

4 CONSTRUCTION OF A ROAD EMBANKMENT

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 (ϕ/c -reduction). It also involves the modelling of drains to speed up the consolidation process.

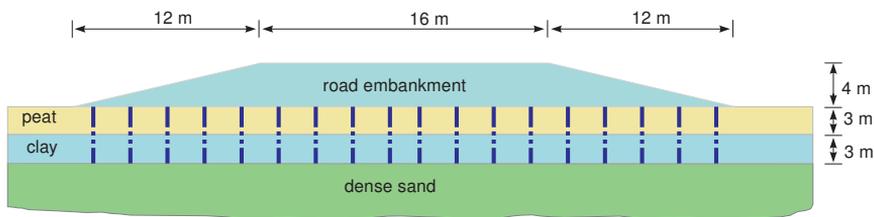


Figure 4.1 Situation of a road embankment on soft soil

Objectives:

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

4.1 GEOMETRY

Figure 4.1 shows a cross section of a road embankment. 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.

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

4.1.1 DEFINITION OF SOIL STRATIGRAPHY

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



Create a borehole by clicking at (0 0 0). The *Modify soil layers* window pops up.

- Define three soil layers as shown in Figure 4.2.
- The water level is located at $z = -1$ m. In the borehole column specify a value of -1 to *Head*.



Open the *Material sets* window.

- Create soil material data sets according to Table 4.1 and assign them to the corresponding layers in the borehole (Figure 4.2).
- Close the *Modify soil layers* window and proceed to the *Structures* mode to define the structural elements.

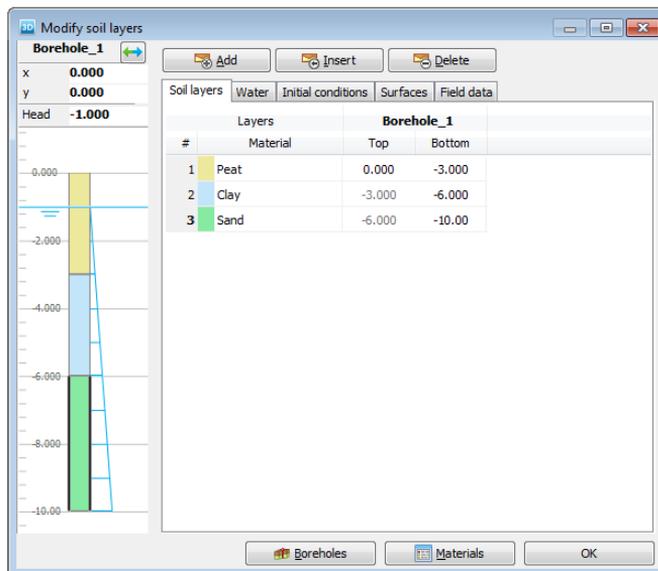


Figure 4.2 Soil layer distribution

Hint: 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.

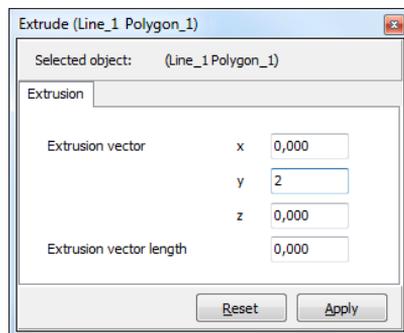
Table 4.1 Material properties of the road embankment and subsoil

Parameter	Name	Embankment	Sand	Peat	Clay	Unit
General						
Material model	<i>Model</i>	Hardening soil	Hardening soil	Soft soil	Soft soil	-
Drainage type	<i>Type</i>	Drained	Drained	Undr. (A)	Undr. (A)	-
Soil unit weight above phreatic level	γ_{unsat}	16	17	8	15	kN/m ³
Soil unit weight below phreatic level	γ_{sat}	19	20	12	18	kN/m ³
Initial void ratio	e_{init}	0.5	0.5	2.0	1.0	-
Parameters						
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.5 \cdot 10^4$	$3.5 \cdot 10^4$	-	-	kN/m ²
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
Power for stress-level dependency of stiffness	<i>m</i>	0.5	0.5	-	-	-
Modified compression index	λ^*	-	-	0.15	0.05	-
Modified swelling index	κ^*	-	-	0.03	0.01	-
Cohesion	c_{ref}'	1.0	0.0	2.0	1.0	kN/m ²
Friction angle	φ'	30.0	33.0	23.0	25.0	°
Dilatancy angle	ψ	0.0	3.0	0.0	0.0	°
Advanced: Set to default	-	Yes	Yes	Yes	Yes	-
Groundwater						
Data set	-	USDA	USDA	USDA	USDA	-
Model	-	Van Genuchten	Van Genuchten	Van Genuchten	Van Genuchten	-
Soil type	-	Loamy sand	Sand	Clay	Clay	-
$< 2\mu m$	-	6.0	4.0	70.0	70.0	%
$2\mu m - 50\mu m$	-	11.0	4.0	13.0	13.0	%
$50\mu m - 2mm$	-	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
Vertical permeability	k_z	3.499	7.128	0.05	0.04752	m/day
Change of permeability	c_k	$1 \cdot 10^{15}$	$1 \cdot 10^{15}$	1.0	0.2	-
Interfaces						
Interface strength	-	Rigid	Rigid	Rigid	Rigid	-
Strength reduction factor	R_{inter}	1.0	1.0	1.0	1.0	-
Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-
Over-consolidation ratio	<i>OCR</i>	1.0	1.0	1.0	1.0	-
Pre-overburden pressure	<i>POP</i>	0.0	0.0	5.0	0.0	kN/m ²

4.1.2 DEFINITION OF EMBANKMENT AND DRAINS

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

-  Reorientate the model such that the front view is displayed by clicking the corresponding button in the toolbar.
-  Create a surface by defining points at (0 0 0), (0 0 4), (8 0 4) and (20 0 0).
-  Create a line passing through (0 0 2) and (14 0 2) to define the embankment layers.
-  Select both the created line and surface by keeping the <Ctrl> key pressed while clicking them in the model.
-  Click the *Extrude object* button.
- Assign a value of 2 to the *y*-component of the extrusion vector as shown in Figure 4.3 and click *Apply*.

Figure 4.3 *Extrusion* window

- Delete the surface and the line with its corresponding points that were created before the extrusion.
- Right-click the volume created by extrusion and point to the *Soil_4* option in the appearing menu.
- A new menu is displayed. Point to the *Set material* option and select *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 in the case with drains.

Drains are arranged in a square pattern, having a distance of 2 m 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:

-  Click the *Create hydraulic conditions* button in the side toolbar.
-  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).
-  Click the *Create array* button to define the drain pattern.
- In the *Create array* window select the *1D, in x direction* in the *Shape* drop-down menu and specify the pattern as shown in Figure 4.4.

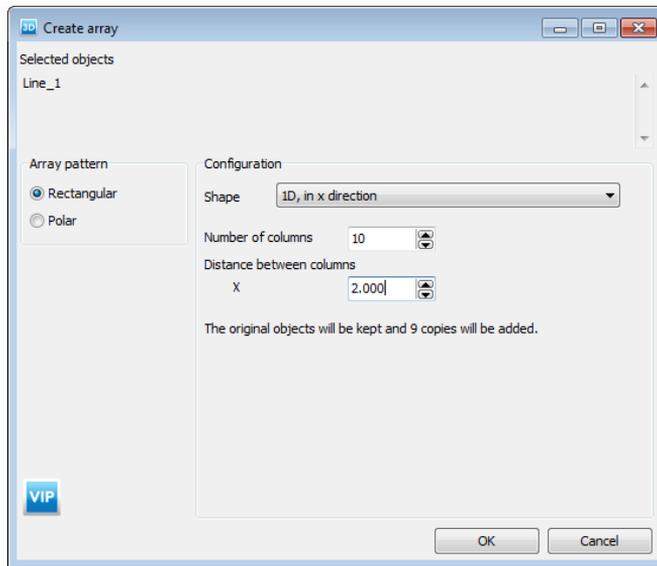


Figure 4.4 Settings of the drain pattern

The model geometry is shown in Figure 4.5.

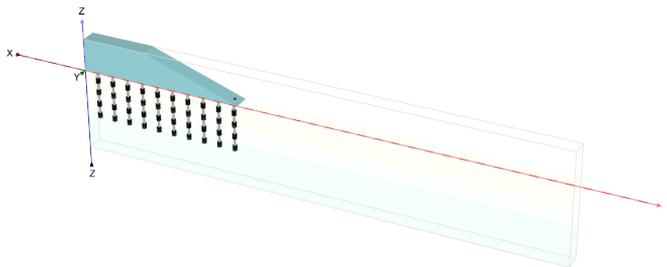


Figure 4.5 Model geometry

4.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Select all volumes, including the embankment and in the *Selection explorer* set the *Coarseness factor* to 0.3.



Click the *Generate mesh* button. Set the element distribution to *Coarse*.



View the generated mesh. The resulting mesh is shown in Figure 4.6.

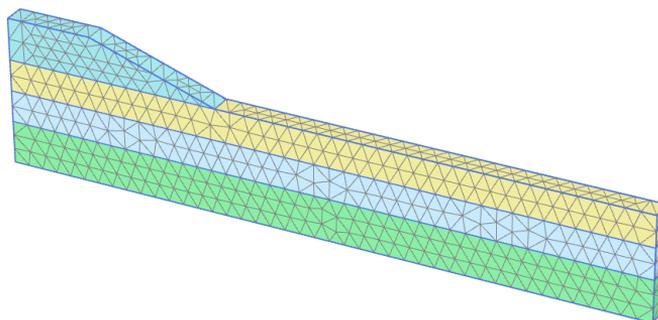


Figure 4.6 The generated mesh

4.3 PERFORMING CALCULATIONS

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

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 and 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 conditions* subtree in the *Model explorer*. 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 Figure 4.7.

4.3.1 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 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:

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

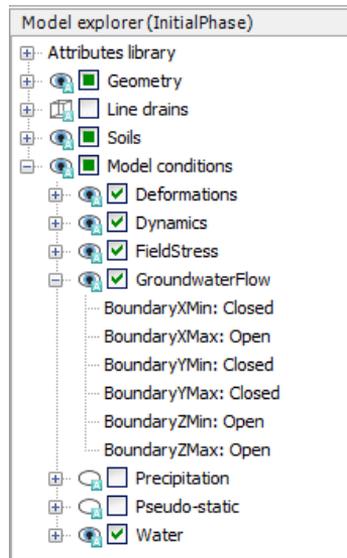


Figure 4.7 Boundary conditions for groundwater flow

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:

Phase 1:

-  Click the *Add phase* button to introduce the first construction phase.
-  In the *General* subtree select the *Consolidation* option in the *Calculation type* drop-down menu.
-  The *Loading type* is by default set to *Staged construction*. This option will be used for this phase.
-  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 *Water* subtree available under the *Model conditions* in the *Model explorer*.
- Specify a value of 2 days to the *Time interval* and click *OK* to close the *Phases* window.
- In the *Staged construction* mode activate the first part of the embankment.

Phase 2: 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.

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

Phase 3:

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

Phase 4: The fourth phase is a *Consolidation* analysis to a minimum excess pore pressure.

-  Click the *Add phase* button to introduce the next calculation phase.
-  Define the calculation type as *Consolidation*.
-  Select the *Minimum excess pore pressure* option in the *Loading type* 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.

-  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)).

-  Start the calculation.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window (Figure 4.8). 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.

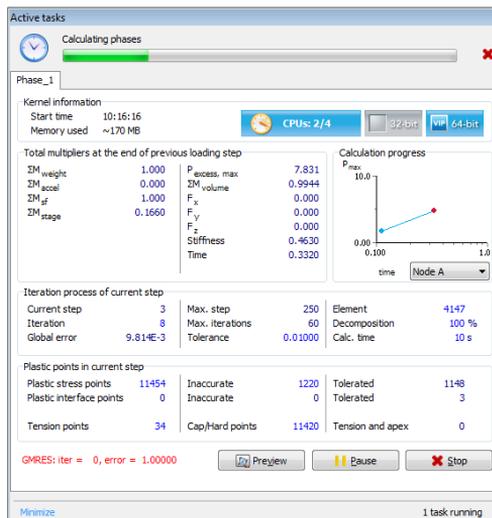


Figure 4.8 Calculation progress displayed in the *Active tasks* window

4.4 VIEWING THE 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 (Figure 4.9). Considering the results of the third phase, the deformed mesh shows the uplift of the embankment toe and hinterland due to the undrained behaviour.

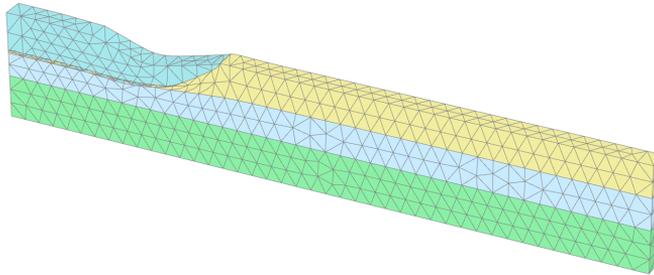


Figure 4.9 Deformed mesh after undrained construction of embankment (Phase 3, true scale)

- In the *Deformations* menu select the *Incremental displacements* $\rightarrow |\Delta u|$.
- ➔ Select the *Arrows* option in the *View* menu or click the corresponding button in the toolbar to display the results arrows.

On evaluating the total displacement increments, it can be seen that a failure mechanism is developing (Figure 4.10).

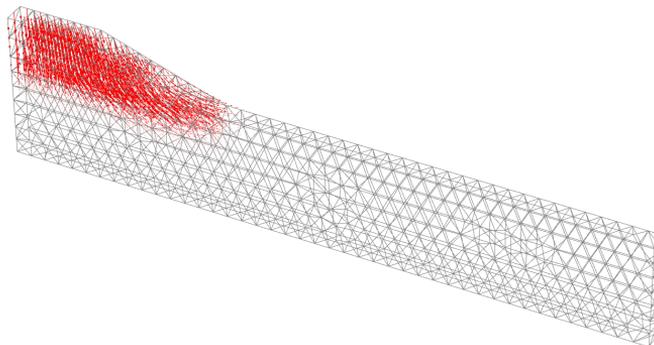


Figure 4.10 Displacement increments after undrained construction of embankment

- Click <Ctrl> + <7> to display the developed excess pore pressures (see Appendix C of 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.
- ✕ Click the *Center principal directions*. The principal directions of excess pressures are displayed at the center of each soil element. The results are displayed in Figure 4.11. It is clear that the highest excess pore pressure occurs under the embankment centre.
- Select Phase 4 in the drop down menu.

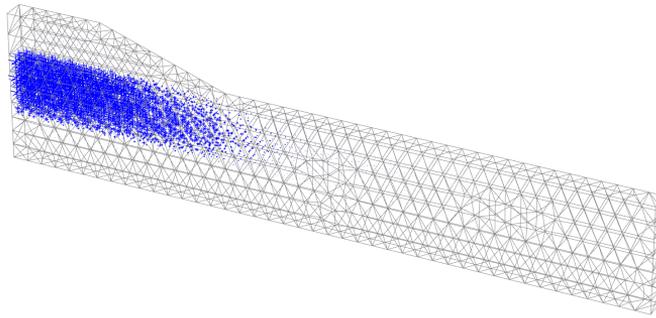


Figure 4.11 Excess pore pressures after undrained construction of embankment (Phase 3)

-  Define a vertical cross section passing through (0 1) and (60 1).
-  Click the *Contour lines* button in the toolbar to display the results as contours.
- In the *View* menu select the *Viewpoint* option. The corresponding window pops up.
- In the *Viewpoint* window select the *Front view* option as shown in Figure 4.12.

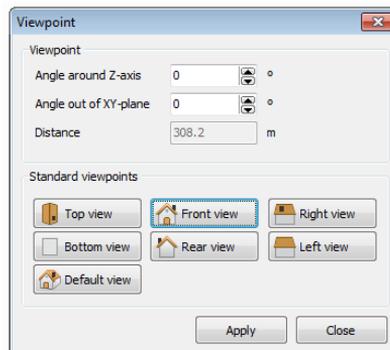


Figure 4.12 Viewpoint window

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

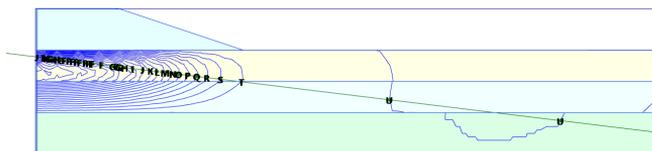


Figure 4.13 Excess pore pressure contours after consolidation to $P_{excess} < 1.0 \text{ kN/m}^2$

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. Figure 4.13 shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below 1.0 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:

- Click the *Curves manager* button in the toolbar. The corresponding window pops up.
 - In the *Charts* tabsheet click *New*. The *Curve generation* window pops up
 - For the *x-axis*, select the *Project* option from the drop-down menu and select *Time* in the tree.
 - 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} .
 - Select the *Invert sign* option for *y-axis*.
 - Click the *Ok* to generate the curve.
- Click the *Settings* button in the toolbar. The *Settings* window will appear displaying the tabsheet of the created curve.
 - Click the *Phases* button and select the phases 1 to 4 in the appearing window.
 - Rename the curve by typing 'Phases 1 - 4' in the *Curve title* cell.
 - Click *Apply* to update the plot.
- Save the chart in Output and save the project in Input.

Hint: To display the legend inside the chart area right-click on the name of the chart, point to the *View* option and select the *Legend in chart* option in the appearing menu.

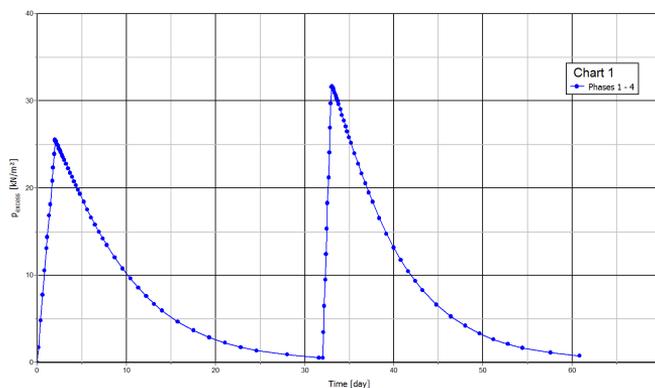


Figure 4.14 Development of excess pore pressure under the embankment

Figure 4.14 clearly 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.

4.5 SAFETY ANALYSIS

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:

$$\text{Safety factor} = \frac{S_{\text{maximum available}}}{S_{\text{needed for equilibrium}}} \quad (4.1)$$

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:

$$\text{Safety factor} = \frac{c - \sigma_n \tan \varphi}{c_r - \sigma_n \tan \varphi_r} \quad (4.2)$$

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 to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:

$$\frac{c}{c_r} = \frac{\tan \varphi}{\tan \varphi_r} = \Sigma M_{sf} \quad (4.3)$$

The reduction of strength parameters is controlled by the total multiplier ΣM_{sf} . This parameter is increased in a step-by-step procedure until failure occurs. The safety factor is then defined as the value of ΣM_{sf} 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.

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

- In the *Phases* window and select Phase 1 in the *Start from phase* drop-down menu.



Add a new calculation phase.



In the *General* subtree, select *Safety* as calculation type.



The *Loading type* is automatically changed to *Incremental multipliers*.

- The first increment of the multiplier that controls the strength reduction process, M_{sf} , is set automatically to 0.1. This value will be used in this tutorial.

 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

- 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.
- 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 4.15.

 Calculate the safety phases.

Hint: The default value of *Max steps* in a *Safety* calculation is 100. In contrast to an *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 $Msf = 0.1$ is an adequate first step to start up the process. During the calculation process, the development of the total multiplier for the strength reduction, ΣMsf , is automatically controlled by the load advancement procedure.

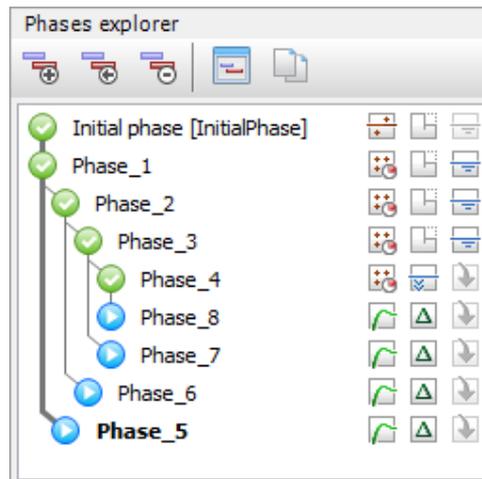


Figure 4.15 *Phases explorer* displaying the *Safety* calculation phases

4.5.1 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:

-  Select the last *Safety* phase (Phase_8) and click the *View calculation results* button.
 - From the *Deformations* menu select *Incremental displacements* → $|\Delta u|$.
-  Change the presentation from *Arrows* to *Shadings*. The resulting plots give a good impression of the failure mechanisms (Figure 4.16). The magnitude of the displacement increments is not relevant.

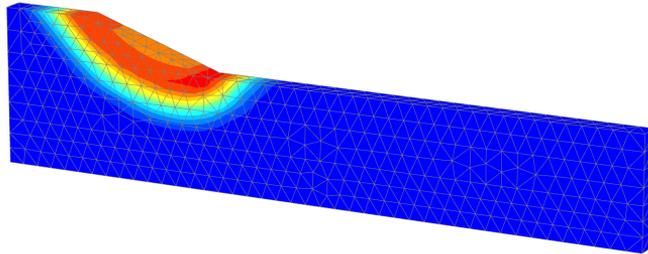


Figure 4.16 Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

The safety factor can be obtained from the *Calculation info* option of the *Project* menu. The value of ΣMsf 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 ΣMsf 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 three situations in this way, follow these steps:

- Click the *Curves manager* button in the toolbar.
- Click *New* in the *Charts* tabsheet.
- In the *Curve generation* window, select the embankment toe (the first node) for the *x*-axis. Select *Deformations* → *Total displacements* → $|u|$.
- 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 Figure 4.17 appears.
- Set *x*-axis interval maximum to 1 in *Chart* tab.

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.

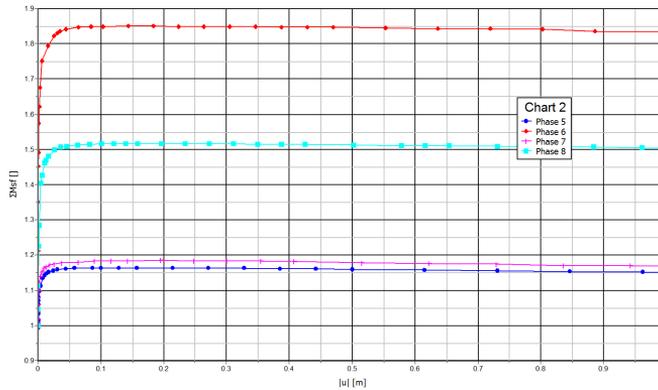


Figure 4.17 Evaluation of safety factor

4.6 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/m² is assigned to the minimum excess pressure (|P-stop|).

 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.

 Open the *Curves manager*.

- 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*.

 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.

Hint: 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. The chart (Figure 4.18) gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate.

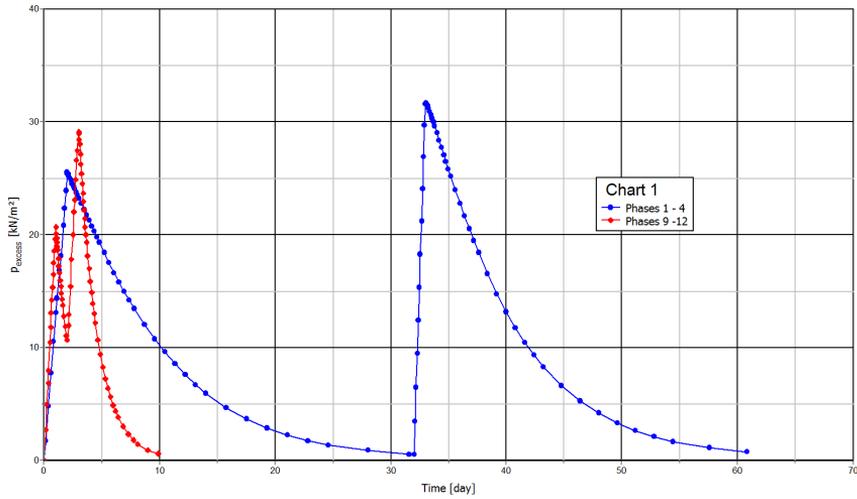


Figure 4.18 Effect of drains