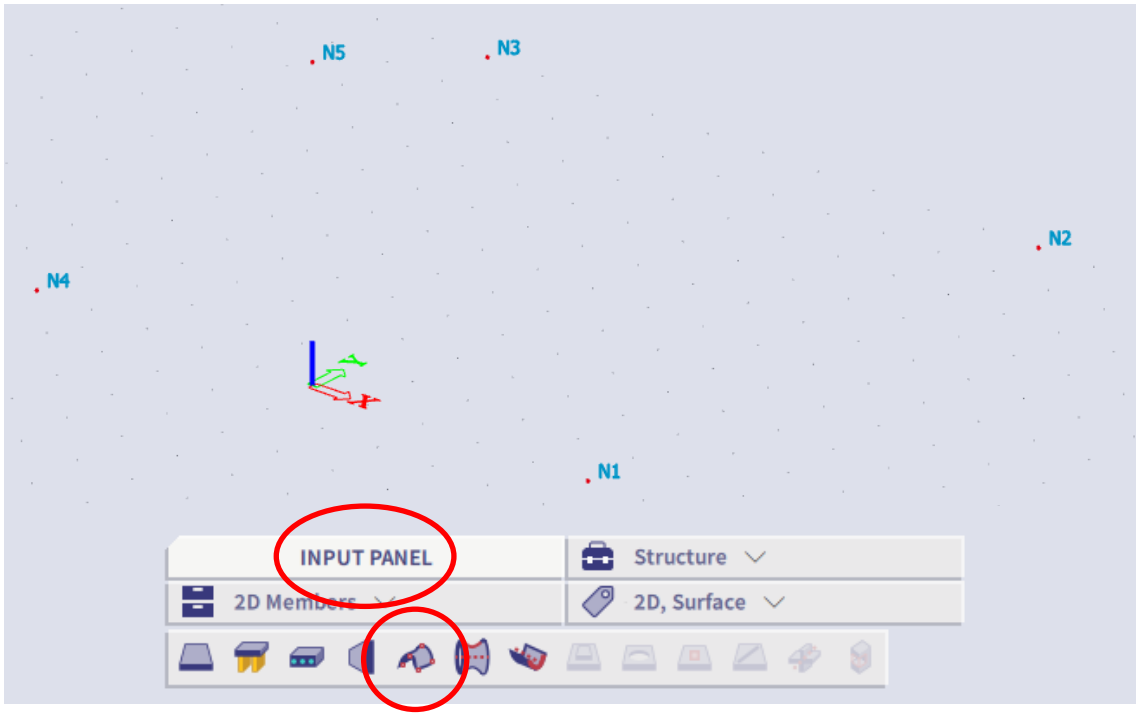




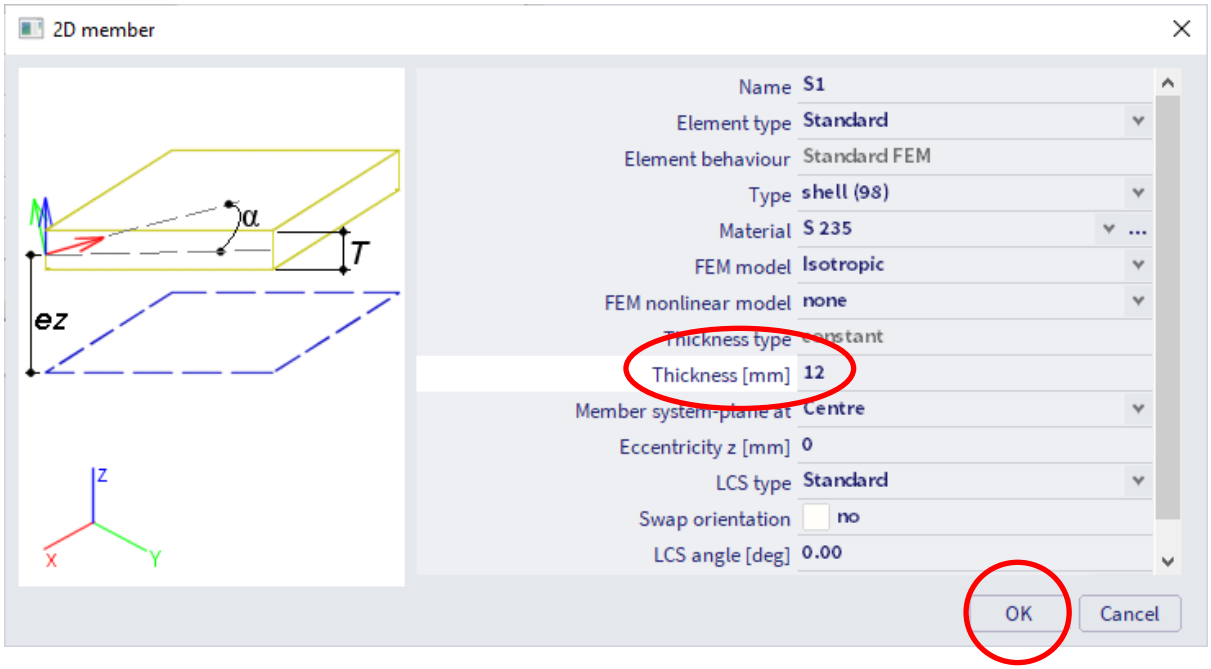




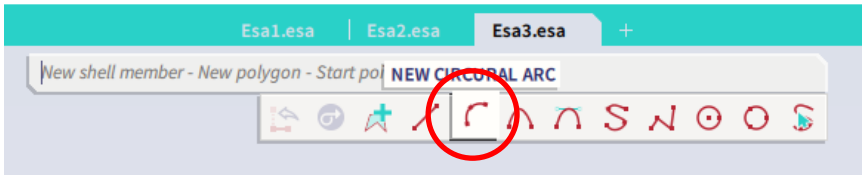




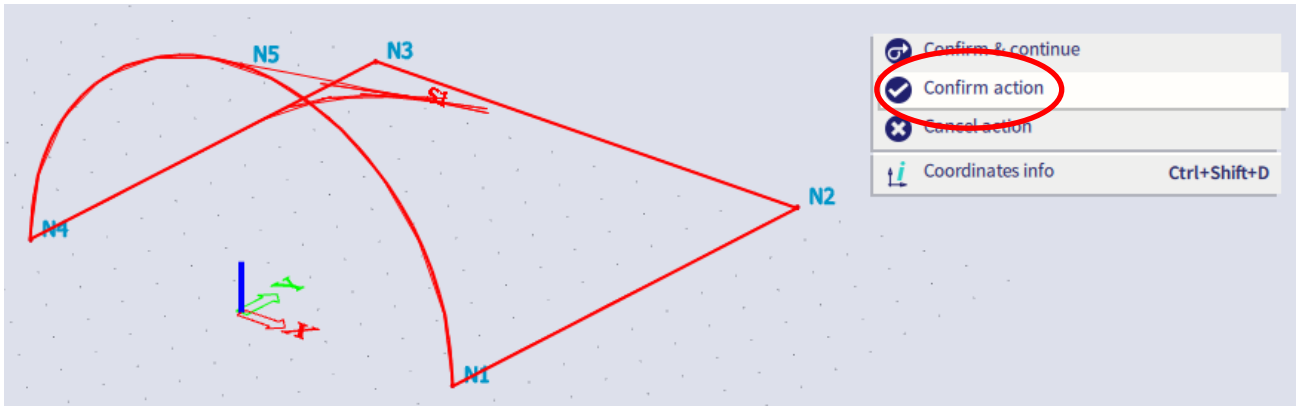
The shell is a 12 mm curved steel plate.



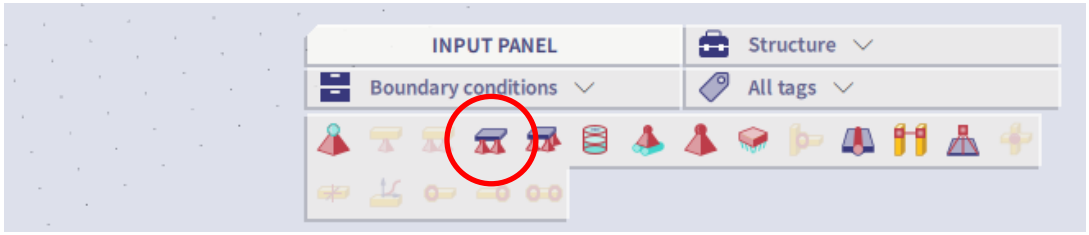
Click on the nodes N1, N2, N3, N4.
Click on “New circular arch”.

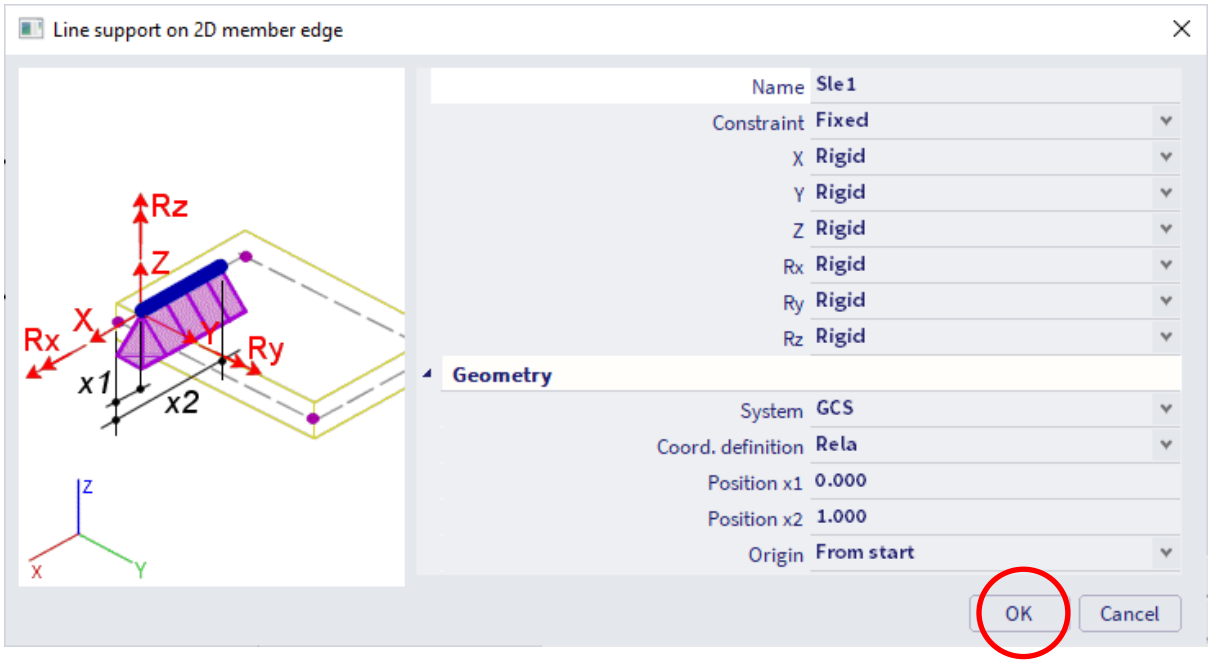


Click on the nodes N5 and N1.
Click right mouse button and click Confirm action.

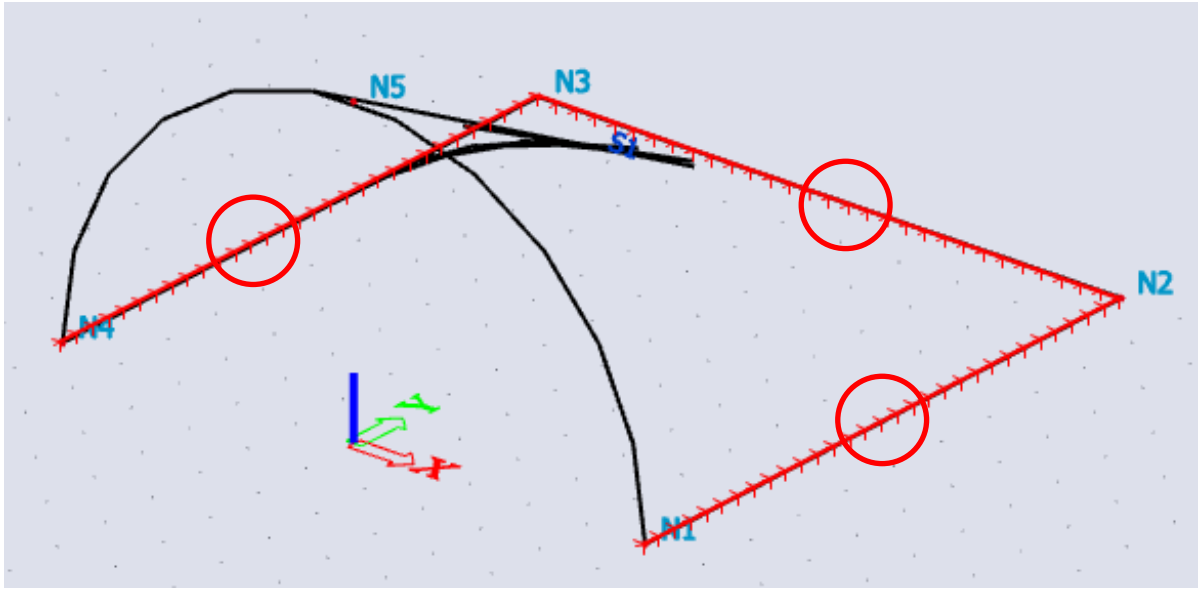


The shell is supported by fixing the straight edges.



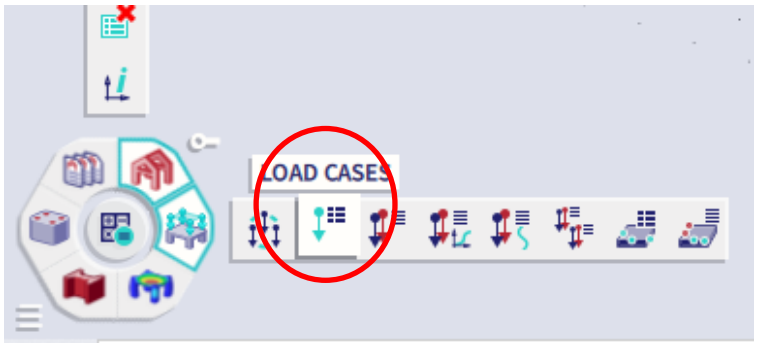


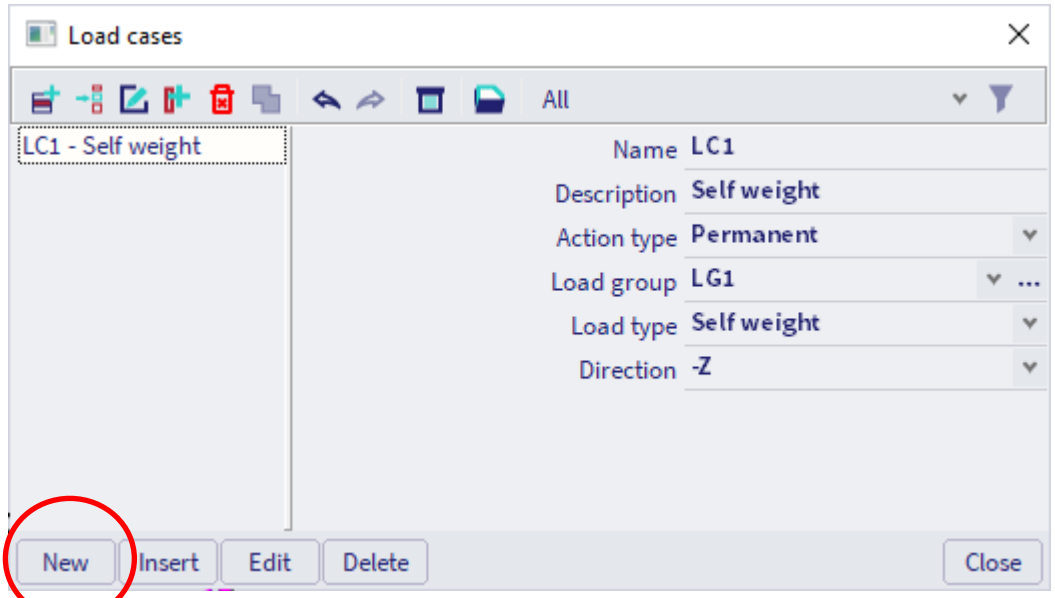
Click on the straight edges.

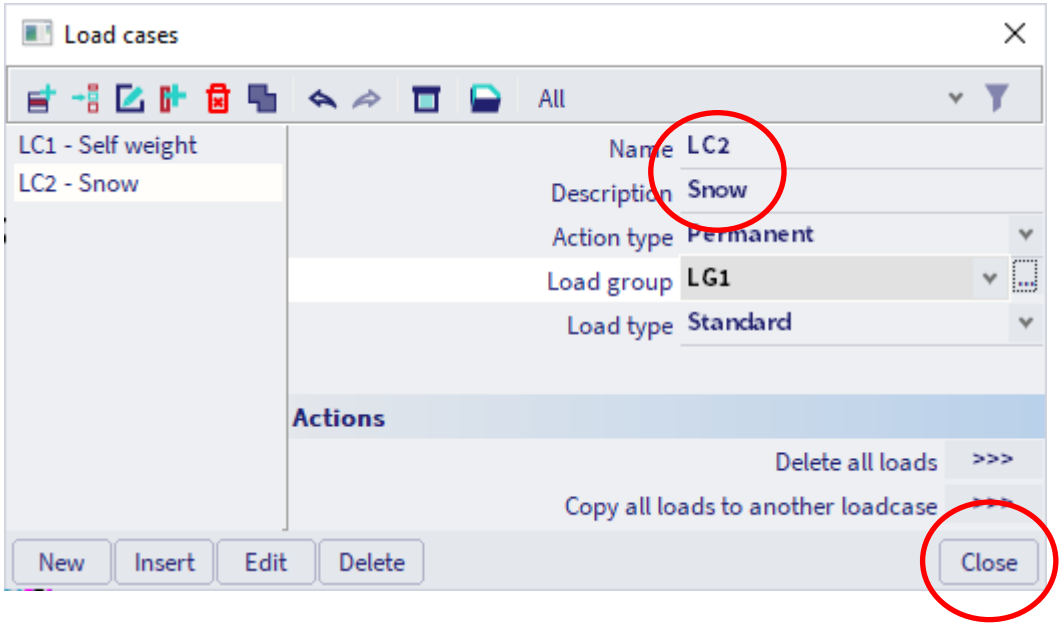


Click right mouse button and click Confirm action.

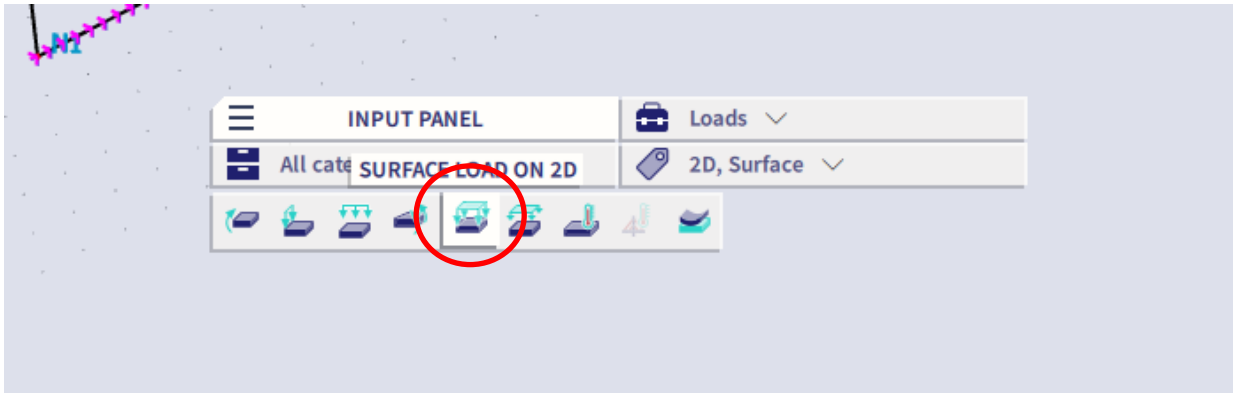
One load case is already present: Self-weight. We add a load case: Snow.

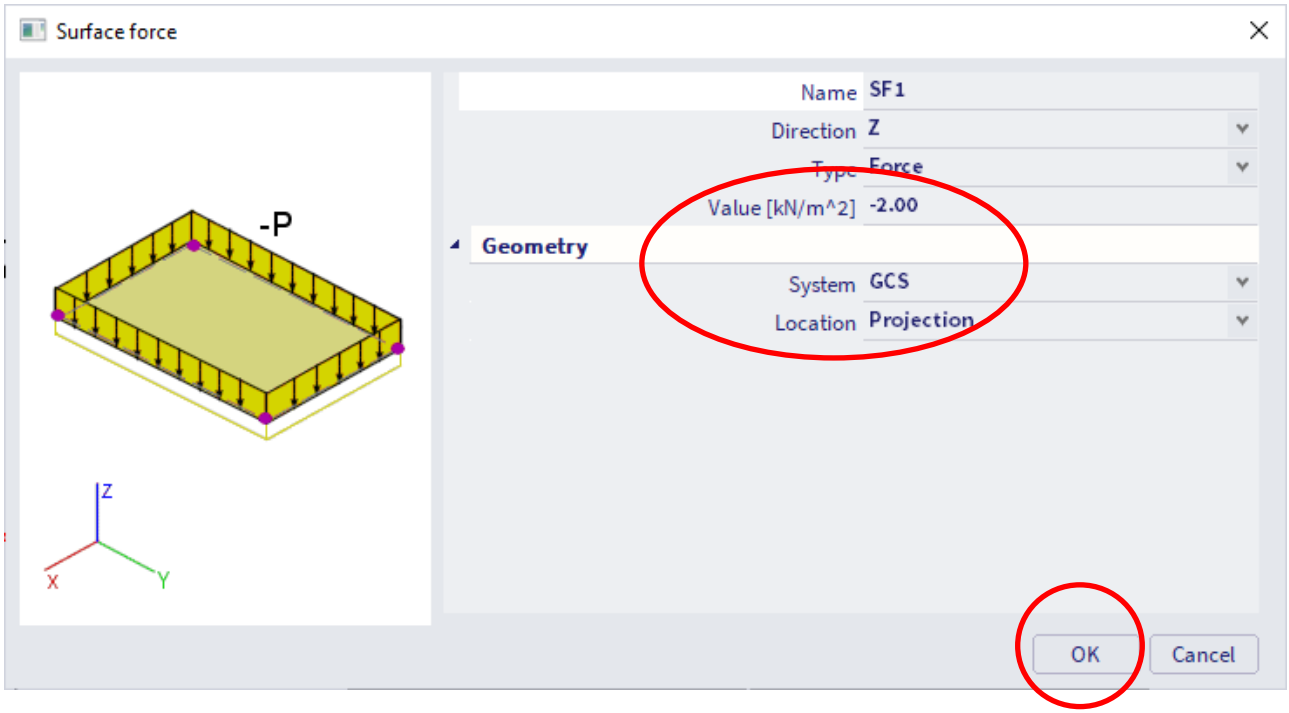


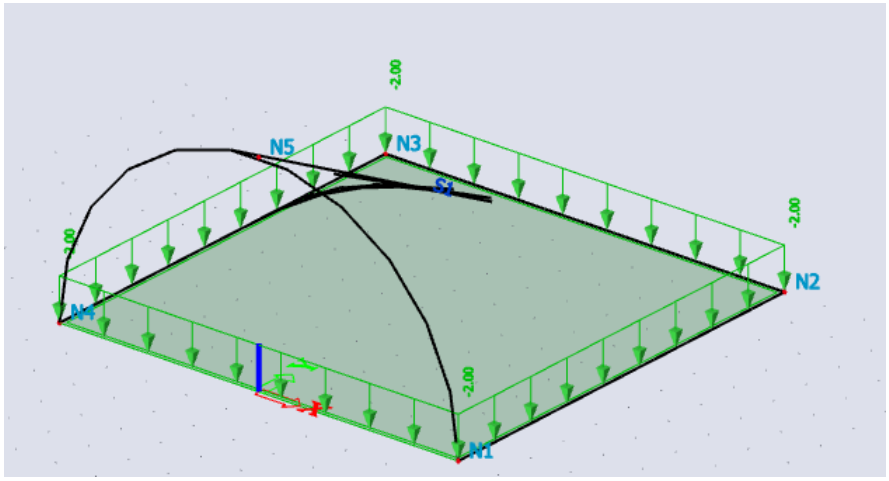




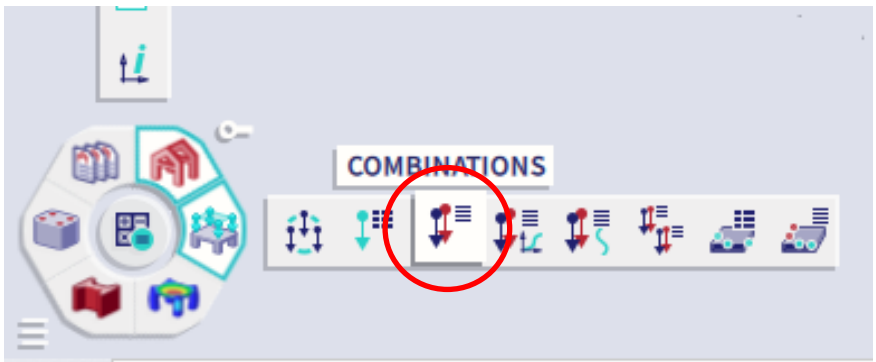
Now we can enter the snow load: 2 kN/m^2

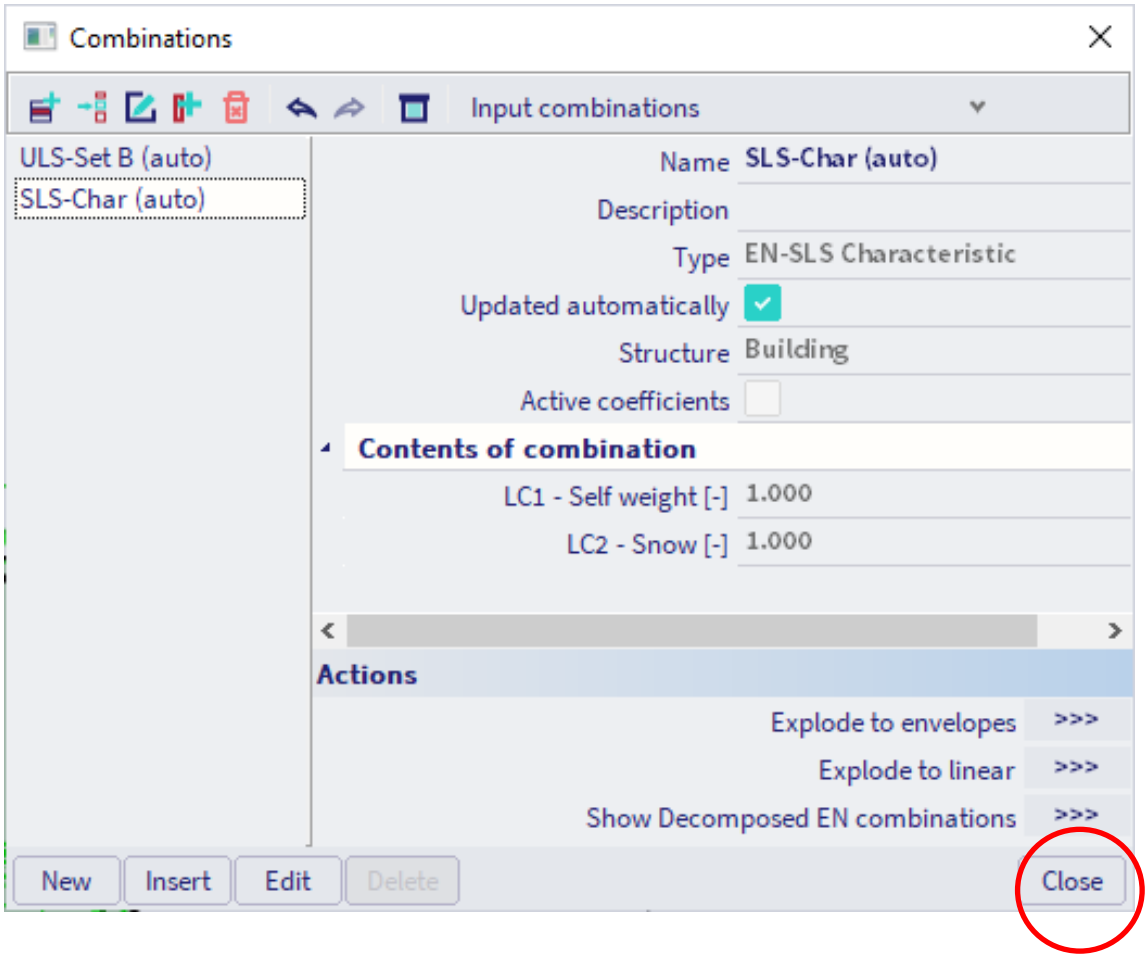






We make a load combination in which both Self-weight and Snow occur at the same time. Safety factors are not used.

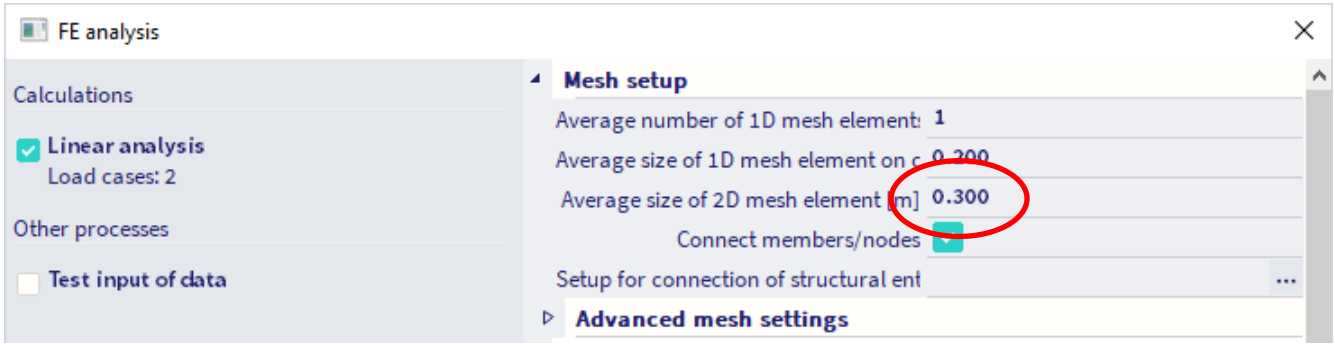


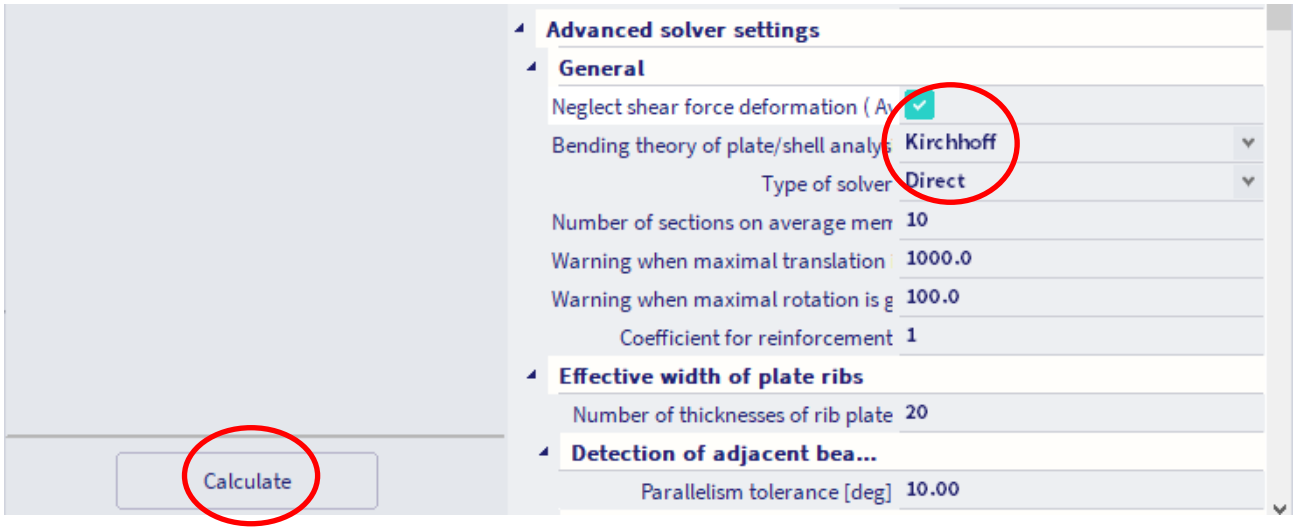


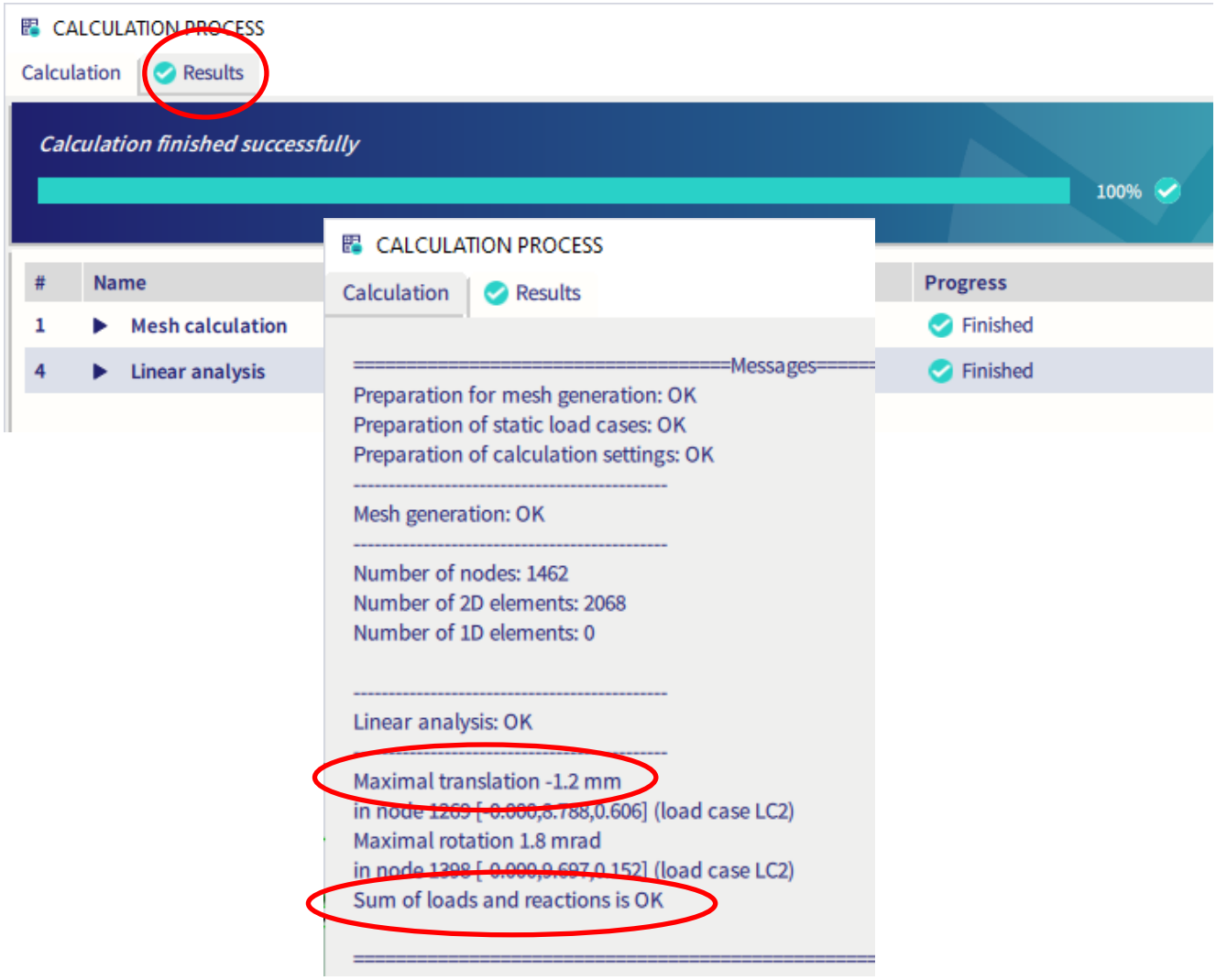
We first perform a **linear elastic analysis**.



The influence length is 2.4*sqrt(5*0.012) = 0.6 m. The element size should be 1/6 of this. Let's set the element size to 0.3 m. This is a bit large but we can always make it smaller later.



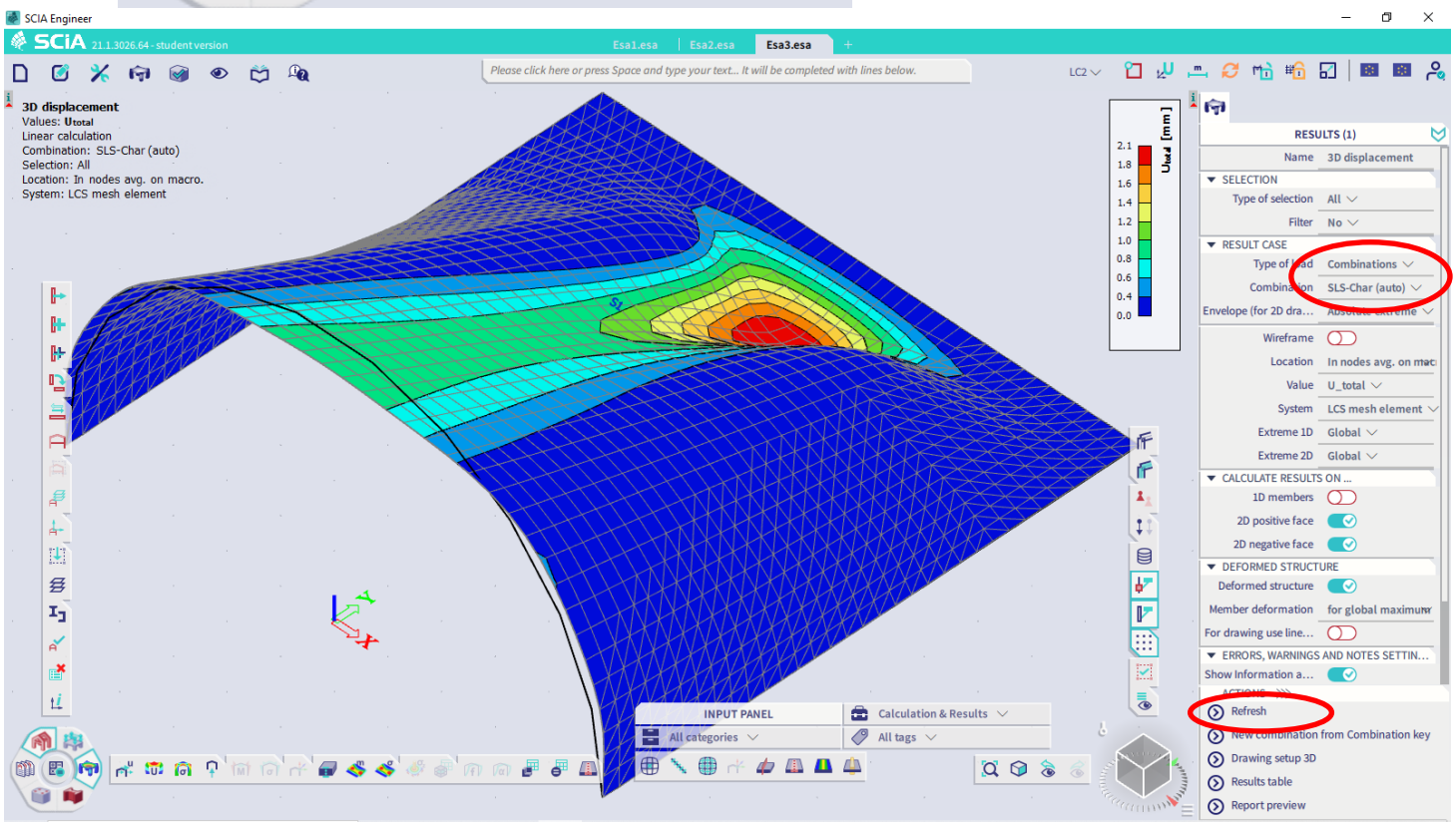




Note that the software has checked equilibrium of the loads and the support reactions.
If this were not OK, the arithmetic accuracy would not be OK.

Let's look at the **results**. So, we do post processing.
Deflection due to the load combination.



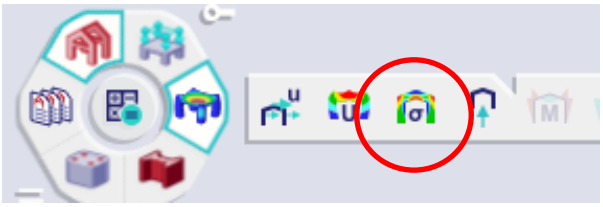


First we check *ourselves*.
The supports did not move ..... OK
The deformation is in the direction of the load ..... OK
The elements may be too large to describe the red peak accurately. For now, we leave the element size as it is because we do not want to increase the computation time ..... OK

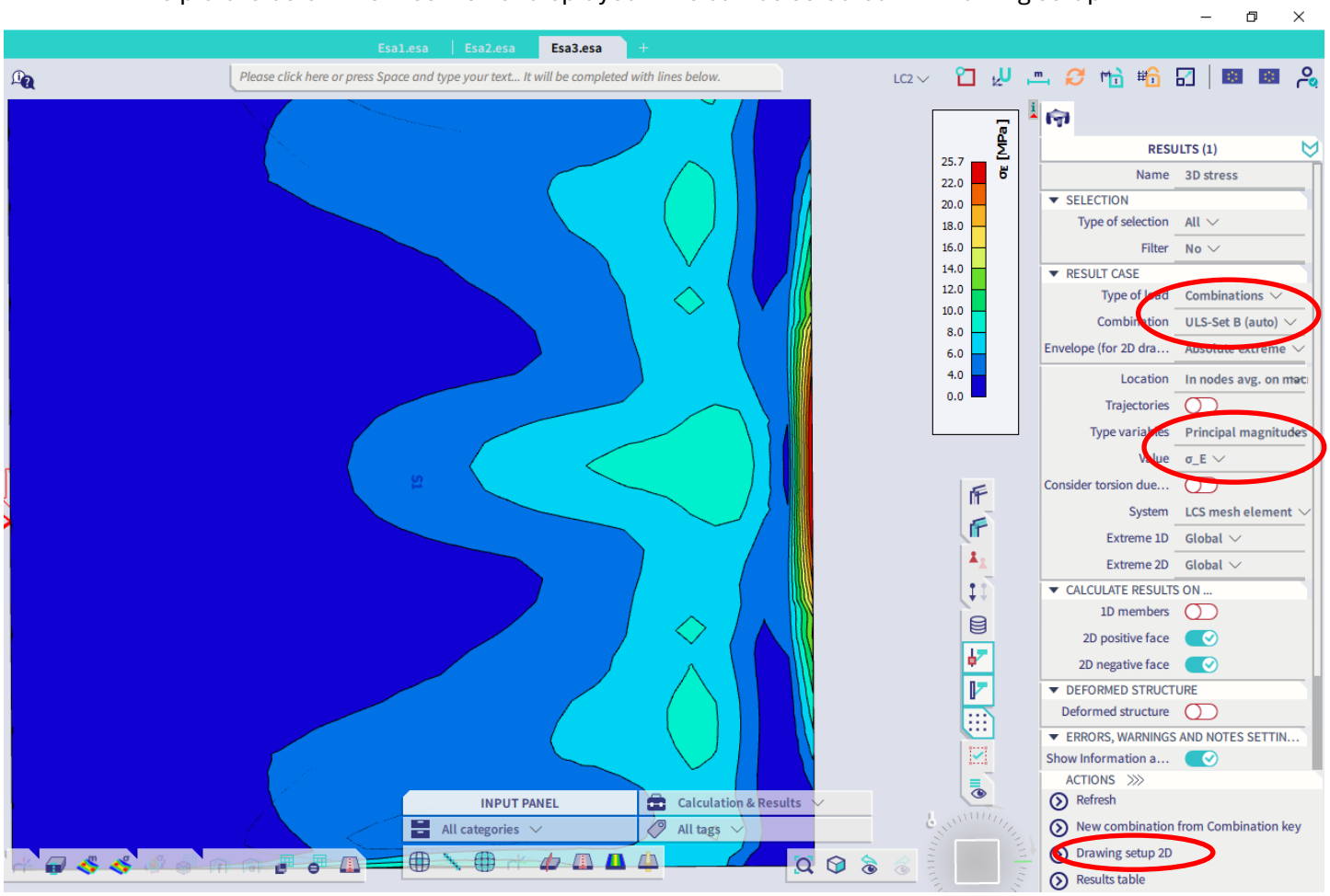
Note that the mesh consists of rectangles and triangles. Triangles are less accurate than rectangles. Why does the program choose triangles?

Second we check the *design*. The deflection is largest in the flat part of the shell. Nonetheless it is just 2.1 mm, which is surely acceptable ..... OK

Let's look at the Von Mises stresses (Equivalent stress)



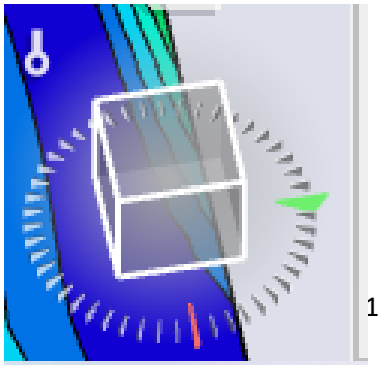
In the picture below the mesh is not displayed. This can be selected in “Drawing setup 2D”.



First we check *ourselves*.
In the free edge the stress is 0 - 4 N/mm^2 ..... OK
Singularities are not visible. We expect one in the transition from free to fixed edge ..... OK
The fixed edges have edge disturbances. The influence length is about 2 elements, which may not be enough. We need to check this later ..... Not OK

Second, we check the *design*.
In most of the shell the stress is almost zero. The largest stresses occur at the edges. A largest stress of 25.7 N/mm^2 is not much. But this will increase when the mesh is refined. (Not to infinity because it is not a singularity.) Nonetheless, perhaps the shell can be designed much thinner.

This view is at the outside surface of the shell. Turn the shell around to see the stresses at the inside surface.



Let's reduce the element size to check accuracy.

Element size:	0.300 m	0.150 m
Largest Von Mises stress	25.7 N/mm^2	26.1 N/mm^2

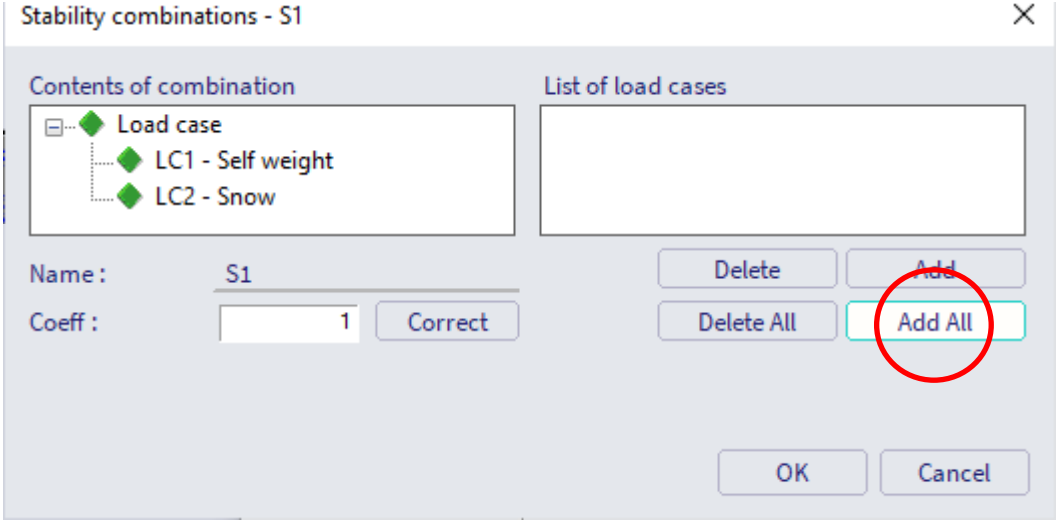
The new linear elastic calculation takes a minute.
The difference between the two computations is just 0.4 N/mm^2. So an estimate of the error that we make with the 0.150 mm mesh is 0.4 N/mm^2 or 2% which is sufficiently small for most applications.

In the following we will work with 0.300 m elements.

Let's check **buckling**. We need to define stability load combinations.

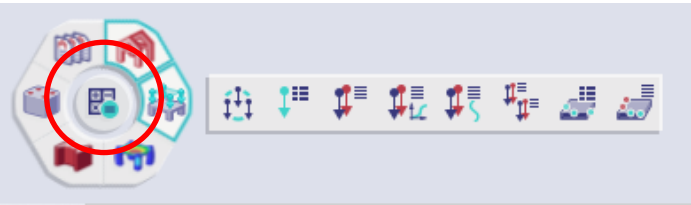


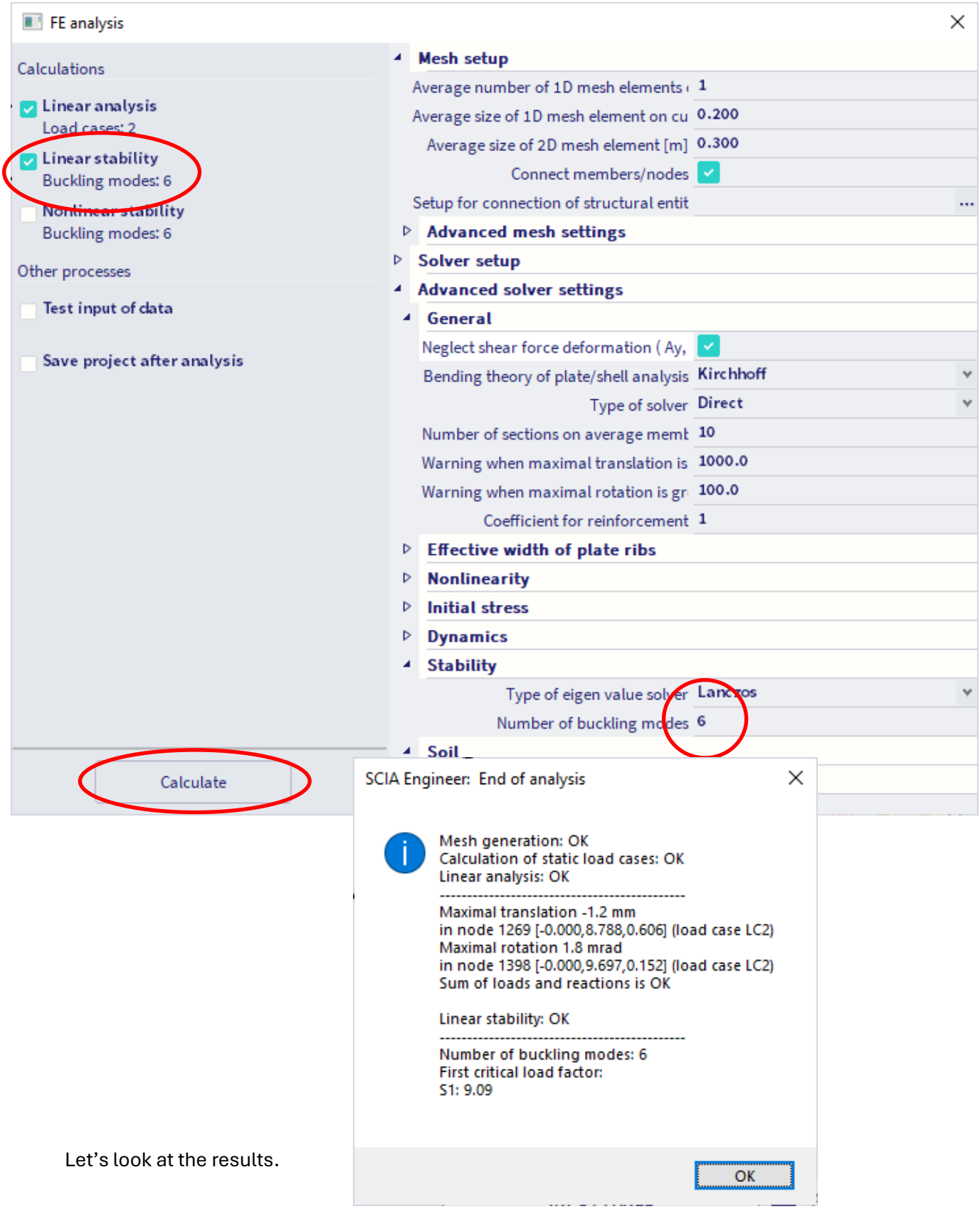
(The teacher does not understand why the software does not use the normal load combinations for stability analysis too. Perhaps this extra step will be removed in future software versions.)



Now we can perform the linear buckling analysis. Let's compute 6 buckling modes.

Note that first a linear analysis is performed. This is to compute the membrane forces, which cause buckling.

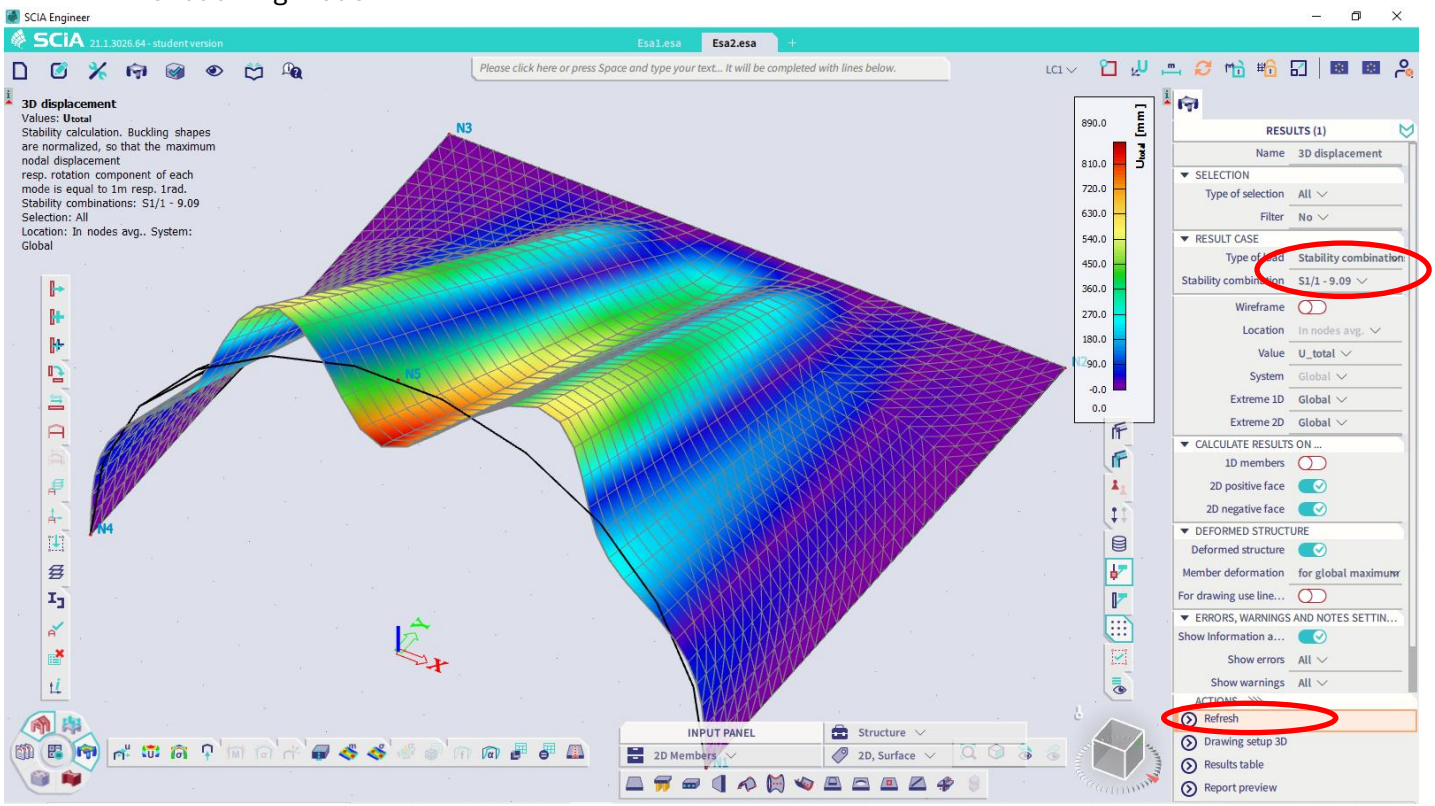




Let's look at the results.



First buckling mode:



First we check *ourselves*.
The edges are correctly fixed and correctly free ..... OK
There are about 6 elements in a halve wave, which gives sufficient accuracy ..... OK

The figure shows that this buckling mode moves mostly inwards. However, the exact same mode is sometimes displayed as moving mostly outwards. Does this make sense?

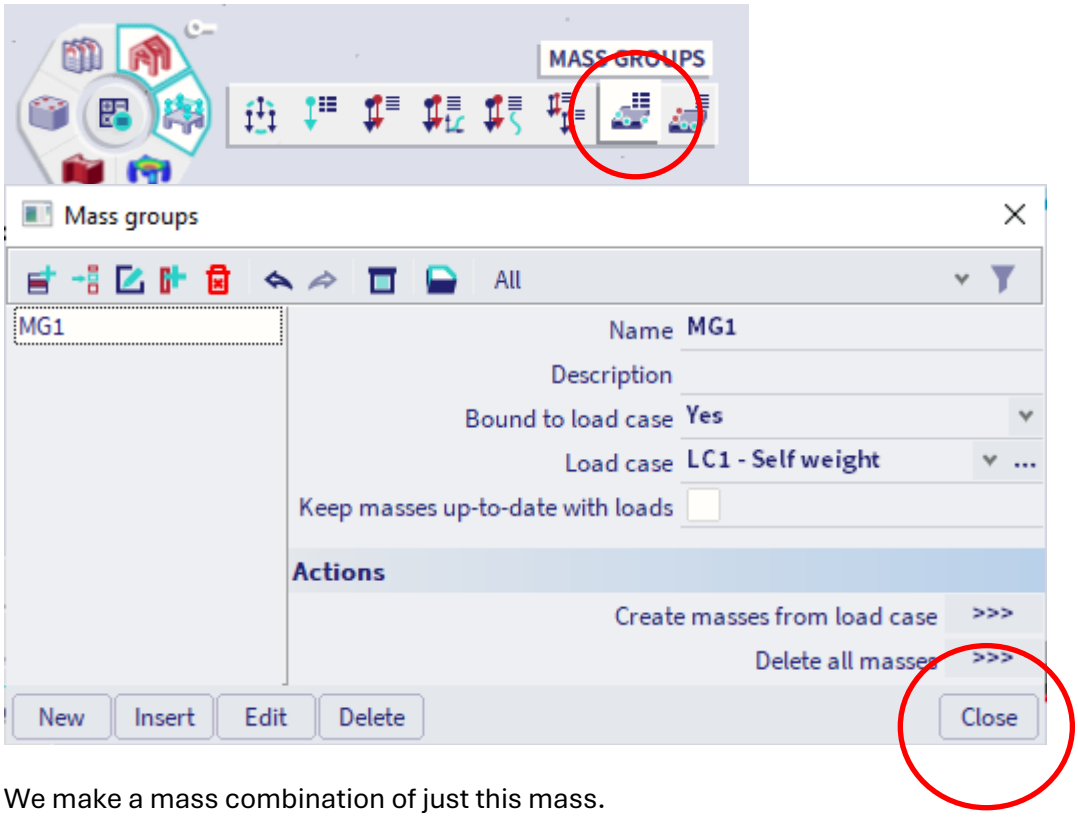
Second we check the *design*.
It buckles in this shape when the load combination is multiplied with a load factor of 9.09. The subsequent buckling load factors are 9.13, 12.37, 12.70, 13.70, 14.24 et cetera.
This is safe, because with a knockdown factor of 1/6, the shell will not buckle. ..... OK

Clearly, an edge beam would supress most buckling modes. With an edge beam the shell could be even thinner.

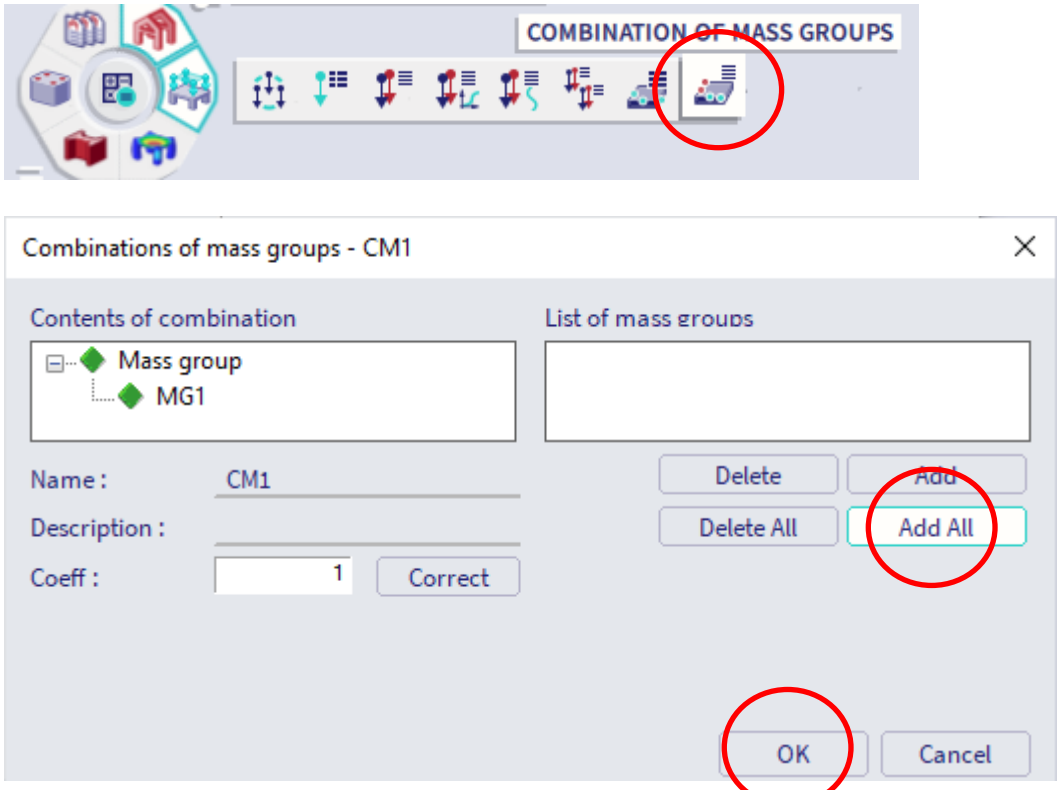
Let's look at the **natural frequencies**. First we need to add mass to the load cases.

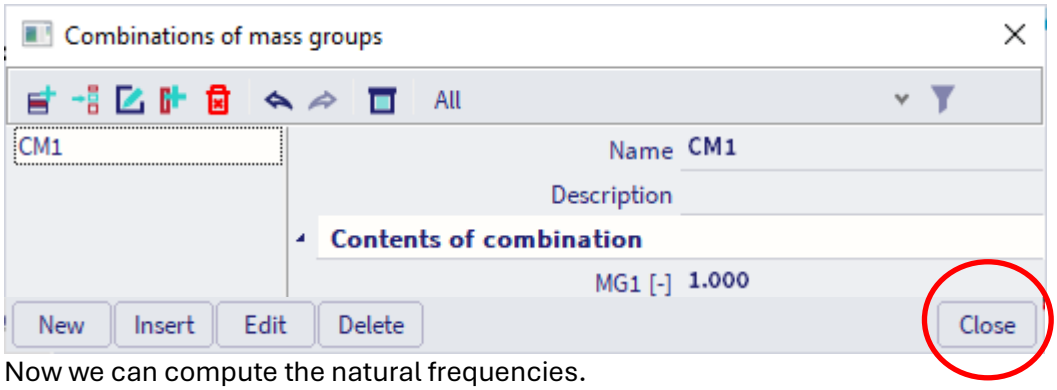
(The program already knows about the mass of the materials but it does not know whether other loads have a mass. For example snow load has a mass and wind load has no mass.)

Let’s study the vibrations without snow. The mass of the materials is in something called MG1.



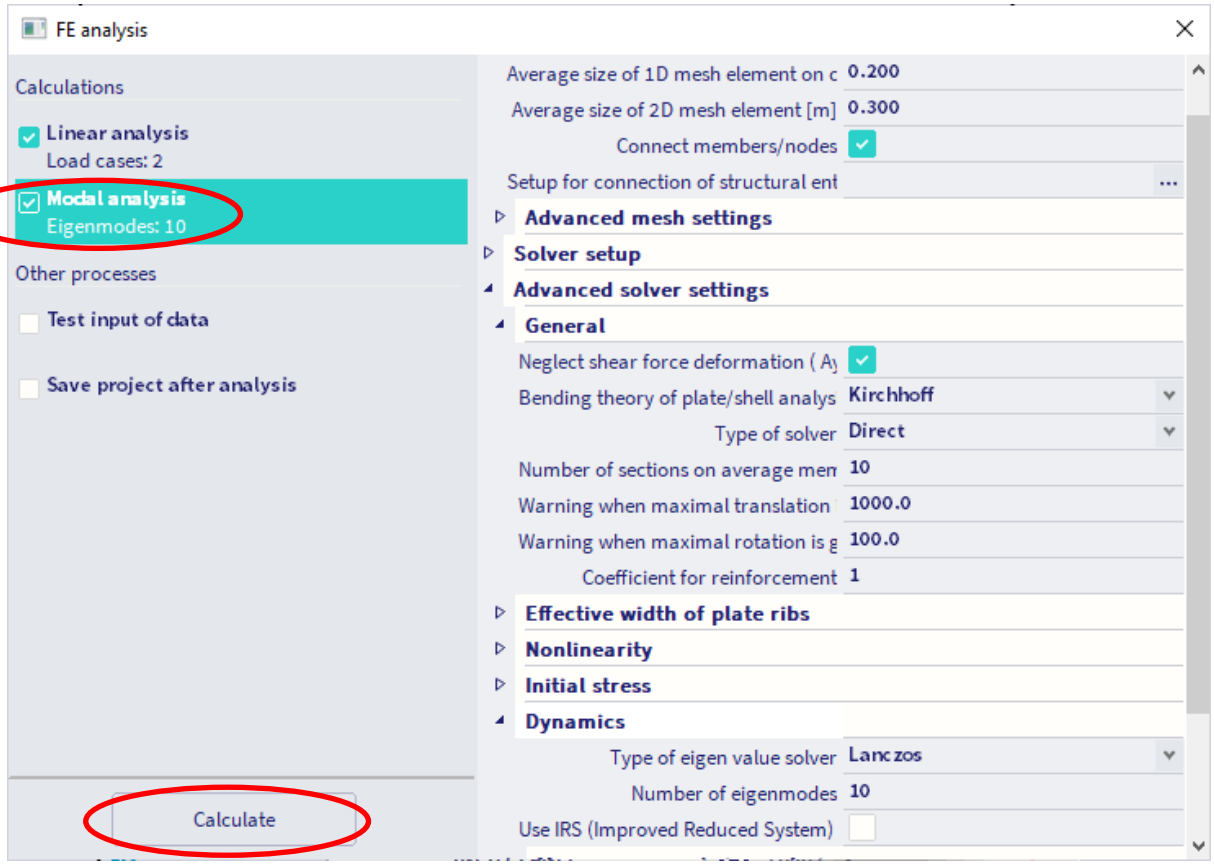
We make a mass combination of just this mass.



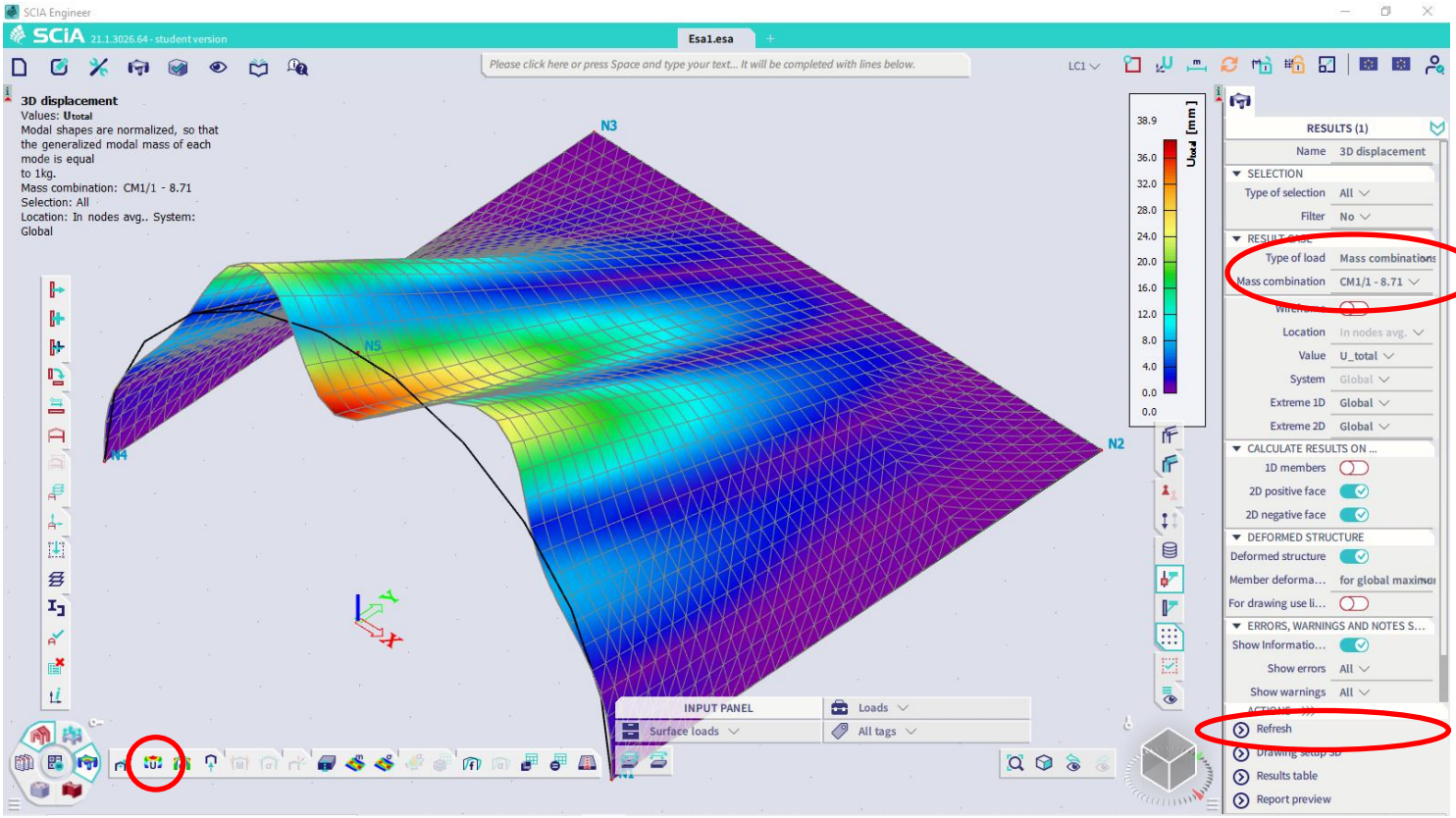


Now we can compute the natural frequencies.





Let's look at the results. First vibration mode: The natural frequency is 8.71 Hz.



The subsequent natural frequencies are 8.71, 12.61, 12.78, 15.37, 15.81, 16.34, 17.01, 18.18, 20.01 et cetera. Note that there is little space between the natural frequencies. This is typical for shell structures.

Frame structures are sometimes designed such that the frequency of the load is no problem because it is in between two natural frequencies. Is this possible for shells too?

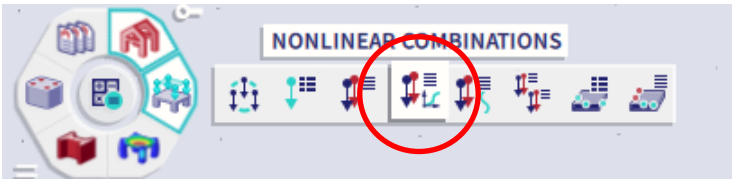
The program has computed the 10 smallest natural frequencies. If you want to see more you can set this just before starting the computation.

This shell is not sensitive to wind load because wind gusts have a frequency of approximately 1 Hz or smaller. This shell is sensitive to earthquakes. It would not be sensitive to earthquakes, if all natural frequencies were larger than 10 Hz.

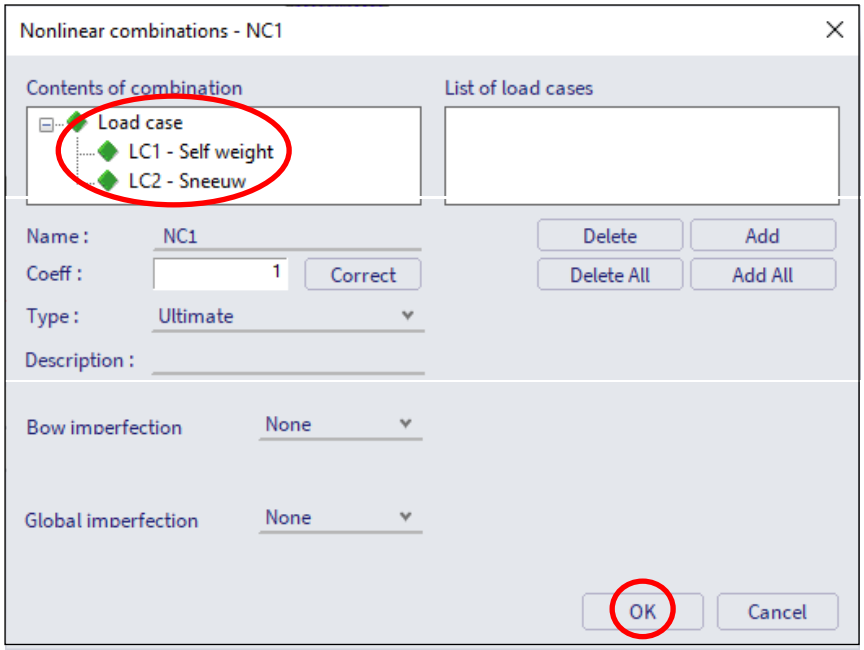
Vibration mode shapes often look like buckling mode shapes. There is a situation in which they are exactly the same. This is when the load is so large that the structure almost buckles. If you give the structure is little push it moves away slowly and comes back slowly. In this situation the natural frequency is almost 0 Hz and the vibration mode shape is the same as the buckling mode shape.

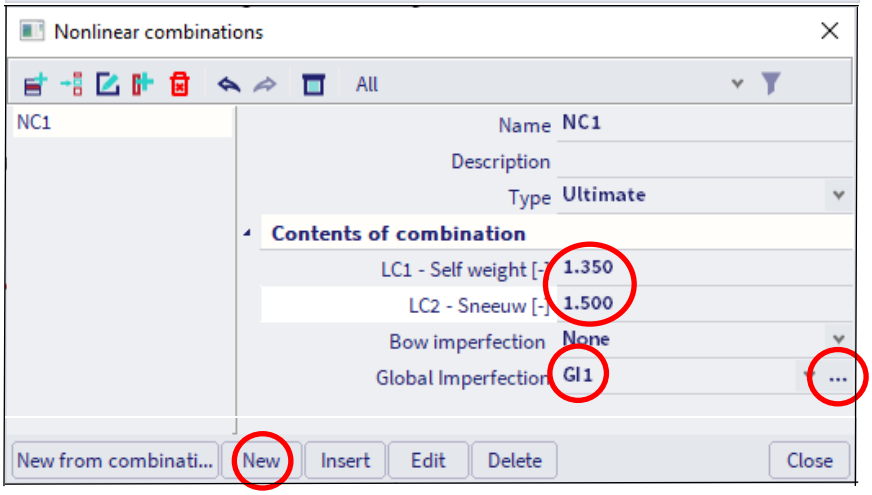
Why does the program always perform linear analyses before computing natural frequencies?

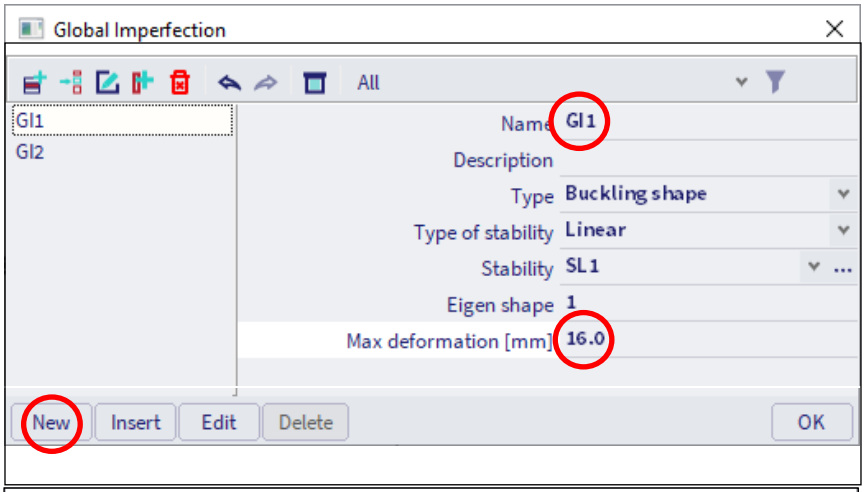
Let's do a **nonlinear analysis**. First, we need to specify nonlinear load combinations. (The teacher thinks that there is no difference between linear and nonlinear load combinations.)

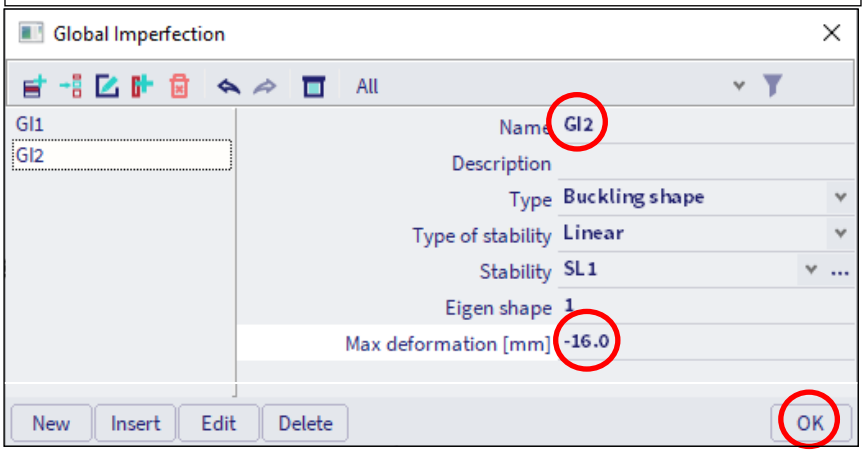


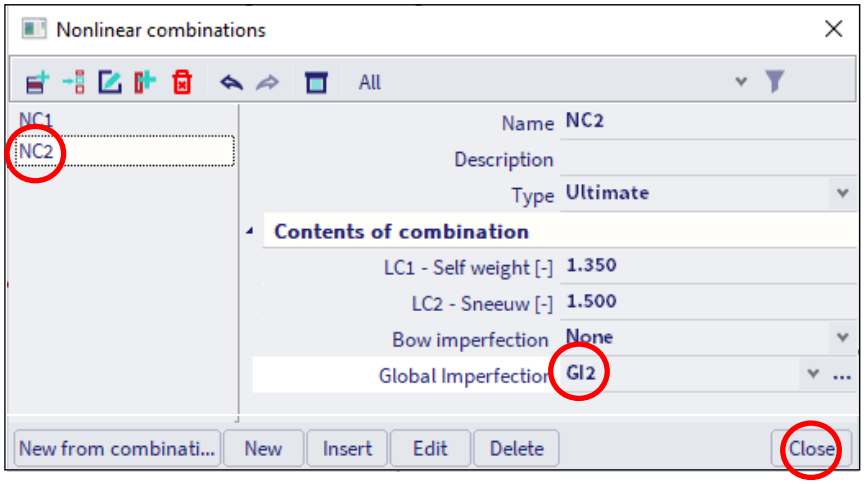
Here we can specify a shape imperfection. We choose an imperfection shaped as the first buckling mode with an amplitude of 16 mm. We also apply an imperfection in the other direction, so an amplitude of -16 mm.



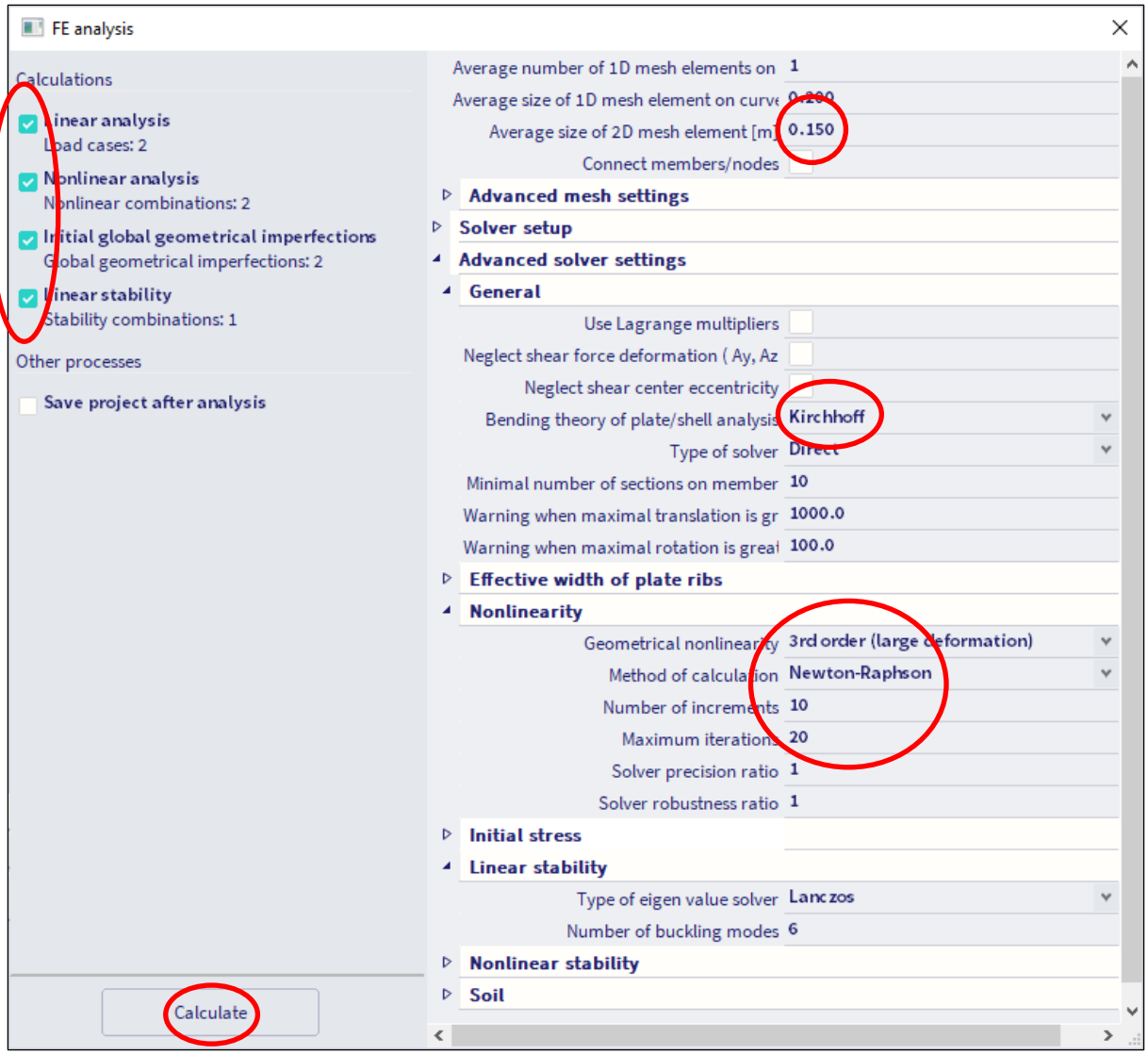




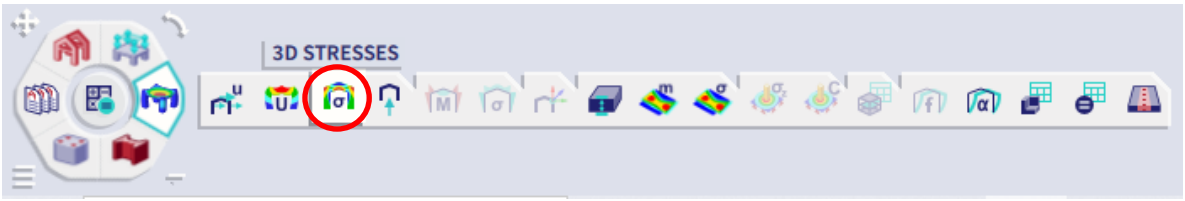


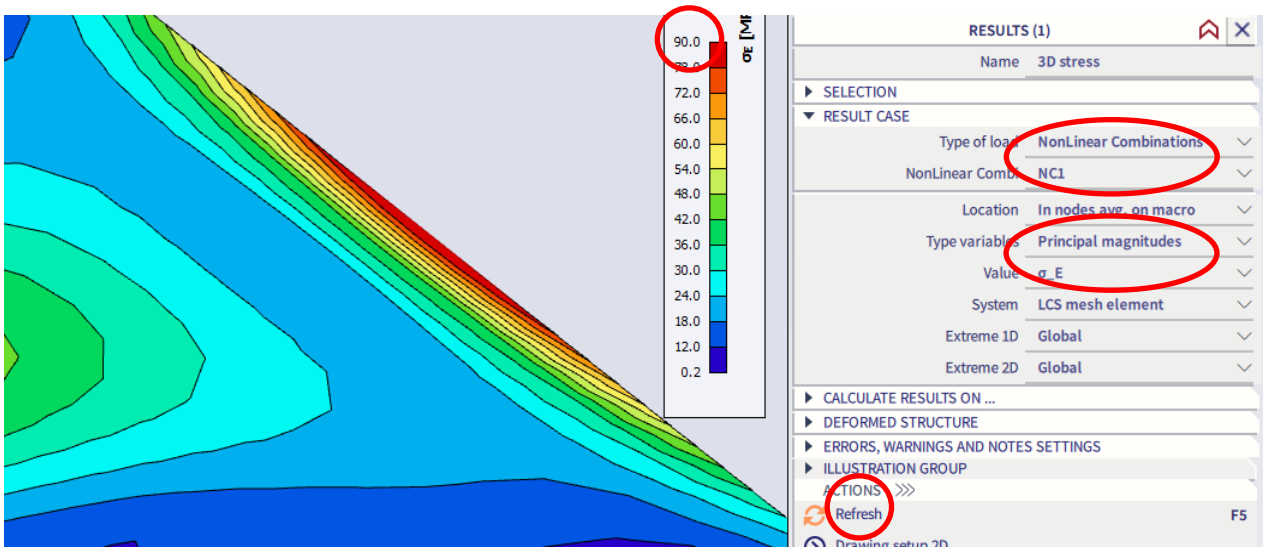


The software applies the load in 10 even steps (increments). After every step, the Newton-Raphson method is used to find the displacements. A few iterations are needed to obtain sufficient accuracy. At 21 iterations the program will stop, regardless of the accuracy. The algorithm is diverging and shell must have buckled.



Let’s look at the computation results.

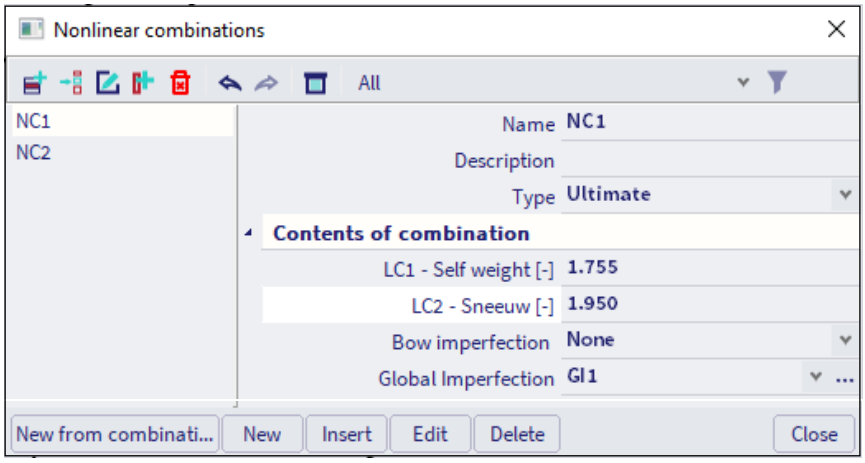




The largest stress is much larger than computed linearly (43,5 N/mm^2). The location of the largest stress is unchanged. The negative imperfection gives a smaller stress.

Question
Which imperfection gives the largest deflection? Compare the deflection of the linear elastic computation with that of the non-linear computation. What causes the difference?

We increase the load by a factor 1.3. This is smaller than the linear buckling load factor of 2.37.



We apply this load in 50 increments. We increase the maximum number of iterations to 40. The computation converges. The deflection is then 40 mm. Subsequently we increase the load by a factor of 1.4. The computation diverges. We conclude that the conoid buckles at a load factor of 1.35. The knockdown factor is 1.35/2.37 ≈ 1/2. This is comparable to the imperfection sensitivity of frame structures. This conoid is not very sensitive to imperfections.

Conclusion

The design can be built. It can be improved. The shell can be a bit thinner. If we design an edge beam, the shell can be even thinner.