2

Drained and undrained stability of an embankment

In this chapter the construction of an embankment on clay is simulated. The clay will first be considered a drained material and then an undrained material. For both cases the factor of safety will be determined. This would for instance give an indication of both the long term and short term stability of the embankment.

Objectives

- Modelling **Drained** and **Undrained** soil behaviour.
- Changing material sets during the calculation.
- Calculating a factor of safety.

Geometry

Figure 28 (on page 39) shows the layout of an embankment . The embankment is 4 m high and the crest of the embankment has a width of 2.0 m. The groundwater level is just below surface, but to simplify the problem it will be defined at ground level.
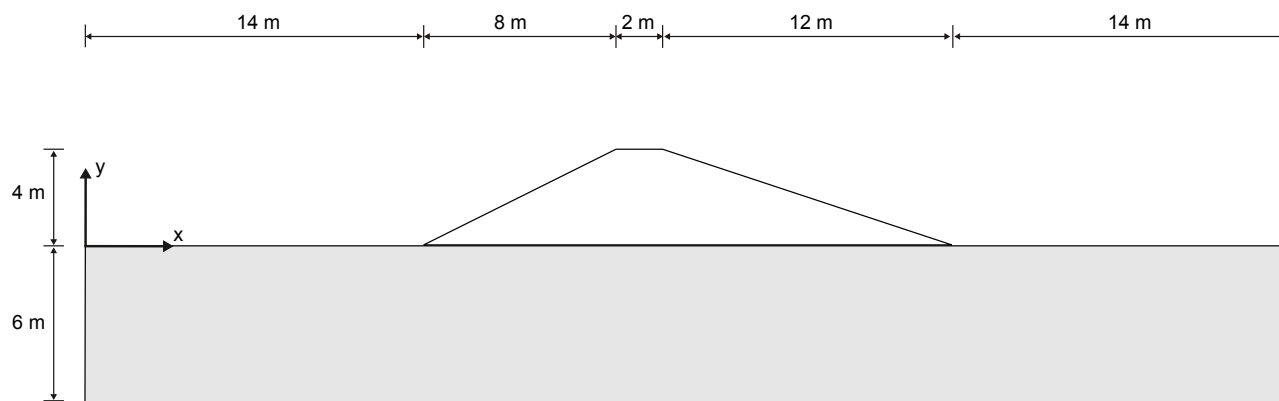

Figure 28: Geometry of the project

2.1 Create new project

To create a new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet keep the default options for **Model** (**Plane strain**), and **Elements** (**15-Noded**).

4. Set the model dimensions to: x_{min} = 0 m, x_{max} = 50 m, y_{min} = -6 m and y_{max} = 4 m.
5. Keep the default values for units, constants and the general parameters and click **OK** to close the **Project properties** window.

2.2 Define the soil stratigraphy

The subsoil profile consists of a single clay layer extending until large depth. Since we are only interested in the stability of the embankment it is not necessary to model the clay layer until very large depth; the model has to be deep enough to allow the failure mechanism to form. Please note that for a deformation analysis a deeper model may be required as deformations due to the construction of the embankment will still occur at considerable depth.

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 .
2. **Add** a single soil layer from **Top** = 0 to **Bottom** = -6.
3. Keep the **Head** for this borehole to 0. Hence, the groundwater level coincides with ground level.
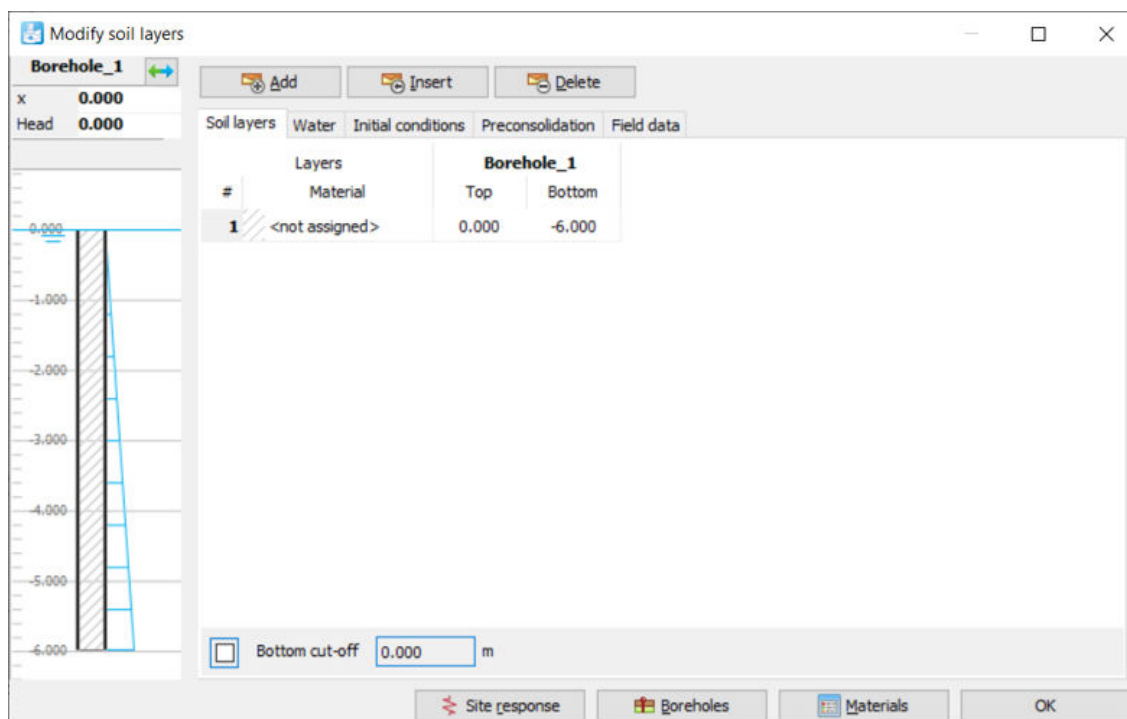

*Figure 29: The soil stratigraphy in the **Modify soil layers** window*

2.3 Create and assign material data sets

For this project three material sets have to be defined. One material to represent the sand of which the embankment will be constructed and two materials representing the subsoil: a drained and an undrained material set representing the clay in long term and short term conditions.

Table 3: Material properties of the soil layer

Parameter	Name	Embankment	Clay layer	Unit
General				
Soil model	-	Hardening soil	Hardening soil	-
Type of material behaviour	-	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	16	13	kN/m^3
Saturated unit weight	γ_{sat}	16	13	kN/m^3
Mechanical				
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	15·10^3	5,600	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	15·10^3	5,000	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	45·10^3	20·10^3	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	1.0	-
Cohesion	c'_{ref}	3	10	kN/m^2
Friction angle	φ'	30	25	°
Initial				
K$_0$-determination	-	Automatic	Automatic	-
Overconsolidation ratio	OCR	1.0	1.2	-

1. Select the **Show materials** button so that the **Materials sets** window shows up.
2. Using the **New** button define the two material sets as defined in the Table 3 (on page 41) .

3. In order to create the undrained material set for the clay layer select the drained material in the **Material sets** window and click the **Copy** button to duplicate the material set. In the copied material set, change the name and set the **Drainage type** to **Undrained (A)**.
4. Assign the material set representing the drained clay to the subsoil.

2.4 Create the embankment

We will now create the embankment. This is done by defining a soil polygon after which the polygon gets assigned the material set representing the embankment material.

1. Go to **Structures mode** .
2. From the tools side bar select the **Create soil polygon** option ⬛ and then from the small popup button menu that appears select the **Create soil polygon** ⬛ option again.
3. Now draw a polygon starting from (x y) = (14 0) continuing to (22 4), (24 4) and finally (36 0).
4. Now assign the embankment material to the polygon. This can either be done by opening the **Material sets** window and then drag and drop the material set onto the polygon, or by selecting the polygon and then in the **Selection explorer** set the **Material** option of the soil polygon to the embankment material.

2.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button 🔲 to generate the mesh.
   The **Mesh options** window appears.
3. Select the **Fine** option in the **Element distribution** list and generate the mesh.
4. Click the **View mesh** button 🔍 to view the mesh and the generated mesh is shown in .
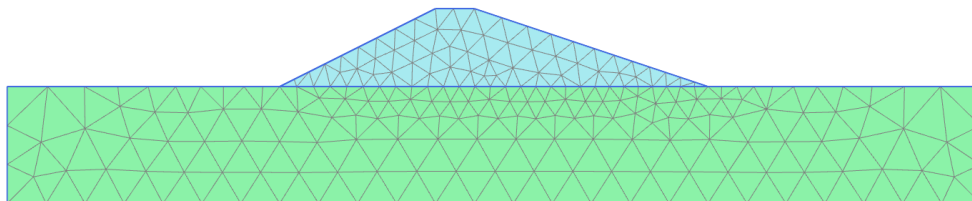

Figure 30: The generated mesh

5. Select the **Close** button on the top left of the Output program to close the mesh view.

2.6 Define and perform the calculation

The embankment construction is done in a single stage here, as we're only interested in the stability. Note that for settlement predictions it is better to divide the construction of the embankment in multiple phases, if necessary with periods of consolidation in between. The embankment construction has to be done both on drained and undrained subsoil, so a total of three calculation phases have to be defined: the initial phase and two construction phases.

2.6.1 Initial phase: Initial conditions

In the initial situation the embankment is not present yet. Since the subsoil consists of only 1 layer with a horizontal ground level the **K0 procedure** can be used to generate the initial stresses. As this is the default option, no changes have to be made to the initial phase. The model in the initial phase is shown in .
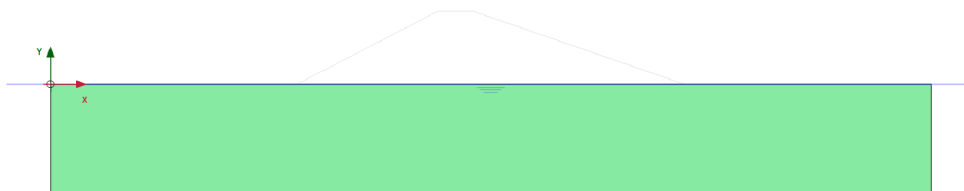

Figure 31: Configuration of the initial phase

2.6.2 Phase 1: Embankment construction on drained subsoil

1. Click the **Add phase** button ⊞ to create a new phase .
   In the new phase the **Calculation Type** > **Plastic** analysis and the **Loading type** > **Staged construction** are used as a default.
2. Right-click on the embankment and from the popup menu select the **Activate** option to activate the soil representing the embankment. The model of phase 1 is shown in .

Figure 32: Configuration of phase 1
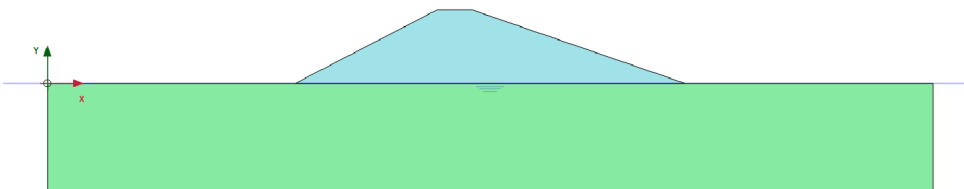
2.6.3 Phase 2: Embankment construction on undrained subsoil

In this second calculation phase the same embankment will be constructed, by now under the assumption of undrained behaviour of the subsoil. This implies that the material set of the subsoil must be changed, but also that this construction phase must start from the initial phase as it is an alternative calculation for phase 1 rather than a continuation of phase 1.

1. In the **Phase explorer** select the **Initial phase** and then create a new phase using the **Add phase** button 🖳. Because the initial phase was the selected phase the newly created phase 2 will start from the initial phase. In case by mistake phase 2 does start from phase 1, this can be changed double-clicking on **Phase 2** in the **Selection explorer** so that the **Phases** window opens. In the **General** section now set the option **Start from phase** to the initial phase and close the **Phases** window again.

2. Now the soil behaviour of the subsoil must be changed to undrained by assigning the undrained material set to the subsoil. There are several ways to do this:

   From the navigation side bar select the **Show materials** button 🔳. From the **Material sets** window that opens drag and drop the material set for undrained clay on the subsoil.

   or

   Right-click on the subsoil and from the popup menu(s) that open consecutively select the options **Soil** (🟩), **Soil** (🟩), **Set material** and finally the material representing the undrained subsoil to assign it.

   or

   Select the subsoil and in the **Selection explorer** change the **Material** under the **Soil** object for the material representing the undrained subsoil.

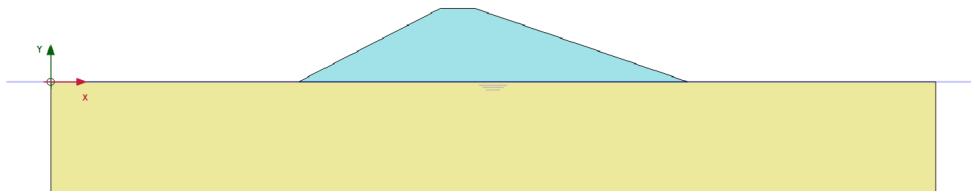3. Finally, activate the embankment, and the resulting model is shown in .

Figure 33: Configuration of phase 2

2.6.4 Calculate

Before starting the calculation it's recommended to choose some nodes or stress points to later evaluate the results in for instance load-displacement or stress-strain curves. In this project we will select a point halfway the left hand side slope to later evaluate the factor of safety.

1. Click the **Select points for curves** button 🔻 on the navigation side bar.
2. Select a **Node** halfway the left hand side slope, hence around (x y) = (18 2).

3. Click the **Update** button on the top left to close the Output program and store the selected point.
4. Click the **Calculate** button ⎇ to start the calculation.

2.7 Results

1. After the calculation ended, select the first phase and click the **View calculations results** button ⎍.
   The Output program now opens showing the deformed mesh after the construction of the embankment on the drained subsoil.
2. From the dropdown list on the button bar at the top now choose to see the results for Phase 2. Now the deformed mesh after construction of the embankment on undrained subsoil is shown.

   Figure 34 (on page 45) shows the deformed mesh for both phases. In case of the drained subsoil the embankment settles everywhere, but in case of the undrained subsoil the embankment settles in the middle but heaves near the toes. This makes perfect sense: the subsoil is now undrained and because of that there can be no volume change. Hence, if the subsoil settles in the middle due to the weight of the embankment it must heave somewhere else, typically just next to the embankment.
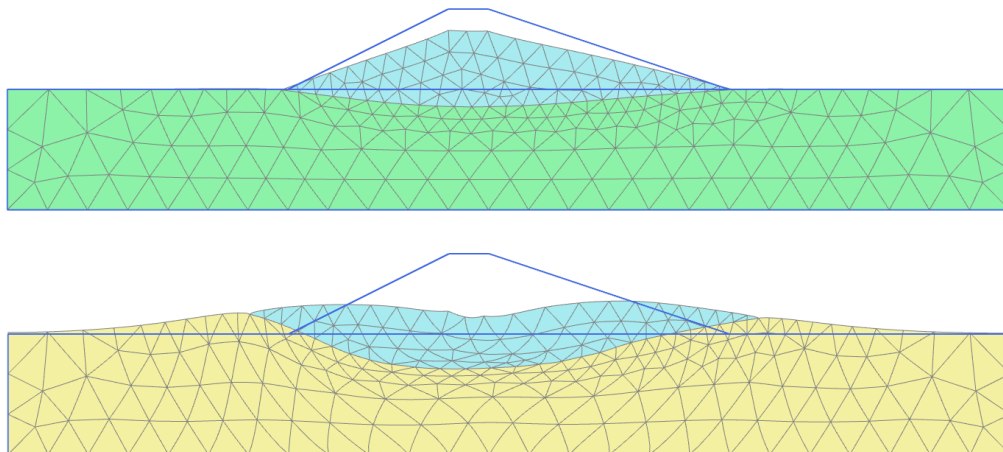
Figure 34: Deformed mesh after phase 1 (top) and phase 2 (bottom)

   It can also be seen that in case of the undrained subsoil the embankment seems to widen leading to a larger settlement at the top of the embankment. This is not immediately obvious and it will be discussed in more detail after performing a factor of safety analysis.

3. From the **Stresses** menu select the option **Pore pressures** and then p_{excess}. This will show the excess pore pressures due to the undrained loading of the subsoil. By default the pore pressures are shown as iso-shadings but with the buttons ⁝ and ⁝ from the horizontal button bar at the top it is possible to see the principal directions of the pore pressures, either for all or a reduced number of stress points.

   Figure 35 (on page 46) shows that excess pore pressures have developed due to the construction of the embankment. The highest excess pore pressures are of course directly underneath the embankment, but also on either side of the toes of the embankment some excess pore pressures occur.
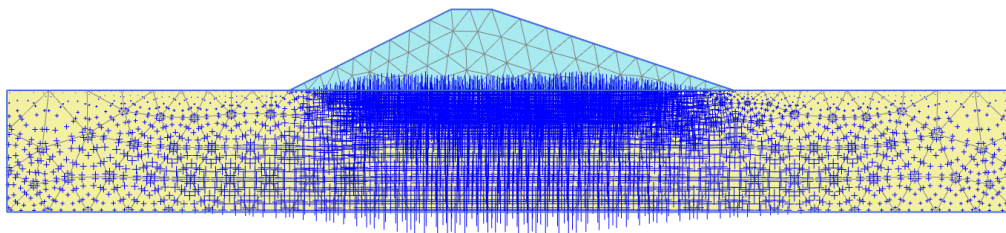
Figure 35: Excess pore pressures after construction on undrained subsoil

4. After viewing the results, close the Output program and return to the Input program.

2.8 Safety analysis

In the design of an embankment it is important to consider not only the settlements, but also the stability in terms of a factor of safety. It could be seen that the settlements are different for the construction of an embankment on drained or undrained subsoil and so it would be interesting to evaluate a global safety factor for both cases.

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

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

Where S represents the shear strength. The ratio of the true strength to the computed minimum strength required for equilibrium (hence, the mobilized strength) is the safety factor that is conventionally used in soil mechanics. For soil models using the standard Coulomb failure condition, the safety factor is obtained as:

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

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 2D to calculate a global safety factor. In this approach the cohesion and the tangent of the friction angle are reduced in the same proportion:

$$\frac{c}{c_r} = \frac{\tan(\varphi)}{\tan(\varphi_r)} = \Sigma Msf$$

The reduction of strength parameters is controlled by a multiplier called ΣMsf that is increased step-by-step 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 with ongoing deformations.

Note that for soil models that do no use the Coulomb failure criterion the concept of strength reduction remains the same, but with reduction of strength parameters specific to that soil model.

The **Safety** calculation option is available in the **Calculation type** drop-down menu in the **General** section of the **Phases** window. If the **Safety** option is selected the **Loading input** on the **Parameters** tabsheet is automatically set to **Incremental multipliers**, this means that the multiplier ΣMsf will be incremented until failure occurs. The other option for the **Loading input** is **Target-ΣMsf**, which means the multiplier ΣMsf will only increase until the specified target value and not until failure. However, this latter option is not used here.

To calculate the global safety factor for the road embankment for both cases, 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_1** drop-down menu.
5. In the **General** subtree, set the **Calculation type** as **Safety** 🖼 from the drop down menu.
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 a new calculation phase that analyses the stability at the end of construction of the embankment on undrained subsoil.
10. Click the **Calculate** button 🖼 to start the calculation .

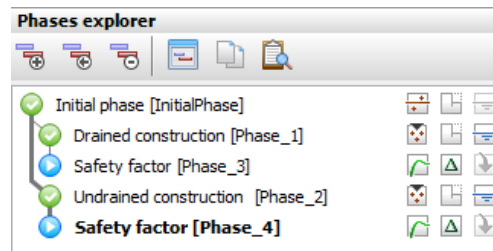The phases explorer window displaying the safety calculation is shown in Figure 36 (on page 47).

Figure 36: Phases explorer displaying the Safety calculation phases

Note:

Note that for a **Safety** phase the option **Use pressures from the previous phase** 🖼 in the **Pore pressure calculation type** drop-down menu is automatically selected and grayed out indicating that this option cannot be changed. A **Safety** calculation always uses the same pore pressures as the phase for which the safety factor has to be calculated.

Note:

The default value of **Max steps** in a **Safety** calculation is 100. In contrast to a **Staged construction** calculation, the specified number of steps is always fully executed. In most **Safety** calculations, 100 steps are sufficient to arrive at a state of failure. If not, the number of steps can be increased to a maximum of 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.

2.8.1 Evaluation of safety analysis results

Due to the ongoing reduction of the soil strength, additional displacements are generated during a **Safety** calculation. The resulting total displacements do not have a physical meaning as they depend on the amount of load steps that was applied: more load steps means the calculation would push further into failure thus generating larger displacement while in reality a failing embankment would re-establish a new equilibrium with limited deformations. However, the incremental displacements in the final step (at failure) are very useful as they give an indication of the likely failure mechanism. The incremental displacements are the change of displacement per load increment. Typically this is a very small value as the load increments are small, but in case of failure the failure zone only needs a very small change of load to generate large changes of displacements, hence the failure zone then has large incremental displacement whereas anywhere else in the model the incremental displacements should be small.

In order to view the mechanism for the embankment on drained subsoil:

1. Select Phase 3, that is the **Safety** phase following on Phase 1, and click the **View calculation results** button .

2. In the Output program select the menu **Deformations** > **Incremental displacements** > **|Δu|**.
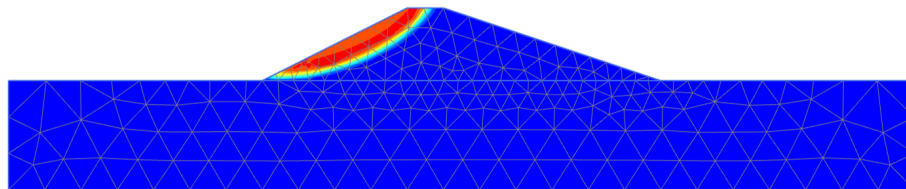

Figure 37: Shadings of the displacement increments indicating the most applicable failure mechanism of the embankment after drained construction

The resulting plot shown in Figure 37 (on page 48) gives a good impression of the failure mechanisms. The magnitude of the displacement increments is not relevant. From the results it can be seen that the slope on the left side of the embankment fails with a classical slip surface .

By choosing phase 4 from the drop down list at the toolbar the failure mechanism for the embankment constructed on undrained subsoil can be evaluated. The failure mechanism is no longer limited to just the embankment, but is in fact mostly a failure of the subsoil underneath the embankment. This also explains the widening of the embankment at the base as was observed earlier: the soil underneath the embankment fails and moves horizontally away from the center of the embankment shown in Figure 38 (on page 49).
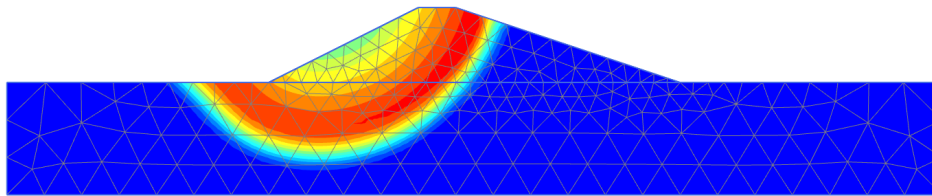
Figure 38: Shadings of the displacement increments indicating the most applicable failure mechanism of the embankment after undrained construction

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, use from the dropdown list the previously selected **Node** 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. Press **OK** to close the window and generate the chart.
6. Right-click on the chart and select the **Settings** option in the appearing menu. The **Settings** window pops up.
7. In the tabsheet corresponding to the curve click the **Phases** button.
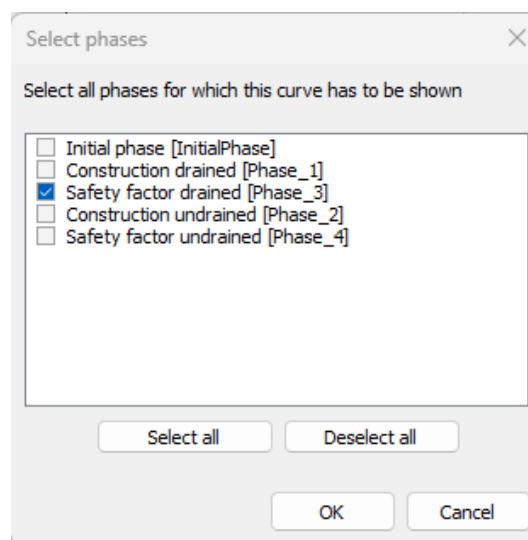8. In the **Select phases** window select only Phase 3 as shown in :

Figure 39: The Select phases window

9. Click **OK** to close the **Select phases** window.
10. In the **Settings** window change the titles of the curve in the corresponding tabsheet.
11. Click the **Add curve** button and select the **From current project** option in the appearing menu. Define the curve for the phase 4 by following the same steps as described for phase 3.
12. In the **Chart** tabsheet shown in Figure 40 (on page 50), set the scaling of the x-axis to **Manual** and set the value of **Maximum** to 0.5:
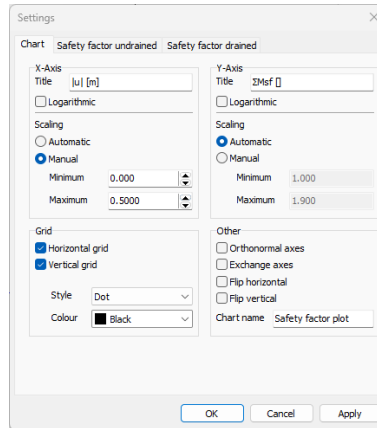

Figure 40: The Chart tabsheet in the Settings window

13. Click **Apply** to update the chart according to the changes made and click **OK** to close the **Settings** window.
14. C
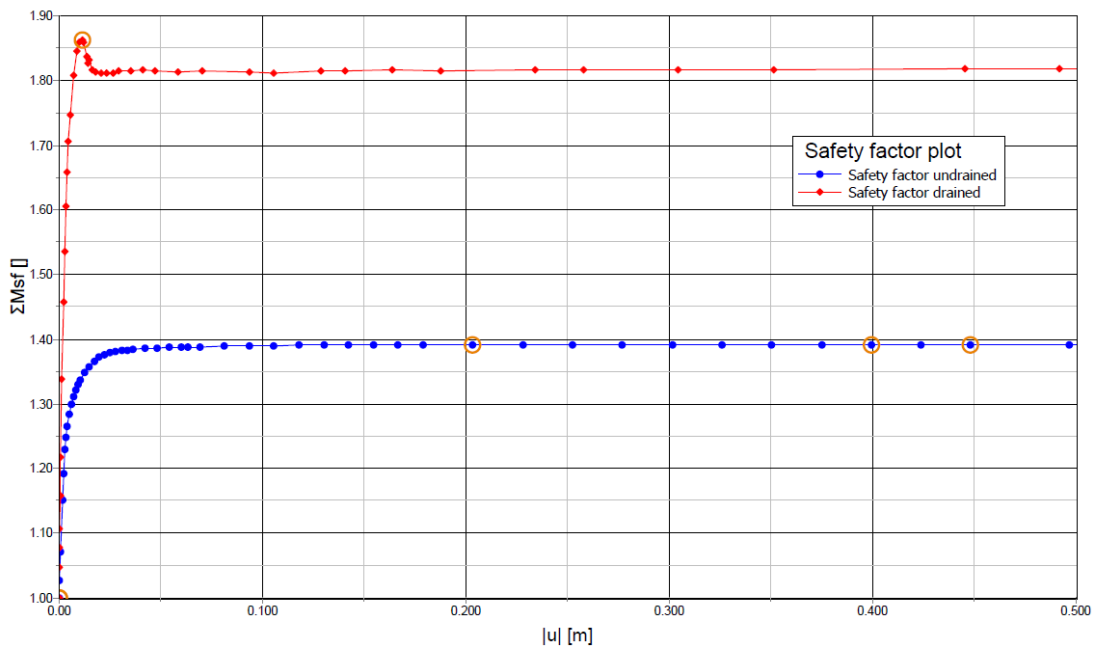15. The plot is shown as follows in Figure 41 (on page 50):

Figure 41: Evaluation of safety factor

The maximum displacements plotted are not relevant. It can be seen that for both curves a more or less constant value of ΣMsf is obtained. Hovering the mouse cursor over a point on the curves, a box shows up with the exact value of ΣMsf as well as the calculation phase. With the latter it can be determined that the upper curve with a factor of safety of 1.8 is phase 3, hence the embankment on drained soil. Similarly the lower curve with a factor of safety of 1.4 is phase 4, the embankment on undrained soil.