

Settlement of a circular silo foundation

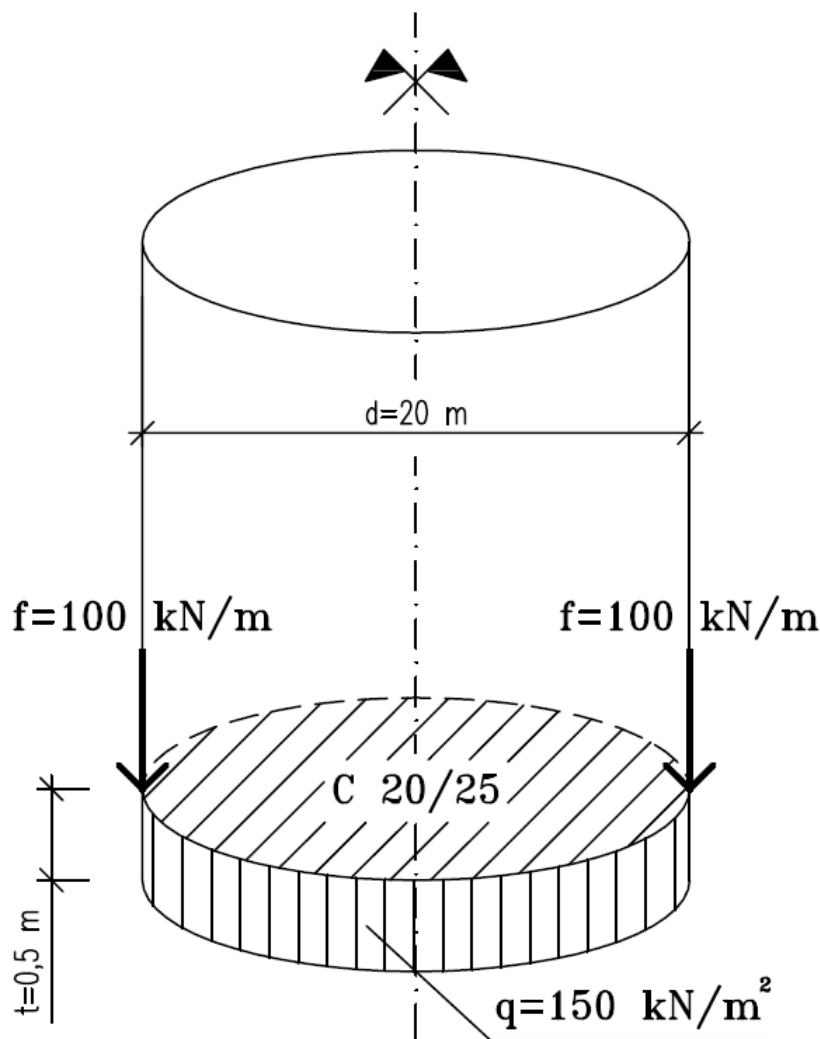
Program: FEM

File: Demo_manual_22.gmk

The objective of this manual is to describe the solution to a circular silo foundation settlement using the Finite Element Method and the Axial Symmetry module.

Task specification

Determine the settlement of a circular silo foundation (thickness of 0.5 m and diameter of 20.0 m) induced by the complete filling of the silo, i.e. the surcharge $q = 150 \text{ kPa}$. Further determine the total settlement of the silo after it is subsequently emptied. The geological profile, including parameters of soils, is identical with the profile described in the previous task (chapter 21. *Analysis of terrain settlement induced by strip surcharge loading*). Apply the **axial symmetry** to this particular case. The circular silo foundation is made from a matured reinforced concrete, class C 20/25.



Task specification chart – a circular silo foundation

In this case, the values of the total vertical deformation, i.e. settlement d_z [mm], will be derived from the Mohr-Coulomb material model only. The comparison of the other material models with various mesh densities was carried out in the previous chapter 21. *Analysis of terrain settlement induced by strip surcharge loading.*

Solution

The analysis will be performed using the GEO 5 – FEM program. The following paragraphs provide a step by step description of the solution:

- Topology: setting and modelling the problem (free points),
- Construction stage 1: primary geostatic stress,
- Construction stage 2: modelling and loading on beam elements, settlement analysis,
- Construction stage 3: unloaded terrain surface settlement (deformation) analysis, internal forces,
- Assessment of results: comparison, conclusion.

Note: To solve this task we shall represent the silo foundation made of reinforced concrete by beam elements free of contact elements thus assuming a perfect bond between the foundation and the soil. The issue of contact elements will be analysed in more detail in Chapter 24. Numerical solution to a sheeting wall structure.

Topology: problem settings

We will select the “Axial symmetry” option as the Project type in the “Settings” frame. The other data will remain the same.

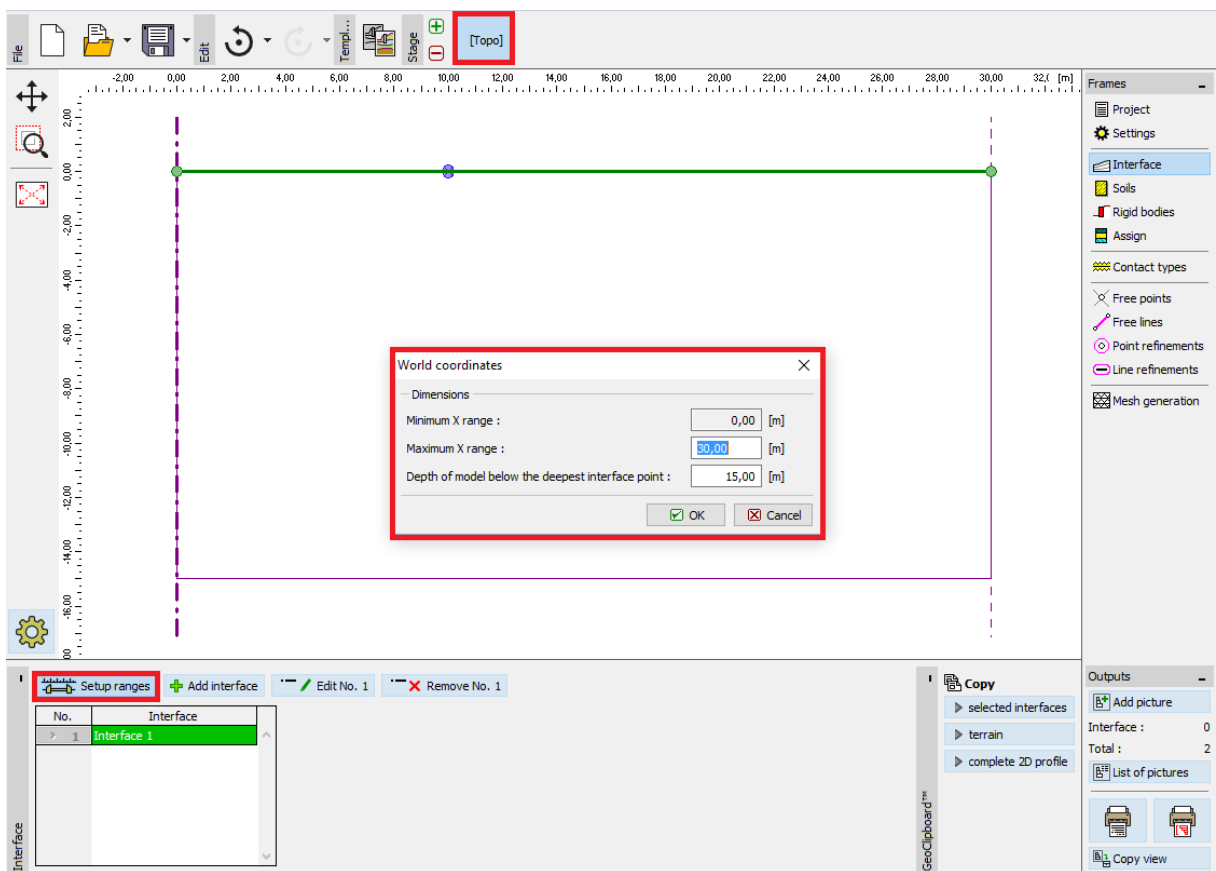
Project parameters Project type : Axial symmetry Analysis type : Stress <input type="checkbox"/> Allow to input water as the result of steady state water flow analysis	Design standards Concrete structures : EN 1992-1-1 (EC2) Calculation of geostatic stress (1st stage) Analysis method : Geostatic stress	Advanced program options <input type="checkbox"/> Advanced mesh generating parameters <input type="checkbox"/> Advanced soil parameters <input type="checkbox"/> Advanced soil models <input type="checkbox"/> Detailed results
--	--	---

“Settings” frame

Note: Axial symmetry is suitable for solving rotationally symmetric problems. This assumption must comply both with the geometrical arrangement of the structure and the loading. The solution to this problem – a circular silo foundation – is therefore a suitable example.

The solution is related to 1rad of the $x(r)$ -radius arch. The axis of symmetry always represents the $x(r)$ coordinate origin. Shear components of deformation in the direction of rotation can be disregarded. The evolution of a circumferential normal component of stress and strain (hoop stress and strain) is also taken into consideration, in addition to the components of stress and strain in the cross-sectional plane (for more details visit Help – F1).

In the “Interface” frame, we will first set the new world coordinates. Then we will set the coordinates of the first point of the interface to [10, 0]. The next point of the interface (at the edges) will be added by the program automatically.



“Interface” frame + “World coordinates” dialogue window

Then we will define the parameters of soils and assign them to the interface region No. 1. In this particular case neither rigid bodies nor contact types will be considered.

Add new soils
✕

Identification

Name :

Material model ?

Material model :

Basic data ?

Unit weight : $\gamma =$ [kN/m³]

Elastic modulus : $E =$ [MPa]

Stiffness according to depth :

Poisson's ratio : $\nu =$ [-]

Uplift pressure ?

Calc. mode of uplift :

Saturated unit weight : $\gamma_{sat} =$ [kN/m³]

Model Mohr - Coulomb ?

Modulus unloading / reloading : $E_{ur} =$ [MPa]

Angle of internal friction : $\varphi_{ef} =$ [°]

Cohesion of soil : $c_{ef} =$ [kPa]

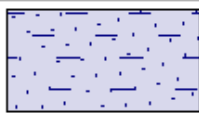
Dilation angle : $\psi =$ [°]

Draw

Color

Pattern category

Pattern



Clayey sand

Classify

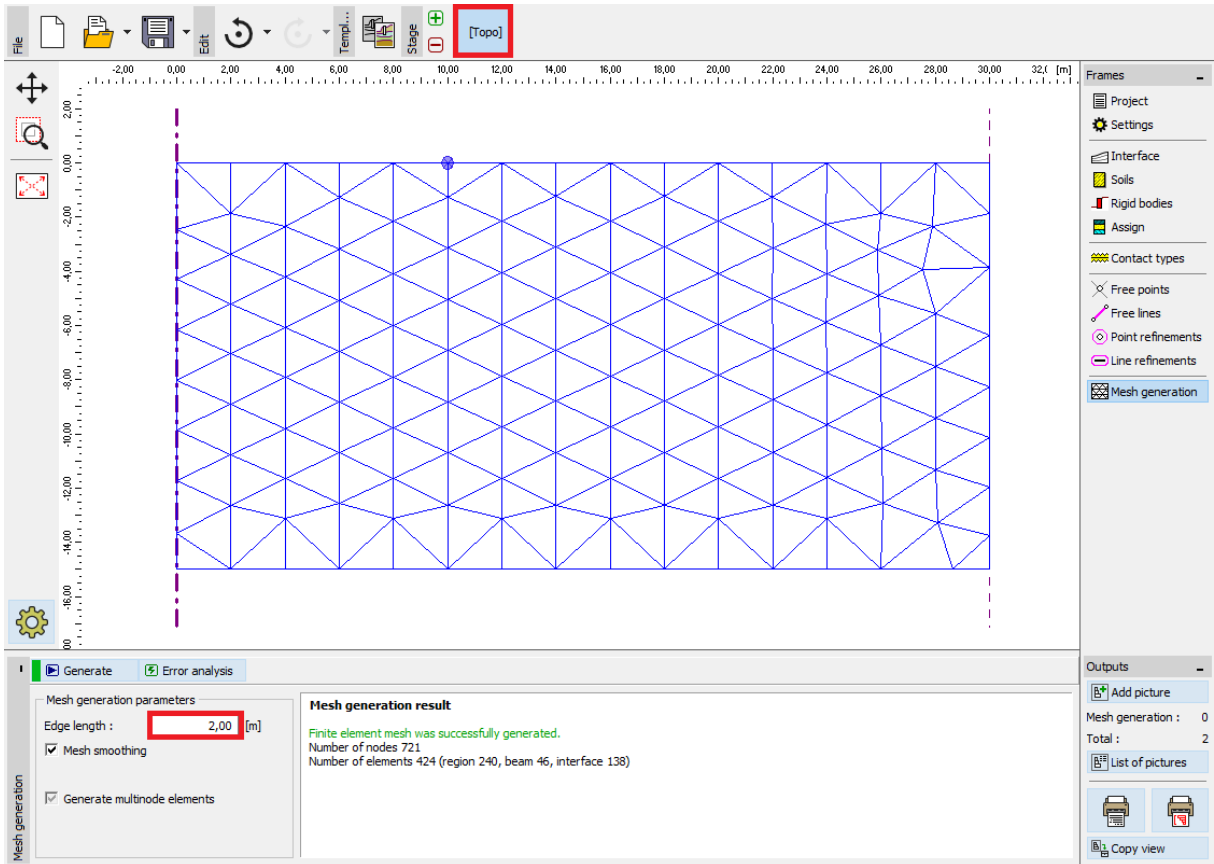
Clear

+ Add

✕ Cancel

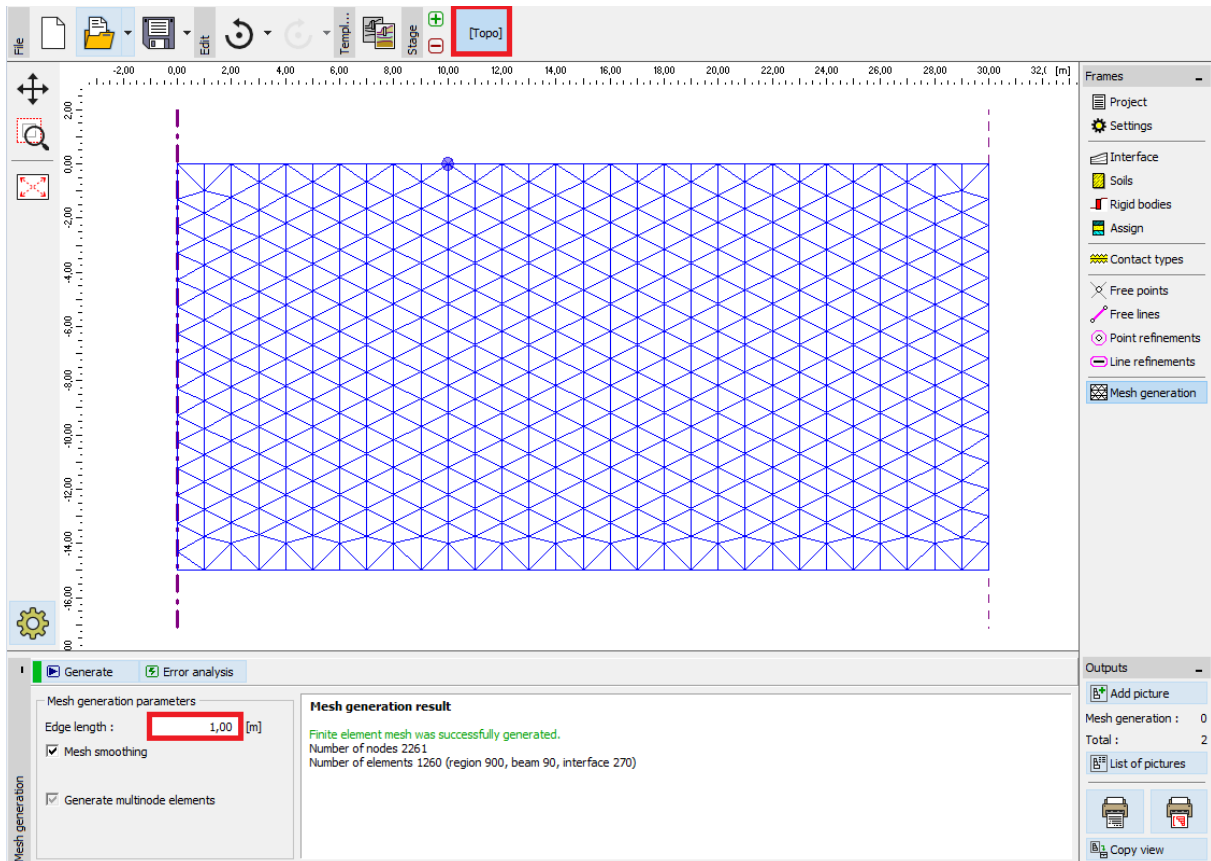
"Add new soils" dialog window

For the mesh generation, we will first set the elements edge length to 2.0 m.



“Mesh generation” frame – Triangular mesh with the element edge length of 2.0 m

Upon examining the generated mesh, we may conclude that for a given problem the resulting mesh is too coarse. For that reason, we will change the length of the edges of the mesh elements to 1.0 m.

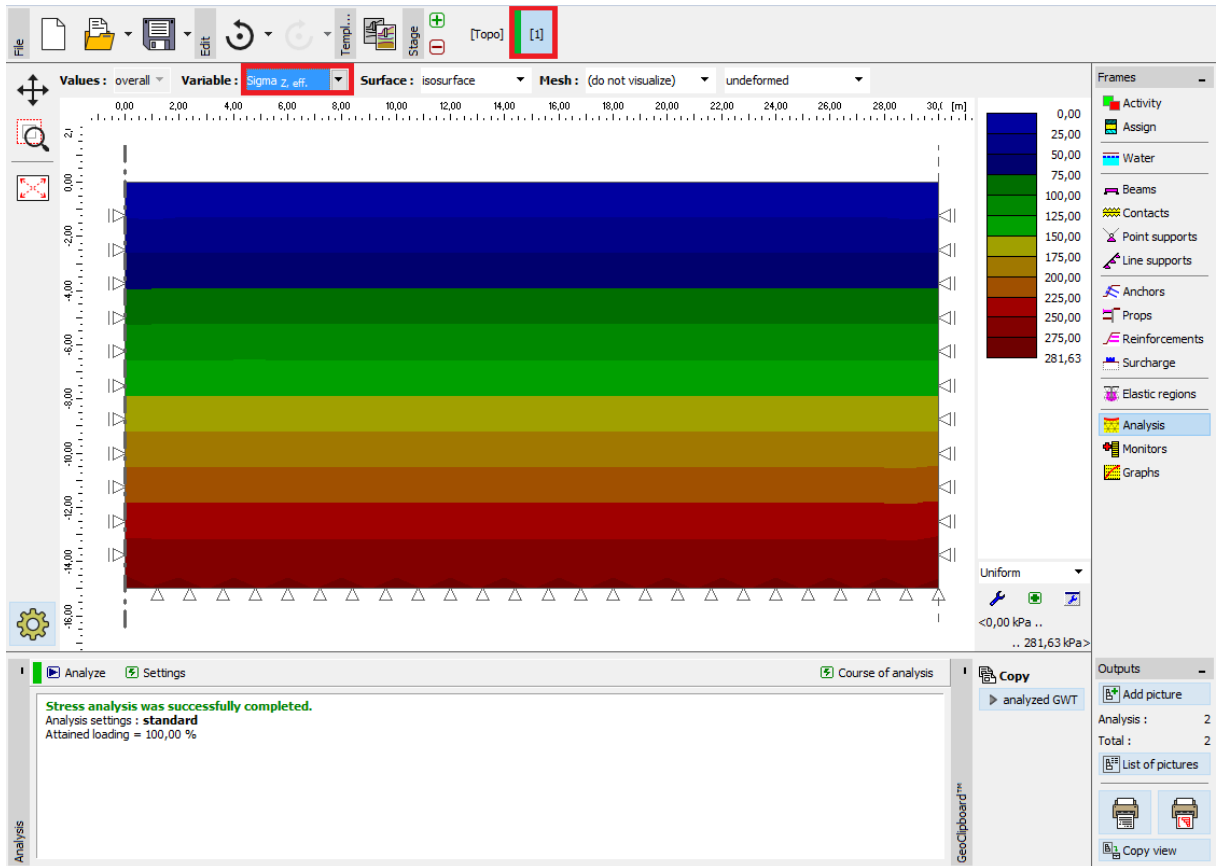


“Mesh generation” frame – Triangular mesh with the element edge length of 1.0 m

Note: It would be reasonable to refine the density of the mesh using line refinement for the area under the circular silo foundation that is being solved (for more details visit Help – F1). We will describe this function in more detail in the following chapter 23. Collector lining analysis.

Construction stage 1: primary geostatic stress

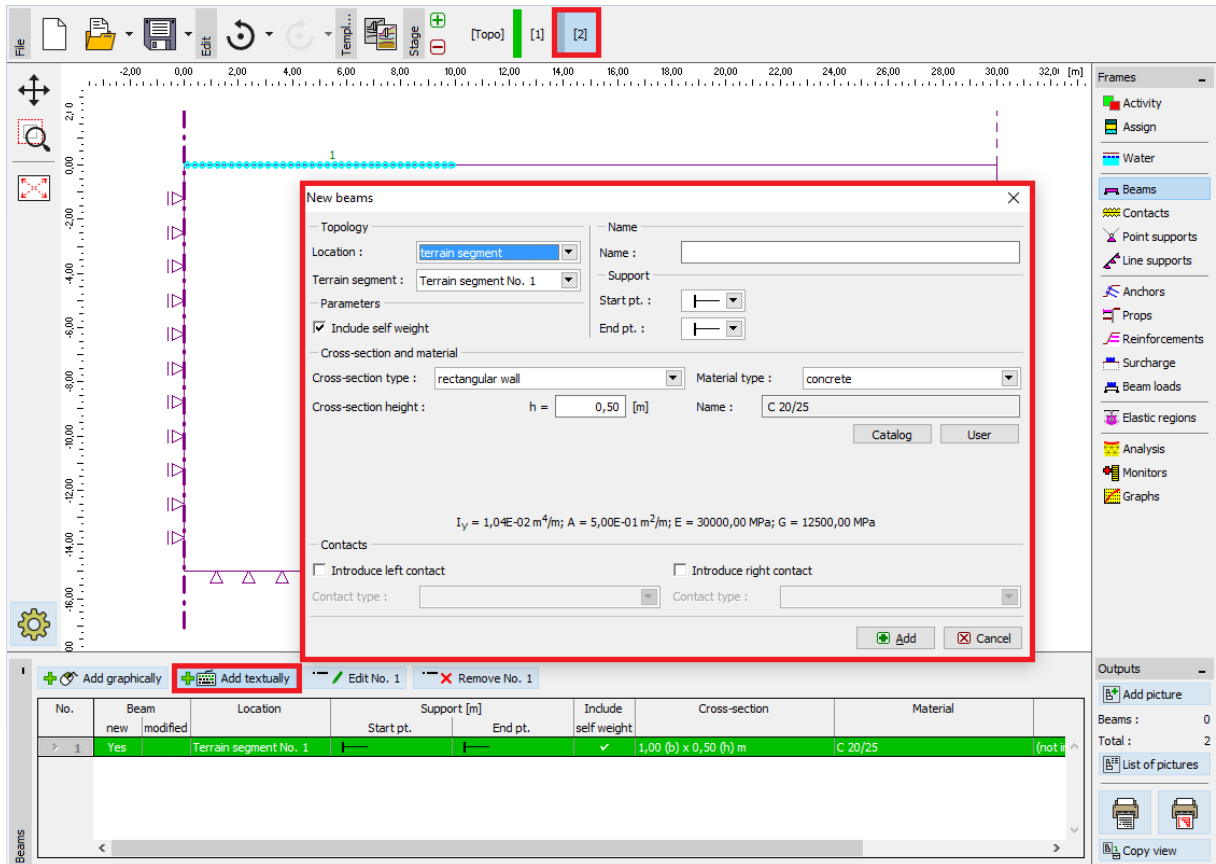
After generating the FE mesh, we will switch over to construction stage 1 and carry out the primary geostatic stress analysis. We will leave the analysis settings as “Standard” (for more details visit Help – F1).



“Analysis” frame – Construction stage 1

Construction stage 2: modelling of and loading on beam elements

In the next step we will add construction stage 2. Then, in the “Beams” frame, we will define the following parameters: beam location, material and concrete class, cross-section height (0.5m) and beam end supports (for more details visit Help – F1).



“New beams” dialogue window – Construction stage 2

Subsequently, we will proceed to the “Beam loads” frame, where we will set the load magnitude $f = 100 \text{ kN/m}$; we will consider it as the weight of the circular silo walls acting on the foundation.

The screenshot displays the GEO5 software interface. A dialog box titled "New beam loads" is open, showing the following settings:

- Loaded beam: Beam No. 1
- Type of load: concentrated force
- Direction: perpendicular to beam
- Angle: $\alpha = 0,00$ [°]
- Point of application: x = 10,00 [m]
- Load magnitude: f = 100,00 [kN/m]

Below the dialog box, the "Beam loads" table is visible, containing the following data:

No.	Beam load new change	Beam	Type of load	Direction	Angle α [°]	Origin x [m]	Length l [m]	Magni f, m, q, q1
1	Yes	Beam No. 1	concentrated force	perpendicular to beam	0,00	10,00		100,00

“New beam loads” dialogue window – loads induced by walls, acting on the circular foundation

Further we will set the value of the uniform continuous load to $q = 150 \text{ kN/m}^2$, representing the filling of the circular silo, acting on its bottom or the upper edge of the foundation.

The screenshot shows the GEO5 software interface. A dialog box titled "New beam loads" is open, with a red border. The dialog box contains the following fields:

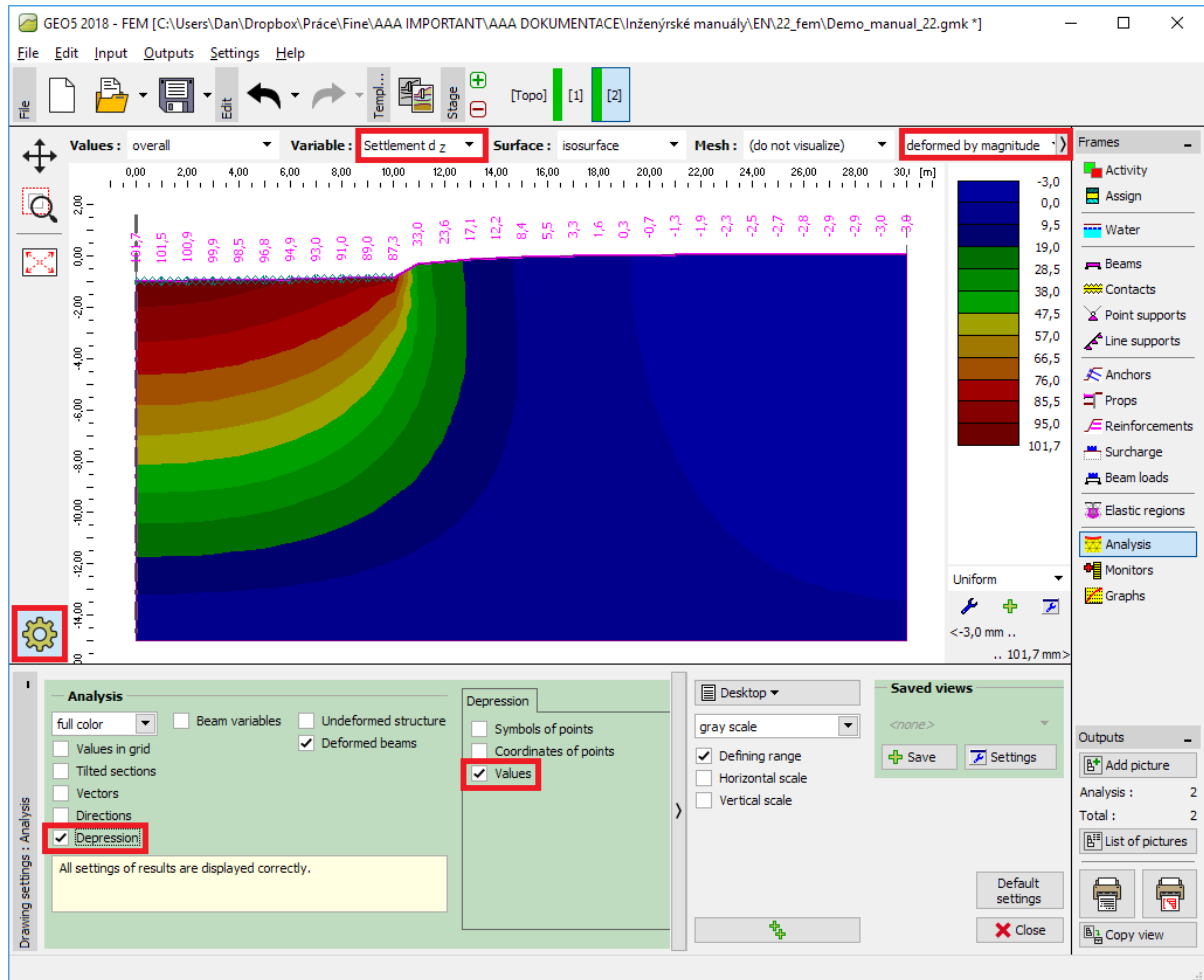
- Loaded beam: Beam No. 1
- Location: Beam No. 1
- Type of load: distr. uniform on the whole beam
- Direction: perpendicular to beam
- Angle: $\alpha = 0,00$ [°]
- Load magnitude: Magnitude: $q = 150,00$ [kN/m²]

Below the dialog box, the "Beam loads" table is visible. The table has the following columns: No., Beam load new change, Beam, Type of load, Direction, Angle α [°], Origin x [m], Length l [m], and Magni f, m, q, q₁. The table contains two rows:

No.	Beam load new change	Beam	Type of load	Direction	Angle α [°]	Origin x [m]	Length l [m]	Magni f, m, q, q ₁
1	Yes	Beam No. 1	concentrated force	perpendicular to beam	0,00	10,00		100,00
2	Yes	Beam No. 1	distr. uniform on the whole beam	perpendicular to beam	0,00			150,00

“New beam loads” dialogue window – load on the circular foundation induced by filling the silo

At this stage, we will again carry out the analysis and will examine the results for settlement d_z [mm]. It follows from the picture that the maximum vertical displacement is 102.0 mm. To better understand the structure behaviour, we will visualise the deformed mesh (the button in the upper part of the screen).



“Analysis” frame – Construction stage 2 (vertical deformation d_z and the depression)

We will click on the “Settings” button and check options “Depression” and “Values” in the “Depression” flag (for more details visit Help – F1).

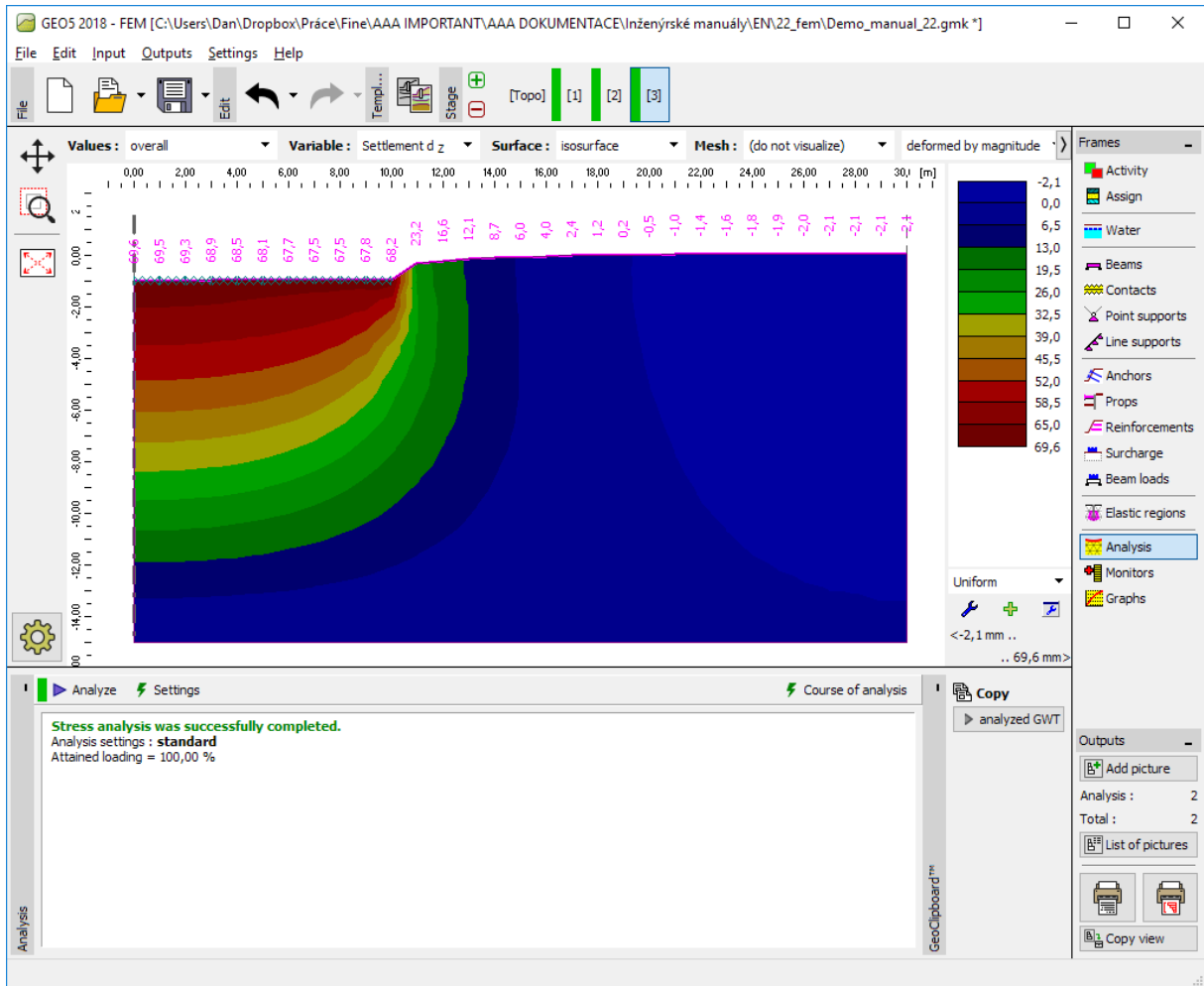
Construction stage 3: unloaded terrain surface settlement, internal forces

In the next step we will add construction stage 3. At this construction stage we will remove the uniform continuous load. Further on we will take into consideration only the beam loading induced by the circular silo walls, which is identical to that determined in the previous construction stage, i.e. $f = 100 \text{ kN/m}$.

No.	Beam load new	Beam load change	Beam	Type of load	Direction	Angle α [°]	Origin x [m]	Length l [m]	f, m, q, q_1	Magni
1	No	No	Beam No. 1	concentrated force	perpendicular to beam	0,00	10,00		100,00	

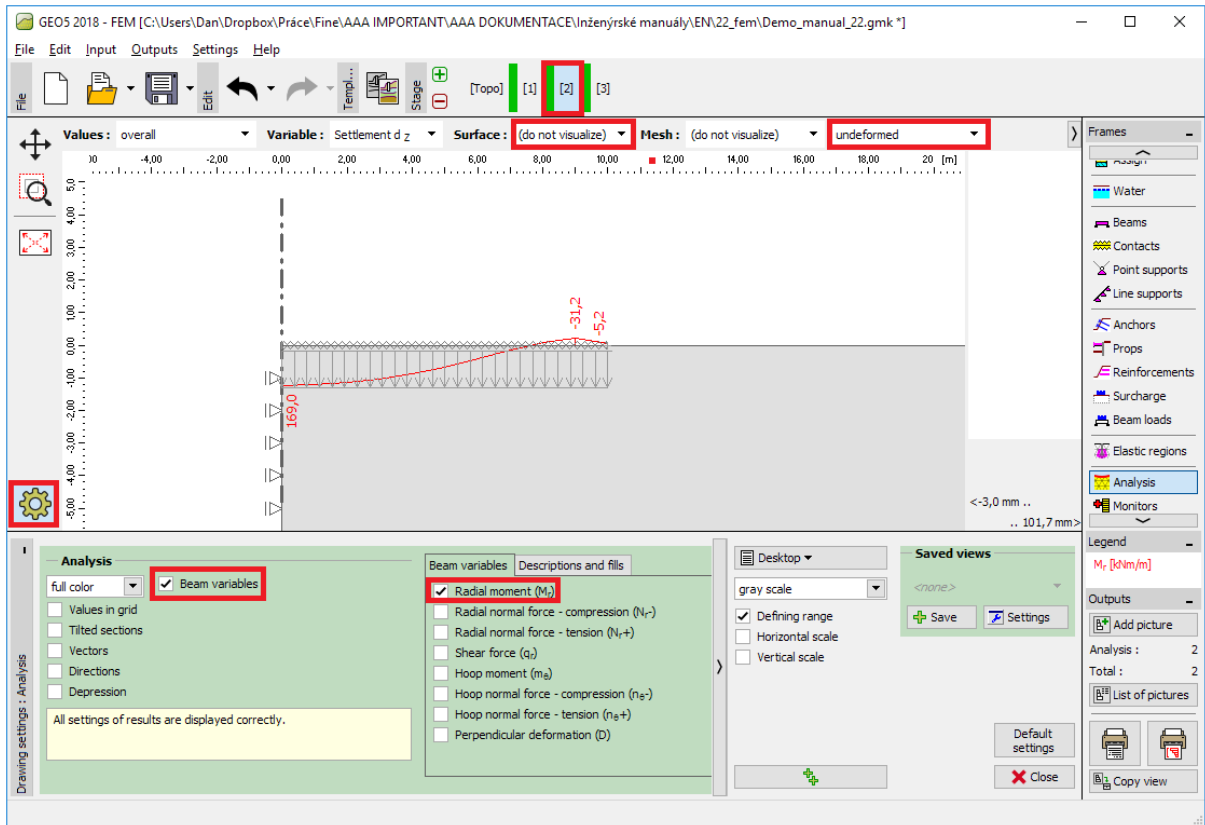
“Beam loads” frame – Construction stage 3

Then we will repeat the analysis and determine the values of displacements. The total settlement value after the terrain surface unloading d_z is 69.6 mm.

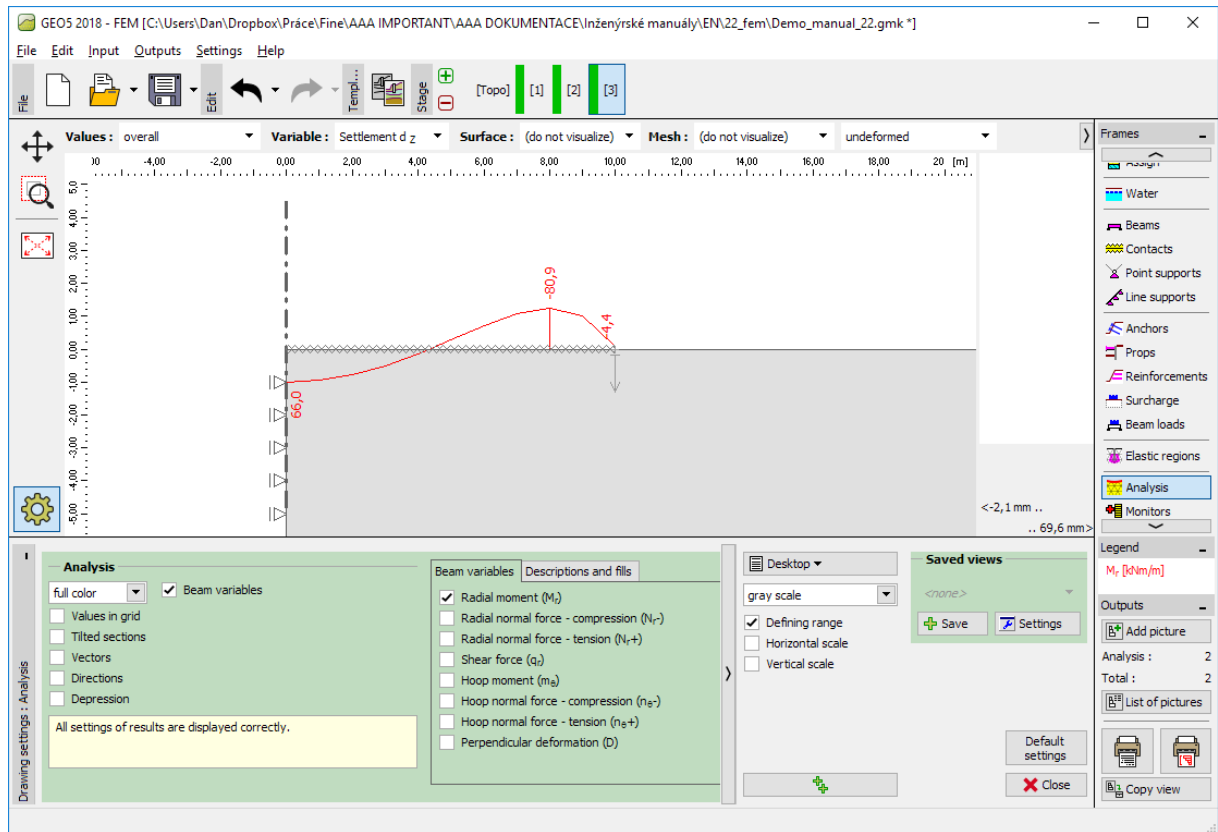


“Analysis” frame – Construction stage 3 (vertical deformation d_z with the settlement trough)

Now we will examine the radial moment diagrams M_r [kNm/m] for construction stages 2 or 3 (using the “Settings” button in the “Distributions” flag) and will record the magnitudes of local extremes in a table. The main structural reinforcement of the circular silo foundation can be designed and assessed for these values in an arbitrary static program (e.g. FIN EC – CONCRETE 2D).



“Analysis frame – Construction stage 2 (variation of radial moment M_r)



“Analysis” frame – Construction stage 3 (variation of radial moment M_r)

Assessment of results

The following table presents the results for the total settlement d_z [mm] and radial moments M_r [kNm/m] for construction stages 2 and 3, in which we modelled the loading or unloading of the circular silo foundation. We carried out the analysis for the Mohr-Coulomb material model with the triangular element mesh having an edge length equal to 1.0 m.

Material Model	Stage 2	Stage 3	Stage 2	Stage 3
	d_z [mm]	d_z [mm]	M_r [kNm/m]	M_r [kNm/m]
Mohr-Coulomb (1.0 m)	101.7	69.6	+ 169.0	+ 66
			- 31.2	-80.9

Results for total settlement d_z and radial moments M_r for individual construction stages

Conclusion

Several following conclusions can be drawn from the results for the quantities being examined:

- When the silo is full (as a result of the action of a uniform continuous load), a positive bending moment prevails along the beam length, where its bottom fibres are stretched.
- After emptying the silo (as a result of subsequent unloading), the circular foundation is loaded only by the silo walls. A negative bending moment prevails along the beam length, where its upper fibres are stretched.