5 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 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 (phi/c-reduction).

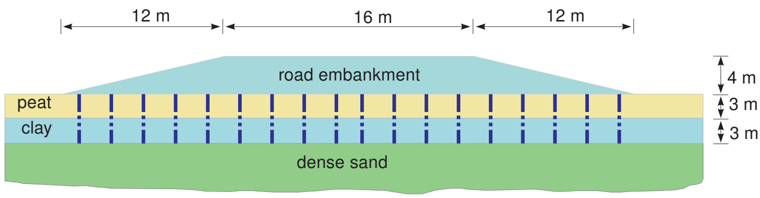

Figure 5.1 Situation of a road embankment on soft soil

Objectives:

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

5.1 INPUT

Figure 5.1 shows a cross section of a road embankment. The embankment is 16.0 m wide and 4.0 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 6.0 m of soft soil. The upper 3.0 m is peat and the lower 3.0 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 4.0 m are considered in the model.

General settings

- 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.
- In the *Model* tabsheet make sure that *Model* is set to *Plane strain* and that *Elements* is set to *15-Noded*.

- Define the limits for the soil contour as x_{min} = 0.0, x_{max} = 60.0, y_{min} = −10.0 and y_{max} = 4.0.

Definition of 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:

- Create a borehole at x = 0. The *Modify soil layers* window pops up.
- Define three soil layers as shown in Figure 5.2.
- The water level is located at y = -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 5.1 and assign them to the corresponding layers in the borehole (Figure 5.2).
- Close the *Modify soil layers* window and proceed to the *Structures* mode to define the embankment and drains.

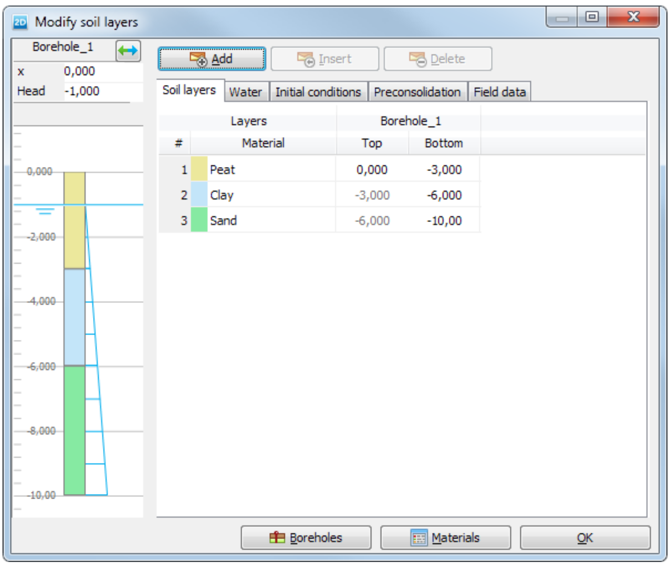

Figure 5.2 Soil layer distribution

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

Table 5.1 Material properties of the road embankment and subsoil

Parameter	Name	Embankment	Sand	Peat	Clay	Unit
General						
Material model	*Model*	Hardening soil	Hardening soil	Soft soil	Soft soil	-
Type of material behaviour	*Type*	Drained	Drained	Undrained (A)	Undrained (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	m	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	33	23	25	°
Dilatancy angle	ψ	0.0	3.0	0	0	°
Advanced: Set to default	Yes	Yes	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	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						
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	*OCR*	1.0	1.0	1.0	1.0	-
Pre-overburden pressure	*POP*	0.0	0.0	5.0	0.0	kN/m^2

5.1.1 DEFINITION OF EMBANKMENT AND DRAINS

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

- Click the *Create soil polygon* button in the side toolbar and select the *Create soil polygon* option in the appearing menu.
- Define the embankment in the drawing area by clicking on (0.0 0.0), (0.0 4.0), (8.0 4.0) and (20.0 0.0).
- Right-click the created polygon and assign the *Embankment* data set to the soil polygon (Figure 5.3).

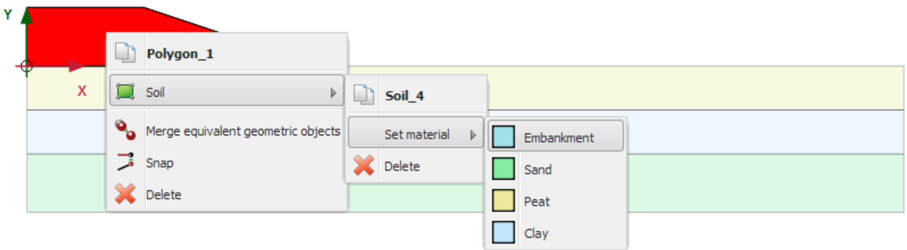

Figure 5.3 Assignment of a material dataset to a soil cluster in the drawing area

- To define the embankment construction level click the *Cut polygon* in the side toolbar and define a cutting line by clicking on (0.0 2.0) and (14.0 2.0). The embankment cluster is split into two sub-clusters.

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.

- Click the *Create hydraulic conditions* button in the side toolbar and select the *Create drain* option in the appearing menu (Figure 5.4).

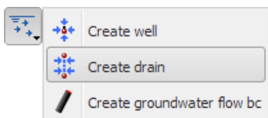

Figure 5.4 The *Create drain* option in the *Create hydraulic conditions* menu

- Drains are defined in the soft layers (clay and peat; y = 0.0 to y = -6.0). 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 (Figure 5.5). The *head* is defined at 0.0 m.

Hint: 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[a].

[a] CUR (1997). 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

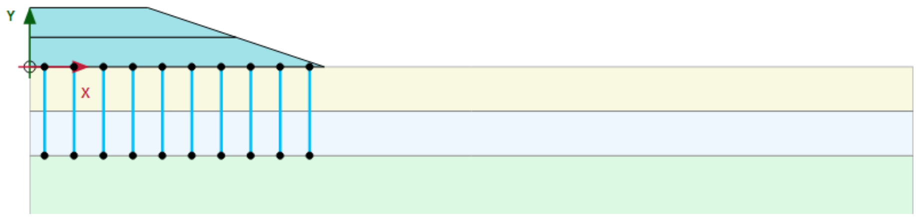

Figure 5.5 Final geometry of the model

5.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Generate the mesh. Use the default option for the *Element distribution* parameter (*Medium*).
- View the generated mesh. The resulting mesh is shown in Figure 5.6.
- Click on the *Close* tab to close the Output program.

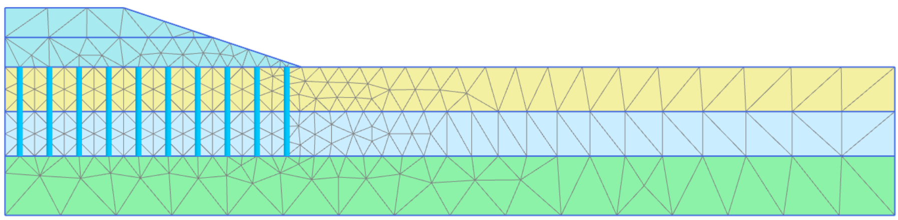

Figure 5.6 The generated mesh

5.3 CALCULATIONS

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.

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 (Figure 5.7).

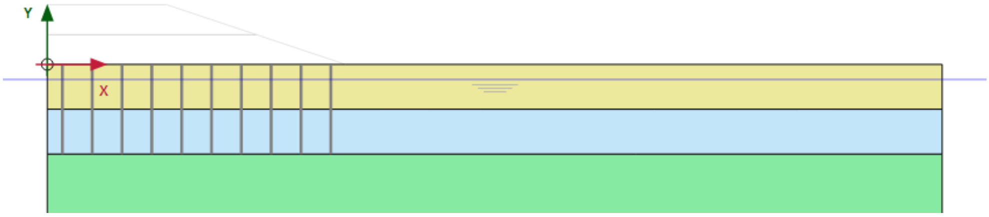

Figure 5.7 Configuration of the initial phase

The initial water pressures are fully hydrostatic and based on a general phreatic level located at y = -1. Note that a phreatic level is automatically created at y = -1, 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:

- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *GroundwaterFlow* subtree and set *BoundaryXMin* to *Closed* and *BoundaryYMin* to *Open* (Figure 5.8).

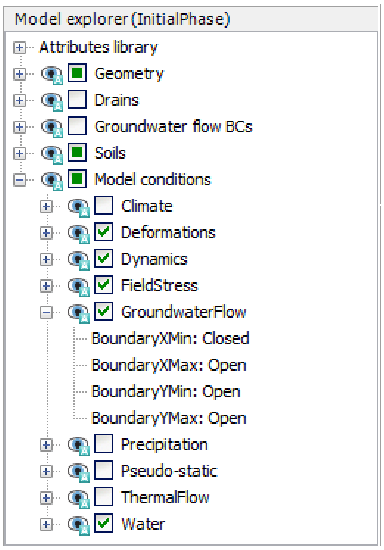

Figure 5.8 The boundary conditions of the problem

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:

- 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: (the way calculation phases are presented here is different than in other chapters) The first calculation stage is a *Consolidation* analysis, *Staged construction*.

- Add a new phase.
- In the *Phases* window select the *Consolidation* option from the *Calculation type* drop-down menu in the *General* subtree.
- Make sure that the *Staged construction* option is selected for the *Loading type*.
- Enter a *Time interval* of 2 days. The default values of the remaining parameters will be used.
- In the *Staged construction* mode activate the first part of the embankment (Figure 5.9).

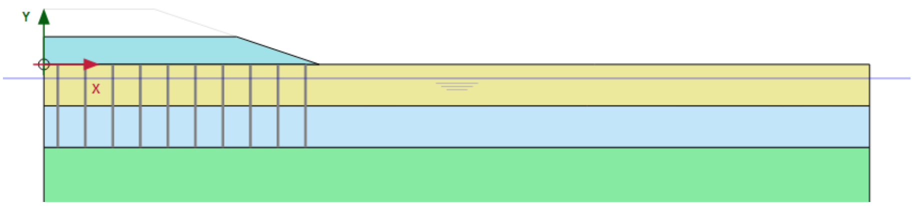

Figure 5.9 Configuration of the phase 1

Phase 2: 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.

- Add a new phase.
- In the *Phases* window select the *Consolidation* option from the *Calculation type* drop-down menu in the *General* subtree.
- Make sure that the *Staged construction* option is selected for the *Loading type*.

- Enter a *Time interval* of 30 days. The default values of the remaining parameters will be used.

Phase 3: The third phase is once again a *Consolidation* analysis, *Staged construction*.

Add a new phase.

In the *Phases* window select the *Consolidation* option from the *Calculation type* drop-down menu in the *General* subtree.

Make sure that the *Staged construction* option is selected for the *Loading type*.

- Enter a *Time interval* of 1 day. The default values of the remaining parameters will be used.
- In the *Staged construction* mode activate the second part of the embankment (Figure 5.10).

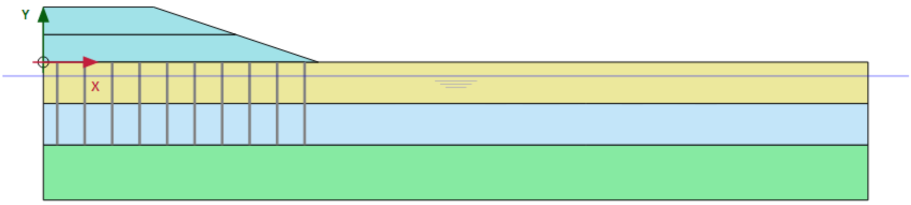

Figure 5.10 Configuration of the phase 3

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

Add a new phase.

In the *General* subtree select the *Consolidation* as calculation type.

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.

Before starting the calculation, click the *Select points for curves* button and select the following points: As the first point, select the toe of the embankment (point at (20.00,0.00)). 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 is needed, close to (but not actually on) the left boundary (point at (0.25, -3.00)). After selecting these points, start the calculation.

During a consolidation analysis the development of time can be viewed in the upper part of the calculation info window (Figure 5.11).

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.

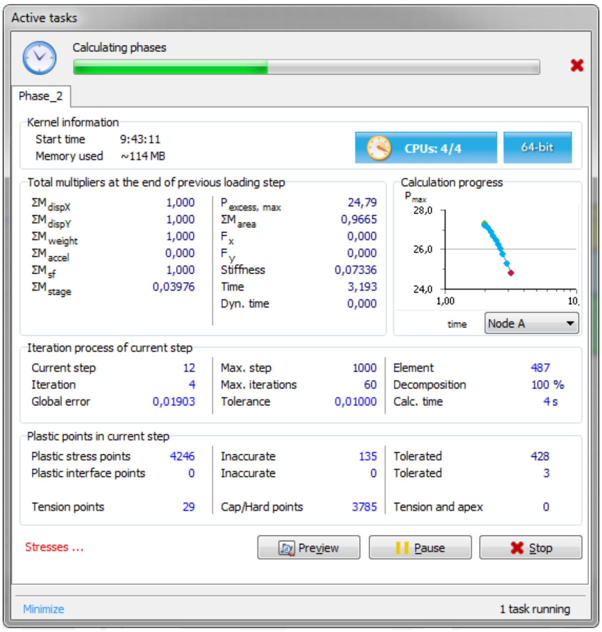

Figure 5.11 Calculation progress displayed in the *Active tasks* window

5.4 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 5.12). Considering the results of the third phase, the deformed mesh shows the uplift of the embankment toe and hinterland due to the undrained behaviour.

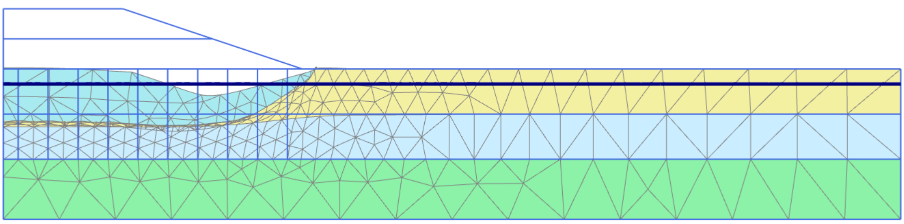

Figure 5.12 Deformed mesh after undrained construction of embankment (Phase 3)

- In the *Deformations* menu select the *Incremental displacements* → $|\Delta u|$.
- 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 5.13).

- 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.

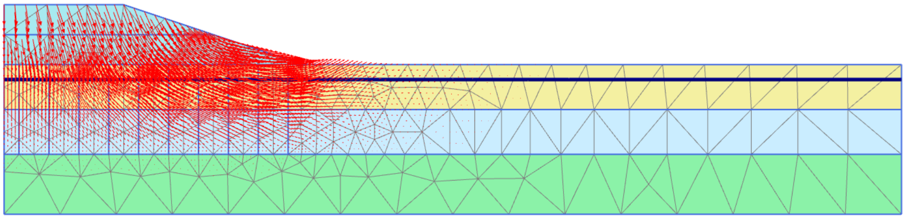

Figure 5.13 Displacement increments after undrained construction of embankment

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

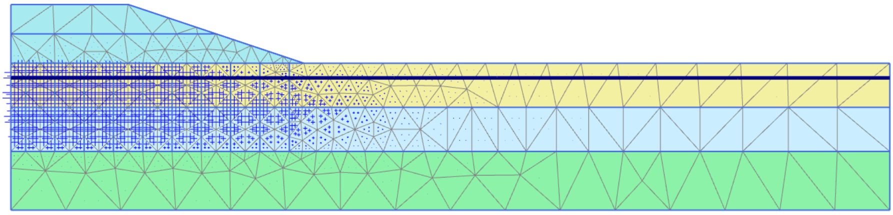

Figure 5.14 Excess pore pressures after undrained construction of embankment

- Select Phase 4 in the drop down menu.

Click the *Contour lines* button in the toolbar to display the results as contours.

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

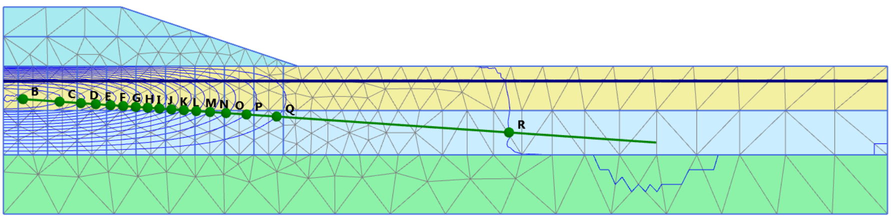

Figure 5.15 Excess pore pressure contours after consolidation to $P_{excess} < 1.0$ kN/m^2

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 5.15 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:

Create a new curve.

- 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 (Point B) from the drop-down menu. In the tree select *Stresses* → *Pore pressure* → p_{excess}.
- Select the *Invert sign* option for y-axis. After clicking the *OK* button, a curve similar to Figure 5.16 should appear.

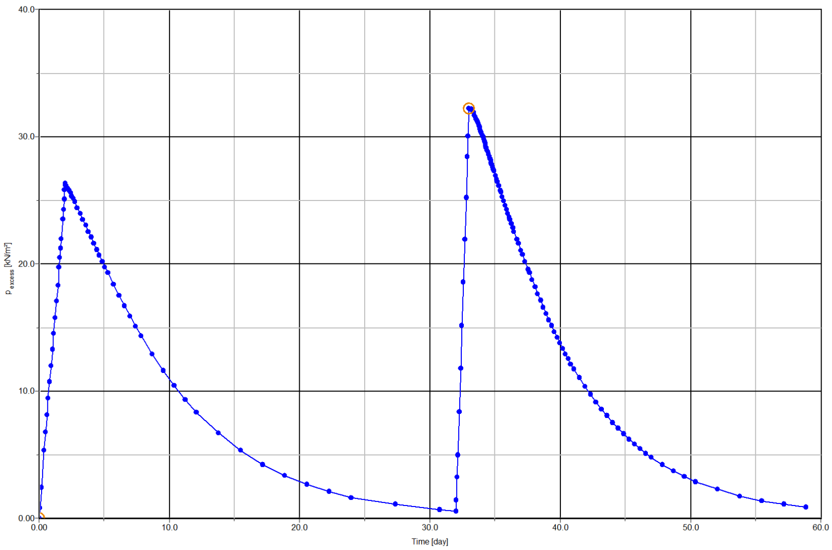

Figure 5.16 Development of excess pore pressure under the embankment

Figure 5.16 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.

5.5 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:

$$\text{Safety factor} = \frac{S_{maximum\ available}}{S_{needed\ for\ equilibrium}} \tag{5.1}$$

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:

$$\text{Safety factor} = \frac{c - \sigma_n \tan \varphi}{c_r - \sigma_n \tan \varphi_r} \tag{5.2}$$

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 the method of *Safety* 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:

$$\frac{c}{c_r} = \frac{\tan \varphi}{\tan \varphi_r} = \Sigma Msf \tag{5.3}$$

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 *General* tabsheet. If the *Safety* option is selected the *Loading input* on the *Parameters* tabsheet is automatically set to *Incremental multipliers*.

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

- Select Phase 1 in the *Phases* explorer.
- Add a new calculation phase.
- Double-click on the new phase to open the *Phases* window.
- In the *Phases* window the selected phase is automatically selected in the *Start from phase* drop-down menu.
- In the *General* subtree, select *Safety* as calculation type.
- 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.
- 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.
- In the *Numerical control parameters* subtree deselect *Use default iter parameters* and set the number of *Max steps* to 50. 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.

> **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 1000.
>
> » 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.

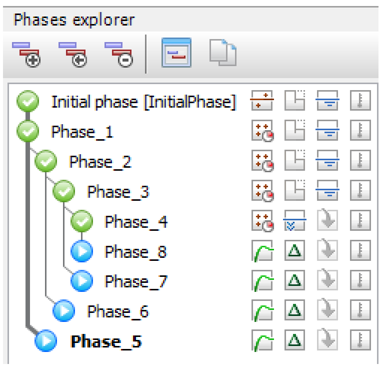

Figure 5.17 *Phases explorer* displaying the *Safety calculation* phases

Evaluation of 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:

- Select one of these phases 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 5.18). The magnitude of the displacement increments is not relevant.

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.

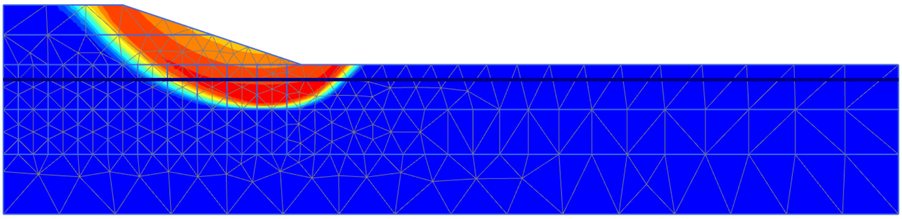

Figure 5.18 Shadings of the total displacement increments indicating the most applicable failure mechanism of the embankment in the final stage

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 (Point A) for the x-axis. Select *Deformations* → *Total displacements* → $|u|$.
- For the y-axis, select *Project* and then select *Multipliers* → ΣMsf. The *Safety* phases are considered in the chart.
- Right-click on the chart and select the *Settings* option in the appearing menu. The *Settings* window pops up.
- In the tabsheet corresponding to the curve click the *Phases* button.
- In the *Select phases* window select Phase 5 (Figure 5.19).

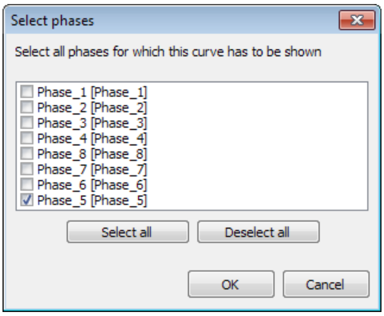

Figure 5.19 The *Select phases* window

- Click *OK* to close the *Select phases* window.
- In the *Settings* window change the titles of the curve in the corresponding tabsheet.
- 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.

- In the *Settings* window click the *Chart* tab to open the corresponding tabsheet.
- In the *Chart* tabsheet specify the chart name.
- Set the scaling of the x-axis to *Manual* and set the value of *Maximum* to 1 (Figure 5.20).

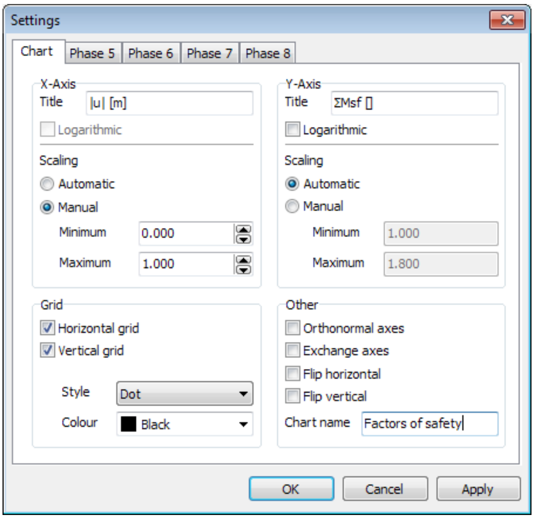

Figure 5.20 The *Chart* tabsheet in the *Settings* window

- Click *Apply* to update the chart according to the changes made and click *OK* to close the *Settings* window.
- To modify the location of the legend right-click on the legend.
- In the appearing menu point at *View* and select the *Legend in chart* option (Figure 5.21).

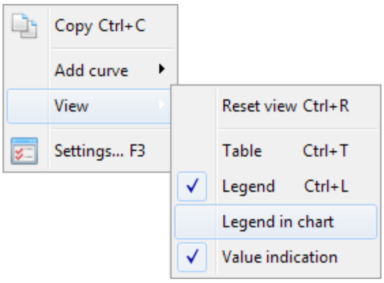

Figure 5.21 Enabling legend in chart

- The legend can be relocated in the chart by dragging it. The plot is shown in Figure 5.22.

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.

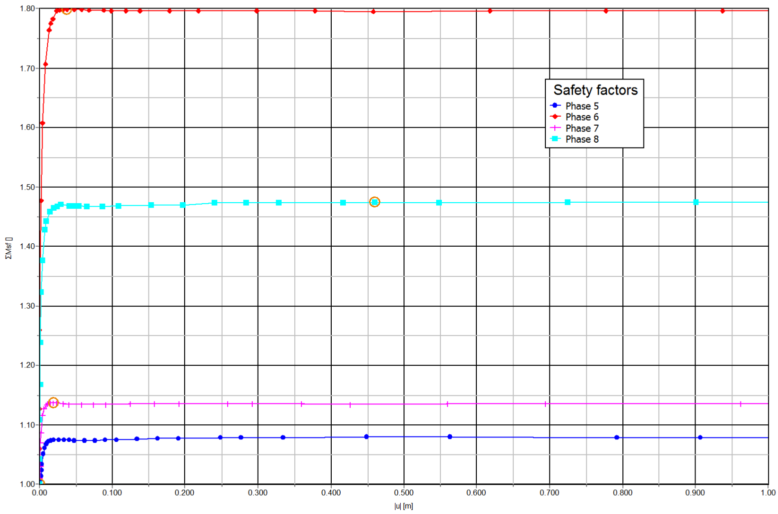

Figure 5.22 Evaluation of safety factor

5.6 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|).

 After the calculation is finished save the project, 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.

 Open the *Curves manager*.

- In the *Chart* tabsheet double click Chart 1 (p_{excess} of the second point (at (0.25,-3.00)) versus time). The chart is displayed. Close the *Curves manager*.
- Double-click the curve in the legend at the right of the chart. 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.
- Select the *Invert sign* option for y-axis and click *OK* to accept the selected options.
- 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* window change the titles of the curves in the corresponding tabsheets.
- In the *Chart* tabsheet specify the chart name.
- Click *Apply* to preview the generated curve and click *OK* to close the *Settings* window. The chart (Figure 5.23) gives a clear view of the effect of drains in the time required for the excess pore pressures to dissipate.

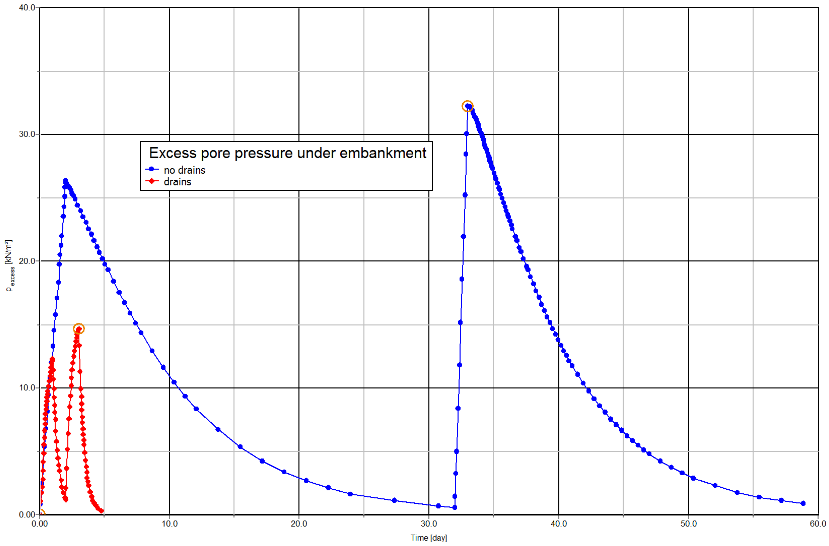

Figure 5.23 Effect of drains

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

5.7 UPDATED MESH + 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 using the *Updated mesh* and *Updated water pressures* options. For the road embankment the effect of using these options will be investigated.

- Select the initial phase in the *Phases* explorer.

 Add a new calculation phase.

- 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.
- Define in the same way the other 3 phases.

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

- 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 (point at (0.25,-3.00)) for the *y*-axis.
- In this curve the results for Initial phase and phases from 1 to 4 will be considered.
- Add a new curve to the chart.
- In this curve the results for Initial phase and phases from 13 to 16 will be considered. The resulting chart is shown in Figure 5.24.

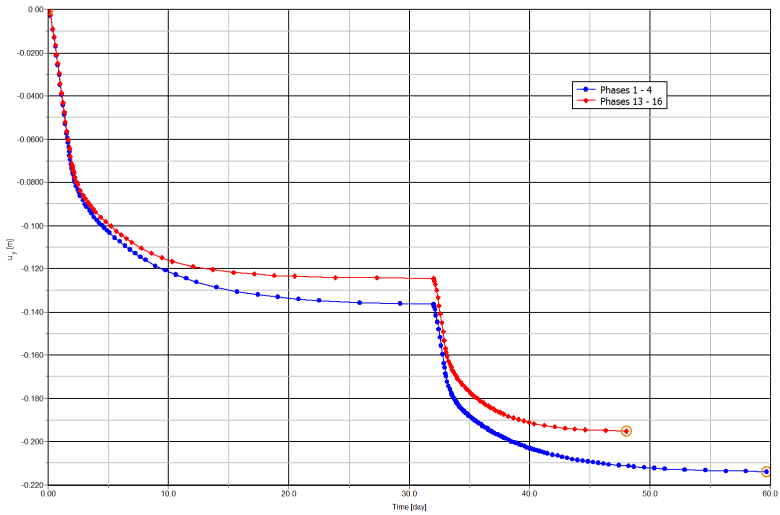

Figure 5.24 Effect of Updated mesh and water pressures analysis on resulting settlements

In Figure 5.24 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.