Embankment – time evolution of settlement (consolidation)

Program: MKP – Consolidation

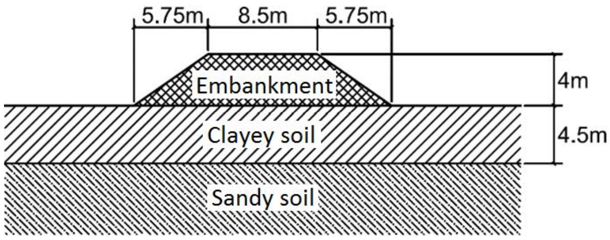
File: Demo_manual_37.gmk

Introduction

This example illustrates the application of the GEO5 FEM – Consolidation module to analyze the time evolution of settlement caused by embankment construction. The objective is to find the evolution of deformation of the embankment and the subsoil caused by a gradual pore pressure redistribution. The outcome of the analysis is the displacement field and pore pressure field at chosen times after the embankment installation.

Task input

The subsoil consists of sandy soil overlaid by 4.5 m thick layer of clayey soil. The cross section of the embankment has trapezoidal shape, being 20 m wide at the base, 8.5 m wide at the top and 4 m high.



The Mohr-Coulomb material model will be used to represent the subsoil and embankment behavior. The model parameters – self weight γ, Young modulus E, Poisson's ratio ν, angle of internal friction φ and cohesion c – are listed in the following table. The parameters $k_{x,sat}$ and $k_{y,sat}$ represent the horizontal and vertical coefficient of permeability of a fully saturated soil. Point out that the implemented numerical solution bases on the assumption of fully saturated soil. Approximate values of the coefficient of permeability for the selected soils are available from the online help for the GEO5 FEM program at http://www.finesoftware.eu/help/geo5/en/coefficient-of-permeability-01/.

	γ	E	ν	φ	c	$k_{x,sat}$	$k_{y,sat}$
	[kN/m^3]	[MPa]	[–]	[°]	[kPa]	[m/day]	[m/day]
Clayey soil	18.5	10	0.4	28	15	10^{-5}	10^{-5}
Sandy soil	19.5	30	0.3	33	2	10^{-2}	10^{-2}
Embankment	20	30	0.3	30	10	10^{-2}	10^{-2}

The initial steady state ground water table is 1 m below the terrain. Our goal is to find the displacement field and pore pressure 7 days, 30 days, 1 year and 10 years after the construction of the embankment.

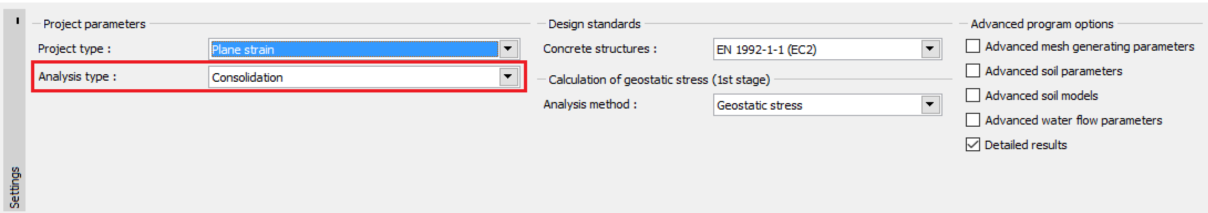
Analysis – entering input data

The project settings, geometry and material parameters are entered in the topology regime [Topo]. The finite element mesh is generated also in this regime. The boundary conditions and the construction of the embankment are introduced subsequently in the calculation stages [1] – [5].

Project settings

In the regime [Topo]->Settings we set the *Plane strain* project type and the *Consolidation* analysis type.

Note: To allow for the visualization of all calculated variables we check also the item Detailed results. In such a case, the program plots, apart from displacements, pore pressures, and flow velocities, also the values of stress and strain components and their invariants.



„Settings" frame

Model geometry

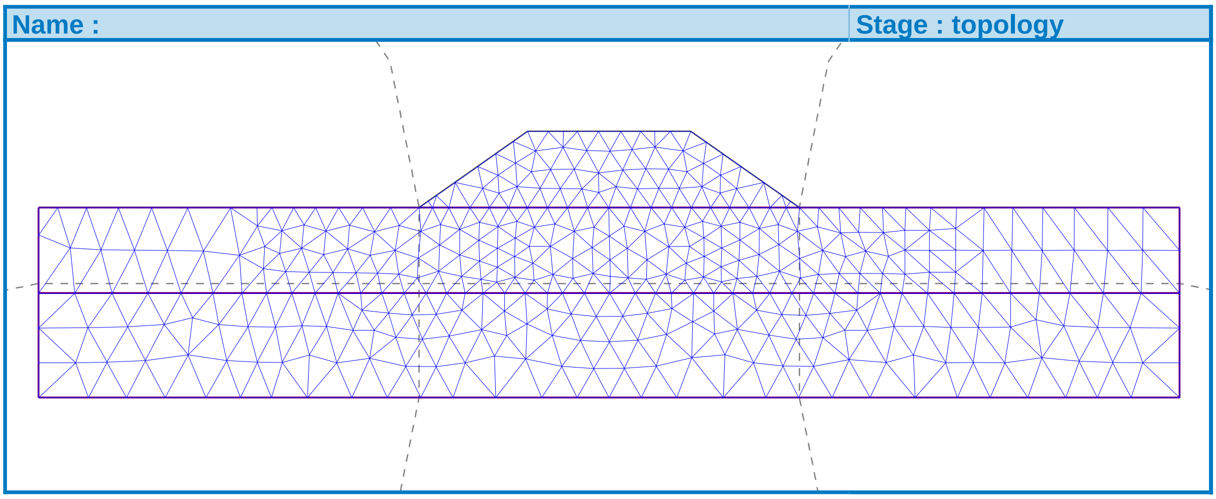
The model dimensions and interfaces between the soils are entered in the [Topo]->Interface frame. Horizontally, the model ranges from -30 to 30 m and it has three interfaces. The first interface locates the original terrain. In his example, it is defined by points having coordinates [-30, 0], [-10, 0], [10, 0] and [30, 0]. The second interface separates the two layers in the subsoil. It is defined by points [-30, -4.5] and [30, -4.5]. The third interface defines the shape of the embankment by points of coordinates [-10, 0], [-4.25, 4], [4.25, 4] and [10, 0]. Finally, we set, in the *Ranges* frame, the depth of the model below the lowest interface to 5.5 m.

Material

The consolidation analysis is a coupled problem controlled by both the mechanical and hydraulic laws. Owing to this, we need to enter both the material parameters used in the standard stress analysis and parameters used in the flow analysis. The material parameters are input in the [Topo]->Soils regime. Here we create three materials adopting the Mohr-Coulomb material model and assign the values from table in section "Task input" to individual model parameters. We consider zero dilatation angle for all materials, $\psi = 0°$. Once created, the materials are assign to their regions in the [Topo]->Assign regime.

Finite element mesh

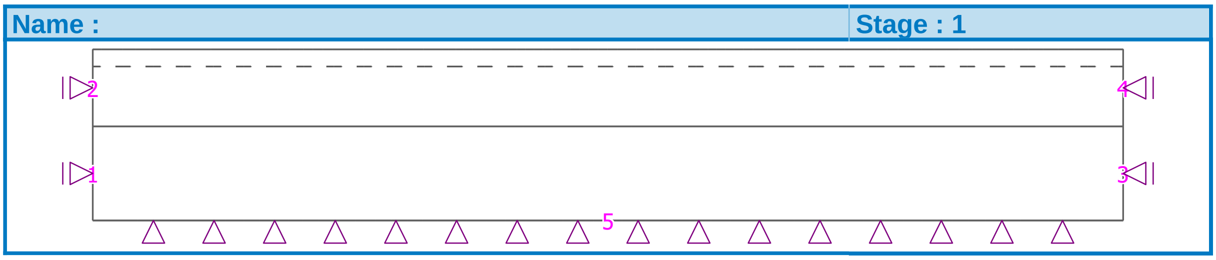
Since the horizontal extent of the model is quite large, it is reasonable to create a mesh with larger elements near the model boundaries where we do not expect a significant deformation. In the [Topo]->Line refinements regime we chose the vertical boundaries and set the element size to 2 m with the radius of 20 m. Similarly, we set the element length to 2 m and the radius to 6 m at the bottom boundary. Finally, in the [Topo]->Mesh generation regime we set the element edge length to 1 m and generate the mesh. The resulting finite element mesh is displayed in the following figure.



Finite element mesh "diluted" along the lateral and the bottom boundaries

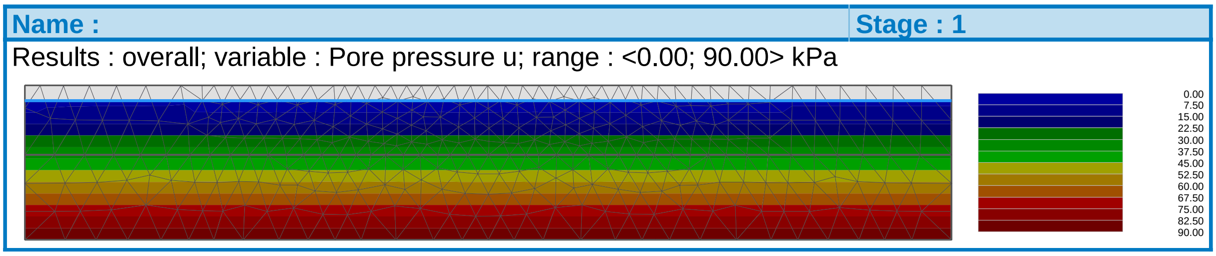
Calculation stage No. 1 – initial geostatic stress

The first calculation stage sets the initial geostatic stress and the initial pore pressure distribution. Only the domain of subsoil is active in this stage while the embankment itself is inactive. This is defined in the Activity frame. Next, we check, in the Assign frame, that the domains are assigned the correct material. The initial ground water table is set in Water frame. Here, the water table is introduced by entering two points having coordinates [-30, -1] and [30, -1]. The boundary conditions are defined in the Line supports frame. Therein we check the item "Generate line supports on project boundaries automatically" to obtain the following supports.



Automatically generated line supports on model boundaries

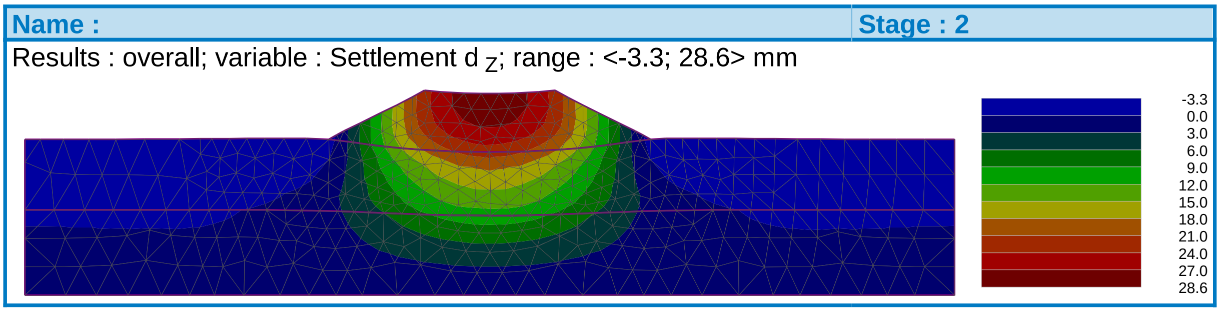
Finally, we run the analysis in the Analysis frame. As a result we obtain the distribution of pore pressure displayed in the following figure. As usual, the displacements are initialized to zero.



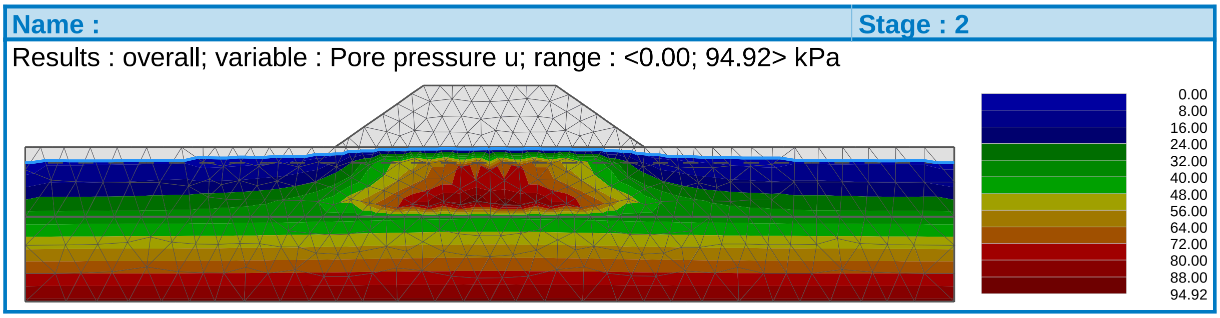
Initial distribution of pore pressure

Calculation stage No. 2 – installation of embankment and analysis of settlement

In the second calculation stage, we simulate the construction of the embankment by activating its domain in the Activity frame. Unlike the first stage, where only the mechanical analysis took place, the second stage requires already to define the hydraulic boundary. These are introduced in the Line flows frame. Here we select the permeable boundary conditions on all boundaries because nothing prevents the ground water to flow in and out of the model region. In the Analysis frame we set the time of the stage duration to 7 days and run the analysis. The resulting distributions of the vertical settlement and pore pressure are displayed in the following two figures.



Distribution of vertical displacements (settlements) 7 days after embankment construction plotted on deformed mesh.

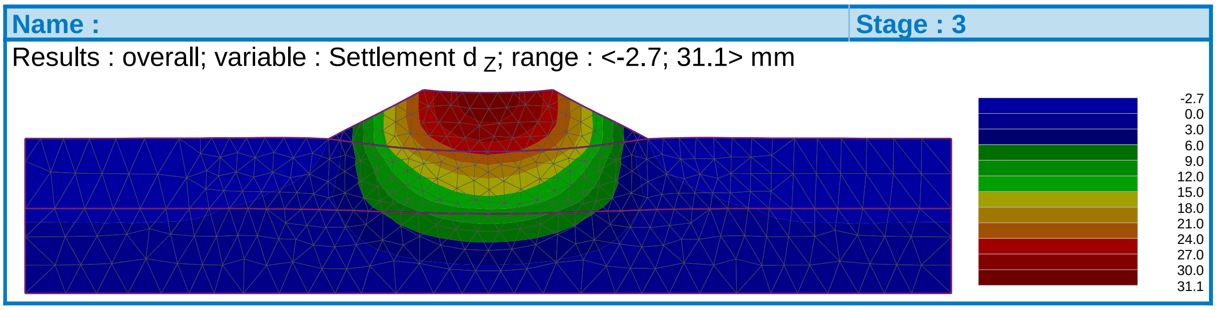


Distribution of pore pressure 7 days after embankment construction plotted on undeformed mesh.

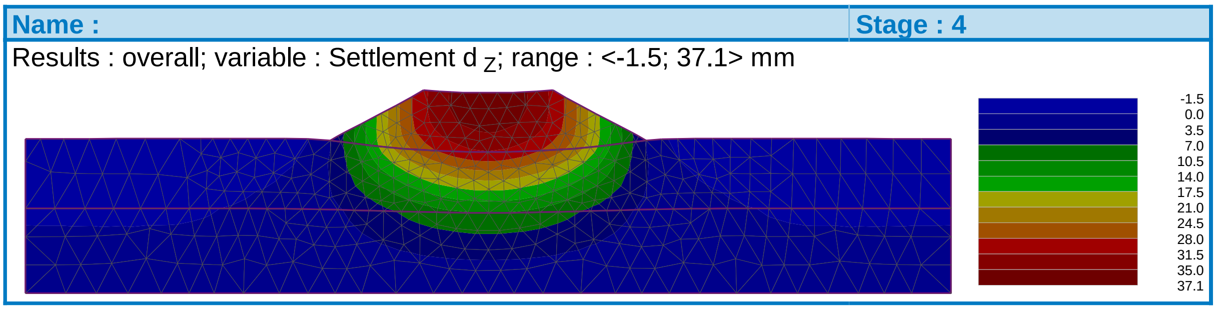
It can be observed that the pore pressure under the embankment has increased. This increased pore pressure helps to carry the increment of vertical stress caused by embankment construction. The increased pore pressure will get redistributed in the following stages causing additional settlement without a change in the vertical load caused by the embankment installation.

Calculation stages No. 3-5 – analysis of subsequent settlement

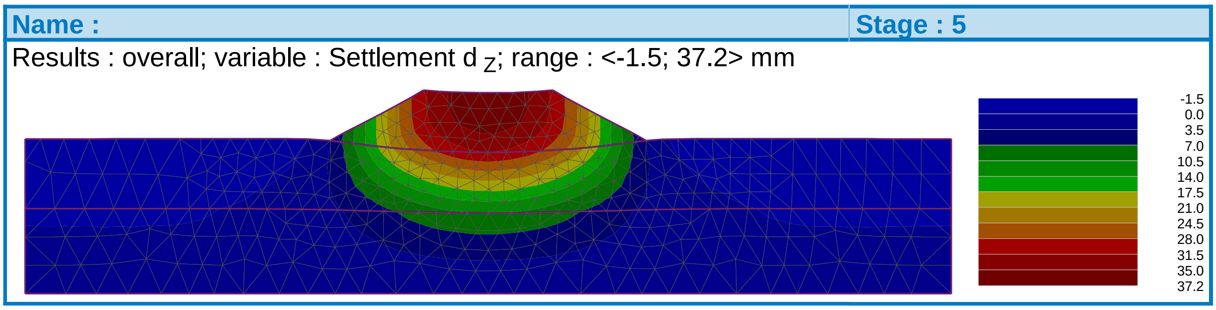
In subsequent stages No. 3, 4 and 5 we shall calculate the distributions of displacements and pore pressures in times 30 days, 365 days and 3650 days after the embankment was installed. The geometry, material and loading remains unchanged and therefore we leave all settings identical to stage No. 2. Prior to running the analysis, we specify the corresponding stage duration. The duration of the stages add up, so we set the duration of the stage No. 3 to 23 days, No. 4 to 335 days and No. 5 to 3285 days, respectively. The resulting evolution of the settlement and the pore pressure are shown in the following figures.



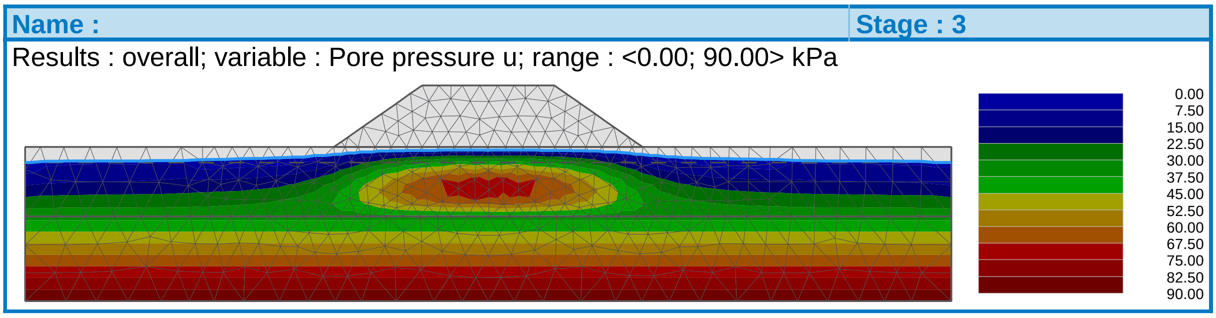
Distribution of vertical displacements (settlements) 30 days after embankment construction plotted on deformed mesh.



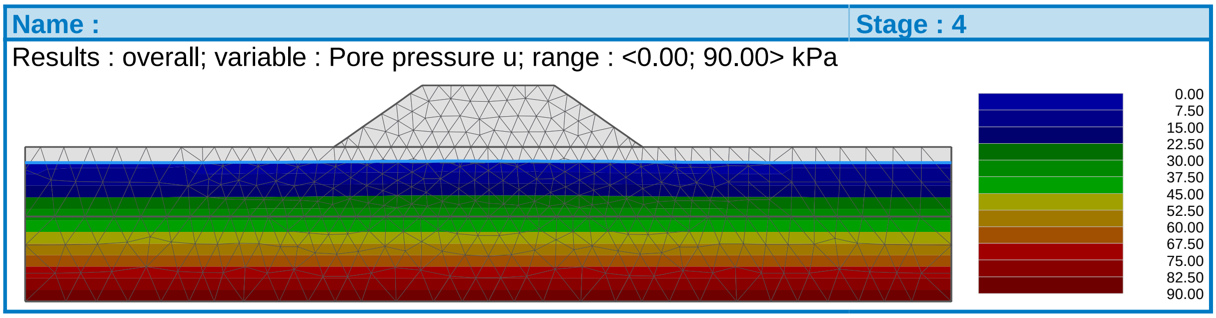
Distribution of vertical displacements (settlements) 1 year after embankment construction plotted on deformed mesh.



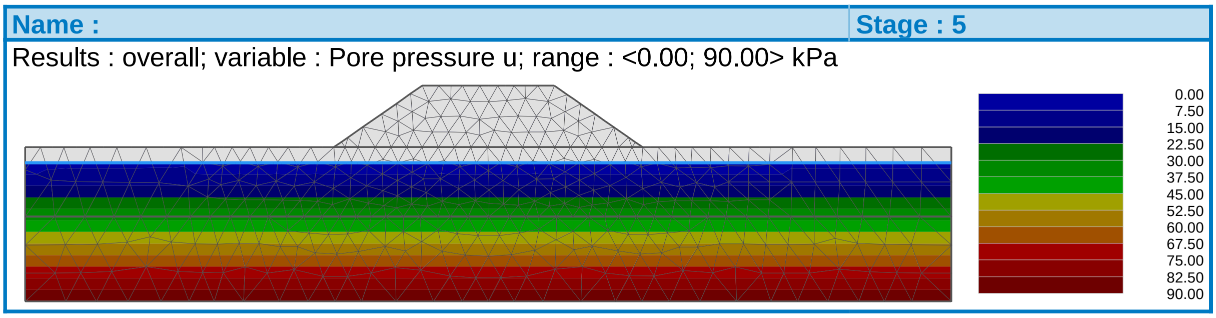
Distribution of vertical displacements (settlements) 10 years after embankment construction plotted on deformed mesh.



Distribution of pore pressure 30 days after embankment construction plotted on undeformed mesh.



Distribution of pore pressure 1 year after embankment construction plotted on undeformed mesh.



Distribution of pore pressure 10 years after embankment construction plotted on undeformed mesh.

Conclusions

The results obtained in stages No. 2 – 5 suggest that the dissipation of pore pressure takes place essentially during the first week after constructing the embankment. The maximum values of the embankment settlement are listed in the following table.

Time	7 days	30 days	1 year	10 years
Settlement	28.6 mm	31.1 mm	37.1 mm	37.2 mm

From the pore pressure distribution, it is clear that 1 year after the embankment is built the ground water table arrives at its initial level. The pore pressure dissipation has already occurred and thus the settlement can be assumed final.