

6

Construction of a road embankment [ADV]

6.1 | Introduction

The construction of an embankment on soft soil with a high groundwater level leads to an increase in pore pressure. As a result of this undrained behaviour, the effective stress remains low and intermediate consolidation periods have to be adopted in order to construct the embankment safely. During consolidation the excess pore pressures dissipate so that the soil can obtain the necessary shear strength to continue the construction process.

This tutorial concerns the construction of a road embankment in which the mechanism described above is analysed in detail. In the analysis two new calculation options are introduced, namely a consolidation analysis and the calculation of a safety factor by means of a safety analysis (phi/c-reduction). It also involves the modelling of drains to speed up the consolidation process.

Objectives

- Modelling drains
- Consolidation analysis
- Change of permeability during consolidation
- Safety analysis (phi-c reduction)

6.2 | Geometry

Figure 6–1 (p. 114) shows a cross section of a road embankment.

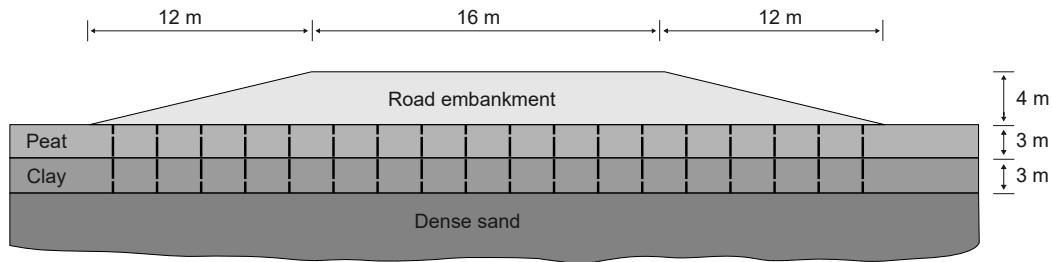


Figure 6–1: Situation of a road embankment on soft soil

6.3 | Create a new project

The embankment is 16 m wide. The slopes have an inclination of 1: 3. The problem is symmetric, so only one half is modelled (in this case the right half is chosen). A representative section of 2 m is considered in the project. The embankment itself is composed of loose sandy soil. The subsoil consists of 6 m of soft soil. The upper 3 m of this soft soil layer is modelled as a peat layer and the lower 3 m as clay. The phreatic level is located 1 m below the original ground surface. Under the soft soil layers there is a dense sand layer of which 4 m are considered in the model.

- 1 Start the Input program and select **Start a new project** from the **Quick select** dialog box.
- 2 In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
- 3 Keep the default units and set the model dimensions to
 - a. $x_{\min} = 0$ and $x_{\max} = 60$.
 - b. $y_{\min} = 0$ and $y_{\max} = 2$.

6.4 | Define the soil stratigraphy

The soil layers comprising the embankment foundation are defined using a borehole. The embankment layers are defined in the **Structures mode**.

- 1 Click the **Create borehole** button  and create a borehole at (0 0 0).

The **Modify soil layers** window pops up.
- 2 Define three soil layers as shown in [Figure 6–2 \(p. 115\)](#).
- 3 The water level is located at $z = -1$ m. In the borehole column specify a value of -1 to **Head**.

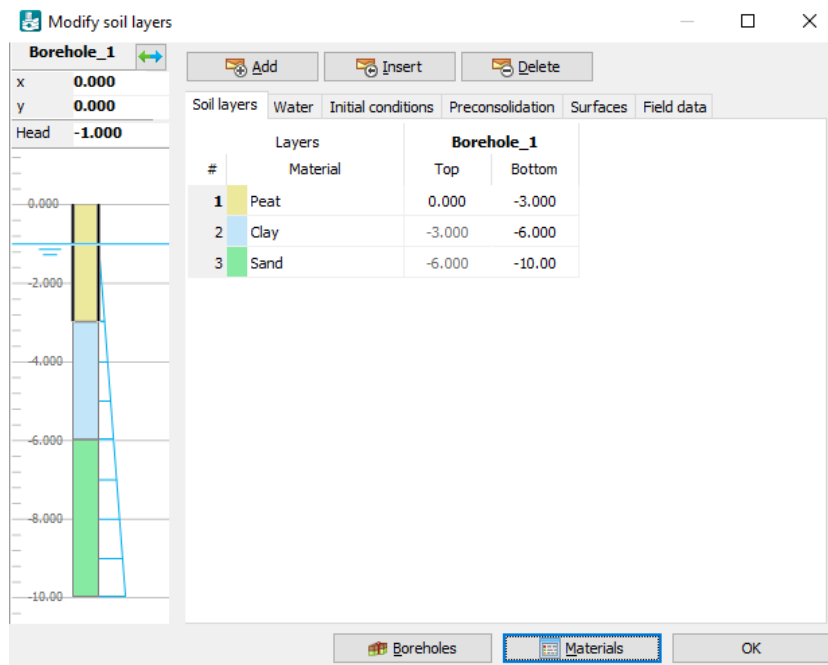


Figure 6-2: Soil layer distribution

6.5 | Create and assign the material data sets

The material properties for the data sets are shown in the table below.

Table 6-1: Material properties of the road embankment and subsoil

Parameter	Name	Embankment	Sand	Peat	Clay	Unit
General						
Soil model	Model	Hardening Soil	Hardening Soil	Soft Soil model	Soft Soil model	-
Type of material behaviour	Type	Drained	Drained	Undrained (A)	Undrained (A)	-
Unsaturated unit weight	γ_{unsat}	16	17	8	15	kN/m ³
Saturated unit weight	γ_{sat}	19	20	12	18	kN/m ³
Initial void ratio	e_{init}	0.5	0.5	2.0	1.0	-
Mechanical						
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.5 \cdot 10^4$	$3.5 \cdot 10^4$	-	-	kN/m ²

Mechanical						
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$2.5 \cdot 10^4$	$3.5 \cdot 10^4$	-	-	kN/m ²
Unloading / reloading stiffness	E_{ur}^{ref}	$7.5 \cdot 10^4$	$1.05 \cdot 10^5$	-	-	kN/m ²
Modified compression index	λ^*	-	-	0.15	0.05	-
Modified swelling index	κ^*	-	-	0.03	0.01	-
Power for stress-level dependency of stiffness	m	0.5	0.5	-	-	-
Cohesion (constant)	c'_{ref}	1.0	0.0	2.0	1.0	kN/m ²
Friction angle	φ'	30	33	23	25	°
Dilatancy angle	ψ	0.0	3.0	0	0	°

Groundwater						
Classification type	-	USDA	USDA	USDA	USDA	-
Model	-	Van Genuchten	Van Genuchten	Van Genuchten	Van Genuchten	-
Soil class	-	Loamy sand	Sand	Clay	Clay	-
<2µm	-	6.0	4.0	70.0	70.0	%
2µm - 50µm	-	11.0	4.0	13.0	13.0	%
50µm - 2mm	-	83.0	92.0	17.0	17.0	%
Use defaults	-	From data set	From data set	None	From data set	-
Horizontal permeability (x-direction)	k_x	3.499	7.128	0.1	0.04752	m/day
Horizontal permeability (y-direction)	k_y	3.499	7.128	0.1	0.04752	m/day

Groundwater						
Vertical permeability	k_z	3.499	7.128	0.05	0.04752	m/day
Void ratio dependency	-	No	No	Yes	Yes	
Change in permeability	c_k	$1 \cdot 10^{15}$	$1 \cdot 10^{15}$	1.0	0.2	-
Interfaces						
Strength determination	-	Rigid	Rigid	Rigid	Rigid	-
Interface reduction factor	R_{inter}	1.0	1.0	1.0	1.0	-
Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-
Pre-overburden pressure	POP	0.0	0.0	5.0	0.0	kN/m ²
Over-consolidation ratio	OCR	1.0	1.0	1.0	1.0	-

Note:

The initial void ratio (e_{init}) and the change in permeability (c_k) should be defined to enable the modelling of a change in the permeability due to compression of the soil. This option is recommended when using advanced models.

- 1 Click the **Materials** button .
- 2 Create soil material data sets according to [Table 6-1 \(p. 115\)](#) and assign them to the corresponding layers in the borehole ([Figure 6-2 \(p. 115\)](#)).
- 3 Close the **Modify soil layers** window and proceed to the **Structures mode** to define the structural elements.

6.6 | Definition of embankment and drains

The embankment and the drains are defined in the **Structures mode**. To define the embankment layers:

1.  Reorientate the model such that the front view is displayed by clicking the corresponding button in the toolbar.

2.  Create a surface by defining points at (0 0 0), (0 0 4), (8 0 4) and (20 0 0).
3.  Create a line passing through (0 0 2) and (14 0 2) to define the embankment layers.
4.  Select both the created line and surface by keeping the **Ctrl** key pressed while clicking them in the model.
5.  Click the **Extrude object** button. Assign a value of 2 to the y-component of the extrusion vector as shown [Figure 6–3 \(p. 118\)](#) and click **Apply**.

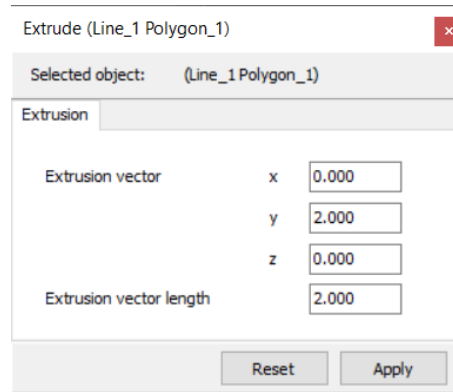


Figure 6–3: Extrusion window

6. Delete the surface and the line with its corresponding points that were created before the extrusion.
7. **Right-click** the volume created by extrusion and select the menu item **Soil_4 > Set material > Embankment**.

In this project the effect of the drains on the consolidation time will be investigated by comparing the results with a case without drains. Drains will only be active for the calculation phases..

Drains are arranged in a square pattern, having a distance of 2m between two consecutive drains in a row (or column). Only one row of drains will be considered in this tutorial. To create the drain pattern:

1.  Click the **Create hydraulic conditions** button in the side toolbar.
2.  Click the **Create line drain** button in the appearing menu. Define a line drain in the model between points (1 1 0) and (1 1 -6).
3.  Click the **Create array** button to define the drain pattern.
4. In the **Create array** window select the menu item **Shape > 1D, in x direction** in the drop-down menu and specify the pattern as shown in [Figure 6–4 \(p. 119\)](#).

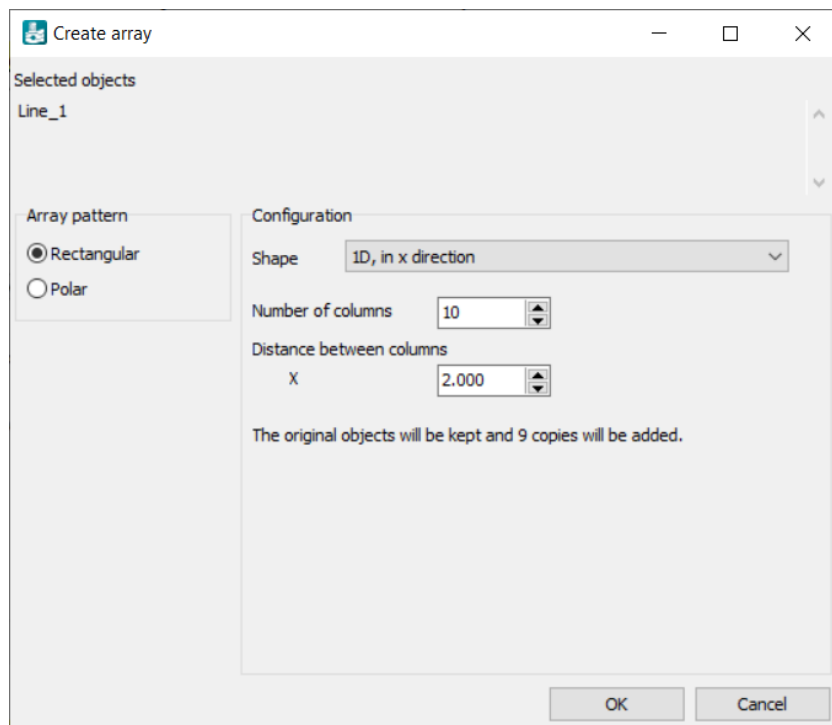


Figure 6–4: Settings of the drain pattern

The model geometry is shown in [Figure 6–5 \(p. 119\)](#):

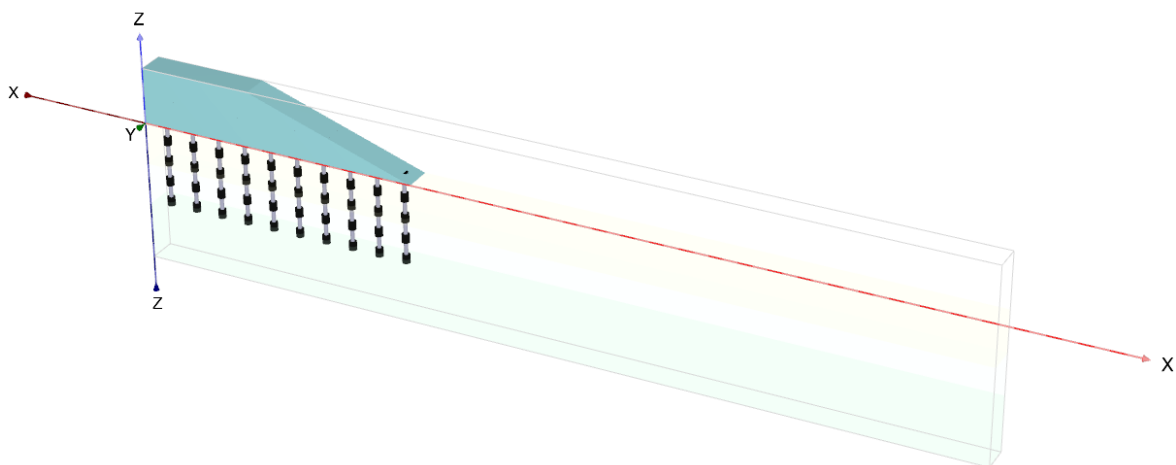


Figure 6–5: Model geometry

6.7 | Generate the mesh

- 1 Proceed to the **Mesh mode**.
- 2 Select all volumes, including the embankment and in the **Selection explorer** set the **Coarseness factor** to 0.3.
- 3  Click the **Generate mesh** button. Set the element distribution to **Coarse**.

4



View the generated mesh.

The resulting mesh is shown in [Figure 6–6 \(p. 120\)](#).

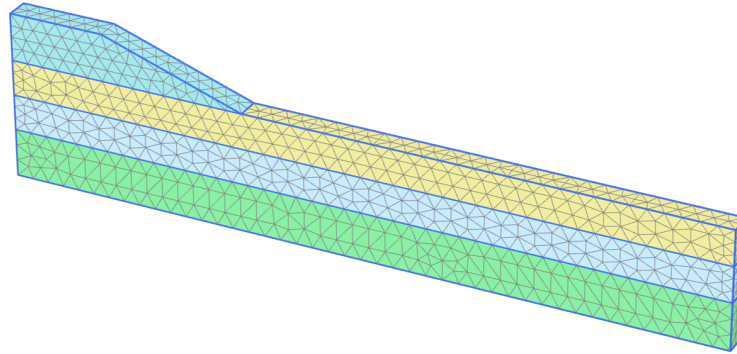


Figure 6–6: The generated mesh

6.8 | Define the calculation

The embankment construction process will be considered twice. In the first calculation the drains will not be considered.

6.8.1 | Initial phase

In the initial situation the embankment is not present. Therefore, the corresponding soil volumes are deactivated in the initial phase. The **K0 procedure** can be used to calculate the initial stresses. The initial water pressures are fully hydrostatic and based on a general phreatic level defined by the **Head** value assigned to the boreholes. For the **Initial phase**, the **Phreatic** option is selected for the **Pore pressure calculation type**. The **Global water level** is set to **BoreholeWaterlevel_1** corresponding to the water level defined by the heads specified for the boreholes.

The boundary conditions for flow can be specified in the **Model explorer** by selecting **Model conditions > Groundwater**. In the current situation the left vertical boundary (Xmin) must be closed because of symmetry, so horizontal flow should not occur. The bottom is open because the excess pore pressures can freely flow into the deep and permeable sand layer. The upper boundary is obviously open as well. The view of the **GroundwaterFlow** subtree after the definition is given in the [Figure 6–7 \(p. 121\)](#).

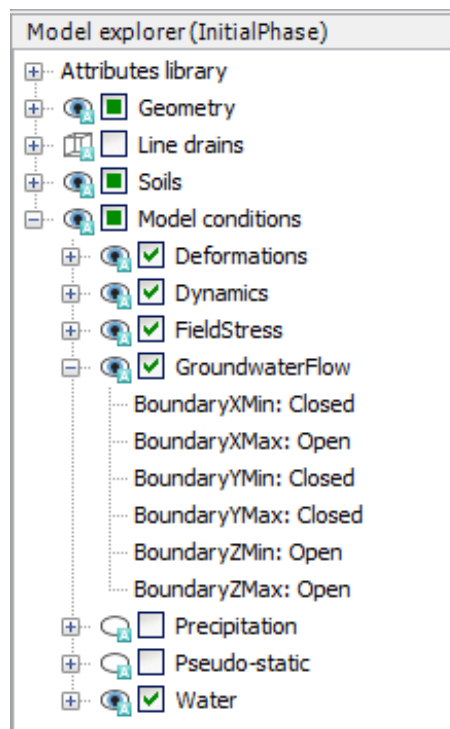


Figure 6-7: Boundary conditions for groundwater flow

6.8.2 | Consolidation analysis

A consolidation analysis introduces the dimension of time in the calculations. In order to correctly perform a consolidation analysis a proper time step must be selected. The use of time steps that are smaller than a critical minimum value can result in stress oscillations. The consolidation option in PLAXIS 3D allows for a fully automatic time stepping procedure that takes this critical time step into account. Within this procedure there are three main possibilities for the **Loading type** parameter:

- Consolidate for a predefined period, including the effects of changes to the active geometry (**Staged construction**).
- Consolidate until all excess pore pressures in the geometry have reduced to a predefined minimum value (**Minimum excess pore pressure**).
- Consolidate until the soil has reached a specified degree of consolidation (**Degree of consolidation**).

6.8.2.1 | Consolidation process - No drains

The embankment construction is divided into two phases. After the first construction phase, a consolidation period of 30 days is introduced to allow the excess pore pressures to dissipate. After the second construction phase another consolidation period is introduced from which the final settlements may be determined.

To define the calculation phases, follow these steps:

6.8.2.1.1 | Phase 1 - First Construction

1.  Click the **Add phase** button to introduce the first construction phase.

2.  In the **General** subtree in the **Calculation type** select the **Consolidation** option from the drop-down menu.
3.  The **Loading type** is by default set to **Staged construction**. This option will be used for this phase.
4.  The **Phreatic** option is automatically selected for the pore pressure calculation type. Note that the global water level for a calculation phase can be defined in the subtree **Model explorer > Model conditions > Water**. . .
5. Specify a value of 2 days to the **Time interval** and click **OK** to close the **Phases** window.
6. In the **Staged construction mode** activate the first part of the embankment.

6.8.2.1.2 | Phase 2 - First consolidation

The second phase is also a **Consolidation** analysis. In this phase no changes to the geometry are made as only a consolidation analysis to ultimate time is required.

1.  Click the **Add phase** button to introduce the next calculation phase.
2.  Define the calculation type as **Consolidation**.
3. Specify a value of 30 days to the **Time interval**. The default values of the other parameters are used for this phase.

6.8.2.1.3 | Phase 3 - First construction

1.  Click the **Add phase** button to introduce the next calculation phase.
2.  Define the calculation type as **Consolidation**.
3. Specify a value of 1 day to the **Time interval**. The default values of the other parameters are used.
4. In the **Staged construction** mode activate the second part of the embankment.

6.8.2.1.4 | Phase 4 - Long term consolidation

The fourth phase is a **Consolidation** analysis to a minimum excess pore pressure.

1.  Click the **Add phase** button to introduce the next calculation phase.
2.  Define the calculation type as **Consolidation**.
3.  In the **Loading type** Select the **Minimum excess pore pressure** option from the drop-down menu. The default value for the minimum pressure ($|P\text{-stop}| = 1.0 \text{ kN/m}^2$) as well as the default values for other parameters are used.

The definition of the calculation phases is complete.

6.8.3 | Execute the calculation

1.  Before starting the calculation, click the **Select points for curves** button and select the following points: As the first node, select the toe of the embankment at (20 0 0). The second node will be used to plot the development (and decay) of excess pore pressures. To this end, a point somewhere in the middle of the soft soil layers is needed, close to (but not actually on) the left boundary (e.g. (0.7 0 -3)).

2



Start the calculation.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window (see [Figure 6–8 \(p. 123\)](#)). In addition to the multipliers, a parameter $P_{\max}$ occurs, which indicates the current maximum excess pore pressure. This parameter is of interest in the case of a **Minimum excess pore pressure** consolidation analysis, where all pore pressures are specified to reduce below a predefined value.

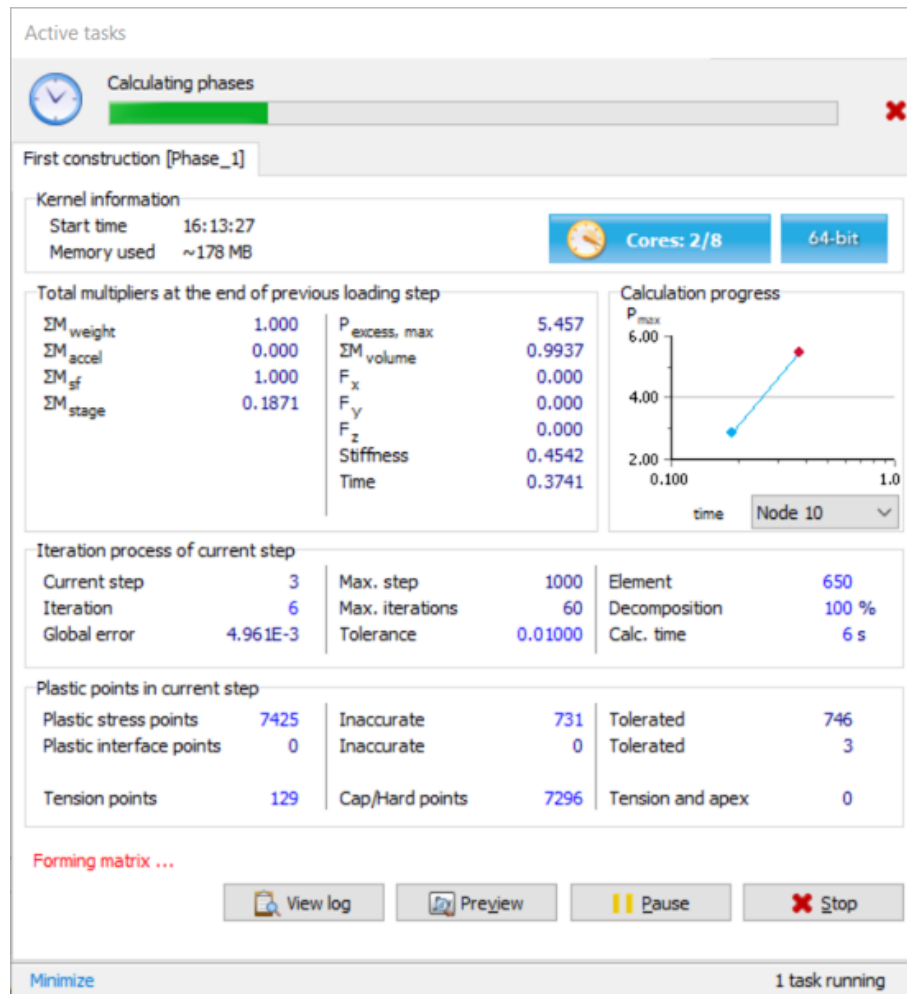


Figure 6–8: Calculation progress displayed in the *Active tasks* window

6.9 | Results

 After the calculation has finished, select the third phase and click the **View calculation results** button. The **Output** window now shows the deformed mesh after the undrained construction of the final part of the embankment. Considering the results of the third phase, the deformed mesh shows the uplift of the embankment toe and hinterland due to the undrained behaviour (see [Figure 6–9 \(p. 124\)](#)).

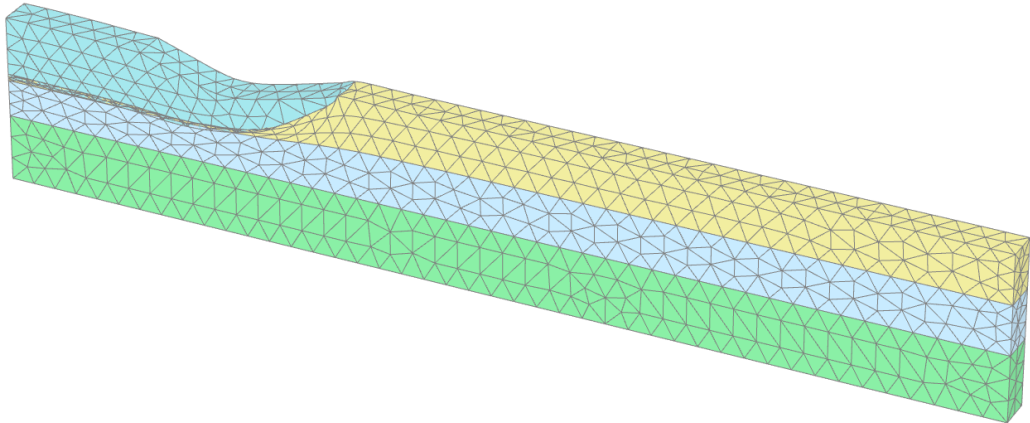


Figure 6–9: Deformed mesh after undrained construction of embankment (Phase 3)

1. Select the menu item **Deformations > Incremental displacements > $|\Delta u|$** .
2.  Select the menu item **View > Arrows** or click the corresponding button in the toolbar to display the results arrows.

On evaluating the total displacement increments, it can be seen in [Figure 6–10 \(p. 124\)](#) that a failure mechanism is developing:

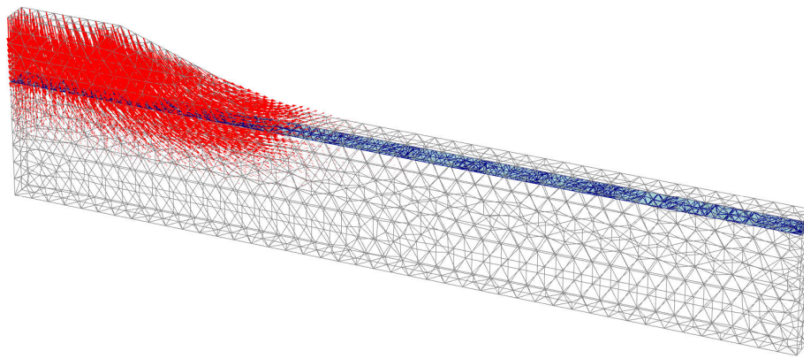


Figure 6–10: Displacement increments after undrained construction of embankment

1. Click **Ctrl** + **7** to display the developed excess pore pressures (see Appendix G of the PLAXIS 3D [Reference Manual](#) for more shortcuts). They can be displayed by selecting the corresponding option in the side menu displayed as the **Pore pressures** option is selected in the **Stresses** menu.
2.  Click the **Center principal directions**. The principal directions of excess pressures are displayed at the center of each soil element. For a coloured visualization, in the general toolbar select **Coloured principal directions**. The results are displayed in [Figure 6–11 \(p.](#)

125). It is clear that the highest excess pore pressure occurs under the embankment centre.

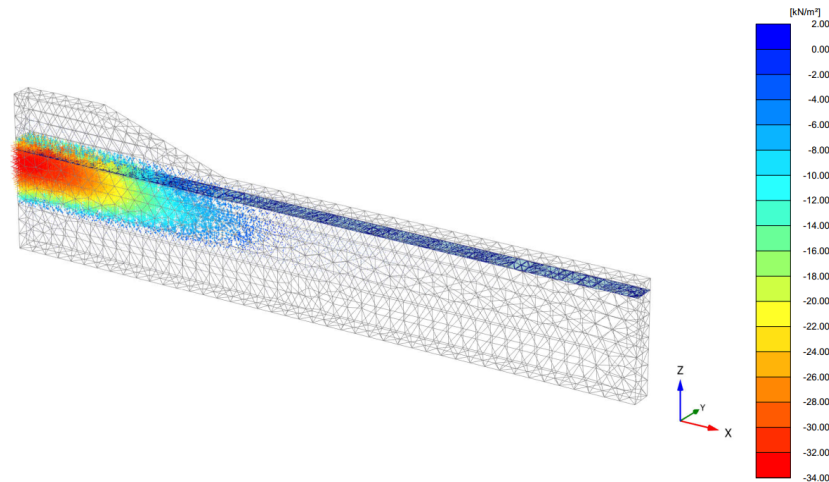


Figure 6–11: Excess pore pressures after undrained construction of embankment (Phase 3)

1. Select Phase 4 in the drop down menu.
2. Define a vertical cross section passing through (0 1) and (60 1).
3. Click the **Contour lines** button in the toolbar to display the results as contours.
4. Select the menu item **View > Viewpoint**. The corresponding window pops up.
5. In the **Viewpoint** window select the **Front view** option as shown in [Figure 6–12 \(p. 125\)](#):

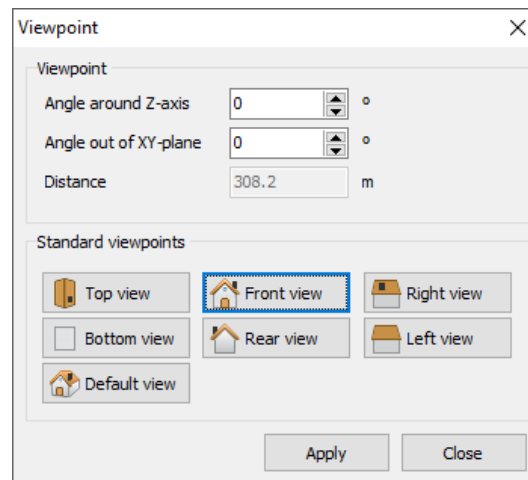


Figure 6–12: Viewpoint window

6. Use the **Draw scanline** button or the corresponding option in the **View** menu to define the position of the contour line labels.

It can be seen that the settlement of the original soil surface and the embankment increases considerably during the fourth phase. This is due to the dissipation of the excess pore pressures (= consolidation), which causes further settlement of the soil. The figure below shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below 1.0 kN/m^2 .

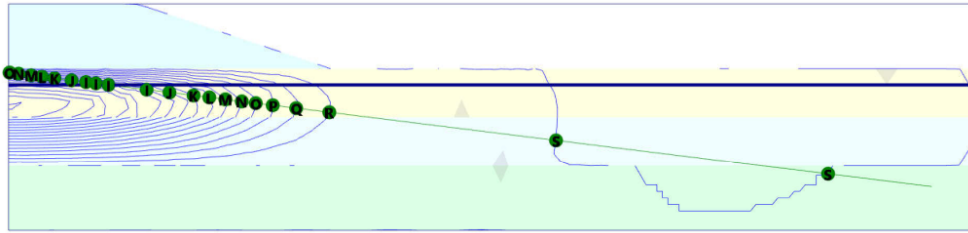


Figure 6–13: Excess pore pressure contours after consolidation to $P_{\text{excess}} < 1.0 \text{ kN/m}^2$

The **Curves manager** can be used to view the development with time of the excess pore pressure under the embankment. In order to create such a curve, follow these steps:

1.  Click the **Curves manager** button in the toolbar. The corresponding window pops up.
2. In the **Charts** tabsheet click **New**. The **Curve generation** window pops up
3. For the x-axis, select the **Project** option from the drop-down menu and select **Time** in the tree.
4. For the y-axis, select the point in the middle of the soft soil layers (the second node (0.7 0.0 -3)) from the drop-down menu. In the tree select **Stresses > Pore pressure > p excess**.
5. Select the **Invert sign** option for y-axis.
6. Click **Ok** to generate the curve.
7.  Click the **Settings** button in the toolbar. The **Settings** window will appear displaying the tabsheet of the created curve.
8. In the tabsheet corresponding to the studied node, click the **Phases** button and select the phases 1 to 4 in the appearing window.
9. Rename the curve by typing Phases 1 - 4 in the **Curve title** cell.
10. Click **Apply** to update the plot.
11.  Save the chart in Output and save the project in Input.

Note: To display the legend inside the chart area right-click on the name of the chart and select the menu item **View > Legend in chart**.

[Figure 6–14 \(p. 127\)](#) shows the four calculation phases. During the construction phases the excess pore pressure increases with a small increase in time while during the consolidation periods the excess pore pressure decreases with time. In fact, consolidation already occurs during construction of the embankment, as this involves a small time interval.

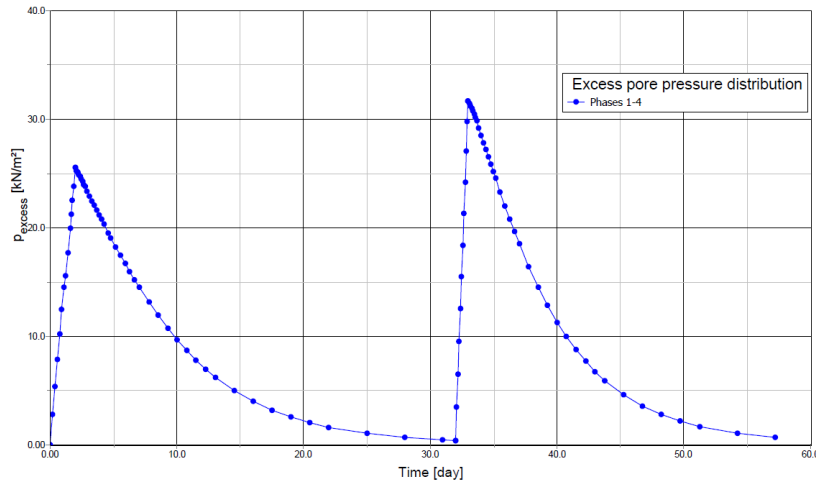


Figure 6-14: Development of excess pore pressure under the embankment

6.10 | Safety analysis

6.10.1 | General considerations

In the design of an embankment it is important to consider not only the final stability, but also the stability during the construction. It is clear from the output results that a failure mechanism starts to develop after the second construction phase.

It is interesting to evaluate a global safety factor at this stage of the problem, and also for other stages of construction.

In structural engineering, the safety factor is usually defined as the ratio of the collapse load to the working load. For soil structures, however, this definition is not always useful. For embankments, for example, most of the loading is caused by soil weight and an increase in soil weight would not necessarily lead to collapse. Indeed, a slope of purely frictional soil will not fail in a test in which the self weight of the soil is increased (like in a centrifuge test). A more appropriate definition of the factor of safety is therefore:

$$Safety\ factor = \frac{S_{\text{maximum available}}}{S_{\text{needed for equilibrium}}}$$

Where S represents the shear strength. The ratio of the true strength to the computed minimum strength required for equilibrium is the safety factor that is conventionally used in soil mechanics. By introducing the standard Coulomb condition, the safety factor is obtained:

$$Safety\ factor = \frac{c - \sigma_n \tan(\varphi)}{c_r - \sigma_n \tan(\varphi_r)}$$

Where c and φ are the input strength parameters and σ_n is the actual normal stress component. The parameters c_r and φ_r are reduced strength parameters that are just large enough to maintain equilibrium. The principle described above is the basis of a **Safety** analysis that can be used in PLAXIS 3D to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:

$$\frac{c}{c_r} = \frac{\tan(\varphi)}{\tan(\varphi_r)} = \Sigma M s f$$

The reduction of strength parameters is controlled by the total multiplier ΣMsf . This parameter is increased in a step-by-step procedure until failure occurs. The safety factor is then defined as the value of ΣMsf at failure, provided that at failure a more or less constant value is obtained for a number of successive load steps.

The **Safety** calculation option is available in the **Calculation type** drop-down menu in the **Phases** window.

6.10.2 | Define the calculation

To calculate the global safety factor for the road embankment at different stages of construction, follow these steps:

1.  Add a new calculation phase.
2. In the **Phases** window in **Start from phase** select Phase 1 from the drop-down menu.
3.  In the **General** subtree as calculation type select **Safety**.
4.  The **Loading type** is automatically changed to **Incremental multipliers**.
5. The first increment of the multiplier that controls the strength reduction process, **Msf**, is set automatically to 0.1. This value will be used in this tutorial.
6.  Note that the **Use pressures from the previous phase** option in the **Pore pressure calculation type** drop-down menu is automatically selected and grayed out indicating that this option cannot be changed
7. In order to exclude existing deformations from the resulting failure mechanism, select the **Reset displacements to zero** option in the **Deformation control parameters** subtree. The default values of all the remaining parameters will be used. The first safety calculation has now been defined.
8. Follow the same steps to create new calculation phases that analyse the stability at the end of each consolidation phase. In addition to selecting **Safety** as calculation type, select the corresponding consolidation phase as the **Start from phase** parameter. The **Phases** explorer displaying the **Safety** calculation phases is shown in [Figure 6–15 \(p. 128\)](#).

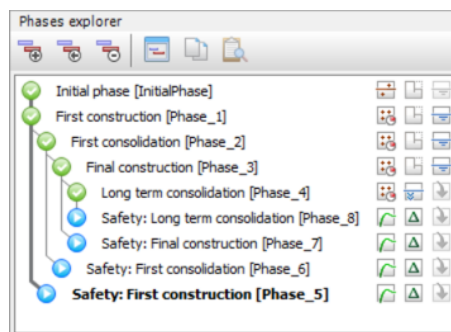


Figure 6–15: Phases explorer displaying the Safety calculation phases

9.  Calculate the safety phases.

Note:

- The default value of **Max steps** in a Safety calculation is 100. In contrast to a **Staged construction** calculation, the specified number of steps is always fully executed. In most **Safety** calculations, 100 steps are sufficient to arrive at a state of failure. If not, the number of steps can be increased to a maximum of 10000.
- For most Safety analyses $M_{sf} = 0.1$ is an adequate first step to start up the process. During the calculation process, the development of the total multiplier for the strength reduction, ΣM_{sf} , is automatically controlled by the load advancement procedure.

6.10.3 | Evaluation of the results - Safety

Additional displacements are generated during a Safety calculation. The total displacements do not have a physical meaning, but the incremental displacements in the final step (at failure) give an indication of the likely failure mechanism.

In order to view the mechanisms in the three different stages of the embankment construction:

1.  Select the last **Safety** phase (Phase_8) and click the **View calculation results** button.
2. Select the menu item **Deformations > Incremental displacements > $|\Delta u|$** .
3.  Change the presentation from **Arrows** to **Shadings**. The resulting plots give a good impression of the failure mechanisms (see [Figure 6-16 \(p. 129\)](#)). The magnitude of the displacement increments is not relevant.

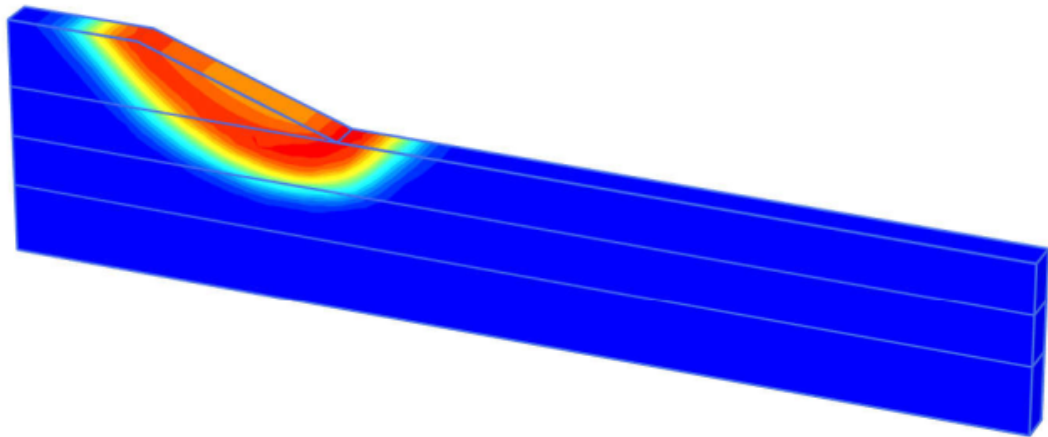


Figure 6-16: Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

The safety factor can be obtained from the **Calculation info** option of the **Project** menu. The value of ΣM_{sf} represents the safety factor, provided that this value is indeed more or less constant during the previous few steps.

The best way to evaluate the safety factor, however, is to plot a curve in which the parameter ΣM_{sf} is plotted against the displacements of a certain node. Although the displacements are not relevant, they indicate whether or not a failure mechanism has developed.

In order to evaluate the safety factors for the four situations, follow these steps:

1. Click the **Curves manager** button in the toolbar.
2. Click **New** in the **Charts** tabsheet.

3. In the **Curve generation** window, select the embankment toe (the first node) for the x-axis. Select the menu item **Deformations > Total displacements > |u|**.
4. For the y-axis, select **Project** and then select **Multiplier > ΣMsf** . The **Safety** phases are considered in the chart. As a result, the curve of appears.
5. Set x-axis interval maximum to 1 in **Chart** tab.

Figure 6–17 (p. 130) displays the safety factor for the calculated phases.

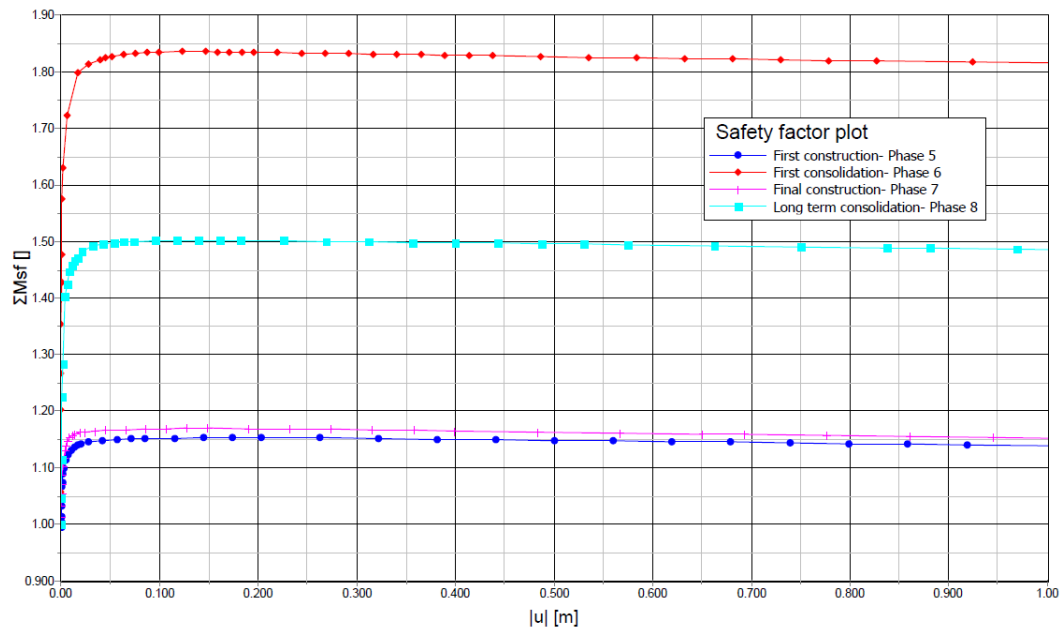


Figure 6–17: Evaluation of safety factor

The maximum displacements plotted are not relevant. It can be seen that for all curves a more or less constant value of ΣMsf is obtained. Hovering the mouse cursor over a point on the curves, a box showing the exact value of ΣMsf can be obtained.

6.11 | Using drains

In this section the effect of the drains in the project will be investigated. The embankment constructions will be redefined by introducing four new phases having the same properties as the first four consolidation phases. The differences in the new phases are:

- The drains should be active in all the new phases.
- The **Time interval** in the first three of the consolidation phases (1 to 3) is 1 day. The last phase is set to **Minimum excess pore pressure** and a value of 1.0 kN/m^2 is assigned to the minimum excess pressure ($|P\text{-stop}|$).

1.  After the calculation is finished, select the last phase and click the **View calculation results** button. The **Output** window now shows the deformed mesh after the drained construction of the final part of the embankment. In order to compare the effect of the drains, the excess pore pressure dissipation in node (0.7 0 -3) can be used.
2.  Open the **Curves manager**.
3. In the **Chart** tabsheet double-click Chart 1 (p_{excess} of node (0.7 0 -3) versus time). Select the displaying phases to Phases 1-4. Close the **Curves manager**.
4.  Click the **Settings** button in the toolbar. The **Settings** window pops up.

- Click the **Add curve** button and select the **Add from current project** option in the appearing menu. The **Curve generation** window pops up.

Note: Instead of adding a new curve, the existing curve can be regenerated using the corresponding button in the **Curves settings** window.

- Select the **Invert sign** option for y-axis.
- Click **OK** to accept the selected options and close the **Curve generation** window.
- In the chart a new curve is added and a new tabsheet corresponding to it is opened in the **Settings** window.
- Click the **Phases** button. From the displayed window select the **Initial phase** and the last four phases (drains) and click **OK**.
- In the **Settings** click **Apply** to preview the generated curve.
- Click **OK** to close the **Settings** window. [Figure 6-18 \(p. 131\)](#) gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate.

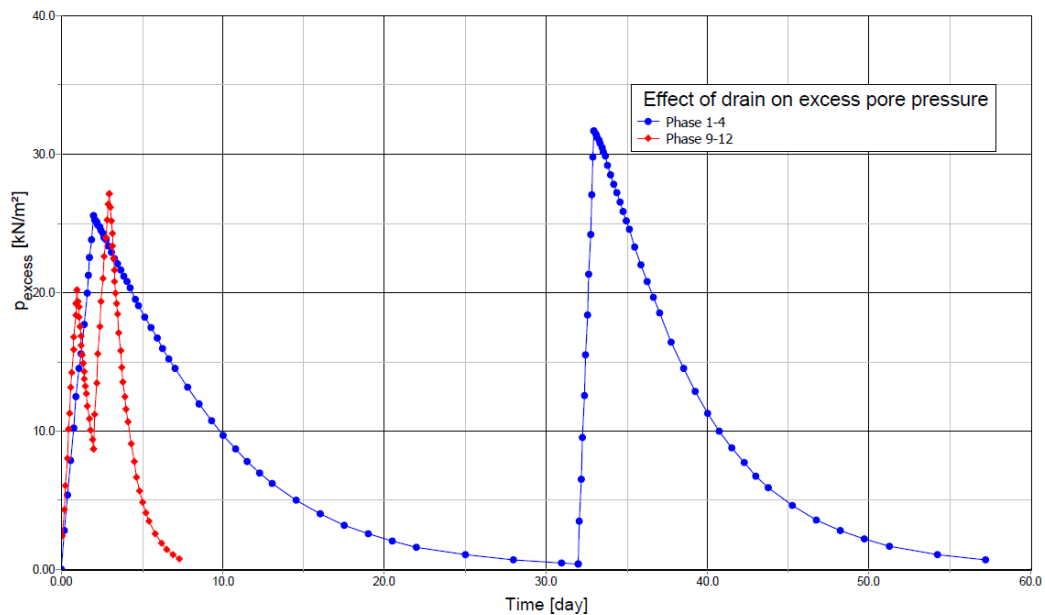


Figure 6-18: Effect of drains