8 Construction of a road embankment [ADV]

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 three new calculation options are introduced, namely a consolidation analysis, an updated mesh analysis and the calculation of a safety factor by means of a safety analysis (strength reduction).

Objectives

- Consolidation analysis
- Modelling drains
- Change of permeability during consolidation
- Safety analysis (strength reduction)
- Updated mesh analysis (large deformations)

Geometry

The embankment is 16 m wide and 4 m high. 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). The embankment itself is composed of loose sandy soil. The subsoil consists of 6 m of soft soil. The upper 3 m is peat and the lower 3 m is 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 which is shown in Figure 106 (on page 127).

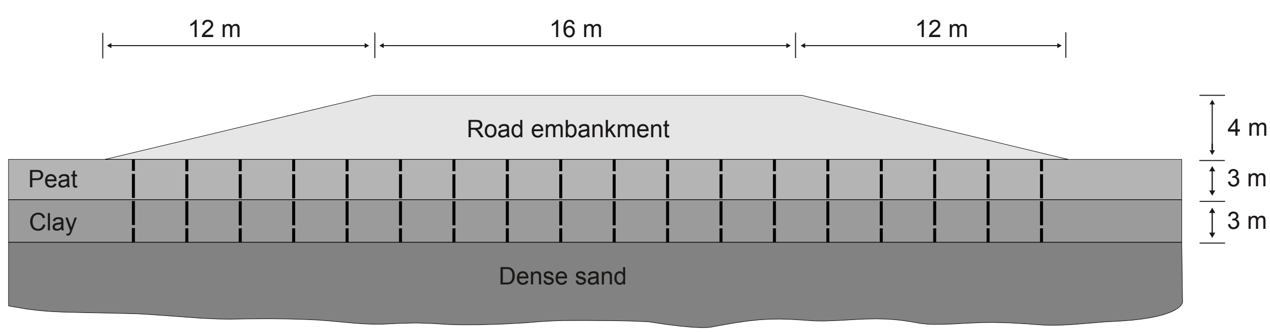

Figure 106: Situation of a road embankment on soft soil

8.1 Create new project

1. Start PLAXIS 2D by double clicking the icon of the Input program.
2. Click **Start a new project**.
3. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
4. In the **Model** tabsheet make sure that **Model** is set to **Plane strain** and that **Elements** is set to **15-Noded**.
5. Set the model **Contour** to x_{min} = 0 m, x_{max} = 60 m, y_{min} = -10 m and y_{max} = 4 m.

8.2 Define the soil stratigraphy

The sub-soil layers are defined using a borehole. The embankment layers are defined in the **Structures mode** .

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up as shown in Figure 107 (on page 128).

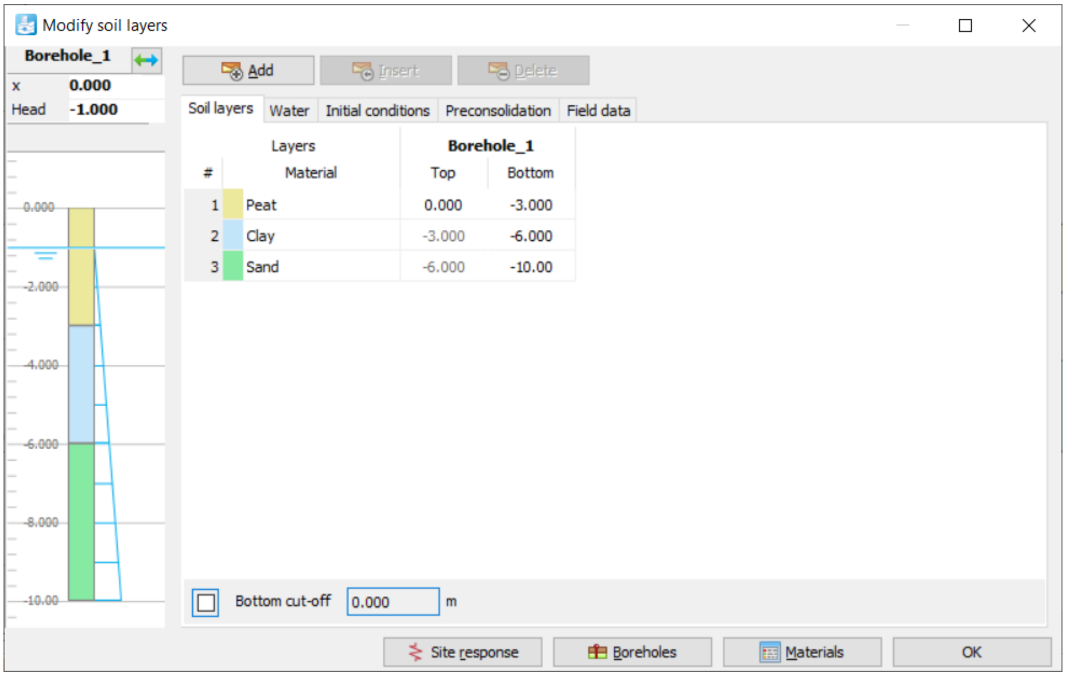

Figure 107: Soil layer distribution

2. Define three soil layers as shown in figure .
3. The water level is located at y = -1 m. In the borehole column specify a value of -1 to **Head**.

8.3 Create and assign material data sets

A number of material set are needed for this tutorial. The properties of the materials are as follows:

Table 20: Material properties of the sand and clay layer and the interfaces

Parameter	Name	Embankment	Sand	Peat	Clay	Unit
General						
Soil model	-	Hardening soil	Hardening soil	Soft soil	Soft soil	-
Drainage 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						
Modified compression index	λ^*	-	-	0.15	0.05	-
Modified swelling index	κ^*	-	-	0.03	0.01	-
Secant stiffness in standard drained triaxial test	${E_{50}}^{ref}$	$25 \cdot 10^3$	$35 \cdot 10^3$	-	-	kN/m^2
Tangent stiffness for primary oedometer loading	${E_{oed}}^{ref}$	$25 \cdot 10^3$	$35 \cdot 10^3$	-	-	kN/m^2
Unloading / reloading stiffness	${E_{ur}}^{ref}$	$75 \cdot 10^3$	$105 \cdot 10^3$	-	-	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	0.5	-	-	-
Cohesion (constant)	c'_{ref}	1	0	2	1	kN/m^2
Friction angle	φ'	30	33	23	25	°
Dilatancy angle	ψ	0	3	0	0	°
Miscellaneous: Set to default		Yes	Yes	Yes	Yes	-

Groundwater						
Classification type	-	USDA	USDA	USDA	USDA	-
SWCC fitting method	-	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	k_x	3.499	7.128	0.1	0.04752	m/day
Vertical permeability	k_y	3.499	7.128	0.05	0.04752	m/day
Change in permeability	c_k	$1 \cdot 10^{15}$	$1 \cdot 10^{15}$	1.0	0.2	-
Interfaces						
Strength determination	-	Rigid	Rigid	Rigid	Rigid	-
Strength reduction factor	R_{inter}	1	1	1	1	-
Initial						
K_0 determination	-	Automatic	Automatic	Automatic	Automatic	-
Pre-overburden pressure	*POP*	0	0	5	0	kN/m^2
Overconsolidation ratio	*OCR*	1.0	1.0	1.0	1.0	-

To create the material sets, follow these steps:

1. Click the **Materials** button  to open the **Material sets** window.
2. Create soil material data sets according to Table 20 (on page 129) and assign them to the corresponding layers in the borehole (see Figure 107 (on page 128)).
3. Close the **Modify soil layers** window and proceed to the **Structures mode** to define the embankment and drains.

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 in a consolidation analysis due to compression of the soil. This option is recommended when using advanced models.

8.4 Define the construction

The embankment and the drains are defined in the **Structures mode** .

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode** .

8.4.1 To define the embankment:

1. Click the **Create soil polygon** button in the side toolbar and select the **Create soil polygon** option.
2. Define the embankment in the drawing area by clicking on (0 0), (0 4), (8 4) and (20 0).
3. Select and right click the created polygon and assign the **Embankment** data set to the soil polygon as shown in Figure 108 (on page 131).

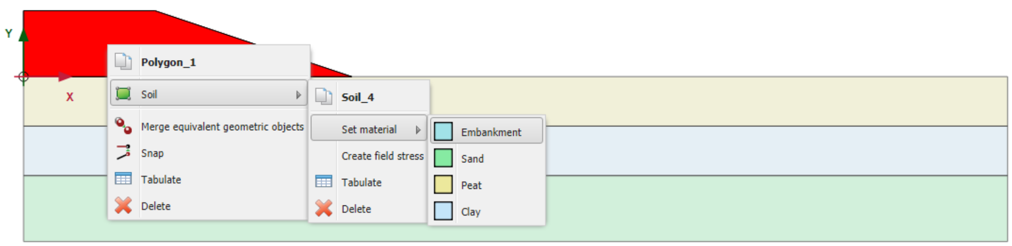

Figure 108: Assignment of a material dataset to a soil cluster in the drawing area

4. To define the embankment construction level click the **Cut polygon** button in the side toolbar and define a cutting line by clicking on (0 2) and (14 2).
   The embankment cluster is split into two sub-clusters.

8.4.2 To define the drains

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.

1. Click the **Create hydraulic conditions** button in the side toolbar and select the **Create drain** option in the appearing menu which is shown in Figure 109 (on page 131).

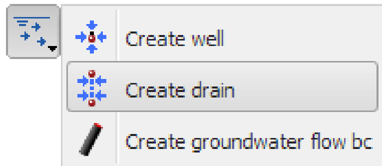

Figure 109: Create Drain option

2. Drains are defined in the soft layers (clay and peat; y = 0 m to y = -6 m). The distance between two consecutive drains is 2 m. Considering the symmetry, the first drain is located at 1 m distance from the model boundary. 10 drains will be created in total. The **head** is defined at 0.0 m. The geometry is shown in Figure 110 (on page 132).

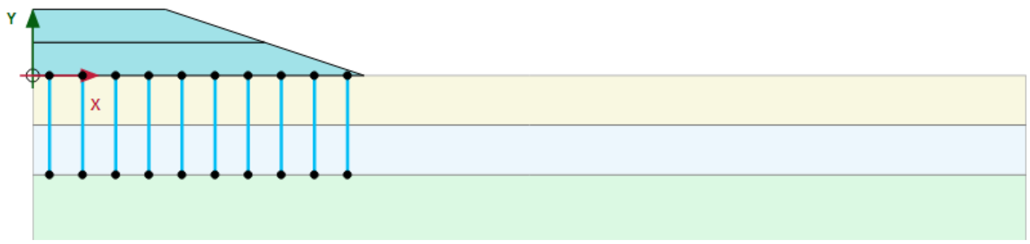

Figure 110: Final geometry of the model

Note:

The modelling of drains in a plane strain model actually involves the use of an equivalent (lateral) permeability in the surrounding soil based on the drain pattern. The latter has been omitted in this simplified example. More information can be in found in literature[1].

8.5 Generate the mesh

In order to generate the mesh, follow these steps:

1. Proceed to **Mesh mode** .
2. Click the **Generate mesh** button in the side toolbar. For the **Element distribution** parameter, use the option **Medium** (default).
3. Click the **View mesh** button to view the mesh as shown in Figure 111 (on page 133).

[1] Achtergronden bij numerieke modellering van geotechnische constructies, deel 2. CUR 191. Stichting CUR, Gouda Indraratna, B.N., Redana, I.W., Salim, W. (2000), Predicted and observed behaviour of soft clay foundations stabilised with vertical drains. Proc. GeoEng. 2000, Melbourne.

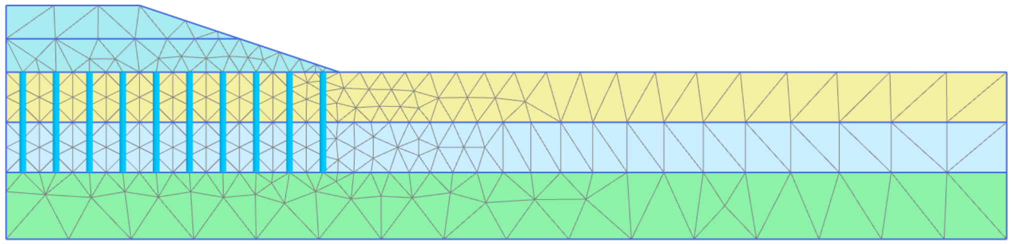

Figure 111: The generated mesh

4. Click the **Close** tab to close the Output program.

8.6 Define and perform the calculation

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. Hence, a total of four calculation phases have to be defined besides the initial phase.

8.6.1 Initial phase: Initial conditions

In the initial situation the embankment is not present.

The remaining active geometry is horizontal with horizontal layers, so the **K0 procedure** can be used to calculate the initial stresses. The geometry of the model for initial phase is shown in Figure 112 (on page 133).

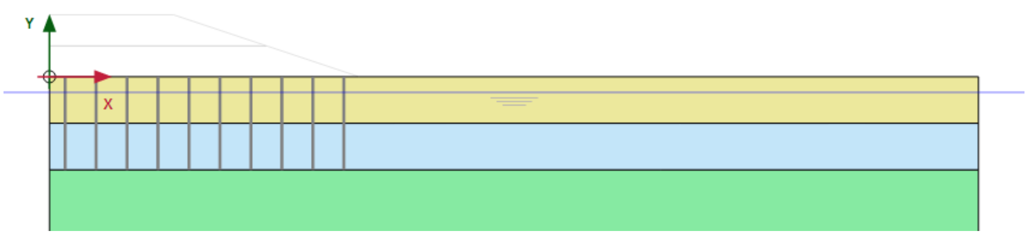

Figure 112: Configuration of the initial phase

The initial water pressures are fully hydrostatic and based on a general phreatic level located at y = -1 m. Note that a phreatic level is automatically created at y = -1 m, according to the value specified for **Head** in the borehole. In addition to the phreatic level, attention must be paid to the boundary conditions for the consolidation analysis that will be performed during the calculation process. Without giving any additional input, all boundaries except for the bottom boundary are draining so that water can freely flow out of these boundaries and excess pore pressures can dissipate. In the current situation, however, the left vertical boundary must be closed because this is a line of symmetry, so horizontal flow should not occur. The remaining boundaries are open because the excess pore pressures can be dissipated through these boundaries. In order to define the appropriate consolidation boundary conditions, follow these steps:

1. Go to the **Staged construction mode** and in the **Model explorer** expand the **Model conditions** subtree shown in Figure 113 (on page 134).
2. Expand the **GroundwaterFlow** subtree and set **BoundaryXMin** to **Closed** and **BoundaryYMin** to **Open**.

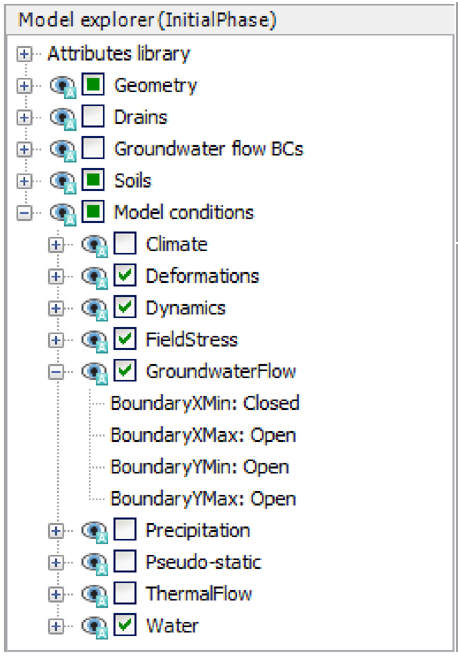

Figure 113: The boundary conditions of the problem

8.6.2 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 2D allows for a fully automatic time stepping procedure that takes this critical time step into account. Within this procedure there are three main possibilities:

	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 a specified degree of saturation is reached (**Degree of consolidation**).

The first two possibilities will be used in this exercise. To define the calculation phases, follow these steps:

Phase 1: First embankment construction

The first calculation stage is a **Consolidation** analysis, **Staged construction**.

1. Click the **Add phase** button to create a new phase and double click.
2. In the **Phases** window select the **Consolidation** option from the **Calculation type** drop-down menu in the **General** subtree.
3. Make sure that the **Staged construction** option is selected for the **Loading type**.
4. Enter a **Time interval** of 2 days. The default values of the remaining parameters will be used.
5. In the **Staged construction mode** activate the first part of the embankment. The model for phase 1 is shown in Figure 114 (on page 135).

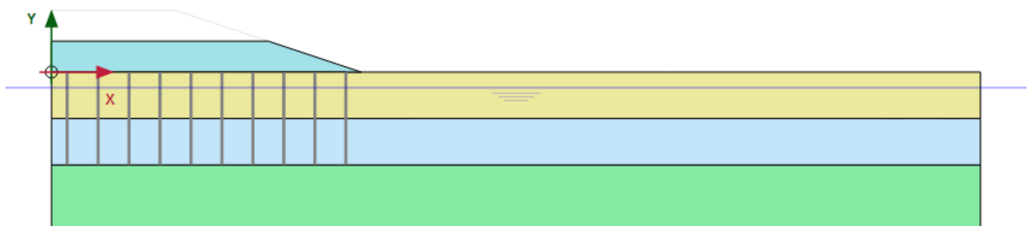

Figure 114: Configuration of the phase 1

Phase 2: First consolidation period

The second phase is also a **Consolidation** analysis, **Staged construction**. 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 create a new phase.
2. In the **Phases** window select the **Consolidation** option from the **Calculation type** drop-down menu in the **General** subtree.
3. Make sure that the **Staged construction** option is selected for the **Loading type**.
4. Enter a **Time interval** of 30 days. The default values of the remaining parameters will be used.

Phase 3: Second embankment construction

1. Click the **Add phase** button to create a new phase.
2. In the **Phases** window select the **Consolidation** option from the **Calculation type** drop-down menu in the **General** subtree.
3. Make sure that the **Staged construction** option is selected for the **Loading type**.
4. Enter a **Time interval** of 1 day. The default values of the remaining parameters will be used.
5. In the **Staged construction mode** activate the second part of the embankment. The model for phase 3 is shown in Figure 115 (on page 136).

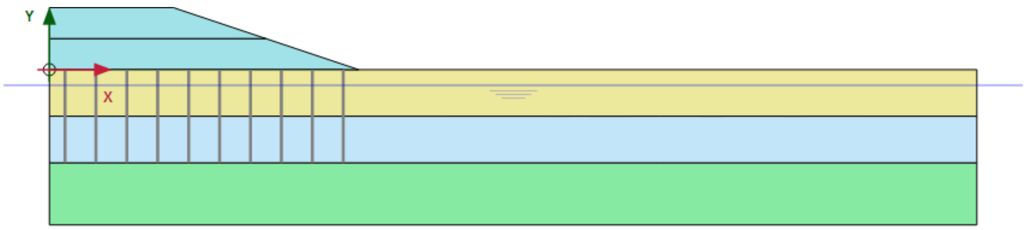

Figure 115: Configuration of the phase 3

Phase 4: End of consolidation

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

1. Click the **Add phase** button to create a new phase.
2. In the **General** subtree select the **Consolidation** option as calculation type.
3. Select the **Minimum excess pore pressure** option in the **Loading type** drop-down menu and accept the default value of 1 kN/m^2 for the minimum pressure. The default values of the remaining parameters will be used.

8.6.3 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.

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

1. Select Phase 1 in the **Phases** explorer.
2. Add a new calculation phase.
3. Double-click on the new phase to open the **Phases** window.
4. In the **Phases** window the selected phase is automatically selected in the **Start from phase** drop-down menu.
5. In the **General** subtree, select **Safety** as calculation type.
6. The **Incremental multipliers** option is already selected in the **Loading input** box. The first increment of the multiplier that controls the strength reduction process, **Msf**, is set to 0.1.
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.
8. The first safety calculation has now been defined.

9. Follow the same steps to create new calculation phases that analyse the stability at the end of each consolidation phase. The various phases after defining safety calculation is shown in Figure 116 (on page 137).

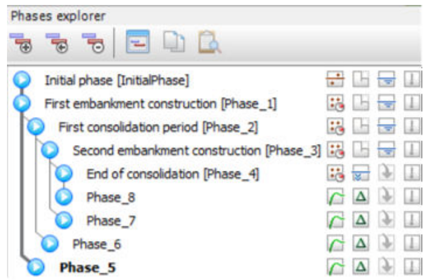

Figure 116: Safety calculation phases

8.6.4 Calculate

Before starting the calculation it is suggested that you select nodes or stress points for a later generation of load-displacement curves or stress and strain diagrams. To do this, follow the steps given below.

1. Click the **Select points for curves** button in the side toolbar.
2. As the first point, select the toe of the embankmentat (20 0).
3. The second point 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 at the left side of the model is needed, hence underneath the middle of the embankment. For instance at (0 -3).
4. Click the **Calculate** button to calculate the project.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window as shown in Figure 117 (on page 138).

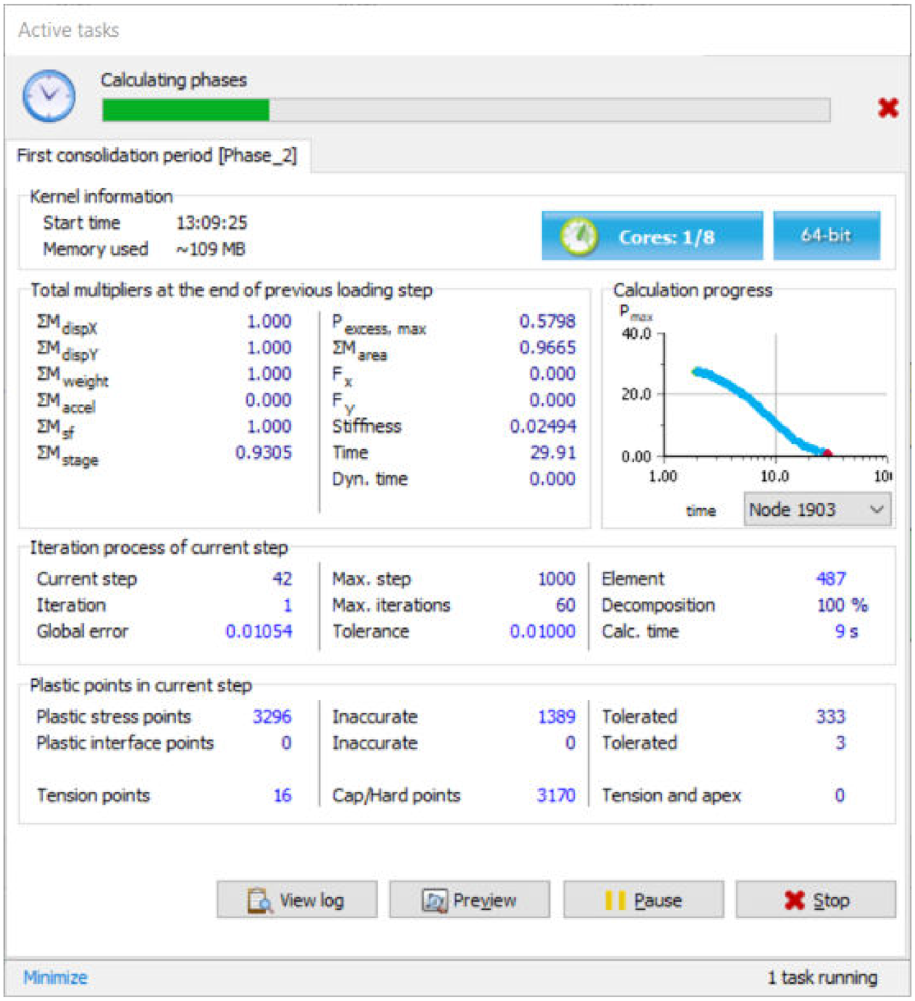

*Figure 117: Calculation progress displayed in the **Active tasks** window*

In addition to the multipliers, a parameter $P_{excess,max}$ 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.

8.7 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 as displayed in Figure 118 (on page 139) shows the uplift of the embankment toe and hinterland due to the undrained behaviour.

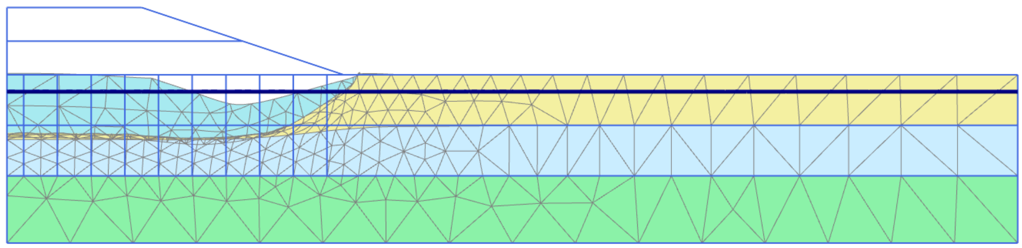

Figure 118: Deformed mesh after undrained construction of embankment (Phase 3)

1. Select the menu **Deformations** > **Incremental displacements** > **|Δ u|**.
2. Select the menu **View** > **Arrows** option in the 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 shown in Figure 119 (on page 139):

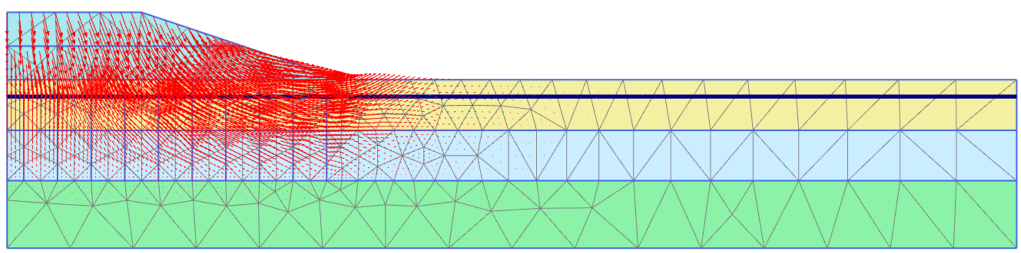

Figure 119: Displacement increments after undrained construction of embankment

1. Press **<Ctrl + 7>** to display the developed excess pore pressures (see Appendix C of the Reference Manual for more shortcuts). They can also be displayed by selecting the menu **Stresses** > **Pore pressures** > **P excess**.
2. Click the **Center principal directions** button . The principal directions of excess pressures are displayed at the center of each soil element. The results are displayed in Figure 120 (on page 139).

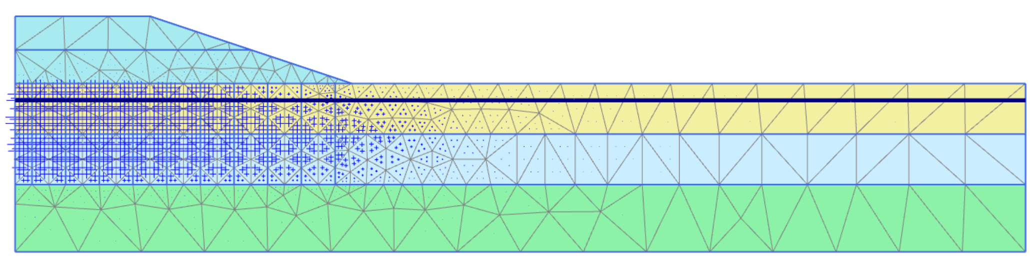

Figure 120: Excess pore pressures after undrained construction of embankment

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

1. Select Phase 4 in the drop down menu.
2. Click the **Contour lines** button in the toolbar to display the results as contours.
3. 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. Figure 121 (on page 140) shows the remaining excess pore pressure distribution after consolidation. Check that the maximum value is below 1.0 kN/m^2.

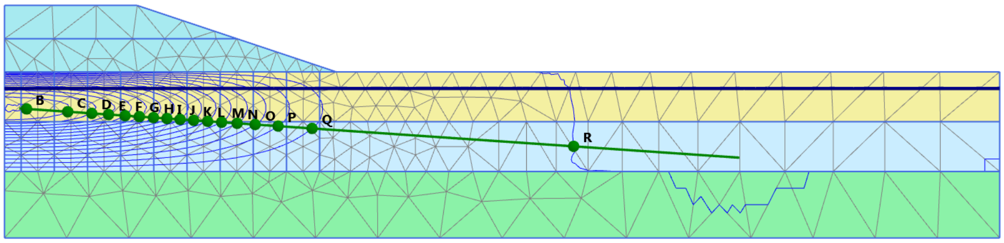

Figure 121: Excess pore pressure contours after consolidation to P_{excess} < 1.0 kN/m^2

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:

1. Create a new curve by clicking the **Curves manager** button .
2. For the x-axis, select the **Project** option from the drop-down menu and select **Time** in the tree.
3. For the y-axis select the point in the middle of the soft soil layers (Point B) from the drop-down menu. In the tree select **Stresses** > **Pore pressure** > **p excess**.
4. Select the **Invert sign** option for the y-axis.
5. Click **OK**.
6. Open the **Curve settings** (F3) and go to the second tabsheet.
7. In the **Show** box click the **Phases** button. By default all phases are selected to show in the curve. For the clarity of the curve, hide the **Safety** phases (phases 5 - 8).
8. Click **OK** to close the **Curve settings** window.

A curve similar to the following one should appear as shown in Figure 122 (on page 141):

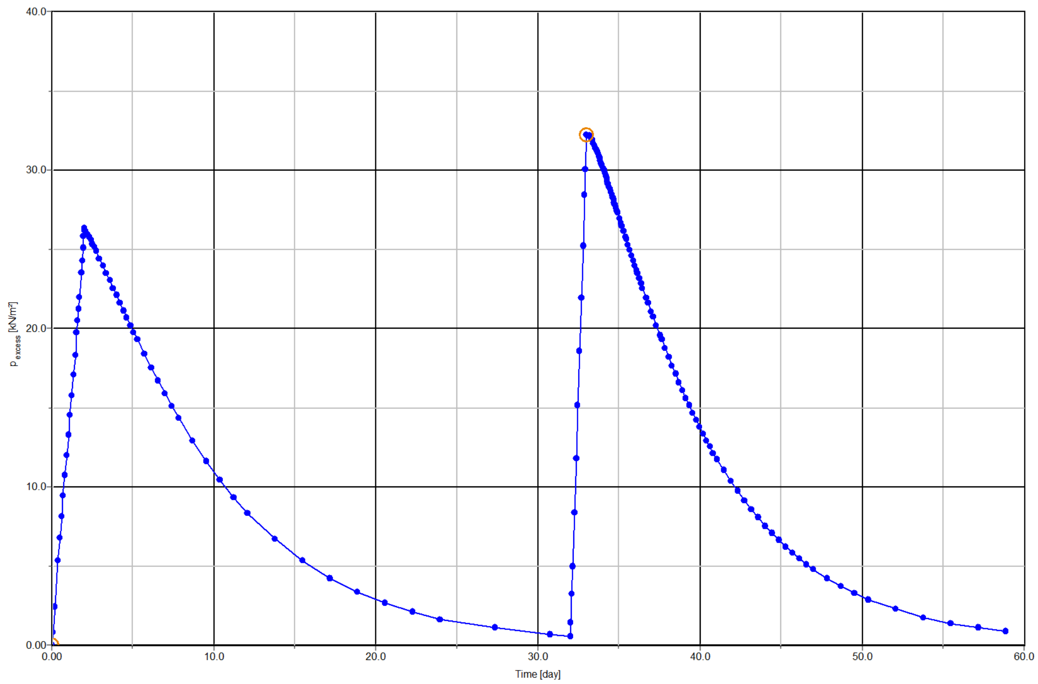

Figure 122: Development of excess pore pressure under the embankment

The figure 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. From the curve it can be seen that more than 50 days are needed to reach full consolidation.

Save the chart before closing the Output program.

8.7.1 Safety analysis results

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 one of these phases and click the **View calculation results** button.
2. Select the menu **Deformations** > **Incremental displacements** > **|Δu|**.
3. Change the presentation from **Arrows** to **Shadings**. The resulting plots shown in Figure 123 (on page 142) gives a good impression of the failure mechanisms. The magnitude of the displacement increments is not relevant.

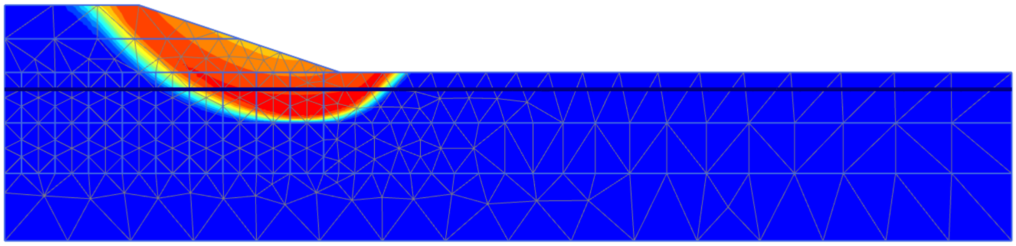

Figure 123: Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

The safety factor can be obtained from the **Calculation info** option of the **Project** menu. The **Multipliers** tabsheet of the **Calculation information** window represents the actual values of the load multipliers. 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:

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 (Point A) for the x-axis. Select **Deformations** > **Total displacements** > **|u|**.
4. For the y-axis, select **Project** > **Multipliers** > **ΣMsf**. The **Safety** phases are considered in the chart.
5. Right-click on the chart and select the **Settings** option in the appearing menu. The **Settings** window pops up.
6. In the tabsheet corresponding to the curve click the **Phases** button.
7. In the **Select phases** window select Phase 5 shown in Figure 124 (on page 142):

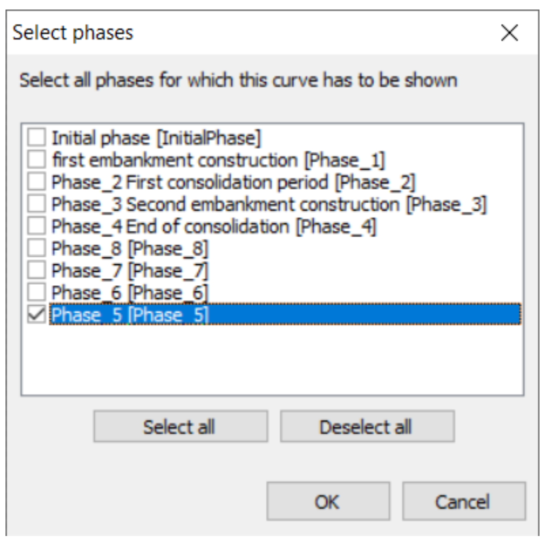

Figure 124: The Select phases window

8. Click **OK** to close the **Select phases** window.

9. In the **Settings** window change the titles of the curve in the corresponding tabsheet.
10. Click the **Add curve** button and select the **From current project...** option in the appearing menu. Define curves for phases 6, 7 and 8 by following the described steps.
11. In the **Settings** window click the **Chart** tab to open the corresponding tabsheet.
12. In the **Chart** tabsheet specify the chart name.
13. Set the scaling of the x-axis to **Manual** and set the value of **Maximum** to 1 as shown in Figure 125 (on page 143):

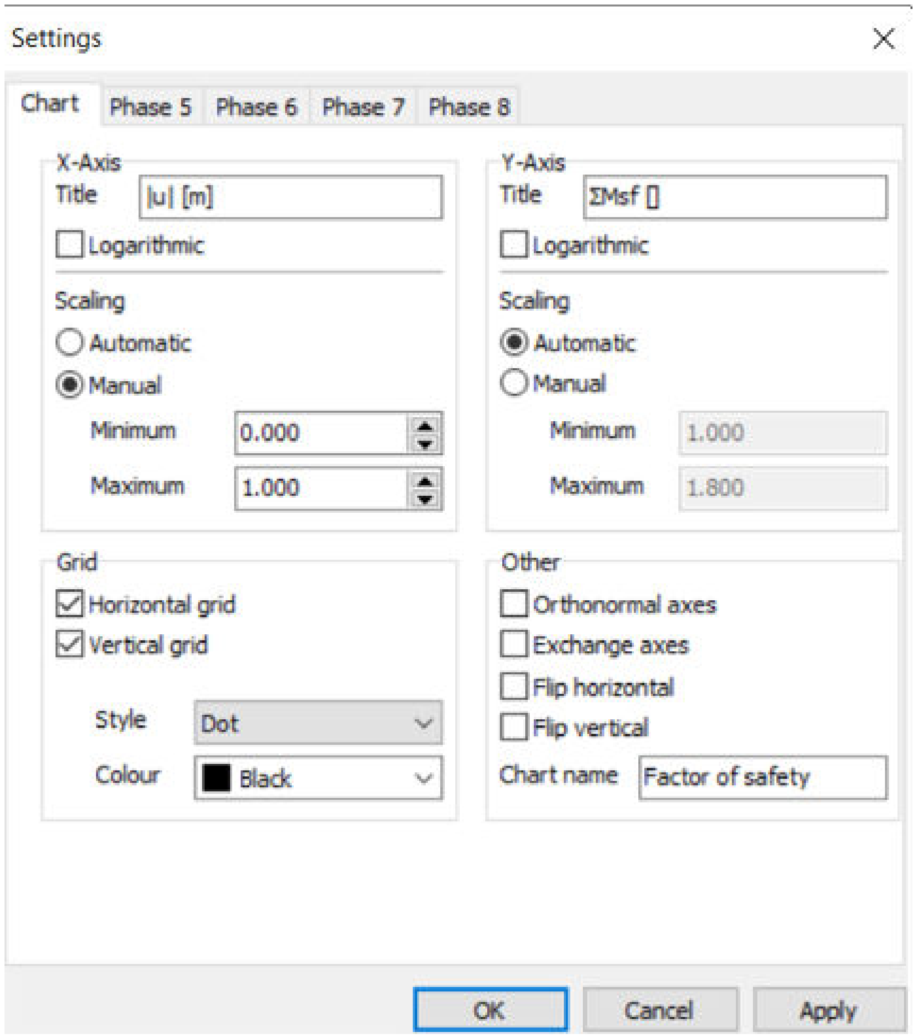

Figure 125: The Chart tabsheet in the Settings window

14. Click **Apply** to update the chart according to the changes made and click **OK** to close the **Settings** window.
15. To modify the location of the legend right-click on the legend.
16. In the context menu select **View** > **Legend in chart**.
17. The legend can be relocated in the chart by dragging it. The plot is shown in Figure 126 (on page 144):

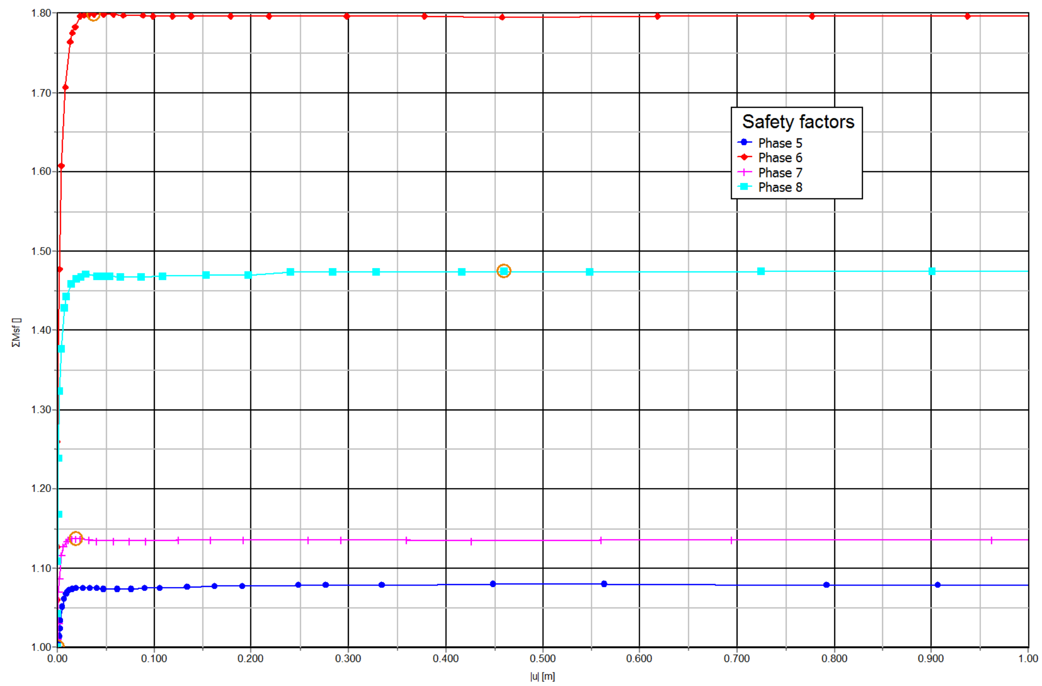

Figure 126: Evaluation of safety factor

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.

8.8 Using drains

In this section the effect of the drains in the project will be investigated. Four new phases will be introduced having the same properties as the first four consolidation phases. The first of these new phases should start from the initial phase. The differences in the new phases are:

- The drains should be active in all the new phases. Activate them in the **Staged construction mode** .
- The **Time interval** in the first three of the consolidation phases (9 to 11) 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 pressure (|P-stop|).

Follow these steps:

1. After the calculation is finished, save the project, then 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 the second point can be used.

2. Click the **Curves manager** button  to open the **Curves manager**.
3. In the **Chart** tabsheet double-click Chart 1 (p_{excess} of the second point at (0 -3) versus time). The chart is displayed. Close the **Curves manager**.
4. Double-click the curve in the legend at the right of the chart. The **Settings** window pops up.
5. Click the **Add curve** button and select the **From current project ...** option in the appearing menu. The **Curve generation** window pops up.
6. Select the **Invert sign** option for y-axis and click **OK** to accept the selected options.
7. 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**.
8. In the **Settings** window change the titles of the curves in the corresponding tabsheets.
9. In the **Chart** tabsheet specify the chart name.
10. Click **Apply** to preview the generated curve and click **OK** to close the **Settings** window. The chart gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate as shown in Figure 127 (on page 145):

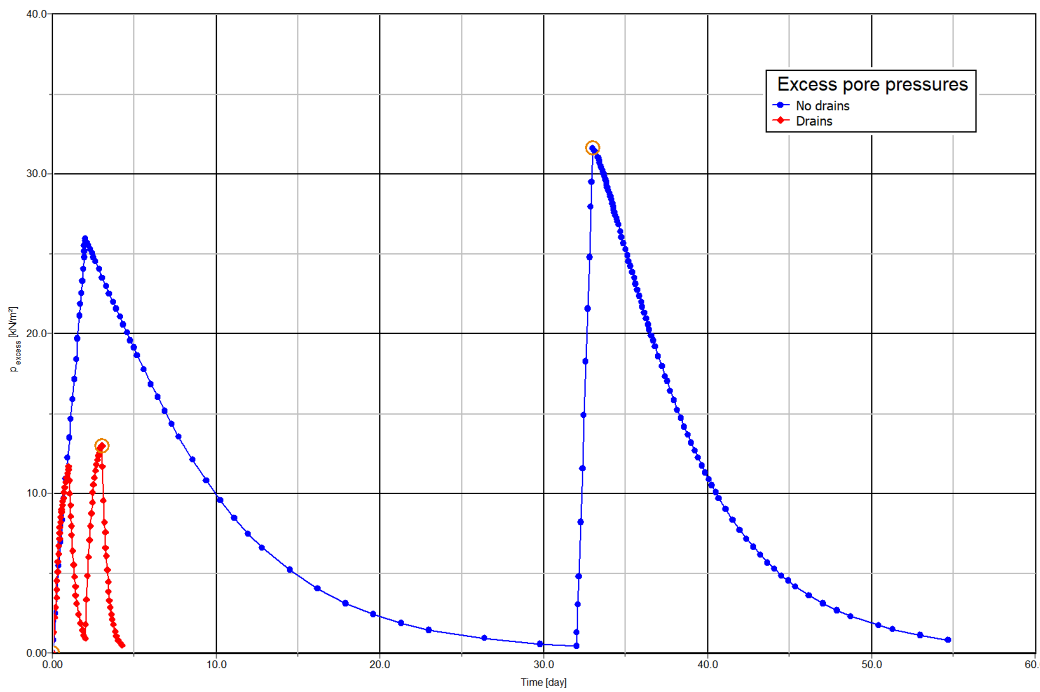

Figure 127: Effect of drains

Note:

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

8.9 Updated mesh and updated water pressures analysis

As can be seen from the output of the **Deformed mesh** at the end of consolidation (stage 4), the embankment settles about one meter since the start of construction. Part of the sand fill that was originally above the phreatic level will settle below the phreatic level.

As a result of buoyancy forces the effective weight of the soil that settles below the water level will change, which leads to a reduction of the effective overburden in time. This effect can be simulated in PLAXIS 2D using the **Updated mesh** and **Updated water pressures** options. For the road embankment the effect of using these options will be investigated.

1. Select the initial phase in the **Phases** explorer.
2. Add a new calculation phase.
3. Define the new phase in the same way as Phase 1. In the **Deformation control parameters** subtree check the **Updated mesh** and **Updated water pressures** options.
4. Define the other 3 phases in the same way.

When the calculation has finished, compare the settlements for the two different calculation methods.

1. In the **Curve generation** window select time for the x-axis and select the vertical displacement (u_y) of the point in the middle of the soft soil layers at (0 -3) for the y-axis.
2. In this curve the results for Initial phase and phases from 1 to 4 will be considered.
3. Add a new curve to the chart.
4. In this curve the results for Initial phase and phases from 13 to 16 will be considered. The resulting chart is shown in Figure 128 (on page 147).

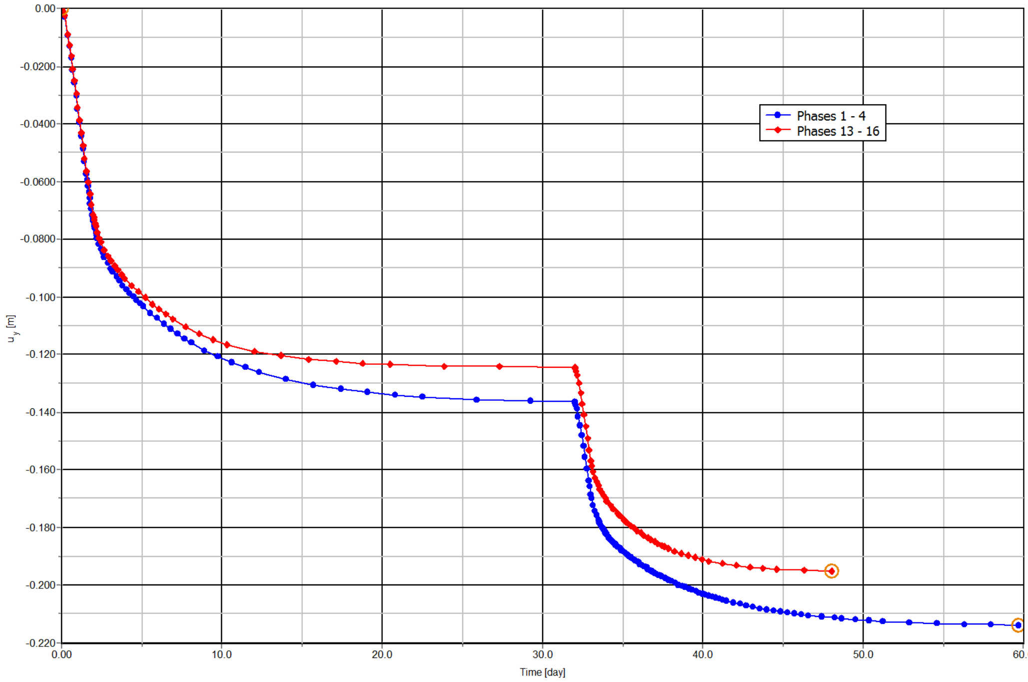

Figure 128: Effect of updated mesh and water pressures analysis on resulting settlements

It can be seen that the settlements are less when the **Updated mesh** and **Updated water pressures** options are used (red curve). This is partly because the **Updated mesh** procedure includes second order deformation effects by which changes of the geometry are taken into account, and partly because the **Updated water pressures** procedure results in smaller effective weights of the embankment. This last effect is caused by the buoyancy of the soil settling below the (constant) phreatic level. The use of these procedures allows for a realistic analysis of settlements, taking into account the positive effects of large deformations.