

Collector lining analysis

Program: FEM

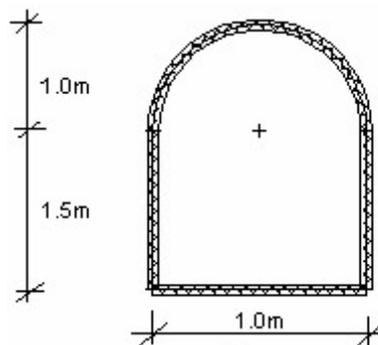
File: Demo_manual_23.gmk

The objective of this engineering manual is to perform the analysis of a mined collector with emphasis on the lining response using the Finite Element Method.

Task specification

Determine the lining response of a mined collector; its dimensions can be seen in the following chart. Determine the internal forces acting on the collector lining. The collector lining (0.1 m thick) is made from reinforced concrete, class C 20/25, the bottom is at a depth of 12.0 m. The geological profile is homogeneous; the soil parameters are as follows:

- Unit weight of soil: $\gamma = 20.0 \text{ kN/m}^3$
- Modulus of elasticity: $E = 12.0 \text{ MPa}$
- Poisson's ratio: $\nu = 0.40$
- Effective soil cohesion: $c_{ef} = 12.0 \text{ kPa}$
- Effective angle of internal friction: $\phi_{ef} = 21.0^\circ$
- Unit weight of saturated soil: $\gamma_{sat} = 22.0 \text{ kN/m}^3$



Task specification chart – mined collector

We will determine the values of displacements and internal forces only for the elastic model because we do not expect the development of plastic deformations. We will subsequently use the Mohr-Coulomb material model for the verification of the yield condition.

Solution

To analyse this task, we will use the GEO 5 – FEM program. The following paragraphs provide a step by step description of the solution:

- Topology: setting and modelling the problem (interface, free points and lines – refining the density)
- Construction stage 1: primary geostatic stress,
- Construction stage 2: modelling beam elements, analysing displacements, internal forces
- Assessment of results: comparison, conclusion.

Topology: problem settings

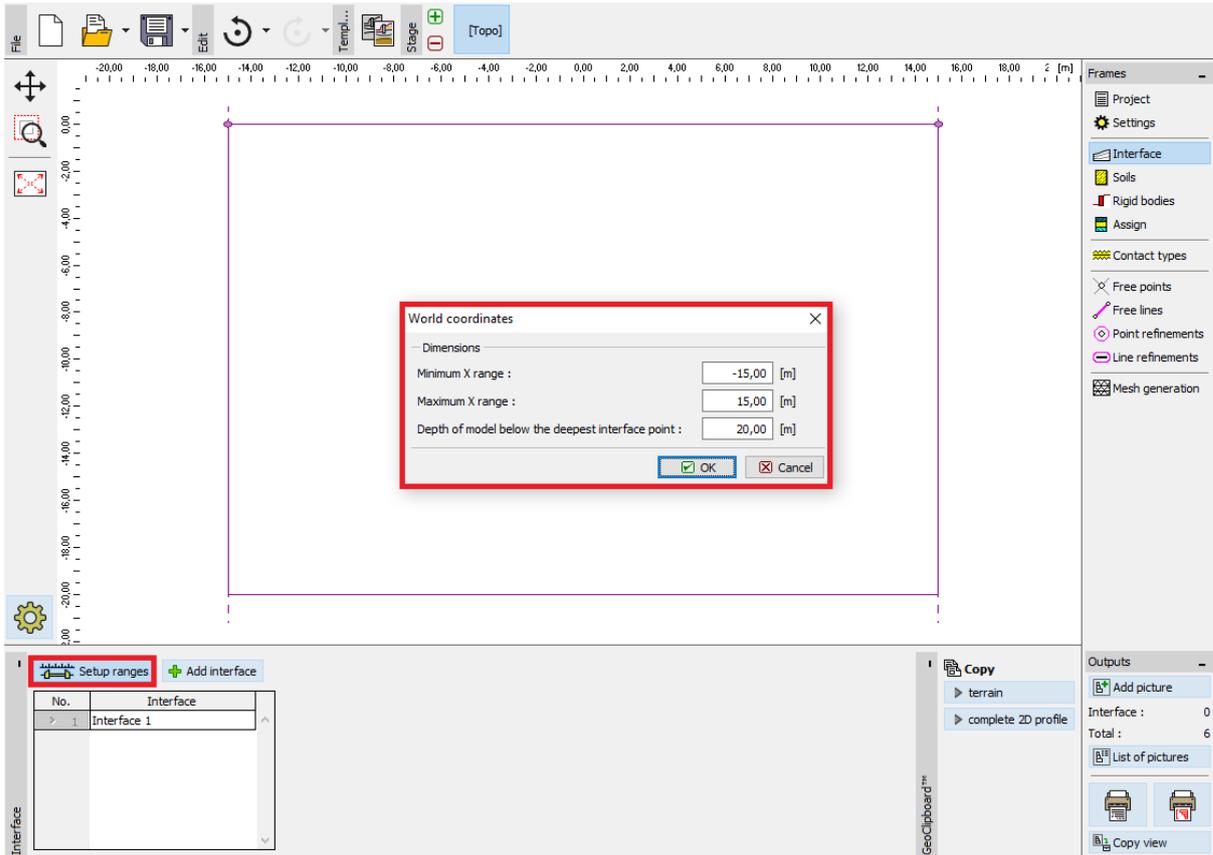
In the “Settings” frame we will select the option “geostatic stress” to perform the analysis of the construction stage 1. We will consider the problem or the analysis type as *plane strain*.

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

“Settings” frame

Further on, we will set the world coordinates and the terrain interface. We will choose sufficiently large world dimensions so that the results are not affected by the conditions at the edge. For our particular problem we will choose model dimensions $(-15\text{ m}; 15\text{ m})$; we will set the depth of the layer to be examined to 20.0 m.

Then we add interface in the terrain level – coordinates of points (x, z) : $[-15, 0]; [15, 0]$ [m]

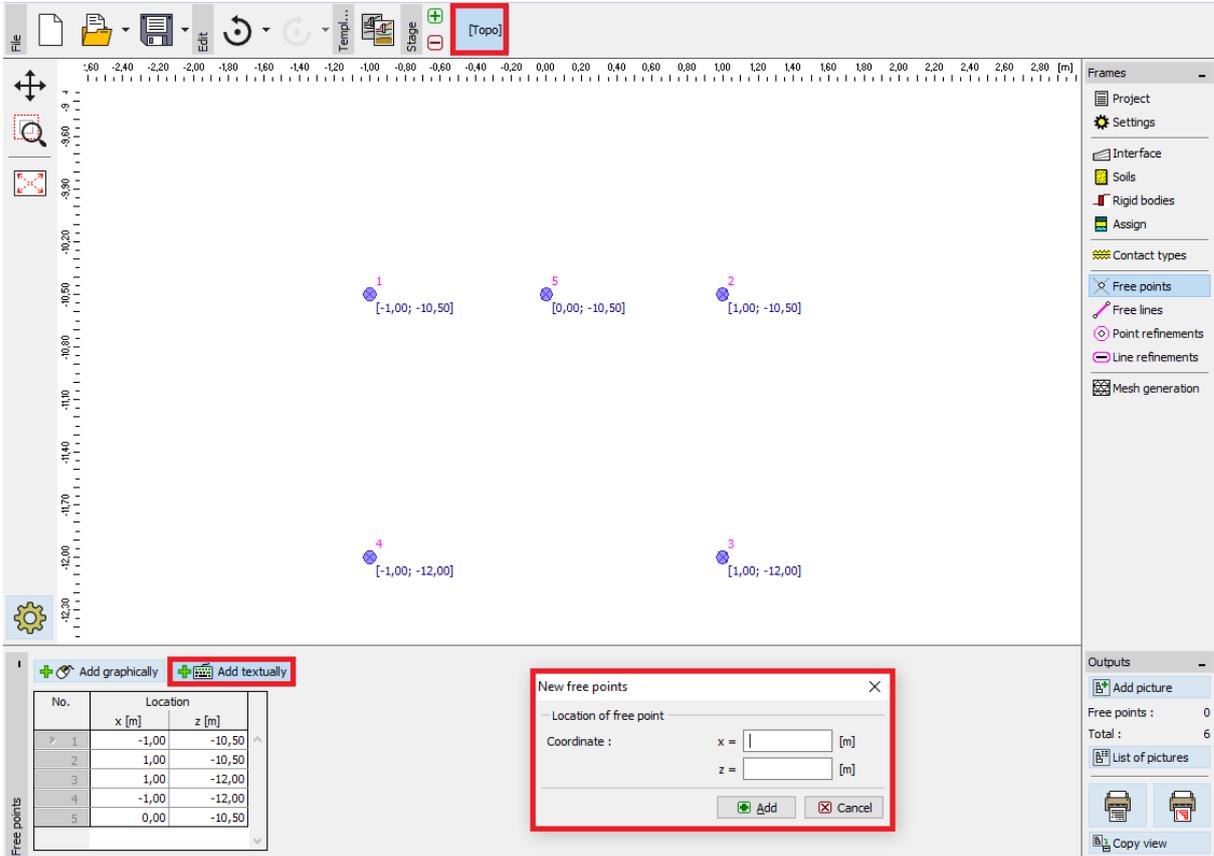


"Interface" frame + "World coordinates" dialogue window

Now we will specify the respective soil parameters including the material model and subsequently we will assign the soil to the created region (for more details visit Help – F1).

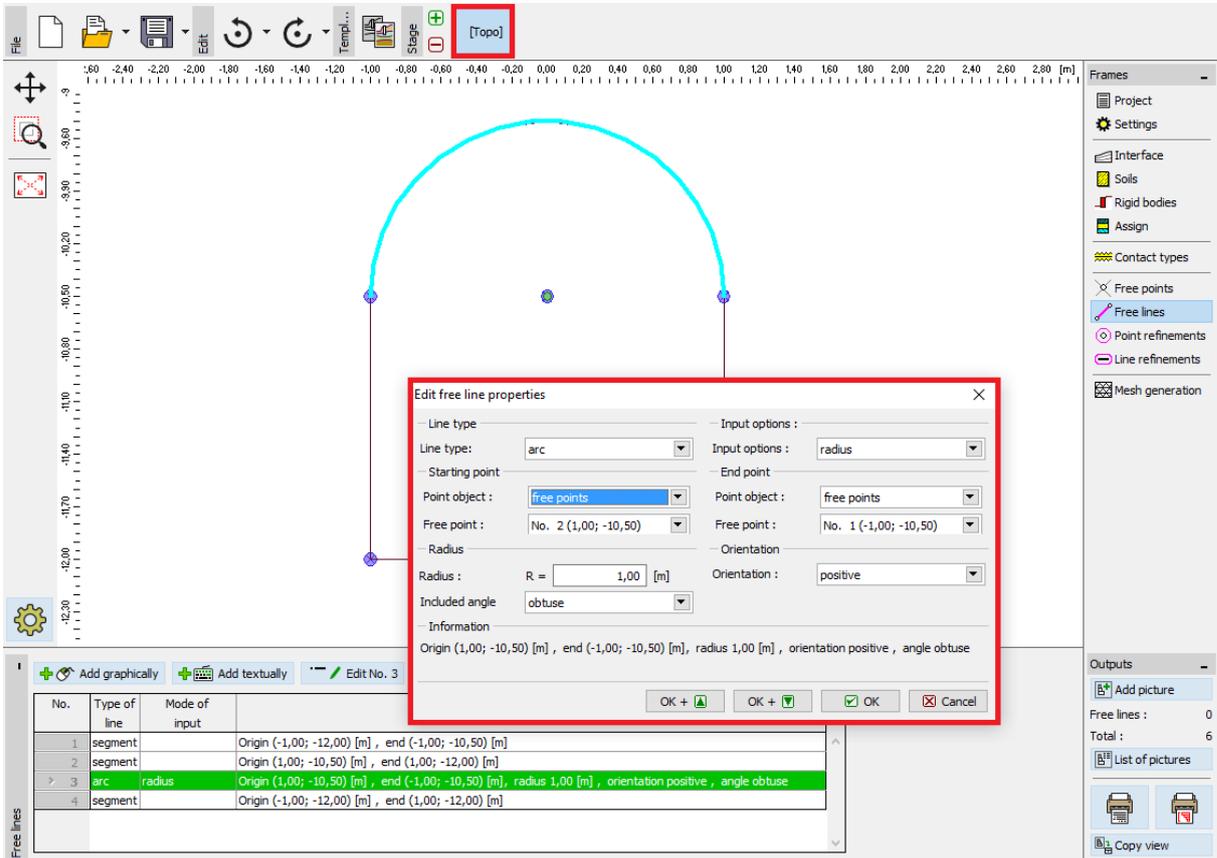
“Add new soils” dialog window

The next step is to set the structure geometry. Firstly, we will define the coordinates of free points (“Add” button), forming the collector corners (for more details visit Help – F1).



“Free points” frame + “New free points” dialogue window

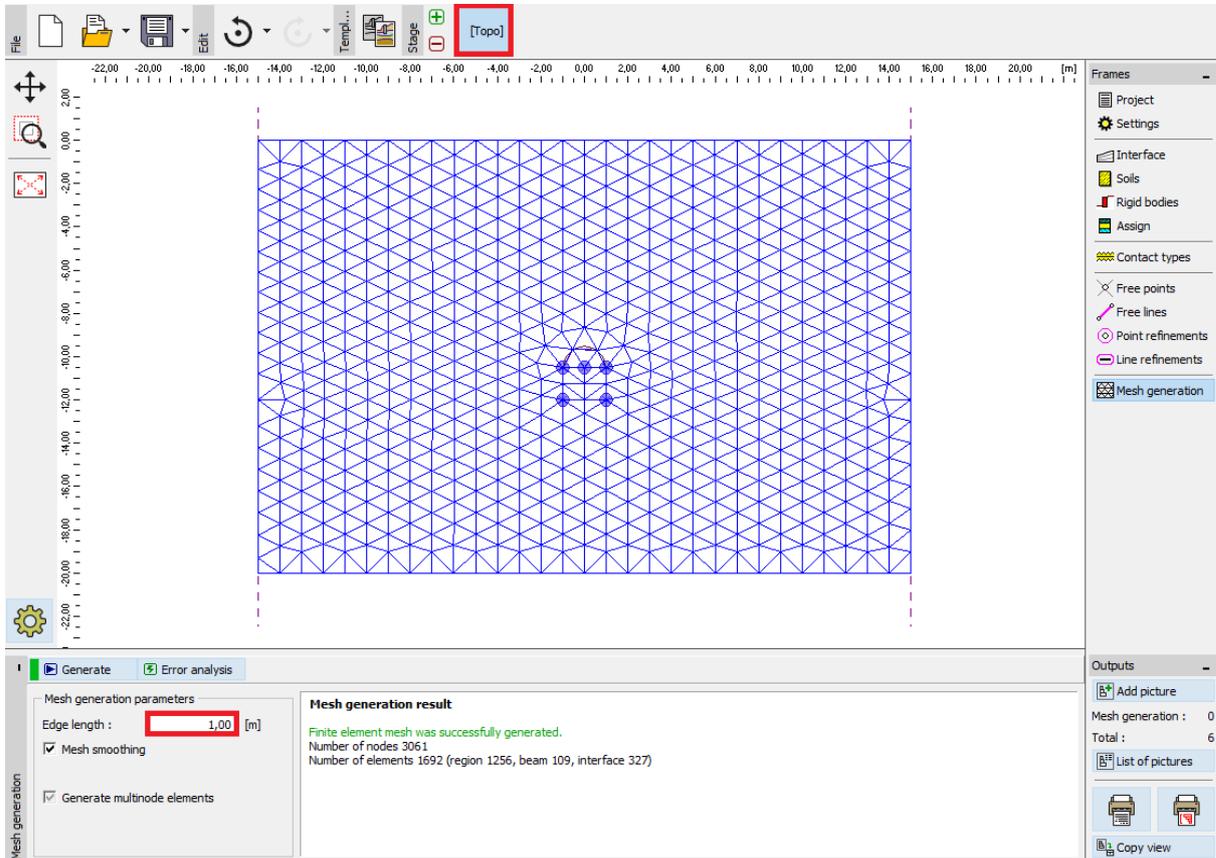
Subsequently, we will click on the “Add” button in the “Free lines” frame and connect the given points on the screen by respective lines using the cursor (for more details visit Help – F1). To set the arc with the radius $R = 1.0\text{ m}$, we must change the line type (using the “Edit” button).



“Modify free line properties” dialogue window

We will examine the resultant structure outline. Through this step, the collector geometry setting will be finished, and we will go over to the FE mesh generation (for more details visit Help – F1).

For the mesh generation parameters, we will choose the element edge length of 1.0 m and push the “**Generate**” button. The program will automatically generate and smoothen the FE mesh.



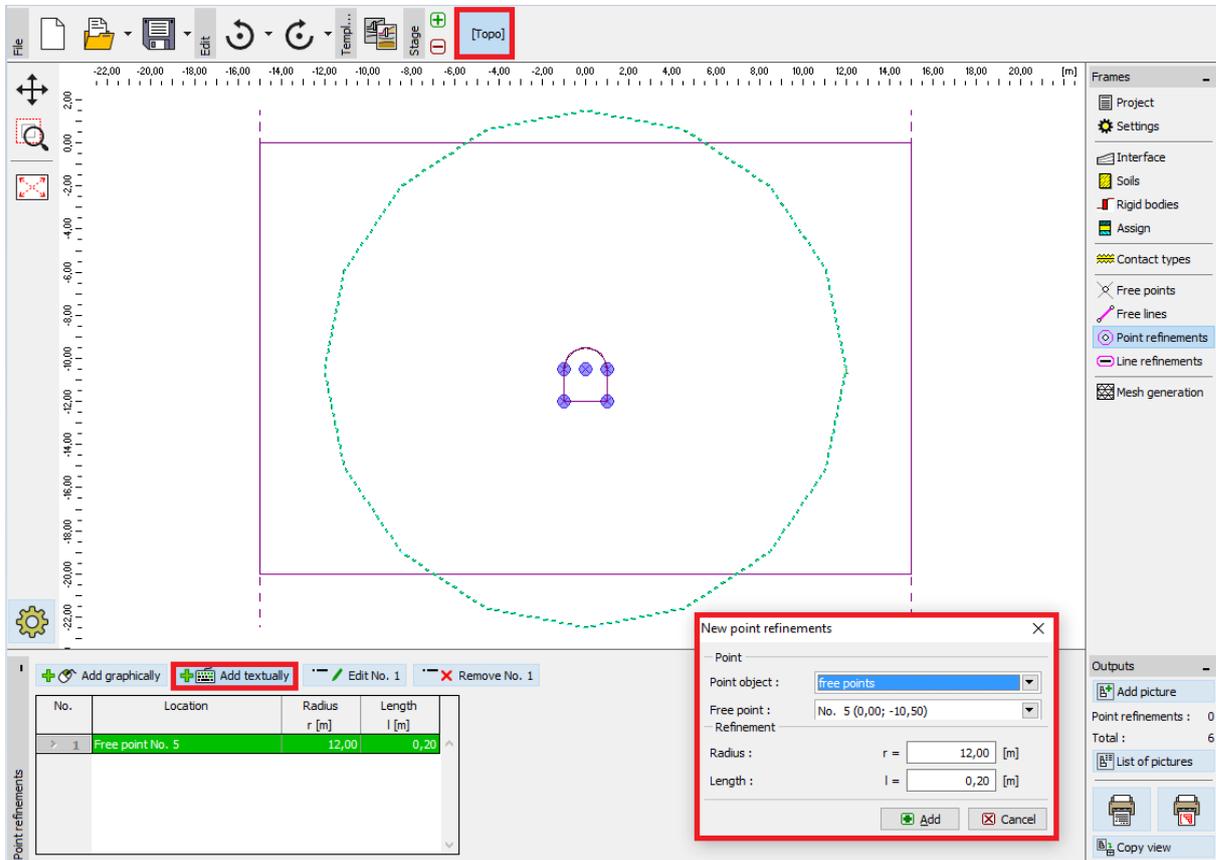
“Mesh generation” frame – element edge length of 1.0 m (without local refinement)

It is obvious at a first glance that the generated mesh in the collector surroundings is very coarse. We will therefore increase the density. We can refine the mesh density either around lines or around free points. The following procedure is suitable for the refining of density around the collector lining (in general the excavation):

- we will specify a free point in the vicinity of the excavation centre,
- we will refine the density around this point.

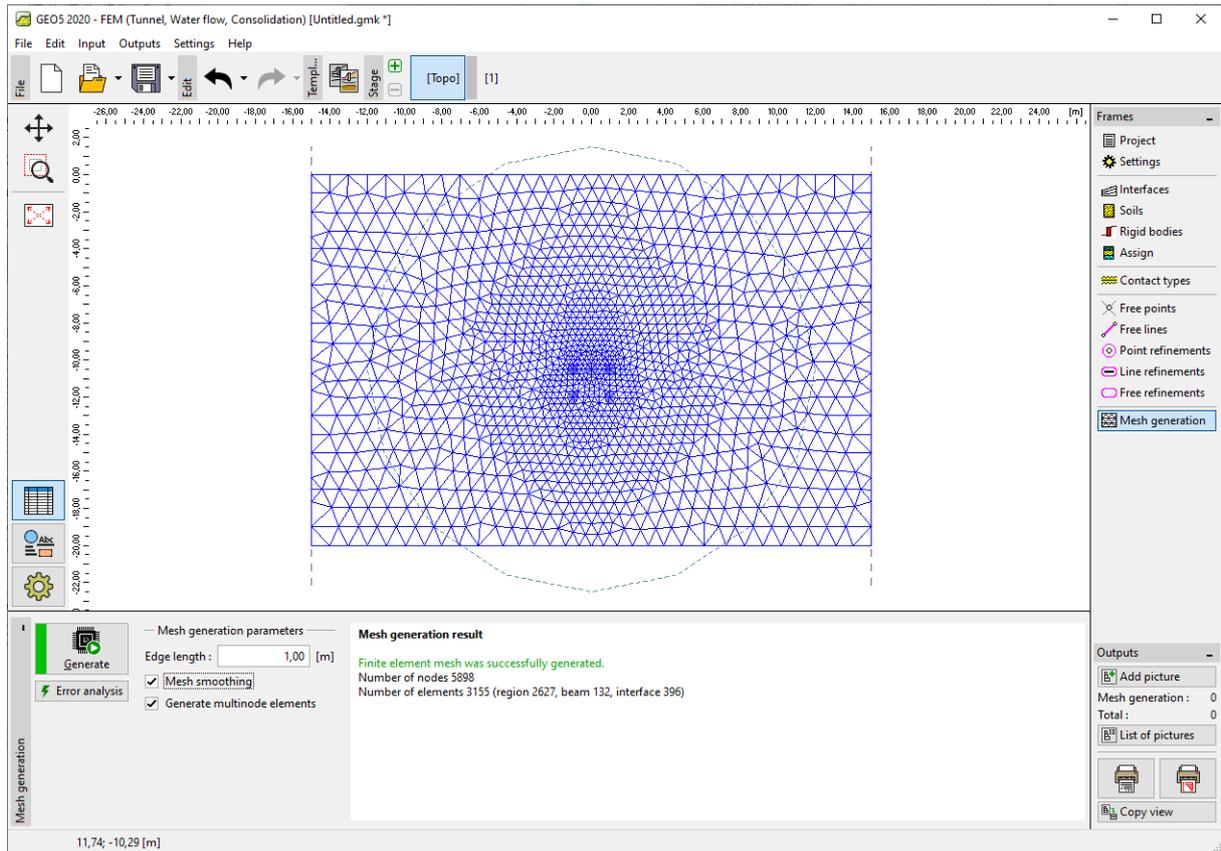
Note: Internal forces in beams are analysed at individual points of the mesh and it is therefore necessary to sufficiently refine the free lines and points of the FE mesh (for more details visit Help – F1).

To refine the finite element mesh density, we will specify the respective radius $r = 12.0 \text{ m}$ and the element edge length $l = 0.2 \text{ m}$. Then we will return to the “Mesh generation” frame and generate the FE mesh again.



“New point refinements” dialogue window

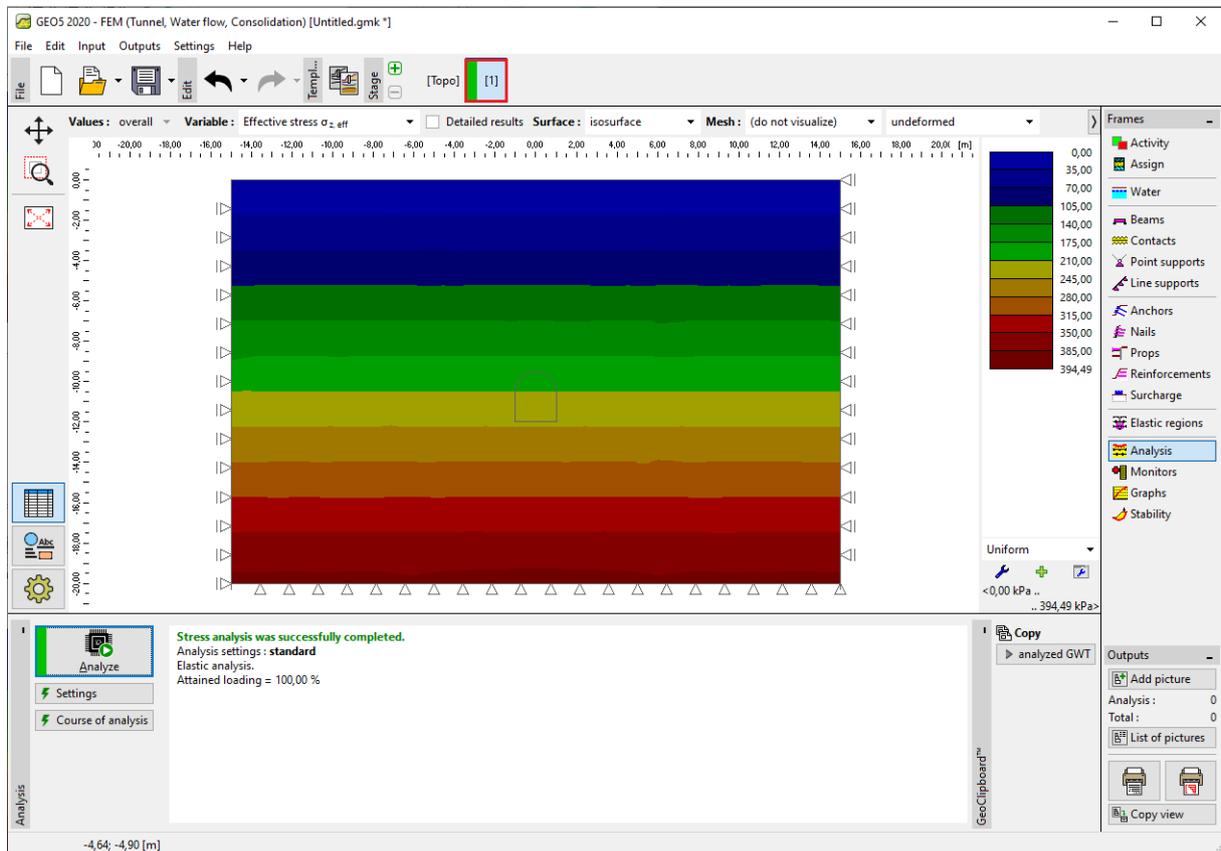
Note: The elements mesh should be sufficiently dense especially in a region where large stress gradients can be expected (point support, sharp corners, underground excavations etc.). It is necessary for this density-refining radius to be at least 3 to 5 times larger than the density in the density refining centre and both values (the density and radius) at the points to be in a reasonable proportion to the mesh density prescribed for the surrounding region. In this way smooth transition between the regions with different density will be secured (for more details visit Help – F1).



“Mesh generation” frame – FE edge length of 1.0 m (with locally increased mesh density in the collector surrounding)

Construction stage 1: primary geostatic stress

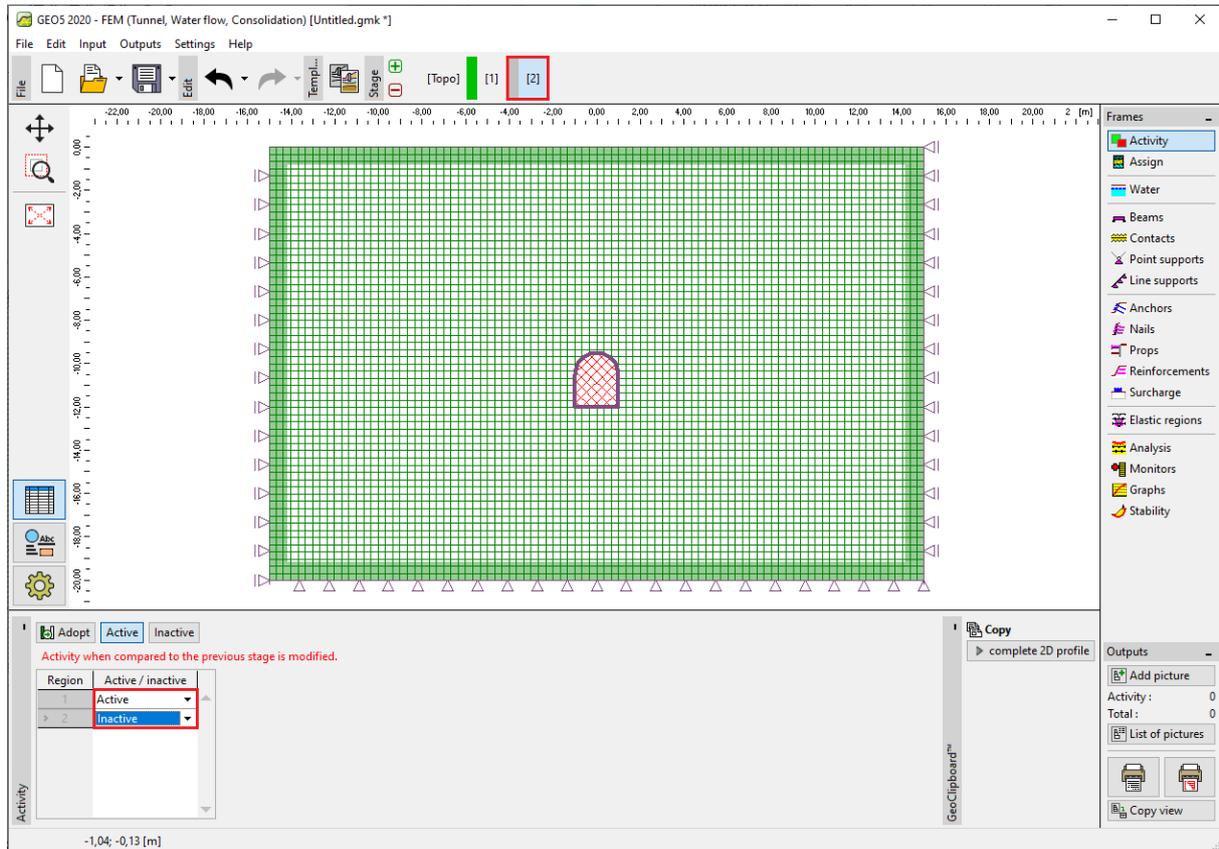
After subsequent generation, the mesh in the collector vicinity looks significantly better. Now we will go over to construction stage 1 and carry out the analysis of the primary geostatic stress. We will maintain the “Standard” analysis setting (for more details visit Help – F1).



“Analysis” frame – Construction stage 1

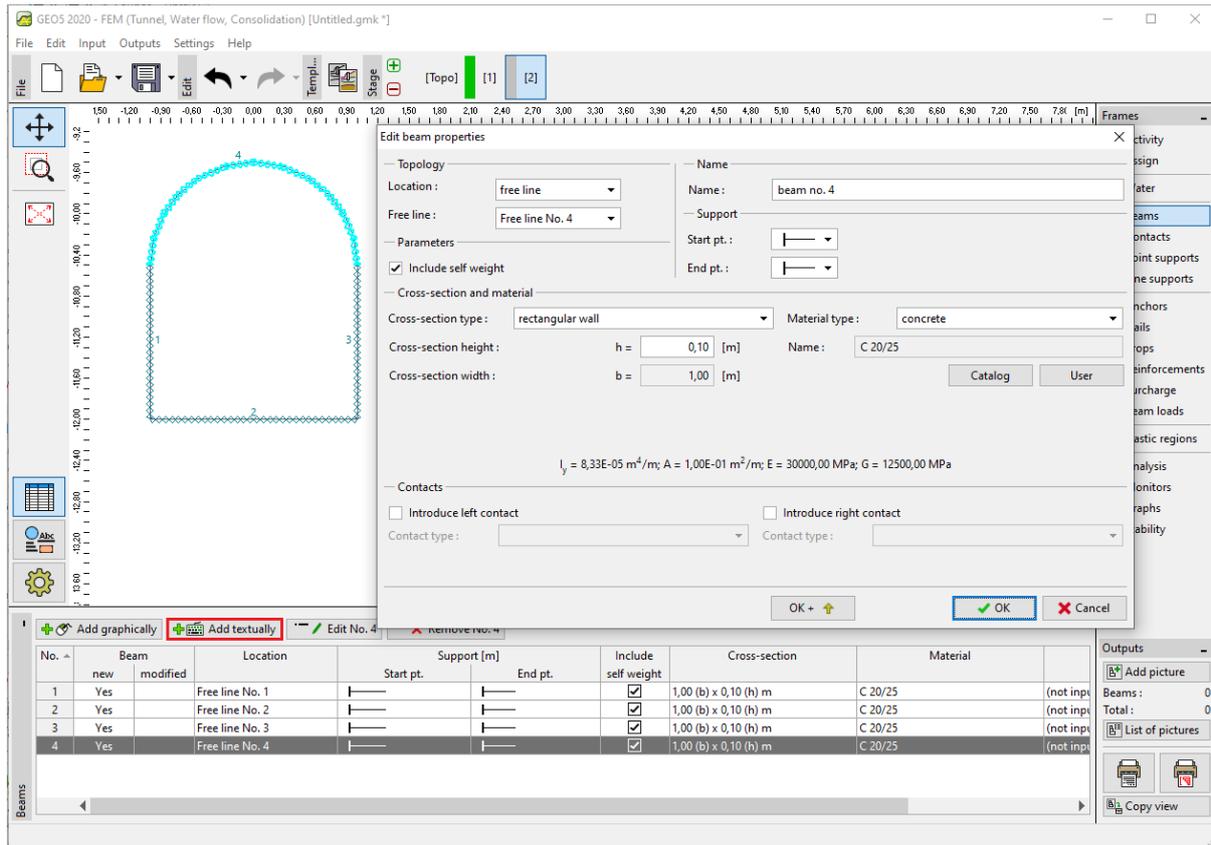
Construction stage 2: modelling of beam elements

In the “Activity” frame, we will first model the excavation of soil from the collector cross-section – we will set the particular region as inactive (for more details visit Help – F1).



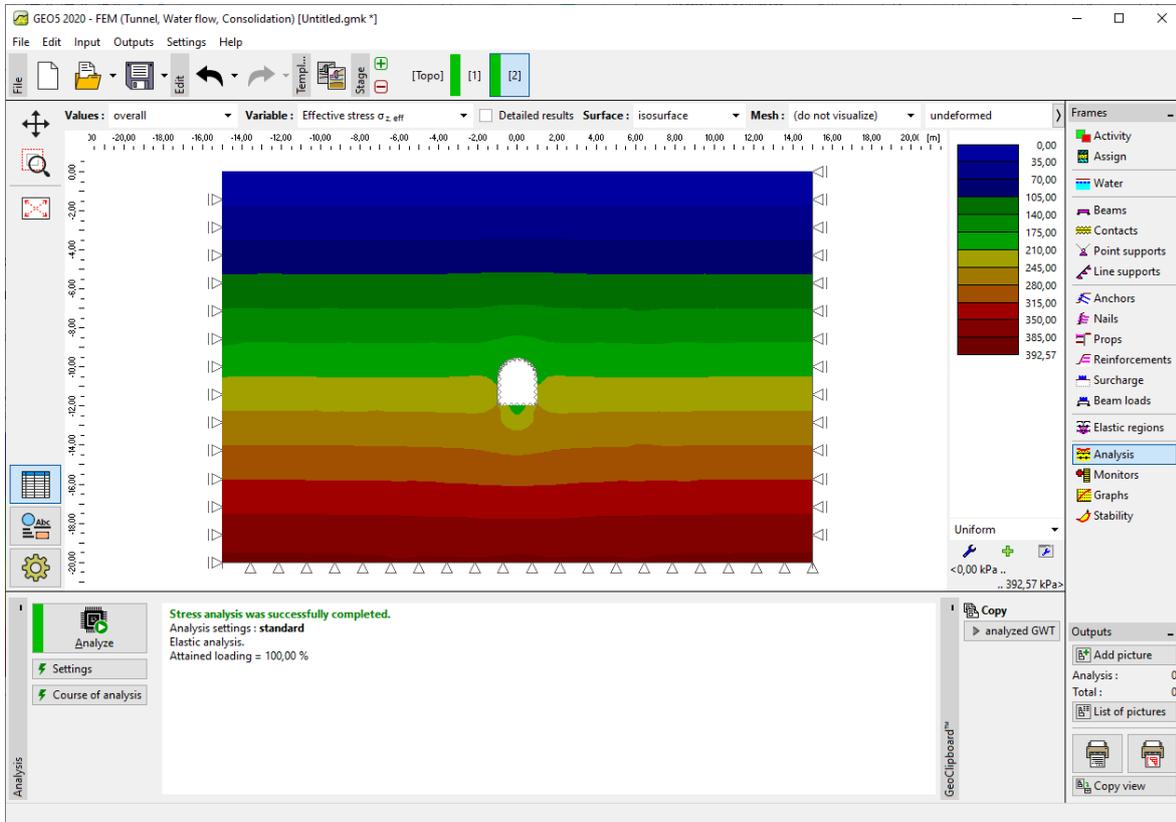
“Activity” frame – Construction stage 2

Then we will go over to the “Beams” frame and model the mined collector lining. We will define the following parameters – the beam location (we take all free lines into consideration), material and concrete class, cross-section height (0.1 m) and the supports of beam ends (for more details visit Help – F1).



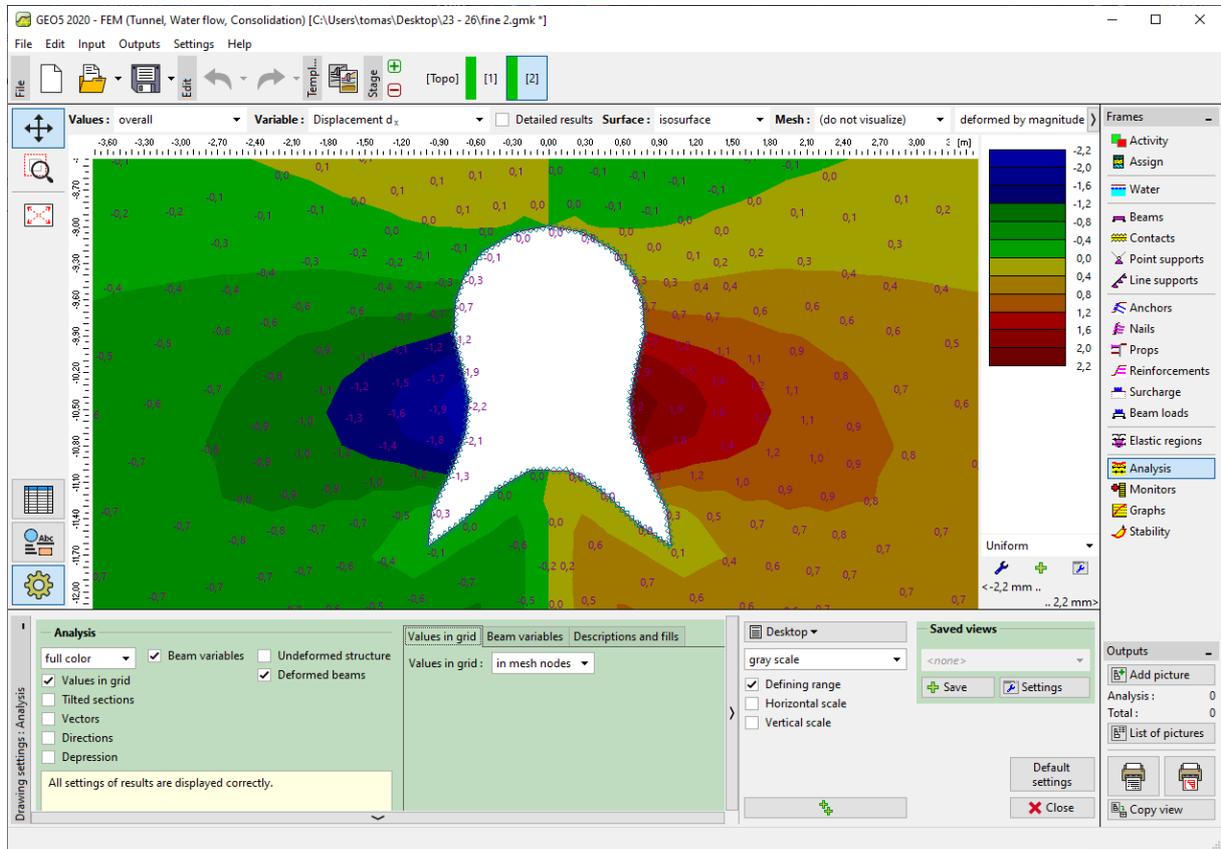
“New beams” dialogue window – Construction stage 2

Now we will carry out the analysis and visualise the results for the vertical geostatic stress $\sigma_{z,ef}$ [kPa], lateral displacement d_x [mm] and internal forces in the collector lining.



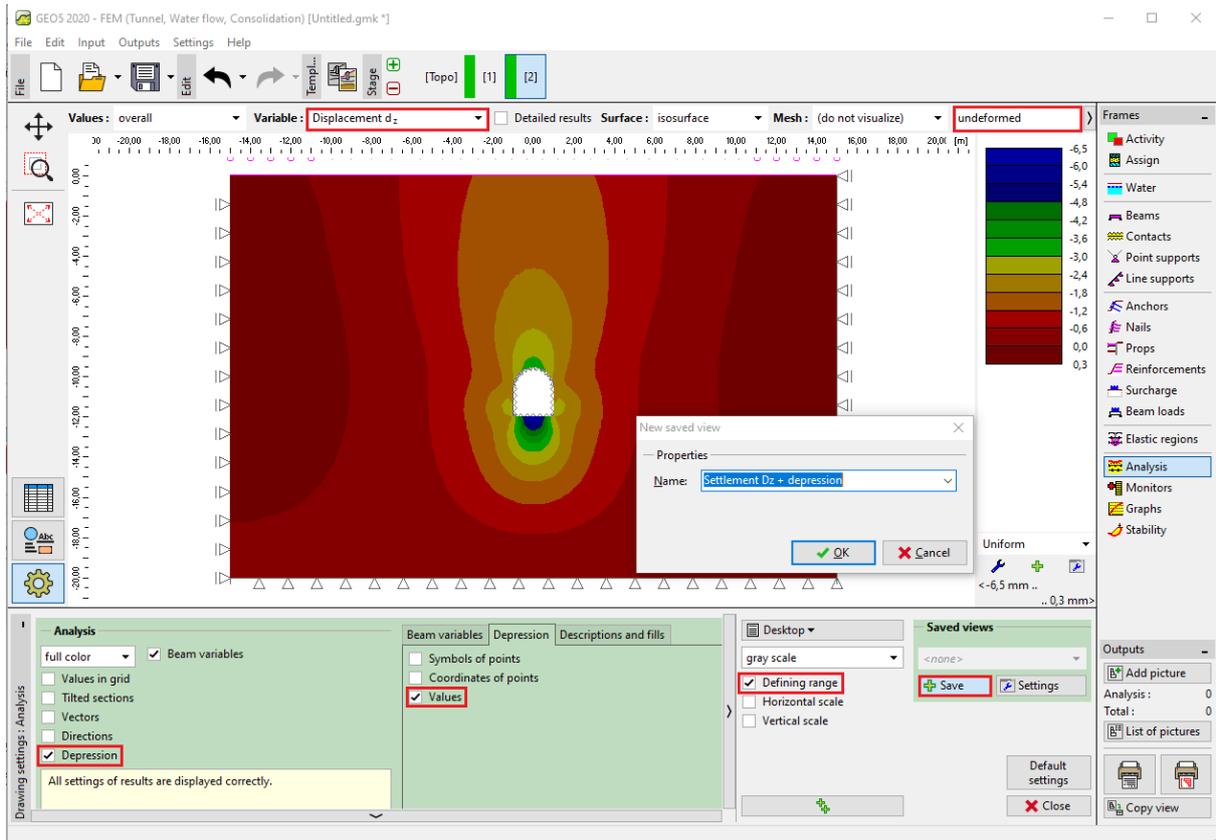
"Analysis" frame – Construction stage 2 (vertical geostatic stress $\sigma_{z,ef}$)

It follows from the picture that the maximum horizontal displacement is 2.2 mm (the collector behaves as a rigid body). To better understand the structure behaviour, we will visualise the deformed mesh (the button in the upper part of screen).



“Analysis” frame – Construction stage 2 (horizontal displacement d_x after excavating the soil)

Note: Individual current views presented on the screen can also be downloaded as independent objects. They can also be managed afterwards. In this way the visualization of results is significantly accelerated (for more details visit Help – F1).

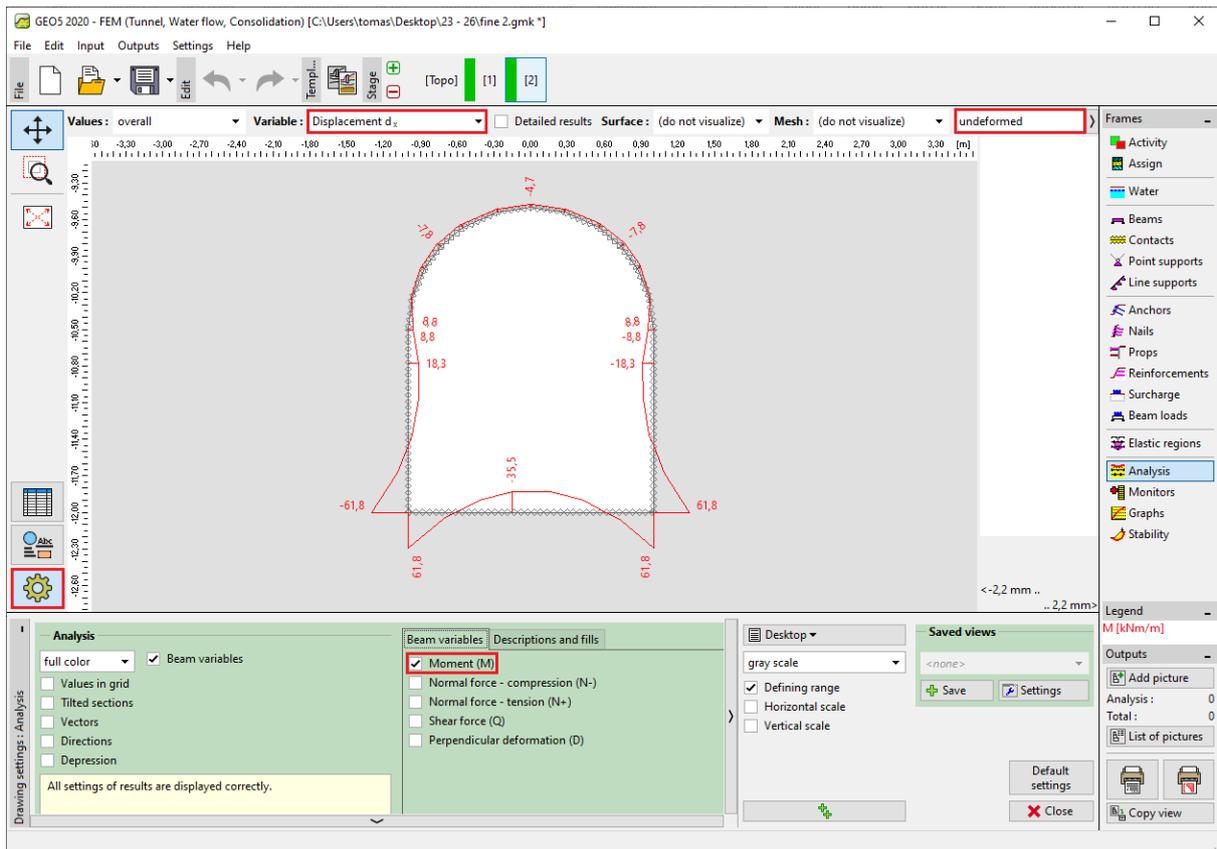


"New picture" dialog window

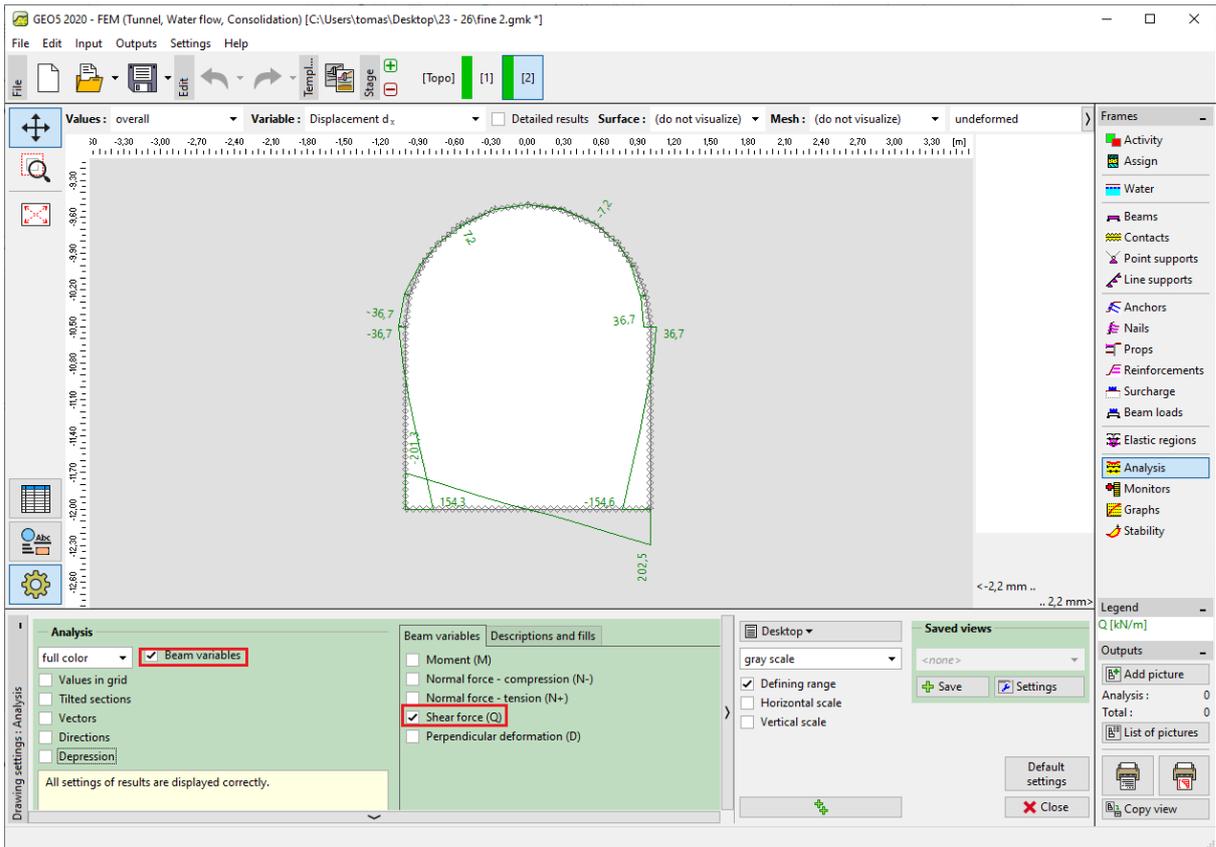
Now we will examine the diagrams for bending moments M [kNm/m], shear forces Q [kN/m] and normal compressive forces N^- [kN/m] for construction stage 2 (using “Settings” button in “Beam variables” flag).

Note: To maintain sufficient clarity and comprehensibility some results cannot be displayed concurrently. It is, for example, impossible to draw a deformed structure concurrently with diagrams for internal forces along the beam; it is always necessary to choose only one option. The program gives a warning at the bottom of the dialogue window if inadmissible combinations of outputs are set (for more details visit Help – F1).

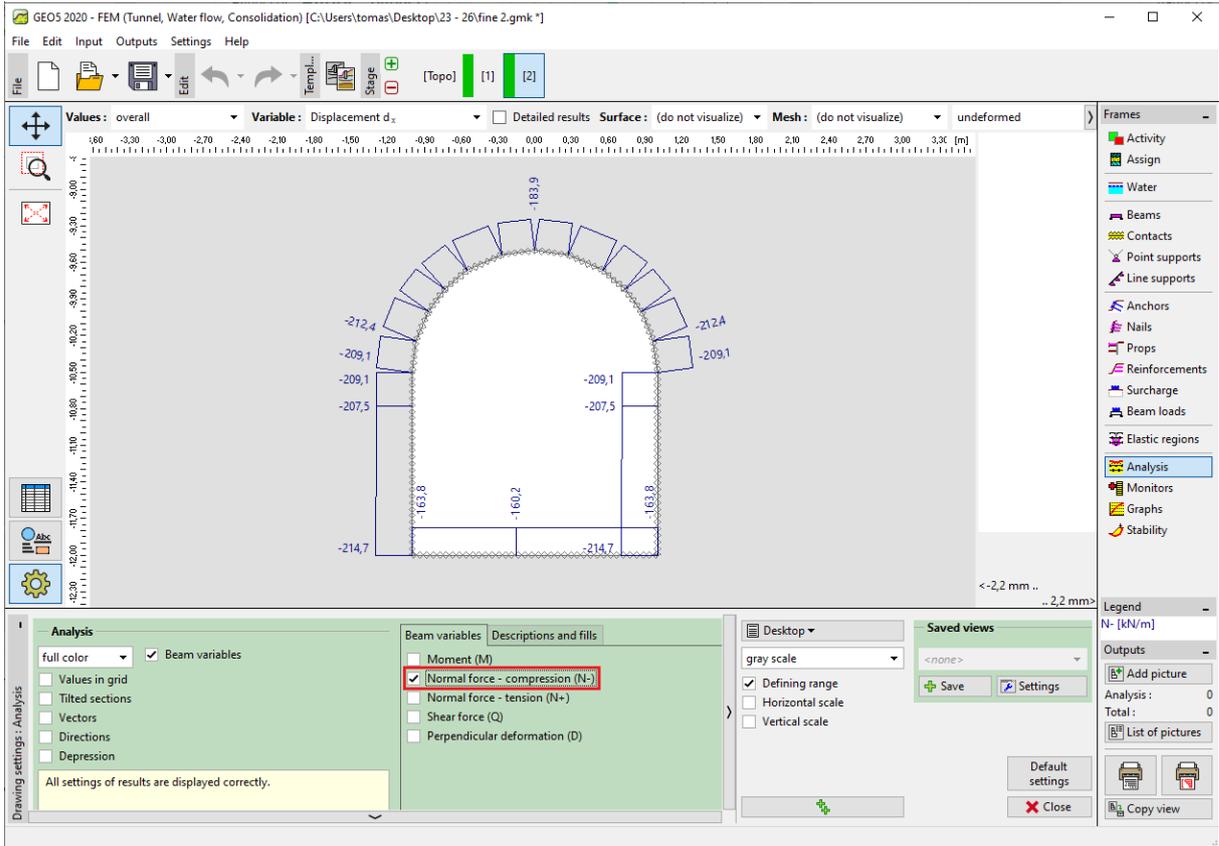
The collector lining reinforcement can be designed and assessed for these values in any arbitrary static program (e.g. FIN EC – CONCRETE 2D). We will record the results in a summary table.



“Analysis” frame – Construction stage 2 (variation of bending moment M)



“Analysis” frame – Construction stage 2 (variation shear force Q)



“Analysis” frame – Construction stage 2 (variation of normal compressive force N^-)

Verification of the yield condition: Mohr-Coulomb material model

Now we will verify, whether plastic strains develop when using non-linear models. We will return to the “Topology” mode and change the material model to “Mohr-Coulomb” in “Soils”.

Edit soil parameters

Identification

Name: zemina č. 1

Material model

Material model: Mohr - Coulomb

Basic data

Unit weight: $\gamma = 20,00$ [kN/m³]

Elastic modulus: $E = 12,00$ [MPa]

Stiffness according to depth: constant

Poisson's ratio: $\nu = 0,40$ [-]

Uplift pressure

Calc. mode of uplift: standard

Saturated unit weight: $\gamma_{sat} = 22,00$ [kN/m³]

Model Mohr - Coulomb

Modulus unloading / reloading: $E_{ur} = 12,00$ [MPa]

Angle of internal friction: $\phi_{ef} = 21,00$ [°]

Cohesion of soil: $c_{ef} = 12,00$ [kPa]

Dilatation angle: $\psi = 0,00$ [°]

Draw

Pattern category: GEO

Search:

Subcategory: Soils (1 - 16)

Pattern: 3 Gravelly silt

Color:

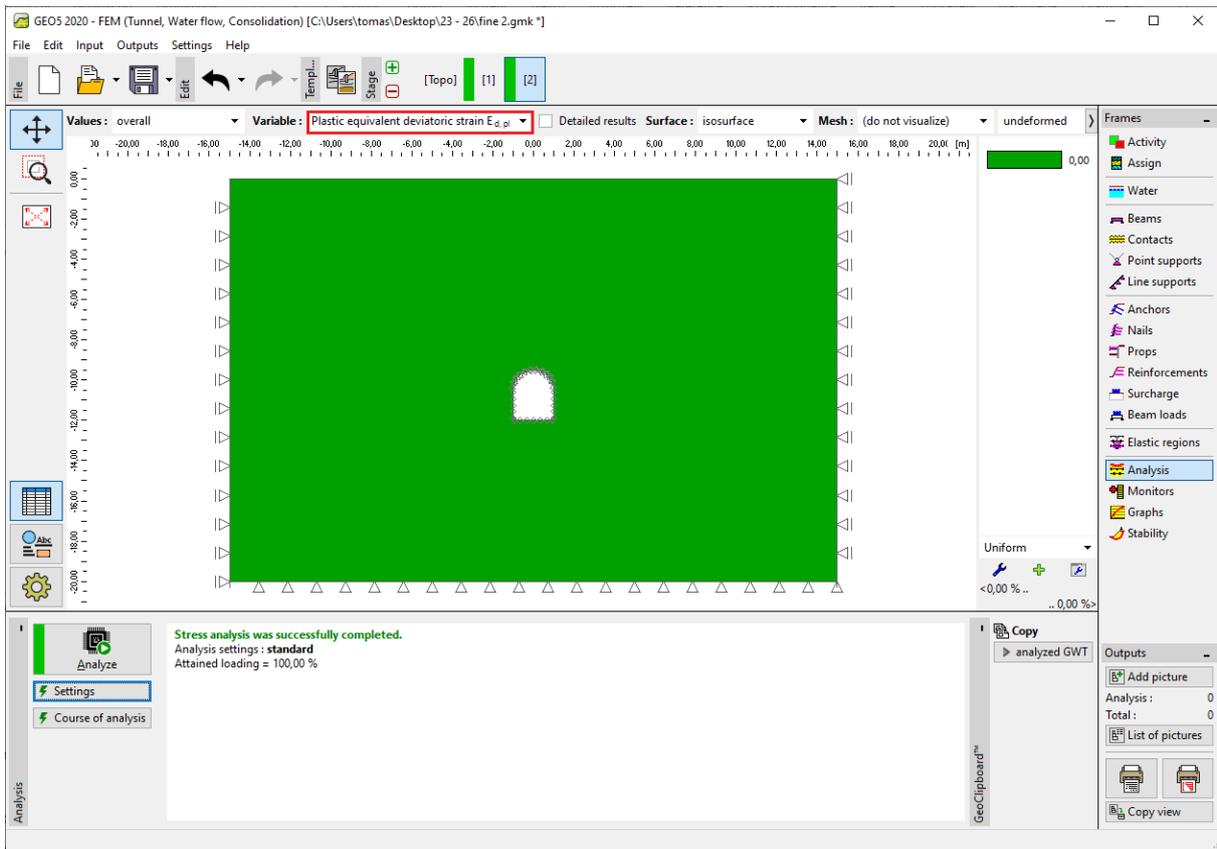
Background: automatic

Saturation <10 - 90>: 40 [%]

Buttons: Classify, Clear, OK, Cancel

Dialog window “Edit soil parameters”

After completing the analysis, we will examine the equivalent plastic strains.



“Analysis” frame – Construction stage 2 (equivalent plastic strains $\epsilon_{eq.,pl.}$ according to the MC model)

It follows from the previous diagram that the yield condition for the Mohr-Coulomb model is not exceeded – equivalent plastic strains $\epsilon_{eq.,pl.}$ are zero, which corresponds to the structure behaviour determined according to the elastic material model. Resulting values of displacements, geostatic stress and internal forces are therefore identical.

Assessment of results

The following table presents the values of extreme internal forces along beams (the collector lining) for construction stage 2 (the values of bending moments, shear forces and normal forces). We carried this analysis out for an elastic material model with locally increased density of triangular elements.

Material model	Construction stage 2		
	N^- [kN/m]	M [kNm/m]	Q [kN/m]
Elastic	- 160.2	+ 61.8	+ 202.5
	- 214.7	- 61.8	- 201.3

Curves for internal forces along beams (extremes) – Construction stage 2

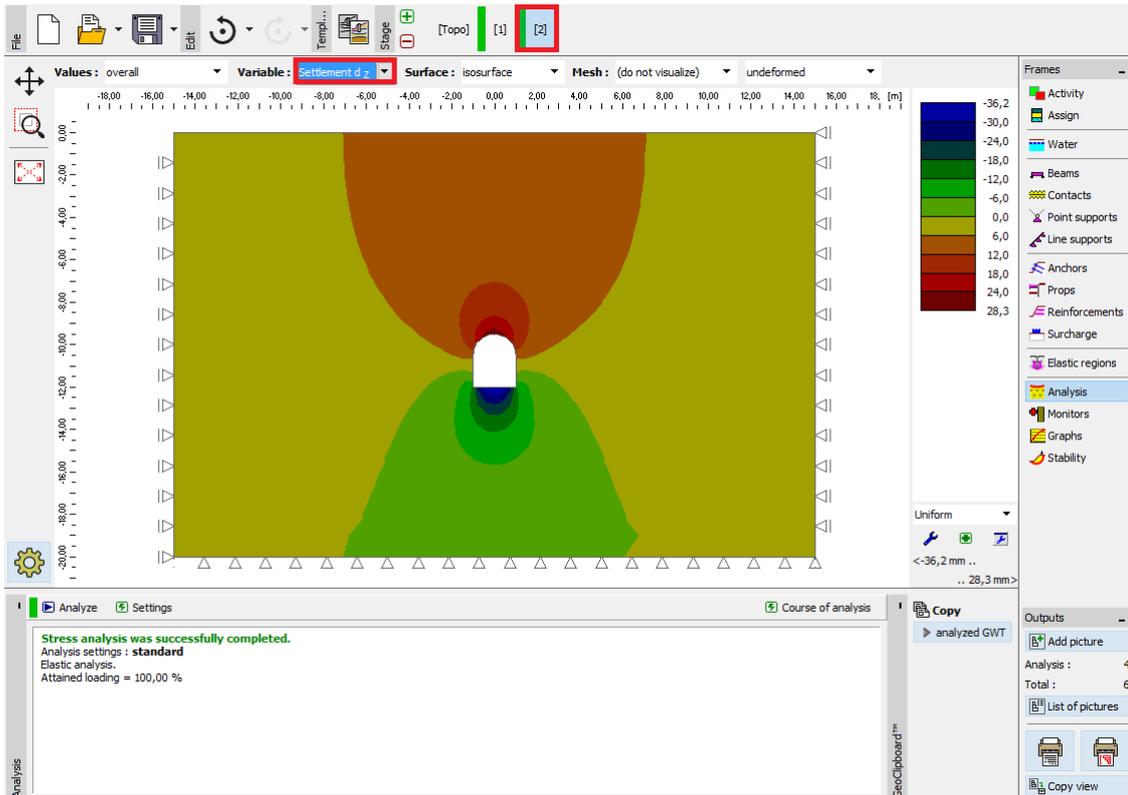
Conclusion

The following conclusions can be drawn from the numerical analysis results:

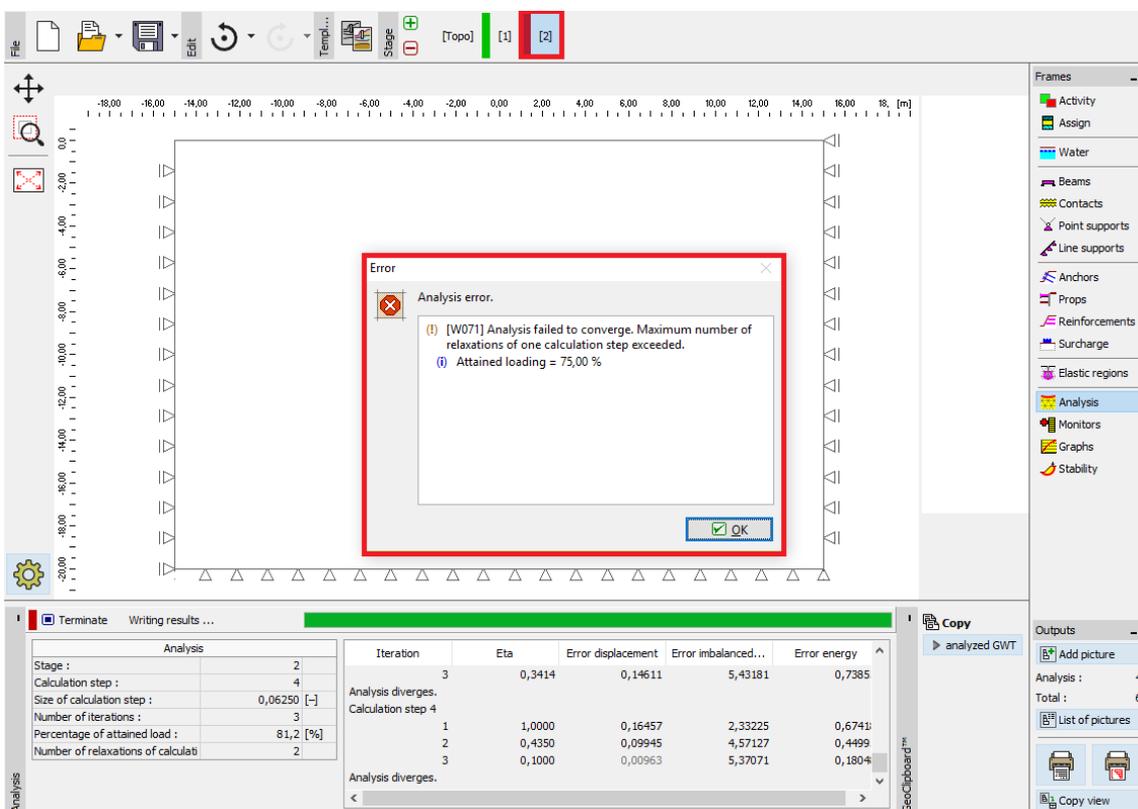
- Local density refinement of finite element mesh leads to more accurate results.
- If non-linear material models (e.g. Mohr-Coulomb) lead to zero values of equivalent plastic strains $\varepsilon_{eq.,pl.}$, the structure behaves elastically and the results of internal forces, displacements and stresses are identical for both model types.

Note: The analysis we have carried out is based on an unrealistic assumption that the lining acts concurrently with excavating of the soil. This procedure would be suitable for structures carried out by jacking through soft soils (jacking of complete structures into soil). In reality the ground mass is unloaded and deformed in the direction of the excavated space when the soil is being excavated. A real example of tunnel modelling is described in Chapter 26. Numerical modelling of tunnel excavation adopting the NATM method.

If, in our particular case, the lining had not been activated immediately (it can be modelled as another stage without specifying beam elements), the excavation would have collapsed; for the elastic model this is presented by large deformations, whilst the program will not find the solution in the case of the non-linear model.



Analysis without beam elements (settlement d_z according to the elastic model)



“Error” dialogue window – analysis without using beam elements (according to the MC model)