Terrain settlement analysis

Program: FEM

File: Demo_manual_21.gmk

This example contains the solution to terrain settlement under surcharge loading using the Finite Element Method.

Task specification

Determine the terrain settlement induced by a strip surcharge loading $q = 250$ kPa along the length of 4.0 m and the total settlement after subsequent unloading. The geological profile is homogeneous; the soil parameters are as follows:

- Unit weight of soil: $\gamma = 19.0 \, \text{kN/m}^3$

- Modulus of elasticity: $E = 15.0$ MPa

- Unloading modulus: $E_{ur} = 45.0$ MPa

- Poisson's ratio: $\nu = 0.35$

- Soil cohesion: $c_{ef} = 8.0$ kPa

- Angle of internal friction: $\phi_{ef} = 29.0$ °

- Unit weight of saturated soil: $\gamma_{sat} = 21.0 \, \text{kN/m}^3$

Regarding the *modified elastic model*, the input parameters of soils will be considered as follows:

- Modulus of soil deformation: $E_{def} = 15.0$ MPa

- Unloading modulus: $E_{ur} = 45.0$ MPa

Compare the analysis of the settlement or the total vertical deformation value d_z $[mm]$ with other material models (we will not take the Clam-Clay model and Hypoplastic model for clays into consideration because the soil mass is formed by cohesionless soil).

Note: The Mohr-Coulomb and Drucker Prager models are used in engineering praxis even for cohesive soils because they are based on shear failure and use common input parameters of soils and rocks (ϕ, c).
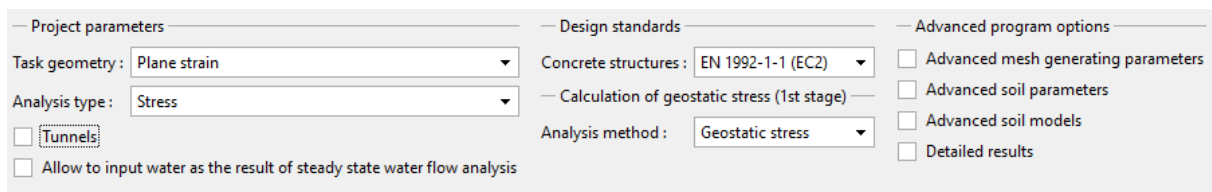
Solution

We will use the GEO5 – FEM program for the analysis. We will describe the solution to this problem step by step in the text below:

- − Topology: setting and modelling the problem,

- − Construction stage 1: geostatic stress analysis,

- − Construction stage 2: introduction of surcharge loading, terrain settlement analysis,

- − Construction stage 3: terrain surface unloading, terrain settlement analysis,

- − Assessment of results (conclusion).

Topology: setting and modelling the problem

Firstly, we will go over to the settings, where we will characterise the type of the problem, the type of the analysis and the primary stress analysis method.

Problem settings – Problem characteristics; primary stress analysis

We will not switch the "Tunnels, "Advanced input and detailed results" options on – they are intended for experienced users of finite elements or for a different type of problems. Their description exceeds the scope and purpose of this manual.

*Note: The **planar problem (plane-strain conditions assumed)** is suitable for solutions to linear structures (a tunnel, embankment, open cut, dam etc.), for which it applies that their longitudinal dimension is by an order of magnitude larger than the lateral dimensions of the area being solved – zero deformations are assumed in the direction of the y-axis. The analysis is carried out under the assumption of plane-strain deformation (for more details visit Help – F1). The other problem type (axial symmetry) is solved in the subsequent chapter.*

*Note: The **Stress** type of the analysis deals with stresses and deformations within the area being solved. It is the basic analysis type; other analysis types and other options (flow, slope stability) will be described separately in other chapters.*

*Note: Two options are available for the **primary stress** analysis (for construction stage 1):*

- *geostatic stress: It is the standard method for geostatic stress analysis, taking into consideration the dead weight of soils and horizontal stresses according to the theory of elasticity. The lateral pressure coefficient is then given by $K_0 = \frac{\nu}{1-\nu}$.*

- K_0 *procedure (according to Jáky, for overconsolidated soils etc.).*
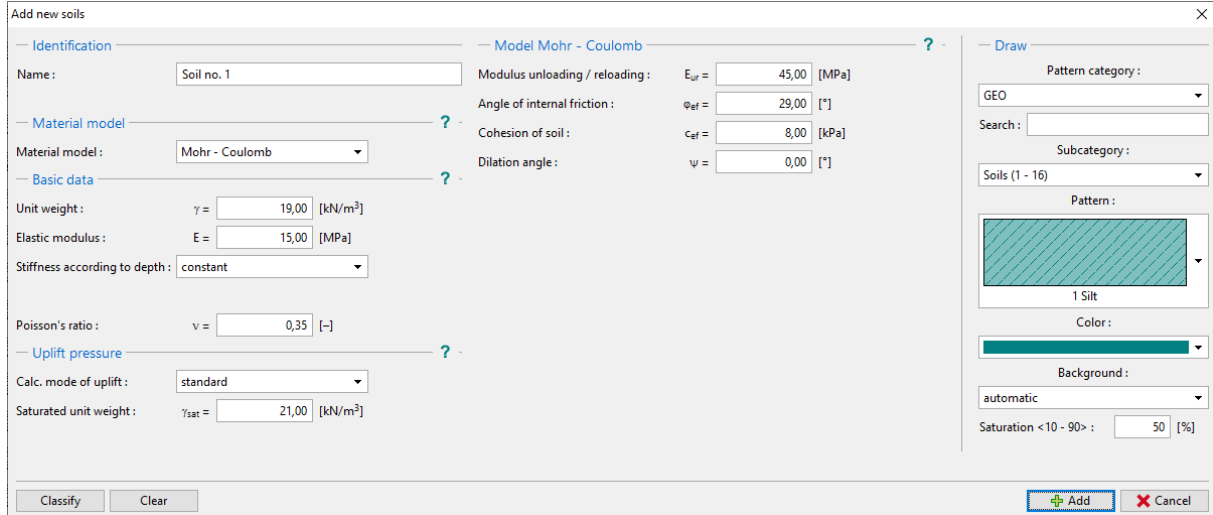
We set the world coordinates (the magnitude of the numerical model of the problem being solved) and the terrain interface in the frame "Interface". We will choose the world coordinates so that the results would not to be affected by conditions at the edge. For our particular problem we will choose model dimensions $\langle -15 \text{ m}; \ 15 \text{ m} \rangle$ and will set the thickness of the layer to be examined to 15.0 m. We will set the terrain coordinates (x, z): [-15, 0]; [15, 0].
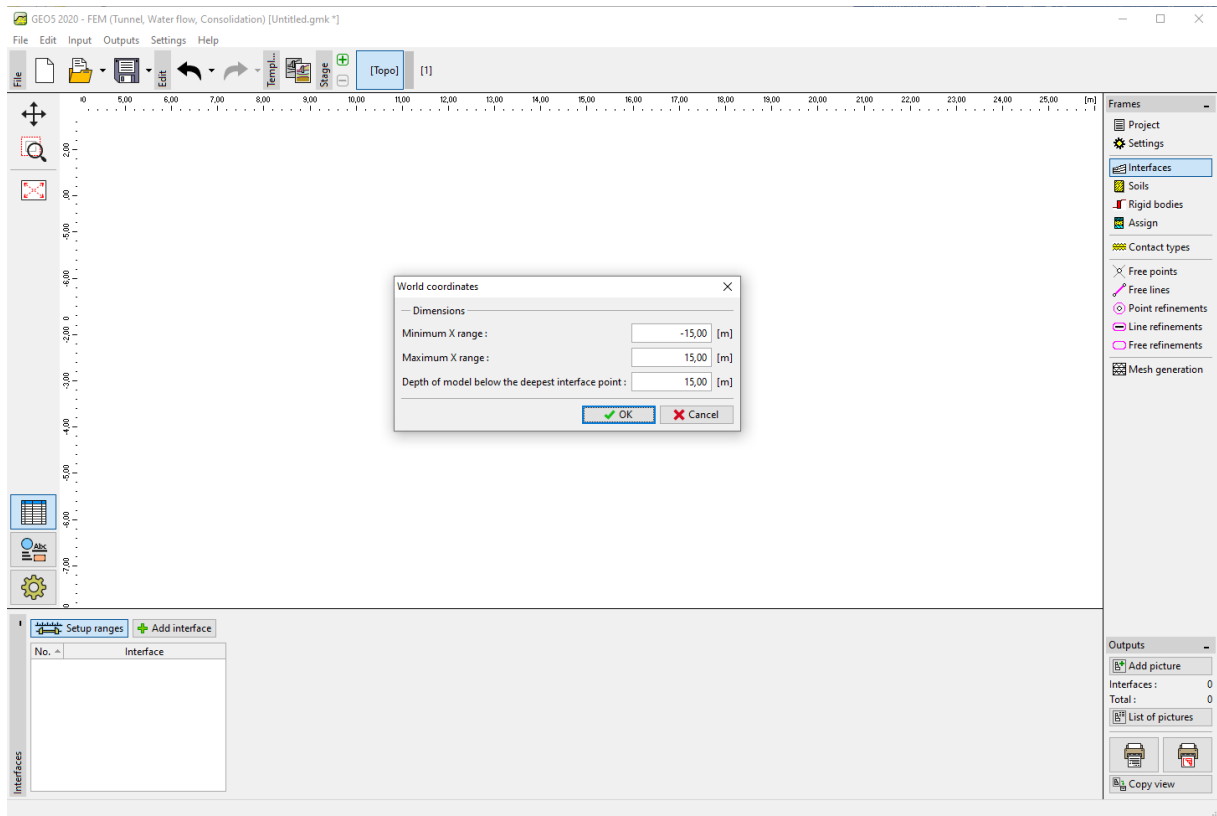
"Interface" frame

Note: The guidance values of recommended dimensions to set the model boundaries for individual solution cases are presented and described in more details in the program Help (for more details visit Help – F1).

For the present analysis, we choose the **Mohr-Coulomb** model of soil (the comparison of various models is presented at the end of this example) and specify the particular soil parameters. This non-linear model will allow us to follow the development of plastic strains or the distribution of potential failure zones.

Setting soil parameters

Note: The elastic model assumes soil behaviour according to Hooke's law (ideally elastic material). The main advantage of this model is that it always analyses the results to the very end. The disadvantage is that the soil behaves in this way only when the loading magnitude is small – it is therefore unsuitable for real structures. On the other hand, it is suitable for modelling areas, in which we do not expect plastic failures of material (e.g. gabion walls, stiff sub-grade etc.) or for the verification of a basic numerical model.

Subsequently, we will assign the soil to the area we have created.
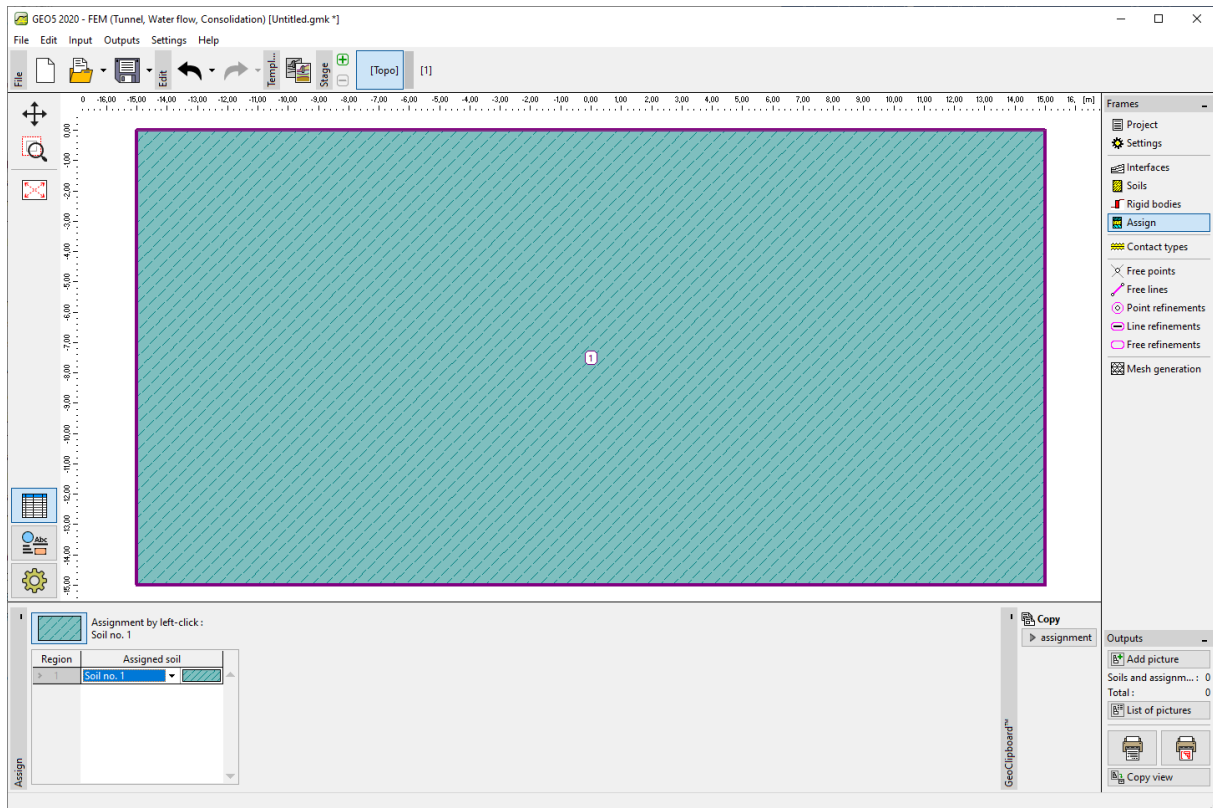
"Assign" frame

We will leave out the other frames for specification of contact types, free points and lines; they are meaningless for the solving of our problem.

The next step is the generation of the Finite Element (hereinafter referred to as FE) mesh. For the mesh generation parameters, we will choose a length of 1.0m for the elements edge (the edge length is chosen depending on the problem dimensions and variability). We will check the option Mesh Smoothing and push the **Generate** button. The program will automatically generate and smoothen the FE mesh. We will verify whether the mesh density is adequate taking into consideration the problem magnitude.
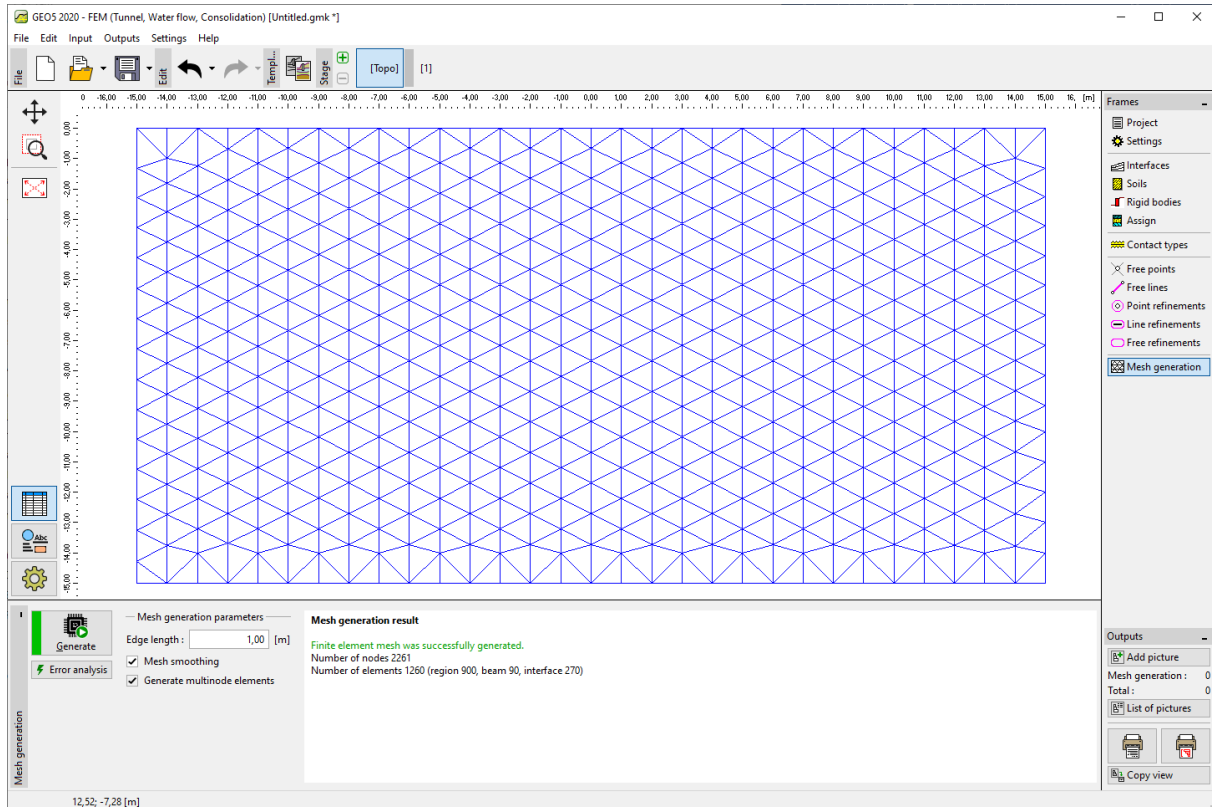
Generation of finite element mesh – Topology (triangular mesh)

Note: The standard triangular mesh with six-node elements is suitable for the majority of geotechnical problems. In the advanced input mode, the program also allows for other mesh types (mixed, triangular) to be generated – this is intended for experienced FEM users.

Note: A correctly generated finite element mesh is the basic condition for achieving results that represent the real behaviour of the structure reasonably well. The FE mesh significantly affects the values obtained because the FEM analysis primarily determines the values of nodal displacements. The remaining variables (stresses, strains) are derived from these values subsequently.

Unfortunately, it is impossible to provide a general rule for a correct mesh density because of the fact that individual problems are different. For beginners to the FEM analysis we recommend that first a courser mesh should be chosen, the problem analysis should be carried out and then several other

options containing the smoothing of the mesh or its parts can be tried. (It is also possible to refine the mesh density around the points or lines – more details are contained in the other chapters of EM). In general, the courser the mesh the stiffer the model behaviour (the resultant settlement value is smaller).

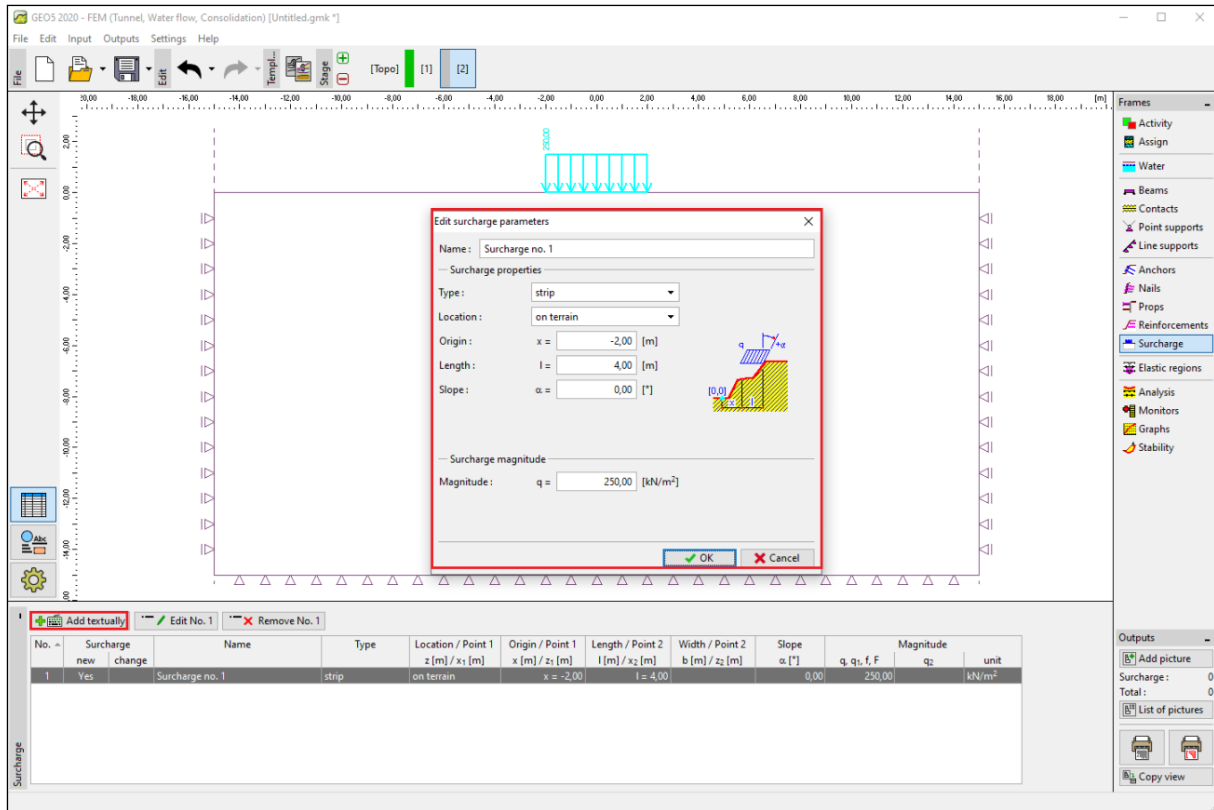## Construction stage 1: primary stress analysis

When the FE mesh generation is finished, we will switch to Phase 1 (using the tool bar on the upper edge of the screen) and carry out the geostatic stress analysis by pushing the "**Analyse**" button. Subsequently, we will examine the results for the effective stress $\sigma_{z,eff}$ [kPa].
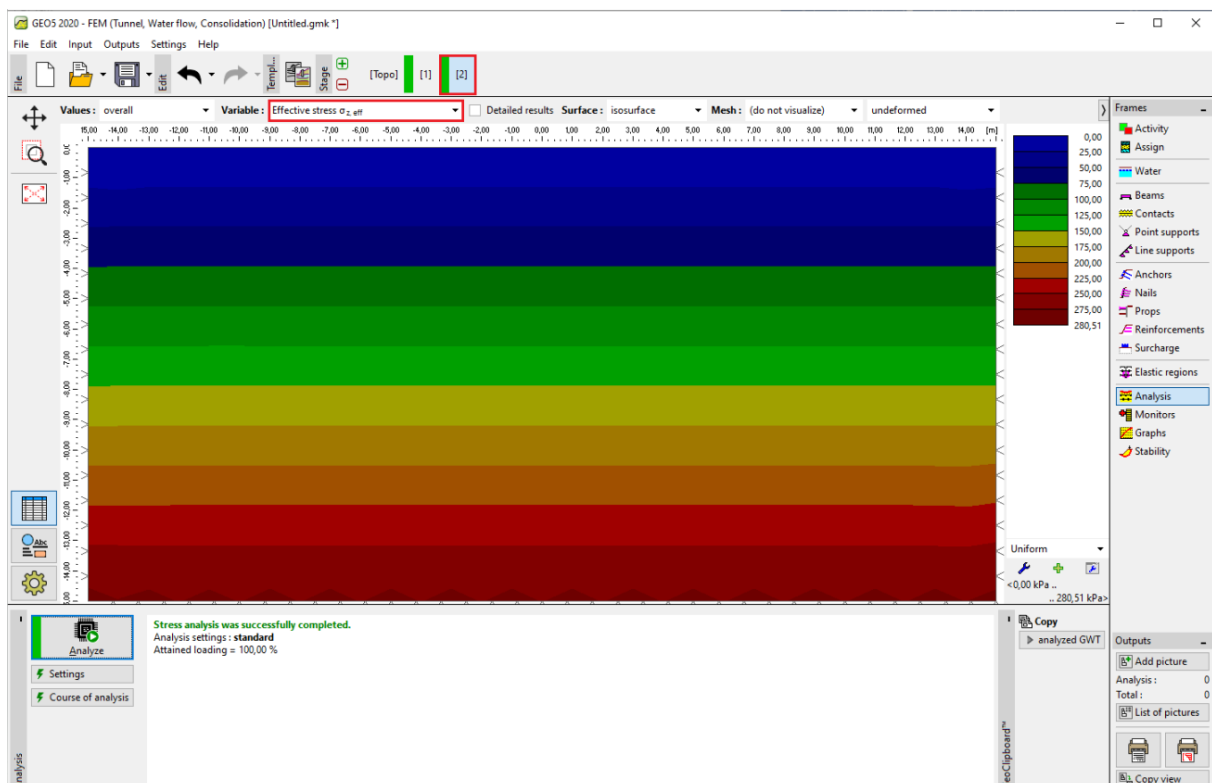
*Construction stage 1 analysis – primary geostatic stress*

Construction stage 2: introduction of surcharge loading

In the next step we will add construction stage 2. Subsequently, we will define the surcharge load acting on the terrain surface and will set relevant characteristics. Then we will confirm everything using the **"Add"** button.


Setting new surcharges

In this construction stage we will again carry out the analysis and examine the results, firstly for the vertical normal stress $\sigma_{z,eff}$ [kPa].
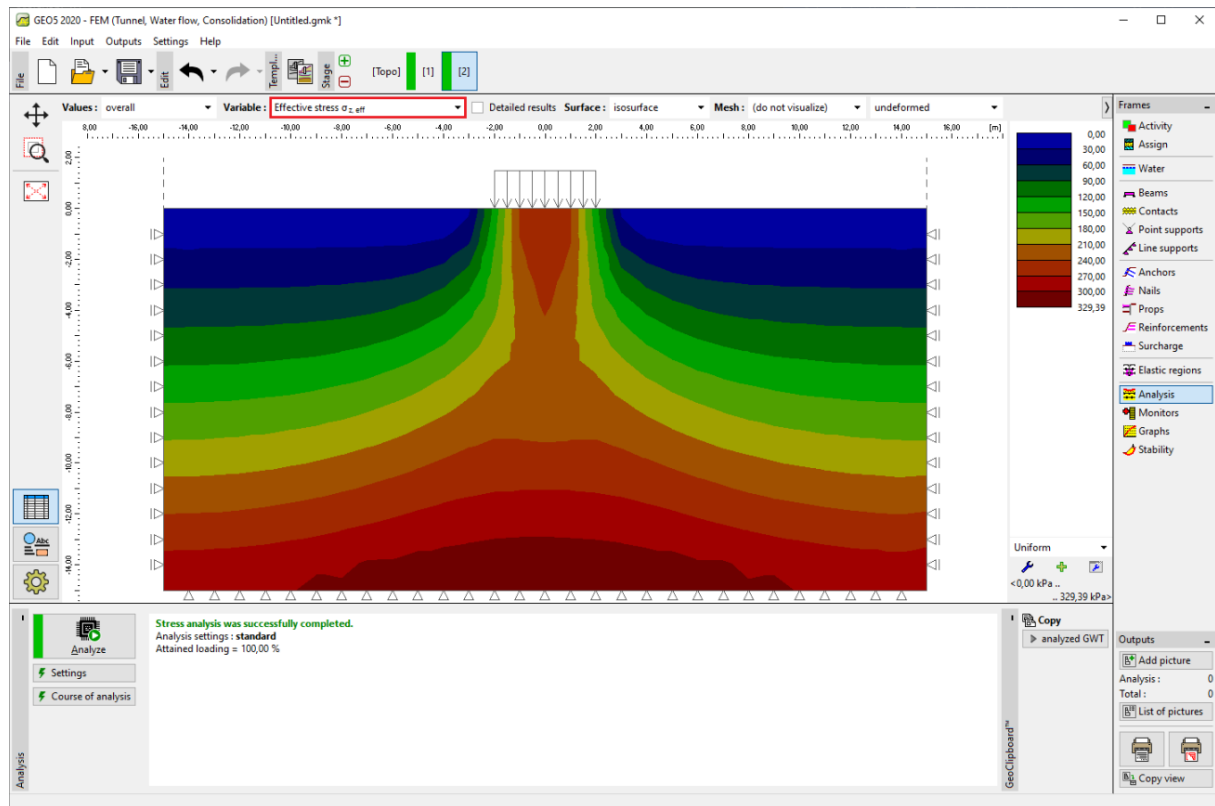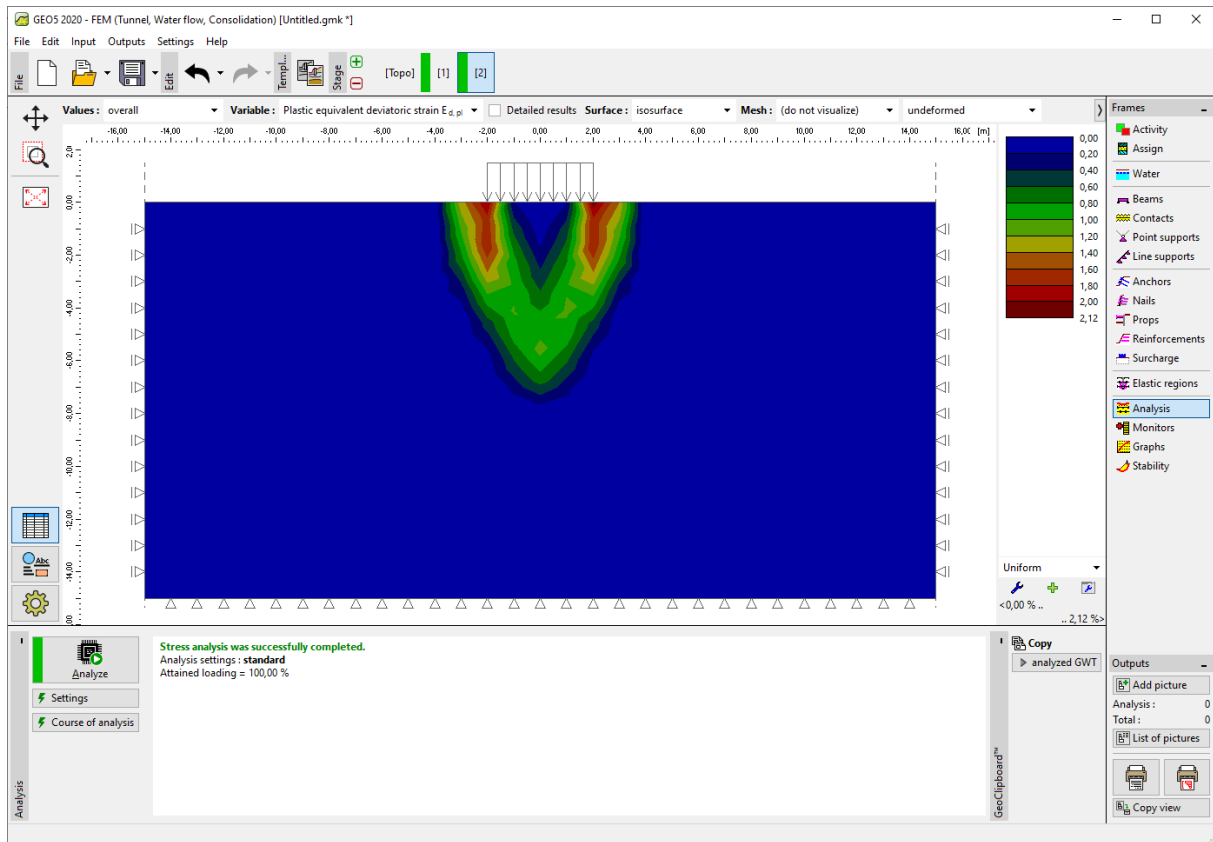

Construction stage 2 analysis – vertical normal stress $\sigma_{z,eff}$ [kPa]

Next, we will switch to the visualisation for drawing the vertical displacement d_z [mm]. It follows from the drawing that the maximum vertical deformation amounts to 102.9 mm.
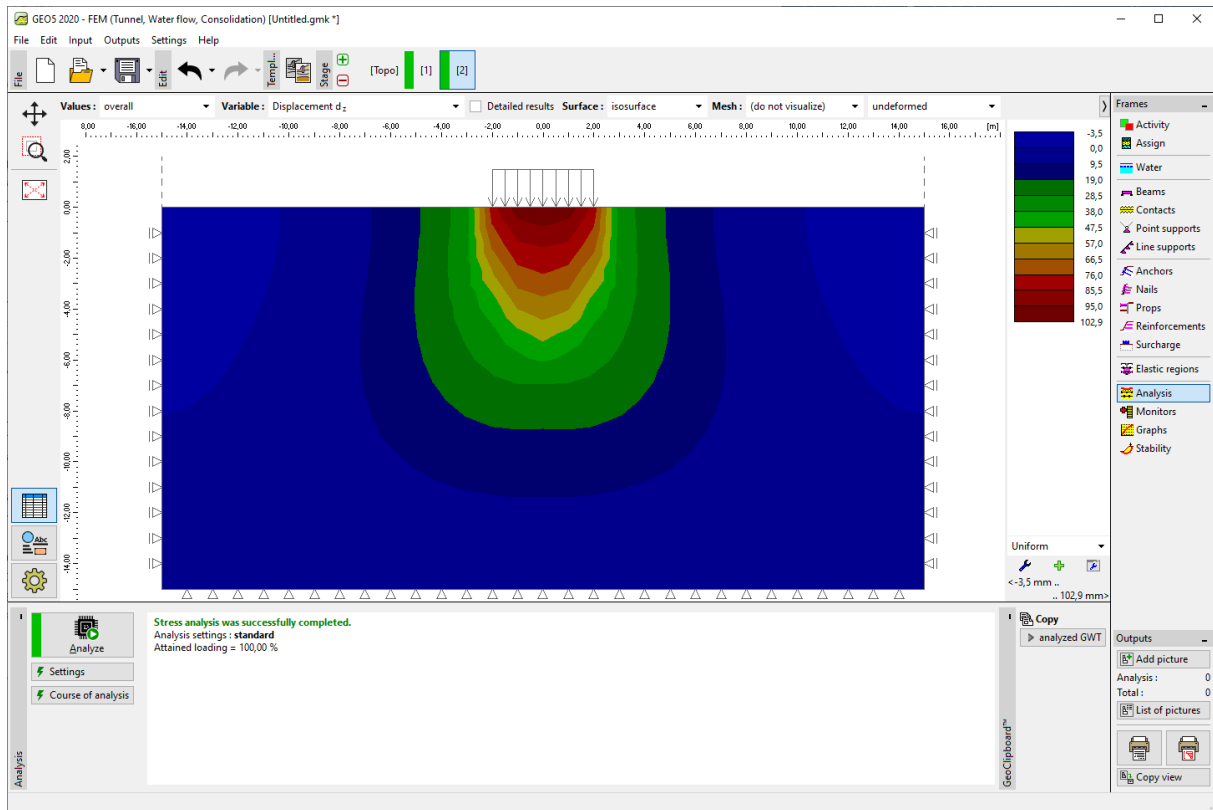
Construction stage 2 analysis – surcharge-induced vertical deformation d_z [mm]

When a FE problem is being examined, an important output parameter is given by equivalent plastic strains (for non-linear models). They represent the locations where the yield condition was exceeded, i.e. the soil is in state of plastic deformation, exhibiting permanent plastic strains.

Construction stage 2 analysis – equivalent plastic strain ratio $\varepsilon_{eq,pl.}$ [%]

Construction stage 3: terrain surface unloading

In the next step we will add construction stage 3. In this construction stage we do not consider the terrain surcharge (we remove it). We will again carry out the analysis and will determine the values of stress and deformations. The total displacement after the terrain surface unloading amounts to 73.3 mm (for a triangular FE mesh).
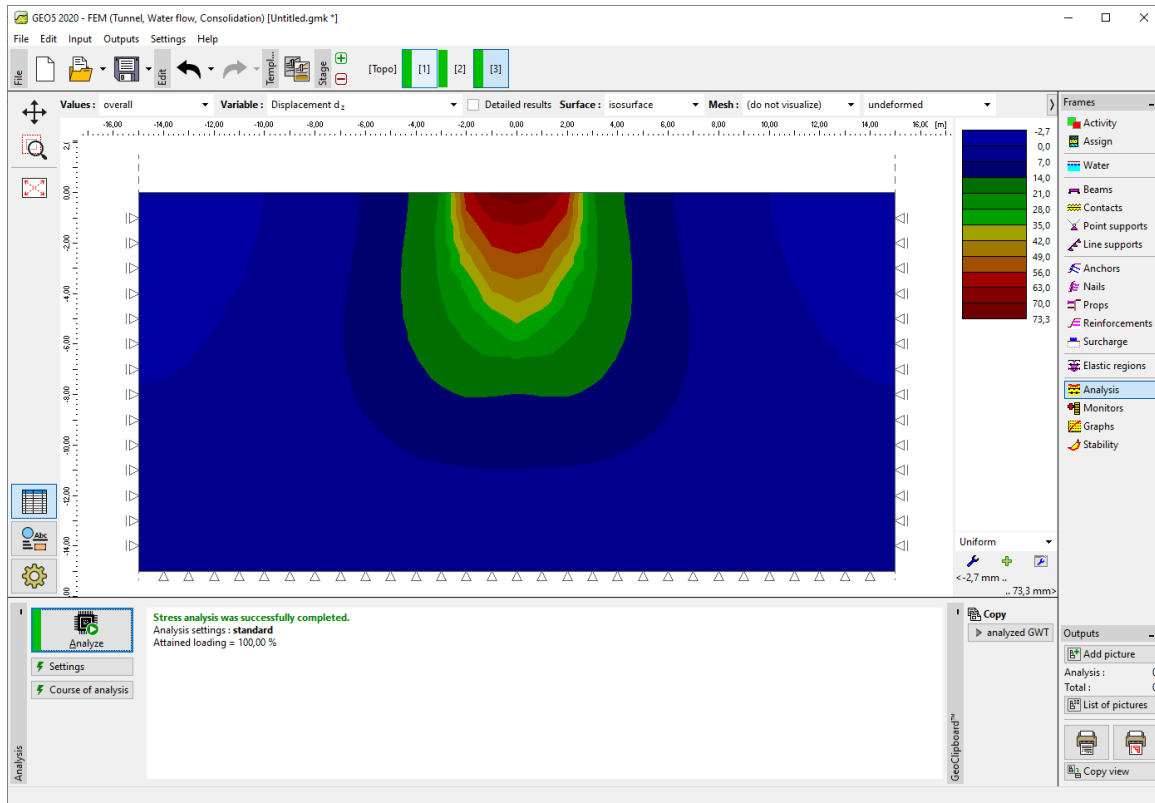
Construction stage 3 analysis – vertical deformation induced by the surcharge d_z [mm]

This completes the basic analysis. We will also carry out other comparative analyses for the other material models.

Assessment of the results

The following table presents the results for the total displacement d_z [mm] at the same example, but for different material models of the GEO5 – FEM program.

Material model / program	Mesh spacing [m]	Stage 2 d_z [mm]	Stage 3 d_z [mm]	Note
Elastic	1.0	88.3	0	---
ELM	1.0	88.2	58.8	---
DP	1.0	114.1	84.8	---
MC	1.0	102.9	73.3	---
MCM	1.0	93.5	64	---
Settlement	---	73.7	---	CSN 73 1001

Total settlement results – summary

*Note: For the purpose of the analytical solution in **GEO5 – Settlement** program we took into consideration the settlement analysis according to the oedometric modulus (in accord with CSN 73 1001 standard) with influence zone restricted by 10% of the initial geostatic stress. We defined the modulus of deformation as $E_{def} = 15.0$ MPa.*

As shown above, the material models in GEO5 FEM allows us to input the modulus for primary loading, denoted as E, and the modulus for unloading and secondary loading, denoted as E_{ur}. On the other hand, GEO5 Settlement accepts only one value of modulus denoted as E_{def}. Since we used GEO5 Settlement to calculate the primary loading, we input the same value as E in FEM.

The FEM analysis with elastic model and the analytical solution in Settlement are both based on theory of linear elasticity. Both models should therefore give similar results. The inevitable difference in the results can be attributed to the following facts:

a) The influence zone in FEM is fixed, given by the model geometry. On the other hand, the influence zone in Settlement depends on the loading and other parameters.

b) In FEM, the stress is calculated from equilibrium and respects vertical and horizontal component of strain. In Settlement, the stress field is calculated a priori with no connection to actual strain field.

c) In FEM the soil under the foundation strip can deform transversally and horizontal displacement occurs. The analytical approach in Settlement uses oedometric modulus and the soil is assumed horizontally constrained.

Conclusions

It is possible to deduce following conclusions from the summary table of the total settlement:

- Drucker-Prager model is in this particular case slightly more compliant than the classical Mohr-Coulomb model or the Modified Mohr-Coulomb material model.

- The settlement computed using the elasto-plastic models is higher than the value obtained using linear model.

- The settlement calculated analytically by GEO5 Settlement roughly correspond to the value computed with the finite element method with linear elastic model. The slight difference in the obtained values can be explained by the different assumptions that these two methods adopt.