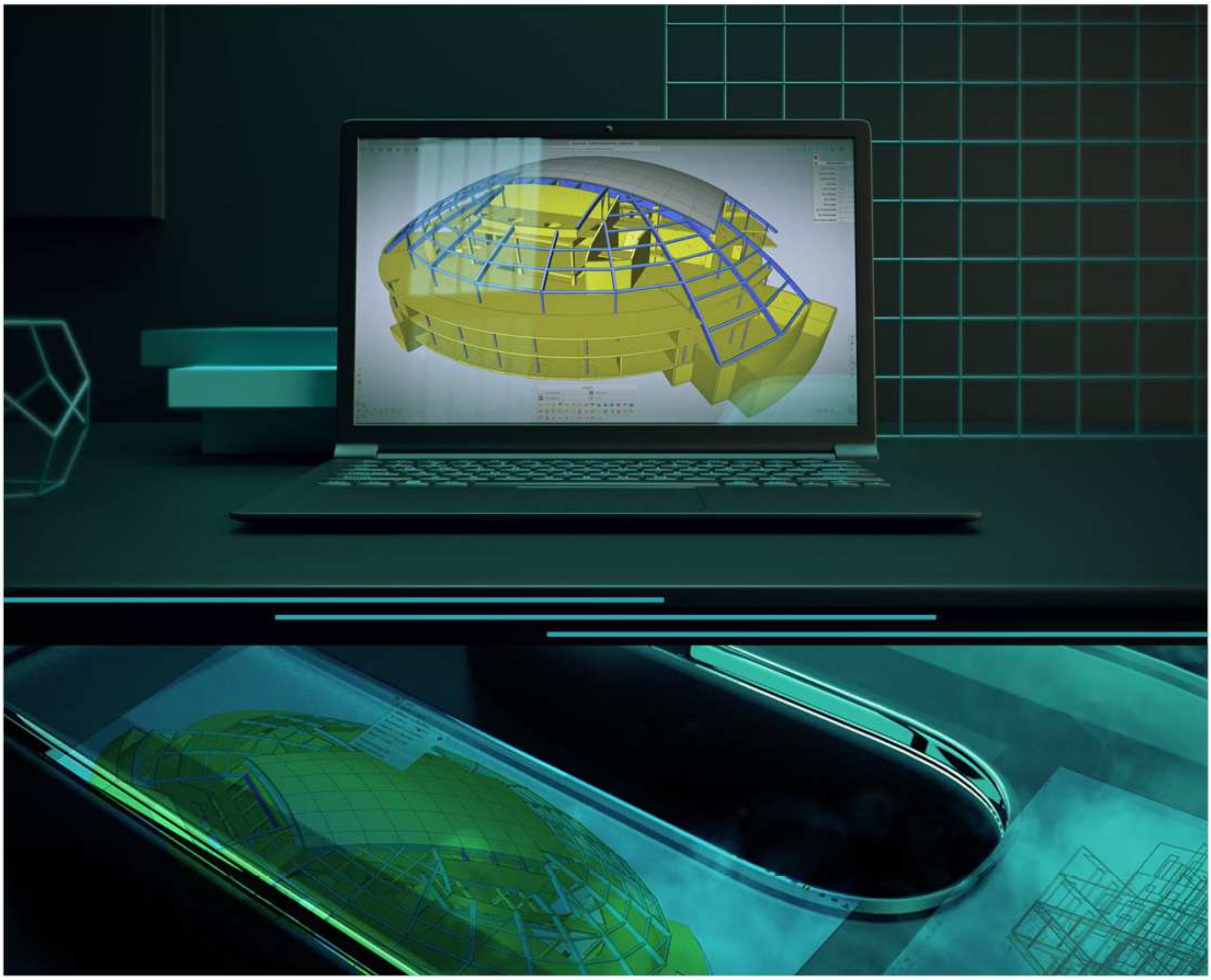


TUTORIAL

CABLES

Table of Contents

Introduction

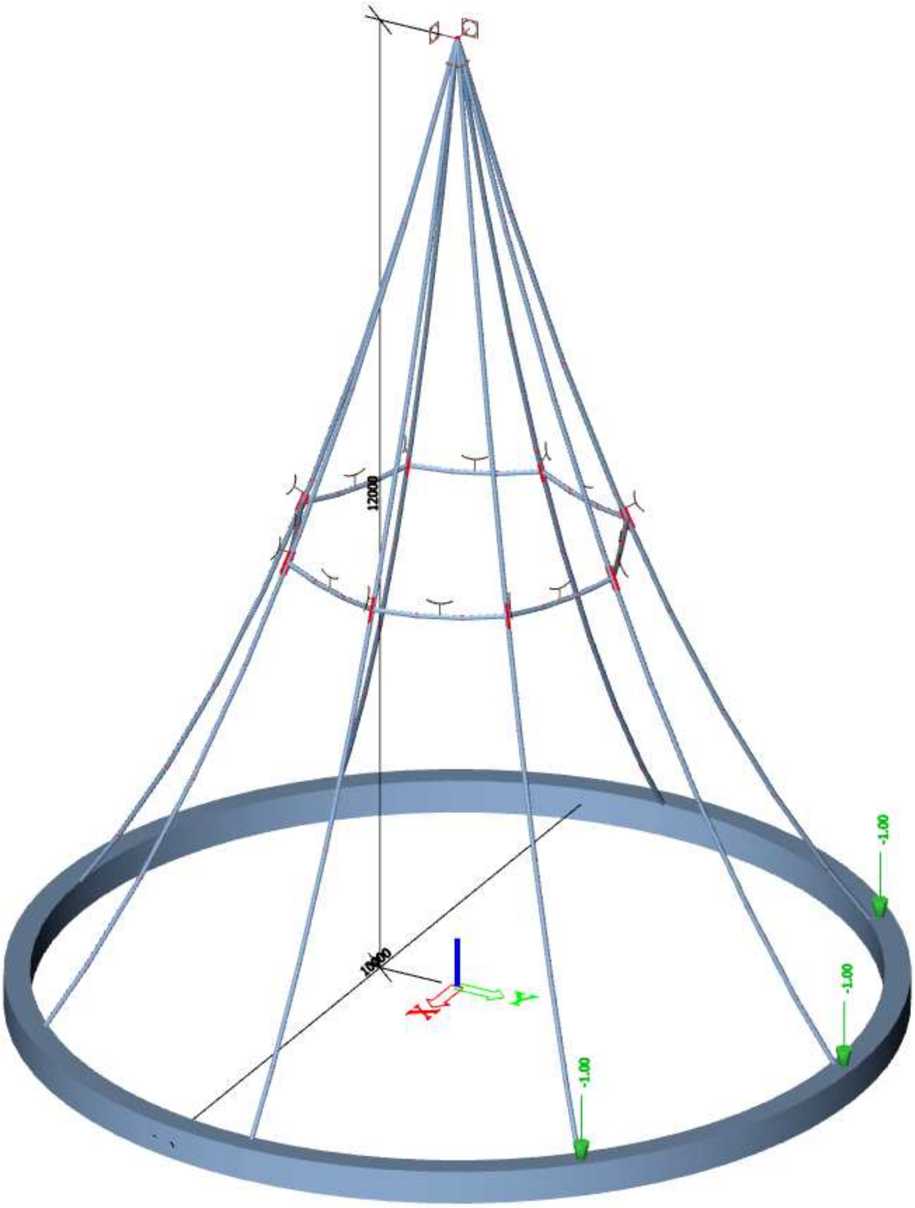
SCIA Engineer allows you to model a structure with cables.

First, we will discuss the theoretical background.

Next, we will explain how to model and load a chandelier structure.

Finally, we will run the (nonlinear) analysis and evaluate some results.

The figure below shows the model of the structure that we will design.



Chapter 1: Theoretical background

Since SCIA Engineer version 21.1, cables are fully supported in the 64bit version in the default post-processing environment. For cable analysis the module sens.03 is necessary. This module is included in the Expert and Ultimate edition.

A cable element is an element without bending stiffness (Iy and Iz ≅ 0). During the solving of the equations this behavior is taken into account, so only axial forces (in tension only) will occur and you will obtain zero values for the internal forces M_x, M_y, M_z, V_y and V_z. The displacements (in the intermediate nodes) have thus been calculated without bending stiffness.

The cable type is a special type of a beam nonlinearity, which can be applied on 1D members in a model.

You will need to create nonlinear combinations to take into account this nonlinear property during the analysis. A geometrical nonlinear analysis should be executed by calculating with a 3rd order analysis with the Newton-Raphson calculation method.

A curve formed by a hanging cable is called a "Catenary". With relatively small bending (height - sag roughly 10% of its length) the curve can be approximated by a parabola (this approach was implemented previously in the 32bit version of SCIA Engineer in the post-processing environment 'v16 and older'), but such approximation however becomes imprecise with greater sag / length ratio (and for different heights of the begin and end nodes of the cable).

The catenary curve is linearly approximated into a polyline. The approximation points (nodes) are calculated from the catenary equation defined by two edge points of the beam and its parameter ***a***, which is defined as a quotient (normal force / cable weight per unit length). The normal force is defined via the nonlinearity parameters and the cable weight per unit length is calculated from the cross-section and material parameters. Formula of the general catenary equation:

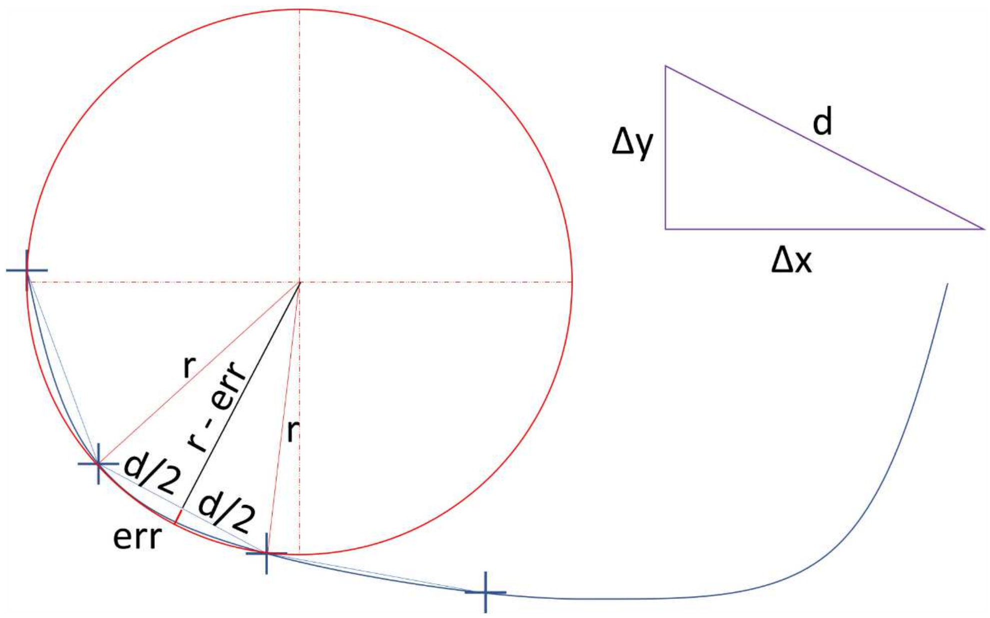
$$y = a \cdot \cosh\left(\frac{x-k}{a}\right) + c$$

Parameters ***k*** and ***c*** represent the horizontal and vertical shifts of the curve, respectively.

The relation between catenary length ***s***, vertical span of two definition points ***v*** and their horizontal span ***H*** is:

$$\sqrt{s^2 - v^2} = 2 \cdot a \cdot \sinh\left(\frac{H}{2a}\right)$$

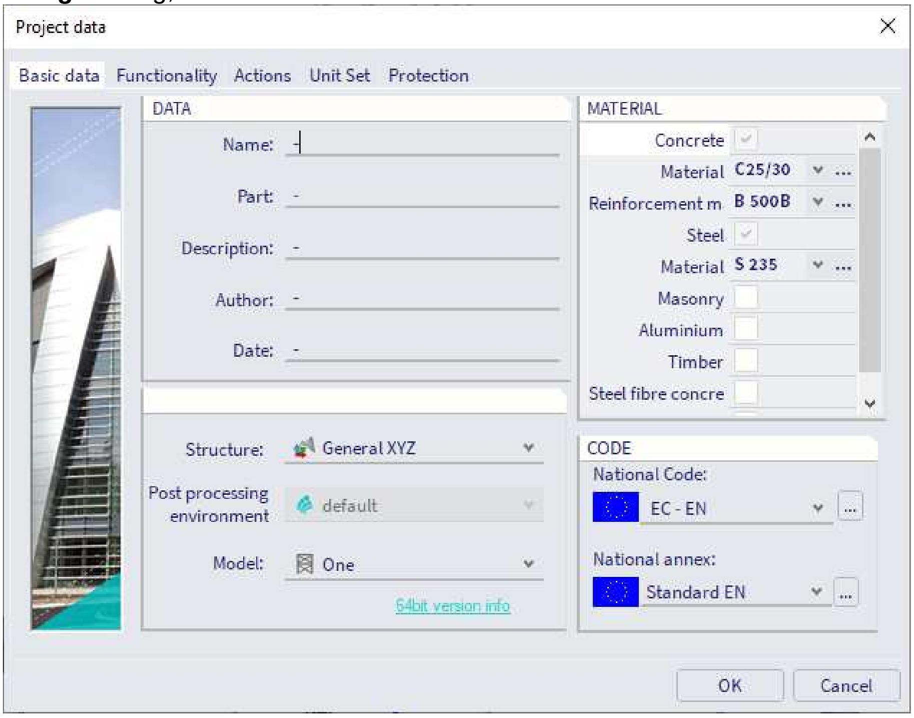
You have full control of the catenary curve approximation precision. A general curvature approach has been utilized. Using the curvature (parameter defining how much the curve bends at the given point) the radius of the circles (of the same curvature as the curve in the touching point) is calculated. Concept of the approximation error is graphically described in the figure below:



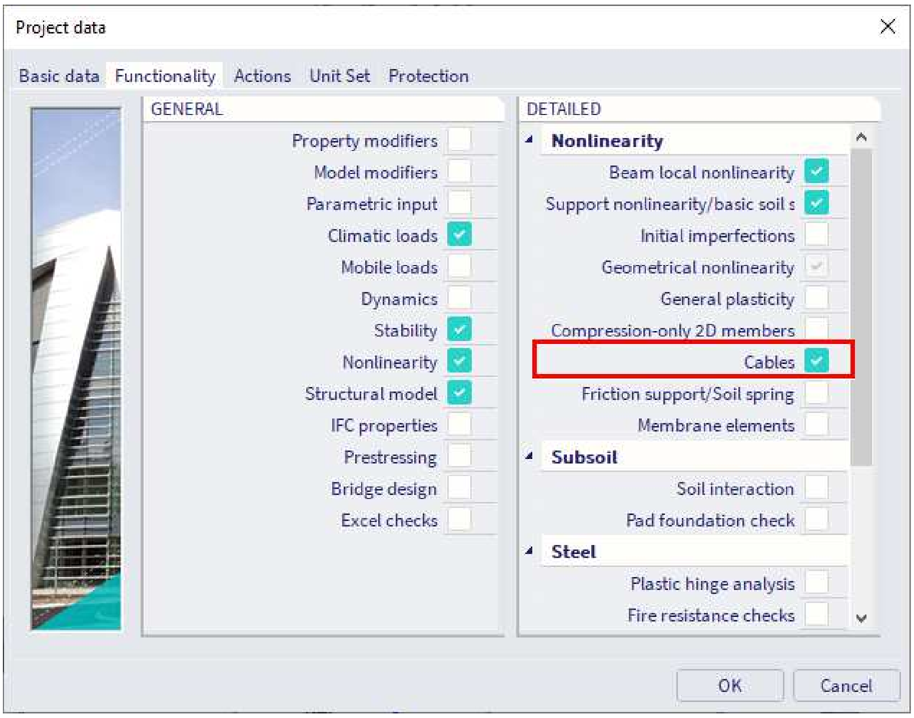
The detailed background (including formula to calculate the value of sag) can be found on our help-pages: https://help.scia.net/webhelplatest/en/#analysis/nonlinear_analysis/cables/cable.htm

Chapter 2: Modelling

In the **Project settings** dialog, activate the **Concrete** and **Steel** material.



On the **Functionality** tab activate the **Cables** functionality:



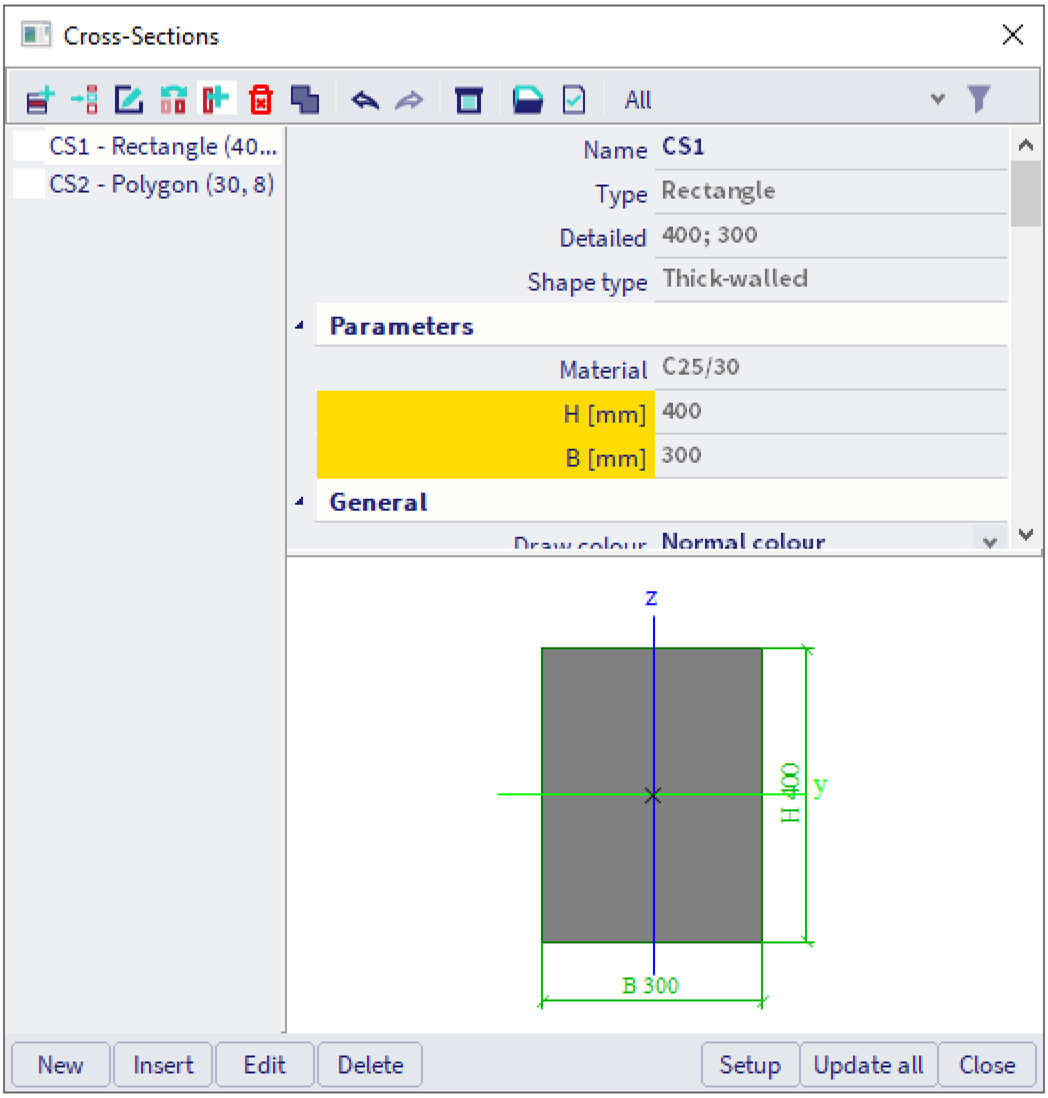
Note that for cable analysis the **module sens.03** is necessary. This module is included in the **Expert** and **Ultimate** edition.

When the Cables functionality is turned on, automatically the functionality **Geometrical nonlinearity** is turned on as well.

2.1. Cross-sections

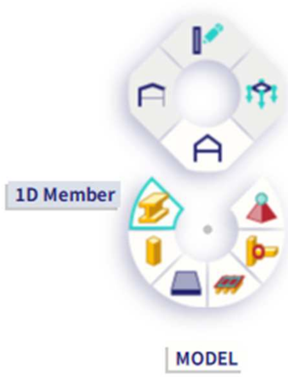
Go to **Menu bar > Libraries > Cross sections (**or with the icon in the **workstation Structure**).

Add a **rectangular** concrete cross-section of **400 x 300 mm** for the circular ring at the bottom of the chandelier. Add a **circular** shape or **polygon** of **diameter 30 mm** for the cable elements.



2.2. Geometry

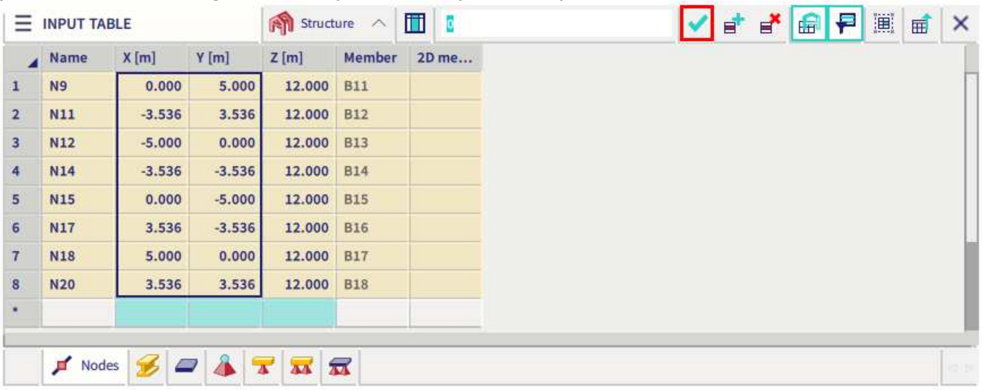
You can open the **Marking menu** by holding the **Alt** button together with the **right mouse click**. In the subgroup **Model** you can choose to add a **1D member**:



Draw the ring as two circular arcs with cross-section CS1.

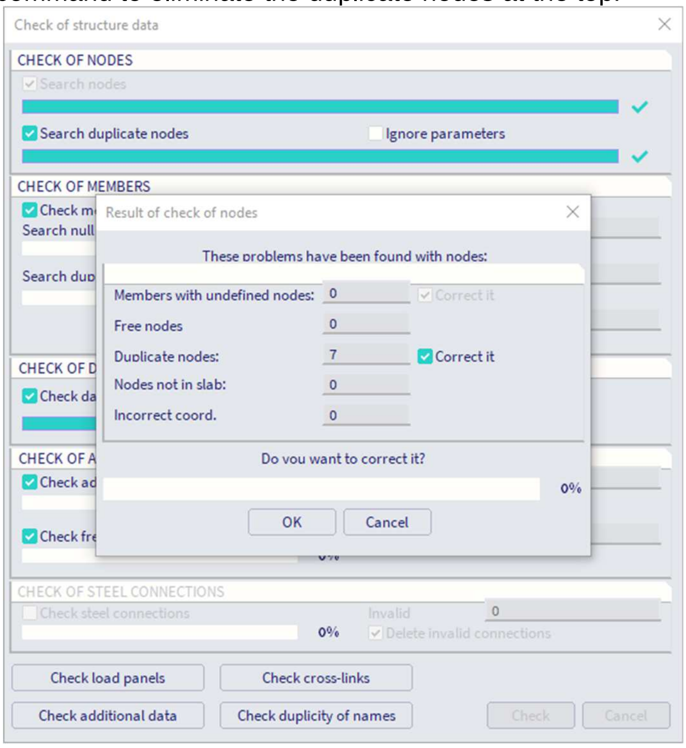
Then you can use the Marking menu again to add 8 **Column** elements with cross-section CS2 and length 12 m.

Then select the top nodes of the column elements. Change the X and Y coordinate of these nodes to 0 m via the property panel or via the **input table** (Tools > Input table):

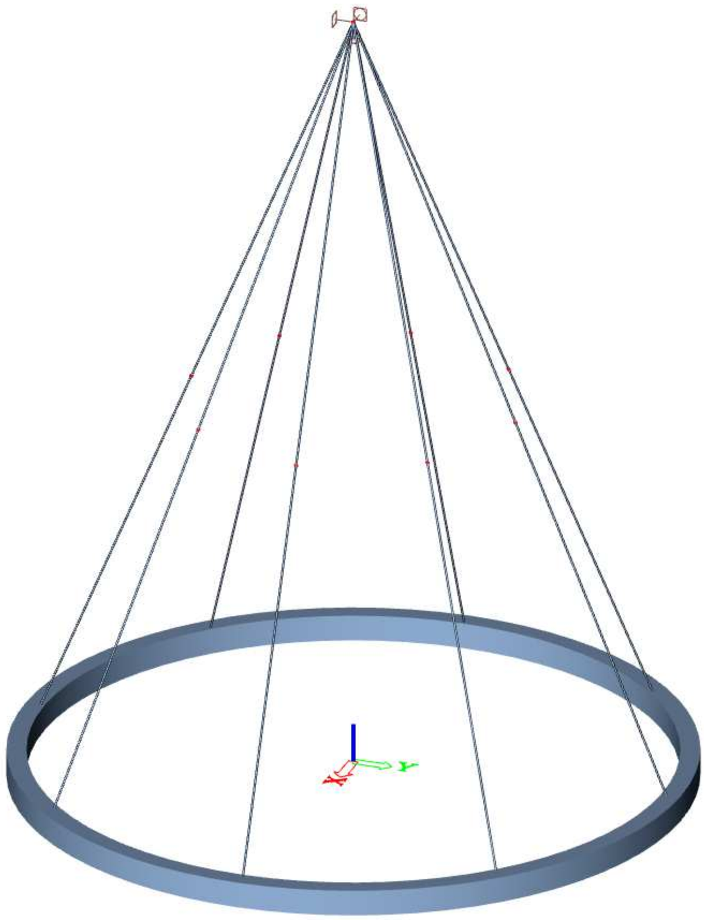


	Name	X [m]	Y [m]	Z [m]	Member	2D me...
1	N9	0.000	5.000	12.000	B11	
2	N11	-3.536	3.536	12.000	B12	
3	N12	-5.000	0.000	12.000	B13	
4	N14	-3.536	-3.536	12.000	B14	
5	N15	0.000	-5.000	12.000	B15	
6	N17	3.536	-3.536	12.000	B16	
7	N18	5.000	0.000	12.000	B17	
8	N20	3.536	3.536	12.000	B18	

Run the Check structure command to eliminate the duplicate nodes at the top:



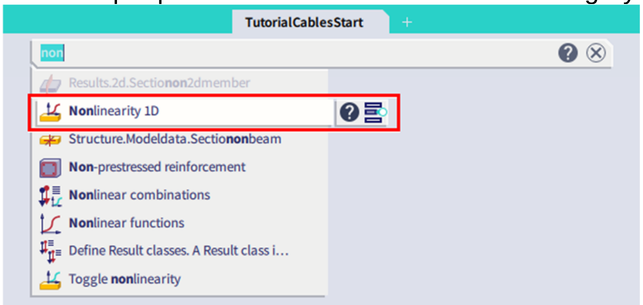
Now we have already following structure:

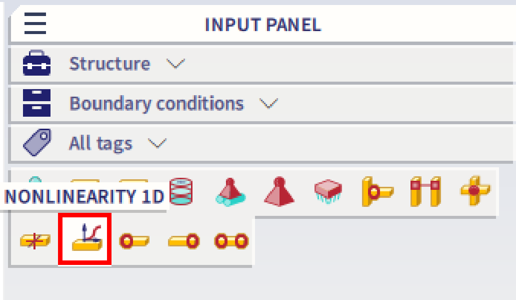


2.3. Boundary conditions

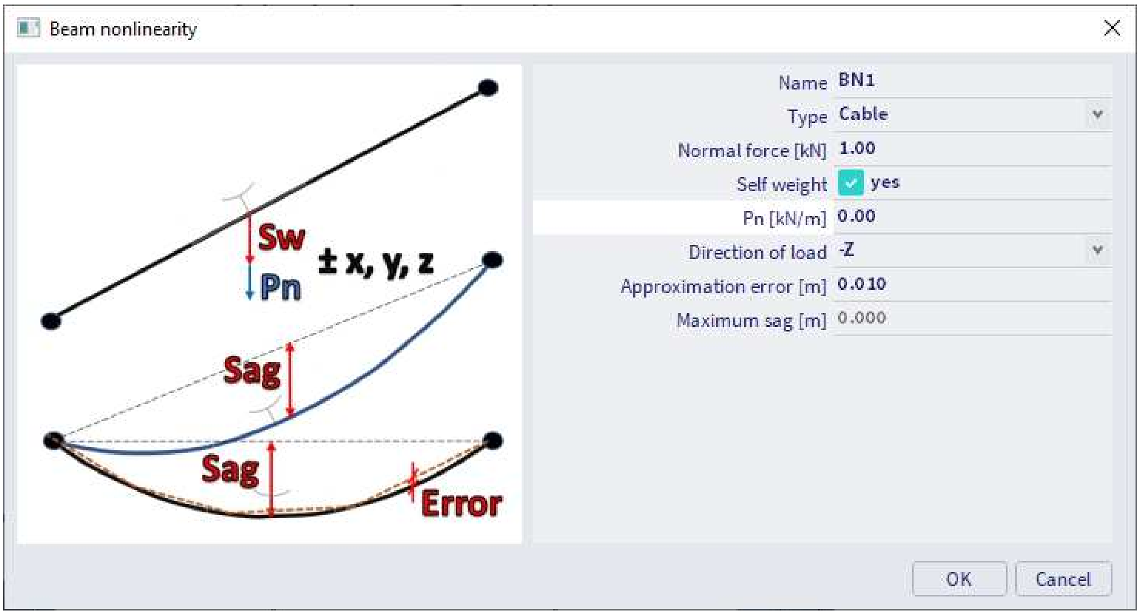
You can use the marking menu to quickly add a fixed support at the top.

Now you can apply the Nonlinearity 1D property to the elements. You can search for it via the SCIA Spotlight or find it via Input panel > workstation Structure > category Boundary conditions.

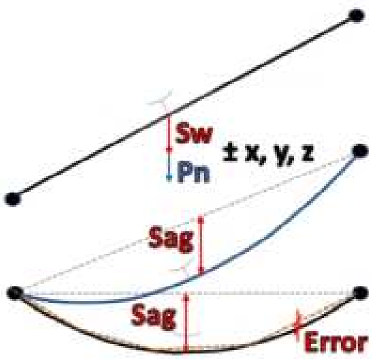




Following parameters are available for the type cable:



- Normal force [kN]: value of the pre-stressing axial force (positive)
- Self-weight: determines, whether the cable is subjected to self-weight
- Pn [kN/m]: specifies the value of the additional load (so other than self-weight)
- Direction of load: specifies the load direction (Pn and self-weight)
- Approximation error [m]: determines the maximal allowed distance between point on the analytically defined catenary curve and the projection of this point (in the direction of load) on the linearly approximated polyline of the cable.
- Maximum sag [m] automatically calculated value, it is the maximal distance (in the direction of load) between a node of the catenary curve and its projection on the straight line connecting both ends of the cable

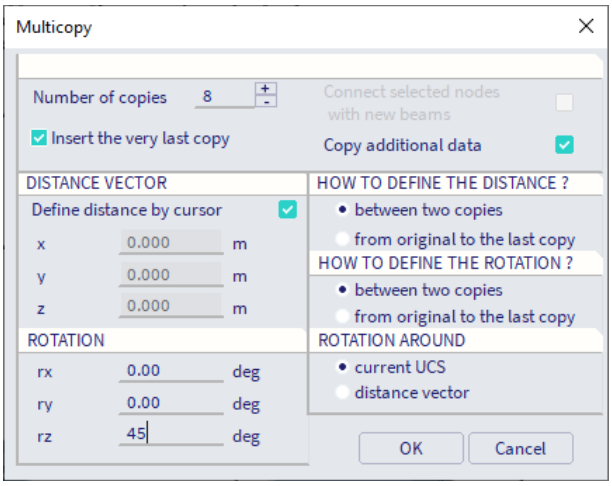


Set 1 kN as normal force, activate self-weight and set 0 kN/m as Pn.

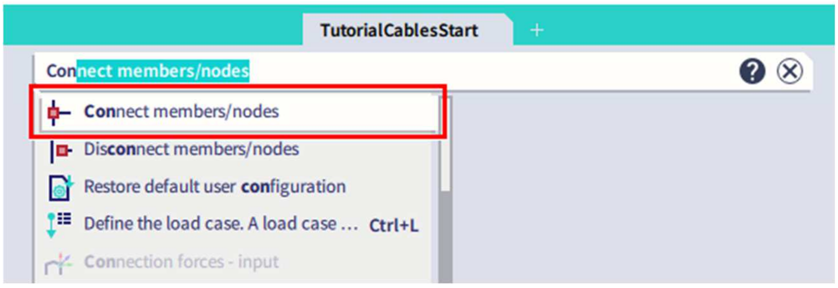
When applying the nonlinearity to the elements automatically the geometry is deformed according to the catenary curve which is calculated with the inputted parameters.

Add an element with cross-section CS1 in the middle of two vertical elements.

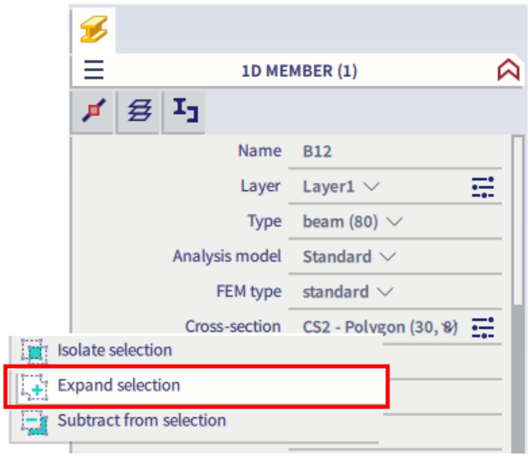
Via the subgroup Modify of the Marking menu you can select the Multicopy command and copy the horizontal element 8 times.



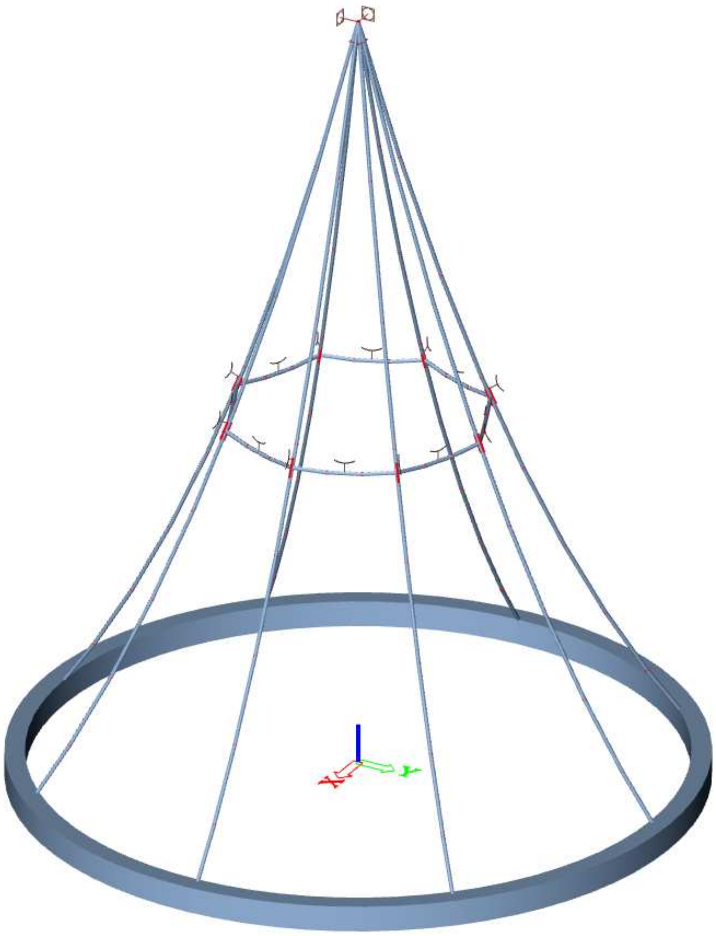
Connect the new elements with the existing vertical elements. When you now would change the nonlinearities (e.g. change the value of the normal force), it will have an effect on the elements with the nonlinearity, but as well on the connected elements.



Select all the new elements via the Expand selection function in the Property panel and add the Cable nonlinearity as well to the elements.

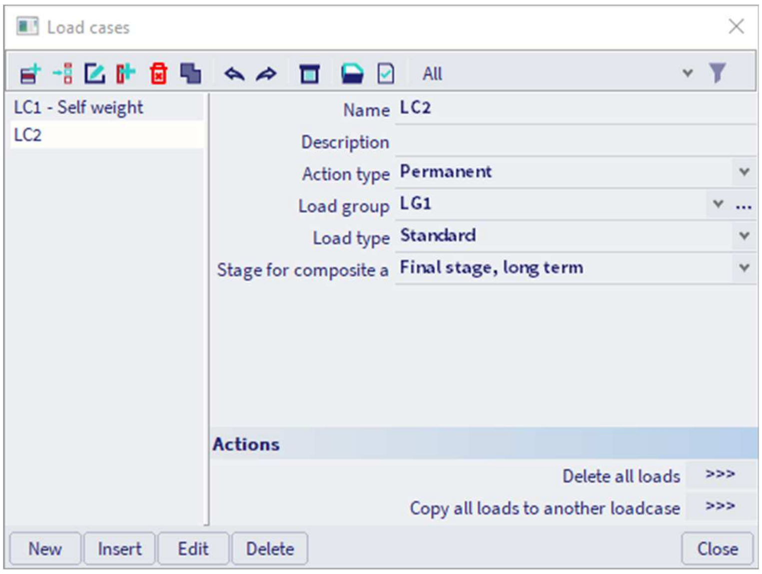


Now the modelling is finished:

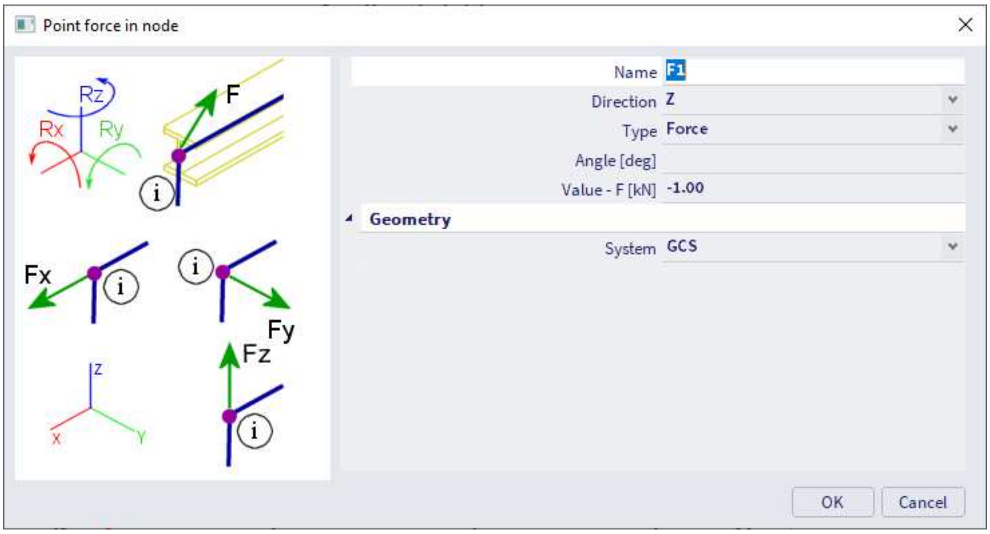


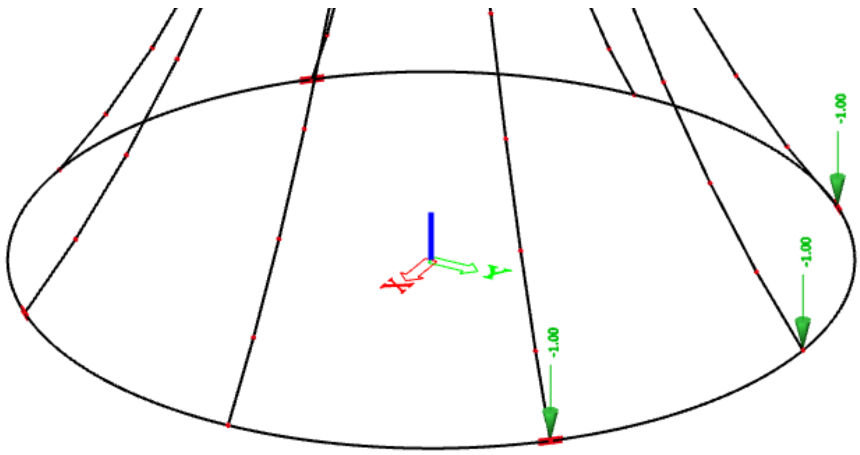
Chapter 3: Loads and combinations

Go to the load cases and create a new load case LC2:



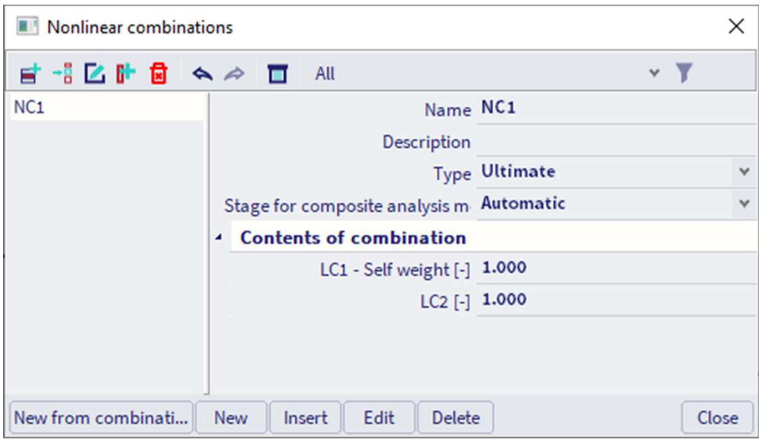
Via the subgroup Load of the Marking menu you can select the Point Load in Node command and add some point loads of 1 kN to the structure:





Open the nonlinear combinations:

And create a nonlinear combination NC1 with load cases LC1 and LC2:

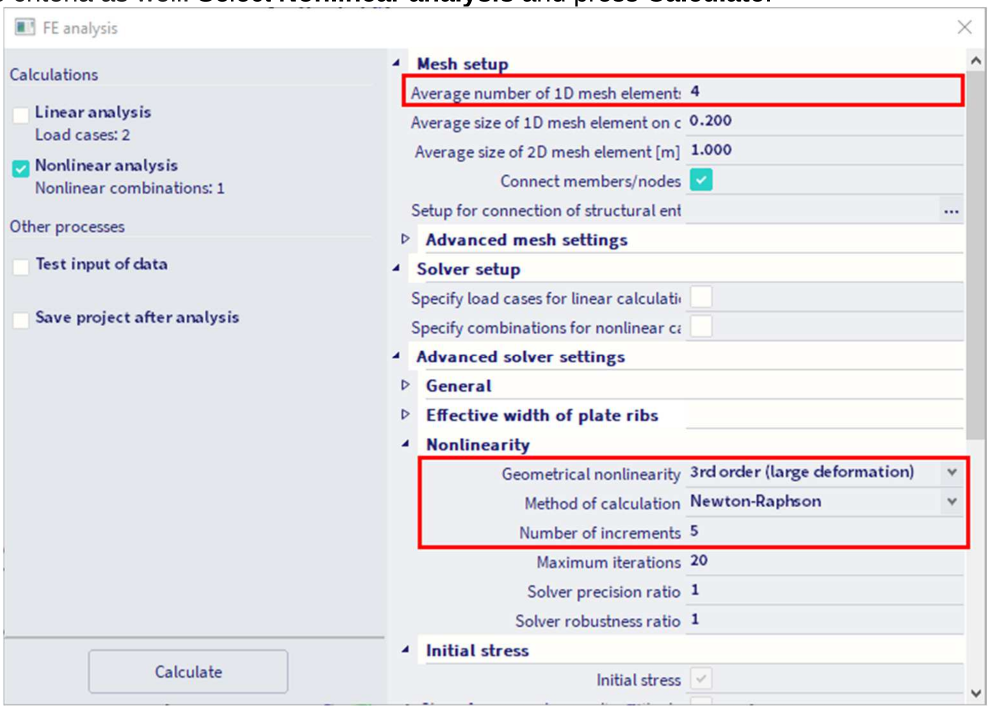


Chapter 4: Analysis

As the calculation model is completely ready, you now can start the calculation.

Use the **Calculate** button in the middle of the process wheel in the **process toolbar** to start the analysis (or via **Menu bar > Tools > Calculation & Mesh > Calculate**).

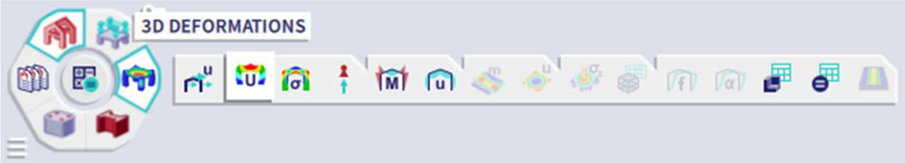
When starting the analysis, check that for the **Mesh setup** the **Average number of 1D mesh elements on straight 1D members** (linear approximation between the nodes of the catenary curve) and in the **Advanced solver settings Geometrical nonlinearity** is set to **3rd order (large deformations)**. If there are problems with convergence, increase the values of **Number of increments**, or **Maximum iterations**. Additionally, the **Solver precision ratio** or **Solver robustness ratio** might be altered to have an influence on the convergence criteria as well. Select **Nonlinear analysis** and press **Calculate**.



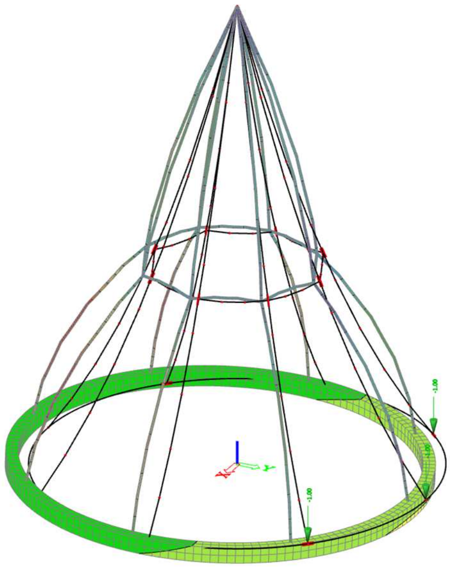
Chapter 5: Results

When the calculation is completed, results can be viewed. The relevant commands in the **workstation Results** become active now.

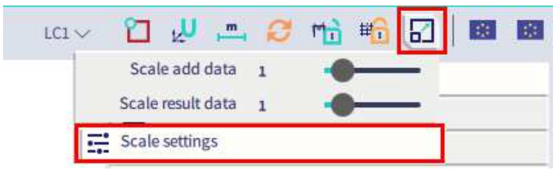
Select the **workstation Results** in the **process toolbar** and click on the command **3D deformations**.

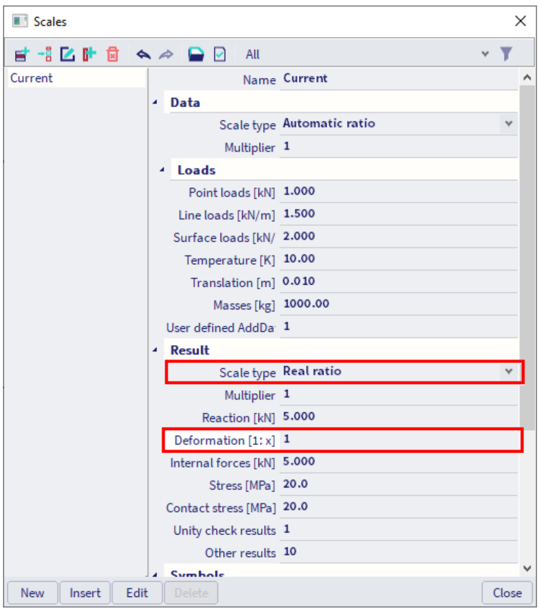


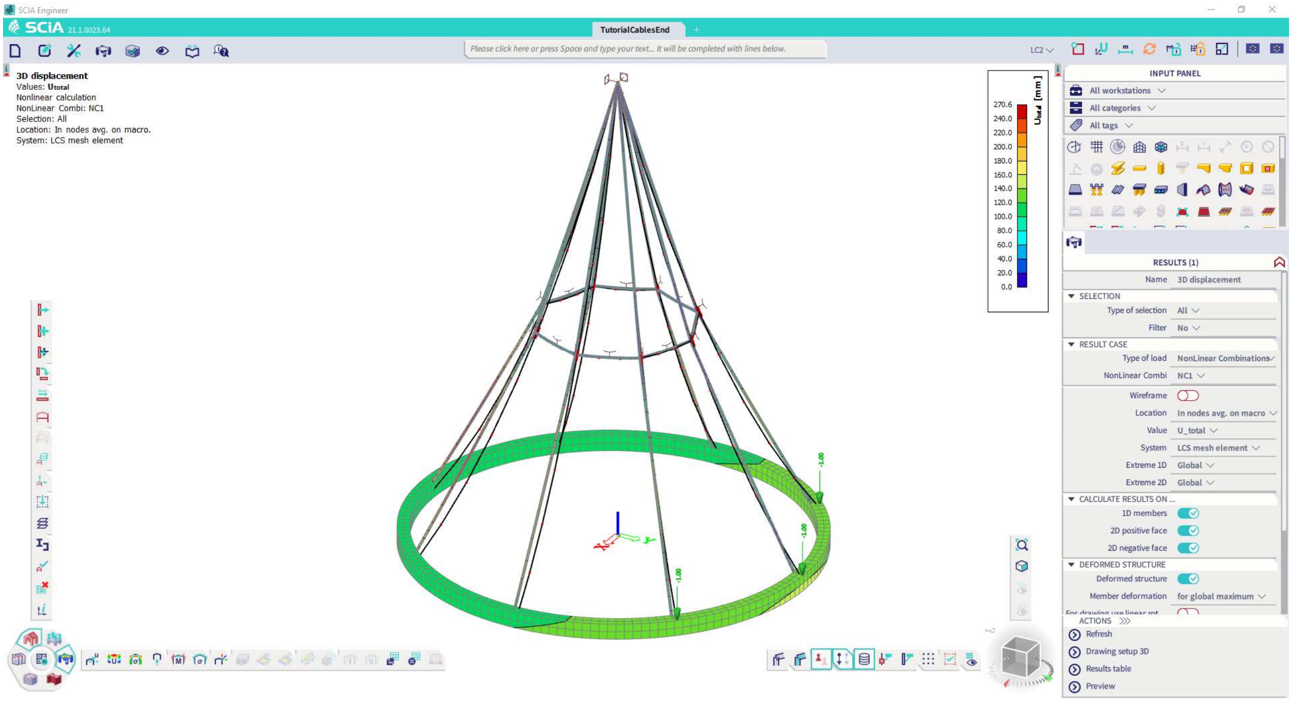
With the 3D deformations you can evaluate the behavior of the cables for the nonlinear combination NC1 and you will see that the cables go upwards. The graphical results look a bit strange because SCIA Engineer presents the biggest deformation as 1 m:



This makes sense for general structures, but for cables it's more convenient to look at the real ratio, which can be edited via the **Scale settings** in the **View bar**. Set **Scale type** to **Real ratio** and **Deformation [1:x]** to **1**.





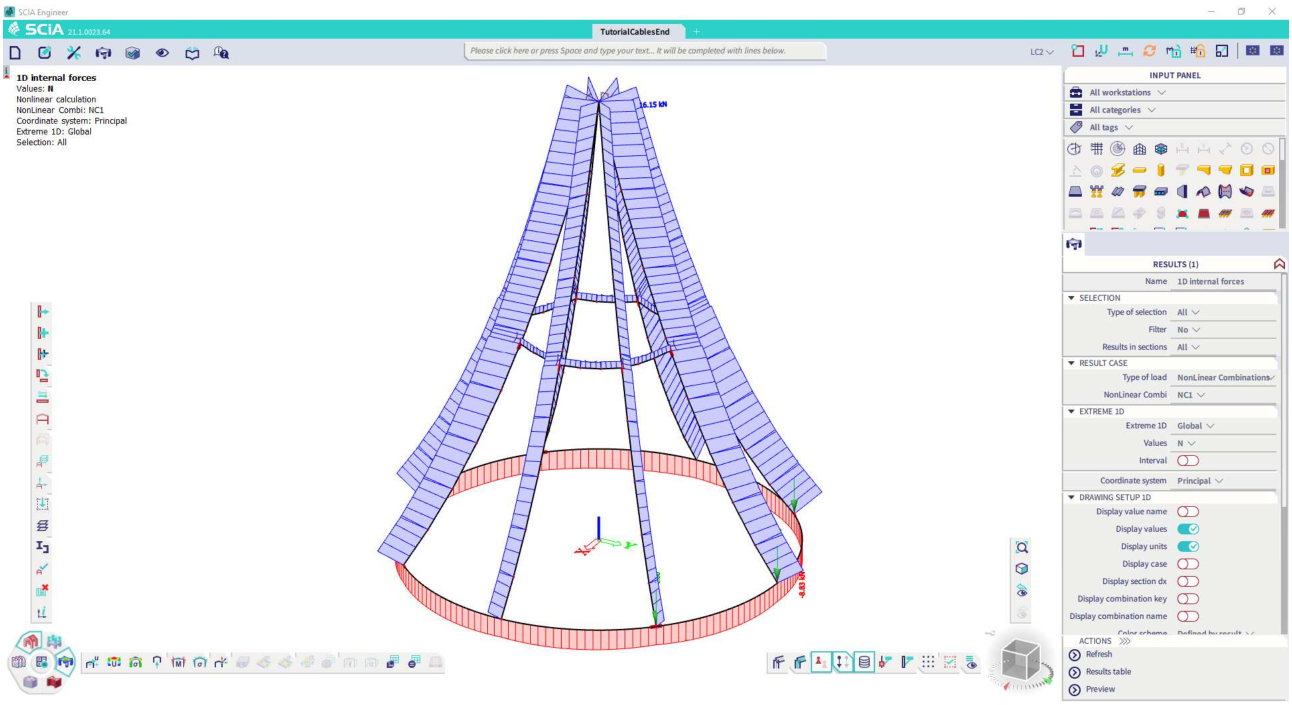


You can notice that the load straightens the curved elements.

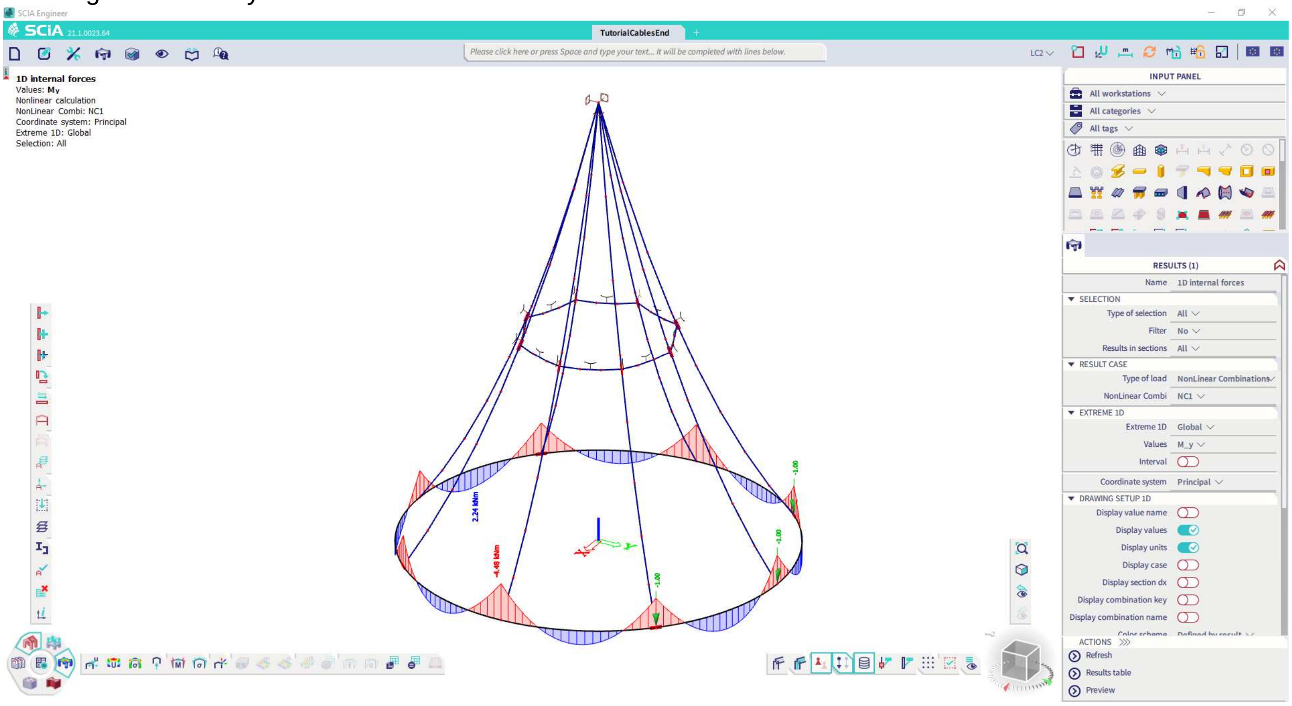
You can also have a look at the 1D internal forces such as normal forces N or bending moments My. Select the **workstation Results** in the **process toolbar** and click on the command **1D internal forces**.

For these results you can put the **Scale type** back to **Automatic ratio** in the Scale settings.

Normal forces N:



Bending moments My:



So in the cables, the only non-zero values of internal forces are for the normal forces N (tension). On the second picture we can see that there are no bending forces present in the cables

Notes:

- The input of the cable element only defines the initial shape / sag. Afterwards the cable can be loaded by real loads. The normal force after calculation will therefore be different to the inputted initial normal force N.
- Adding a cable into the model will adapt the mesh also for the linear calculations. This means that the linear calculation may become unstable when a cable is added, and a nonlinear calculation becomes necessary to obtain results.