11

Flow through an embankment [ULT]

In this chapter the flow through an embankment will be considered. Flow takes place from the left side (river) to the right side (polder). As a result seepage will take place at the right side of the embankment. The position of the phreatic level depends on the river water level, which varies in time.

Objectives

- Performing **Flow only** analysis
- Using cross section curves

Geometry

The figure below shows the layout of the embankment problem where free surface groundwater flow occurs. The crest of the embankment has a width of 2.0 m. Initially the water in the river is 1.5 m deep. The difference in water level between the river and the polder is 3.5 m.
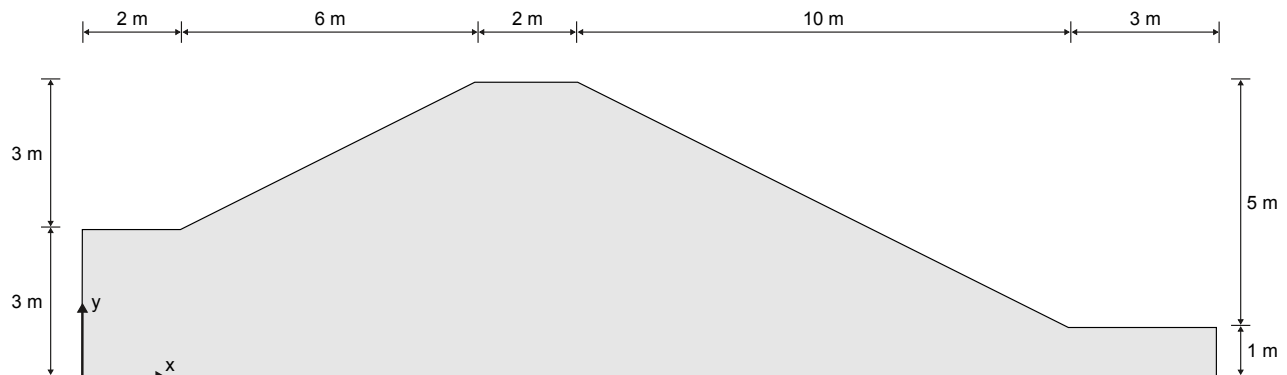
Figure 133: Geometry of the project

11.1 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 select** 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-Node**).
4. Set the model dimensions to: x_{min} = 0 m, x_{max} = 23 m, y_{min} = 0 m and y_{max} = 6 m

5. Keep the default values for units, constants and the general parameters and click **OK**.
   The **Project properties** window closes.

11.2 Define the soil stratigraphy

A number of boreholes has to be defined according to the information in the table below.

Table 26: Information on the boreholes in the model

Borehole number	Location (x)	Head	Top	Bottom
1	2	4.5	3	0
2	8	4.5	6	0
3	10	4.0	6	0
4	20	1.0	1	0

To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 2.
   The **Modify soil layers** window pops up.
2. Specify the head value as 4.5.
3. Add a soil layer in the borehole. Set the top level to 3. No change is required for the bottom boundary of the layer.
4. Create the rest of the required boreholes according to the information given in Table 26 (on page 160).

11.3 Create and assign material data set

A material data set needs to be created for the soil layer.

The sand layer has the following properties:

Table 27: Material properties of the embankment material (sand)

Parameter	Name	Sand	Unit
General			
Material model	-	Linear elastic	-
Drainage type	-	Drained	-

Parameter	Name	Sand	Unit
Soil unit weight above phreatic level	γ_{unsat}	20	kN/m^3
Soil unit weight below phreatic level	γ_{sat}	20	kN/m^3
Parameters			
Young's modulus	E'	$10{\cdot}10^3$	kN/m^2
Poisson's ratio	ν'	0.3	-
Groundwater			
Data set	-	Standard	-
Soil Type	-	Medium fine	-
Flow parameters - Use defaults	-	From data set	-
Horizontal permeability	k_x	0.02272	m/day
Vertical permeability	k_y	0.02272	m/day

To create the material set, follow these steps:

1. ![icon] Define the soil material according to the table above and assign the material dataset to the cluster. Skip the **Interfaces** and **Initial** tabsheets as these parameters are not relevant.
2. After assigning the material to the soil cluster close the **Modify soil layers** window.

11.4 Generate the mesh

1. Proceed to the **Mesh** mode.
2. Select the two lines that form the left hand side slope and river bed as shown in the figure below. In the **Selection Explorer** specify a **Coarseness factor** of 0.5.
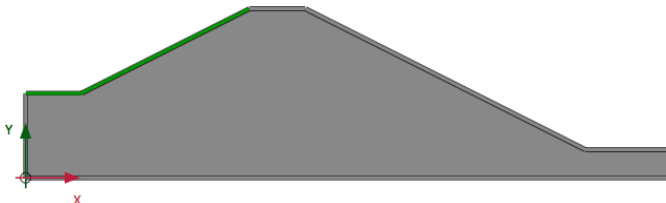
3. Click the **Generate mesh** button ![icon] to generate the mesh.
   The **Mesh options** window appears.
4. Select the **Fine** option in the **Element distribution** drop-down menu and generate the mesh.

5. Click the **View mesh** button to view the mesh.
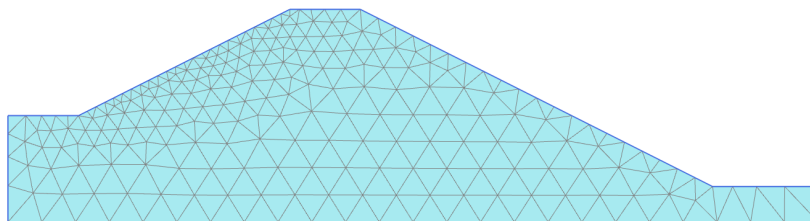

Figure 134: The generated mesh

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

11.5 Define and perform the calculation

In this project only the flow related behaviour will be analysed. The calculation process consists of three phases that will be defined in the **Staged construction** mode. In the initial phase, the groundwater flow in steady state is calculated for an average river level. In Phase 1, the transient groundwater flow is calculated for a harmonic variation of the water level. In Phase 2, the calculation is similar as in Phase 1, but the period is longer.

Click the **Staged construction** tab to proceed to the corresponding mode. A global level is automatically created according to the head values specified for each borehole ([Table 26](navigation) (on page 160)). The model in the **Staged construction** mode is shown in the figure below.
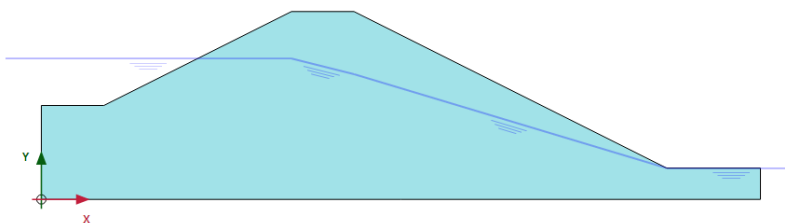

*Figure 135: The model in the **Staged construction** mode*

Note: Note that the 'internal' part of the global water level will be replaced by the result of the groundwater flow calculation.

11.5.1 Initial phase

1. Double-click the initial phase in the **Phases** explorer.
2. In the **General** subtree select the **Flow only** option as the **Calculation type**.
3. The default values of the remaining parameters are valid for this phase. Click **OK** to close the **Phases** window.
4. In the **Model explorer** expand the **Model conditions** subtree.
5. In the **Model conditions** expand the **GroundwaterFlow** subtree. The default boundary conditions are relevant for the initial phase. Check that only the bottom boundary is closed.
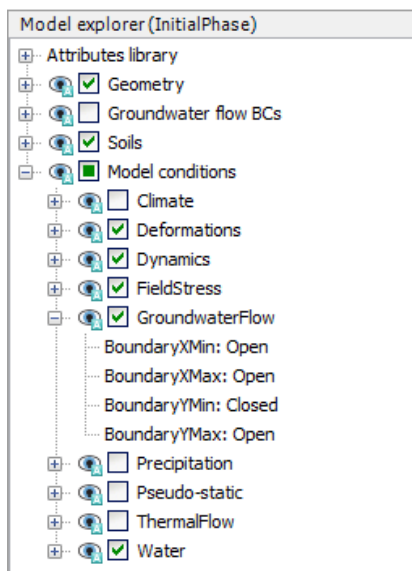
Figure 136: The groundwater flow boundary conditions for the initial phase

6. In the **Model explorer** expand the **Groundwater flow BCs** subtree. The boundary conditions at the extremities of the model are automatically created by the program and listed under the **GWFlowBaseBC**.

Note: Note that when the boundary conditions under the **Groundwater flow BCs** subtree are active, the model conditions specified in the **GroundwaterFlow** are ignored.

11.5.2 Phase 1

1. Click the **Add phase** button to create a new phase.

2. In the **Phases explorer** double-click the current phase.

3. In the **General** subtree select the **Transient groundwater flow** option as pore pressure calculation type.

4. Set the **Time interval** to 1 day.

5. In the **Numerical control parameters** subtree set the **Max number of steps stored** parameter to 50. The default values of the remaining parameters will be used.

6. Click **OK** to close the **Phases** window.

7. Click the **Select multiple objects** button in the side toolbar.

8. Click **Select lines** > **Select water boundaries**.

9. Select the hydraulic boundaries as shown in the following figure.
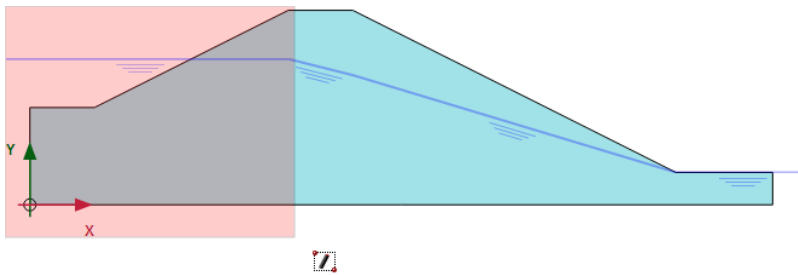
10. Right-click and click **Activate**.
11. In the **Selection explorer** set the **Behaviour** parameter to **Head**.
12. Set h_{ref} to 4.5 m.
13. Select the **Time dependent** option in the **Time dependency** drop-down menu.
14. Click on the **Head function** parameter.
15. Click the **Add** button ➕ to add a new head function.
16. In the **Flow functions** window select the **Harmonic** option in the **Signal** drop-down menu. Set the amplitude to 1 m, the phase angle to 0° and the period to 1 day.
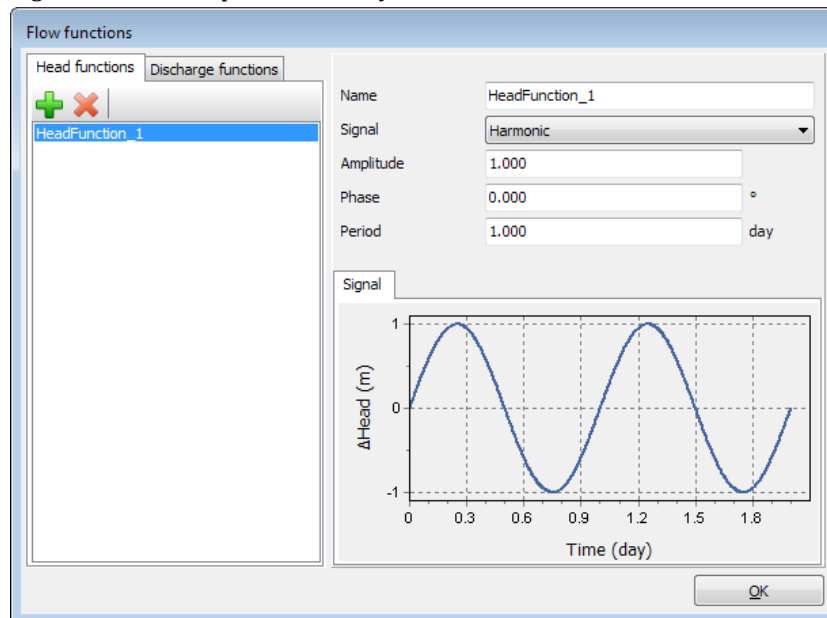

Figure 137: The flow function for the rapid case

17. Click **OK** to close the **Flow functions** window.

11.5.3 Phase 2

1. Click the **Add phase** button 🔲 to create a new phase.
2. In the **Phases explorer** double-click the current phase.
3. In the **General** subtree select the **Initial phase** in the **Start from phase** drop-down menu.

4. Select the **Transient groundwater flow** option as **Pore pressure calculation type**.
5. Set the **Time interval** to 10.0 day.
6. In the **Numerical control parameters** subtree set the **Max number of steps stored** parameter to 50. The default values of the remaining parameters will be used.
7. Click **OK** to close the **Phases** window.
8. Make sure that the same boundaries are selected as in Phase 1.
9. In the **Selection explorer** click on the **Head function** parameter.
10. Click the **Add** button ⊞ to add a new head function.
11. In the **Flow functions** window select the **Harmonic** option in the **Signal** drop-down menu. Set the amplitude to 1 m, the phase angle to 0° and the period to 10 day.
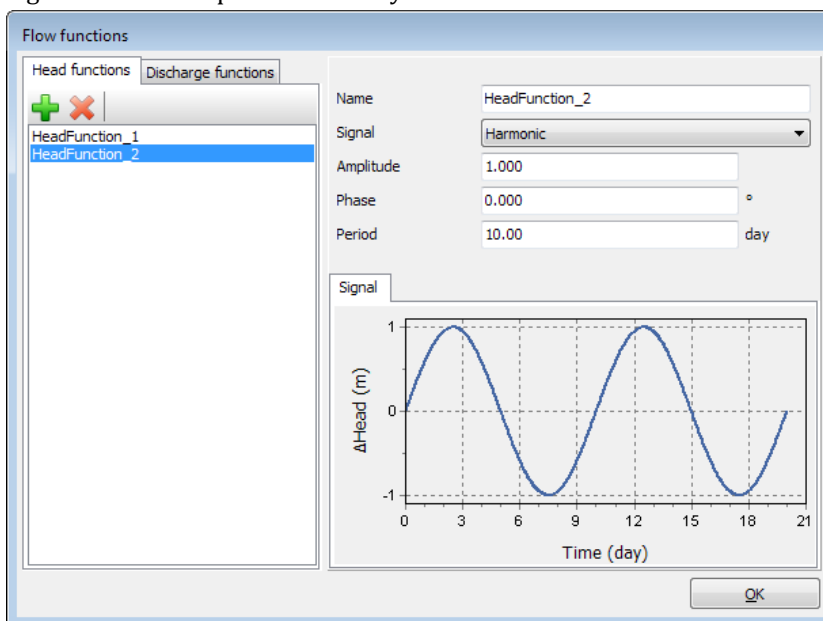

Figure 138: The flow function for the slow case

12. Click **OK** to close the **Flow functions** window.

11.5.4 Execute the calculation

To select points to be considered in curves:

1. In the **Staged construction** mode click the **Select point for curves** button ▽ in the side toolbar. The **Connectivity plot** is displayed in the Output program.
2. In the **Select points** window select nodes located nearest to (0 3) and (8 2.5) to be considered in curves.
3. Click **Update** to close the output program.
4. Click the **Calculate** button to calculate the project.
5. Save the project after the calculation has finished.

11.6 Results

In the Output program the **Create animation** tool can be used to animate the results displayed in the Output program. To create the animation follow these steps:

1. Click the menu **Stresses > Pore pressures > Groundwater head**.
2. Select the menu **File > Create animation**. The corresponding window pops up.
3. Define the name of the animation file and the location where it will be stored. By default the program names it according to the project and stores it in the project folder. In the same way animations can be created to compare the development of pore pressures or flow field.
4. Deselect the Initial phase and Phase 2, such that only Phase 1 is included in the animations and rename the animation accordingly. The **Create animation** window looks like this:
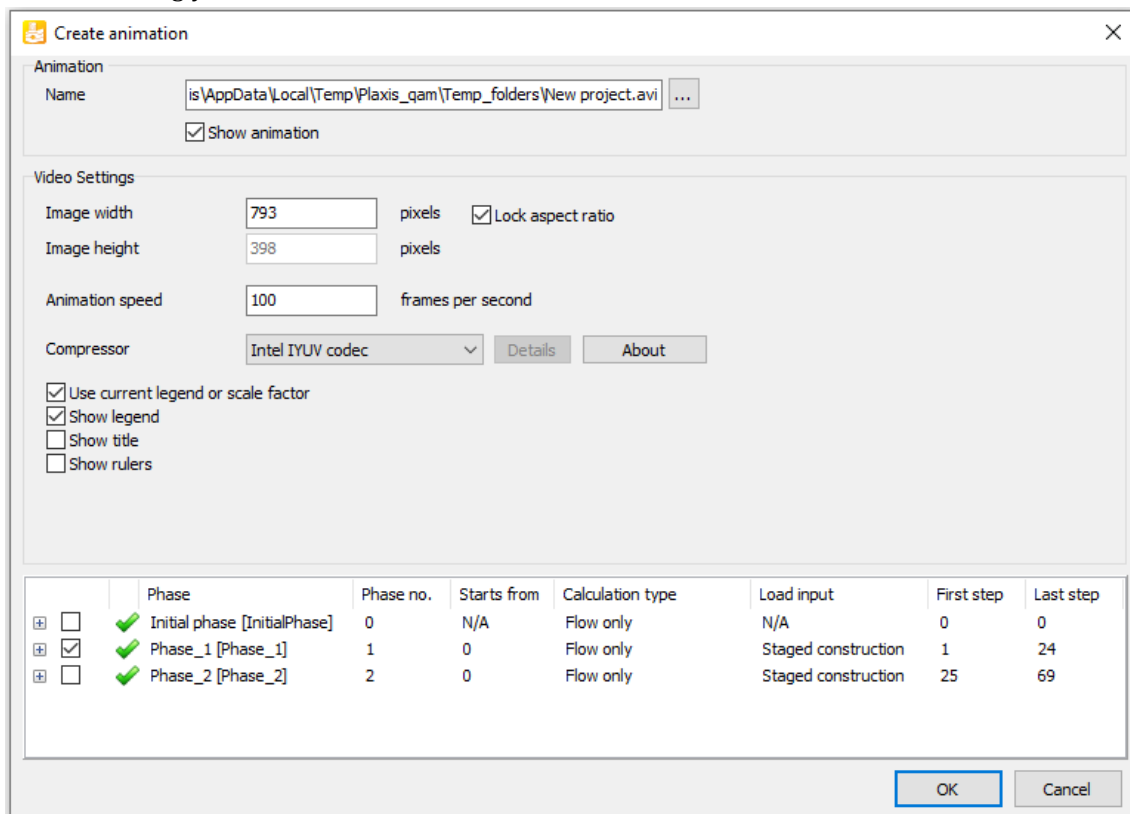

*Figure 139: **Create animation** window*

To view the results in a cross section:

1. Click the **Cross section** button in the side toolbar.

    The **Cross section points** window pops up and the start and the end points of the cross section can be defined.

    Draw a cross section through the points (2.0 3.0) and (20.0 1.0). The results in the cross section are displayed in a new window.

2. In the **Cross section** view select the menu **Stresses** > **Pore pressures** > **p active**.

3. Select the **Cross section curves** option ▲ in the **Tools** menu. After the curves window pops up, select the menu **Selection style** > **Individual steps**.

4. Select Phase 1. The variation of the results in the cross section is displayed in a new window.

5. Do the same for Phase 2. This may take about 30 seconds.

6. The variation of the results due to different time intervals in harmonic variation at a specific cross section can be compared, see the figures below.

It can be seen that the slower variation of the external water level has a more significant influence on the pore pressures in the embankment and over a larger distance.
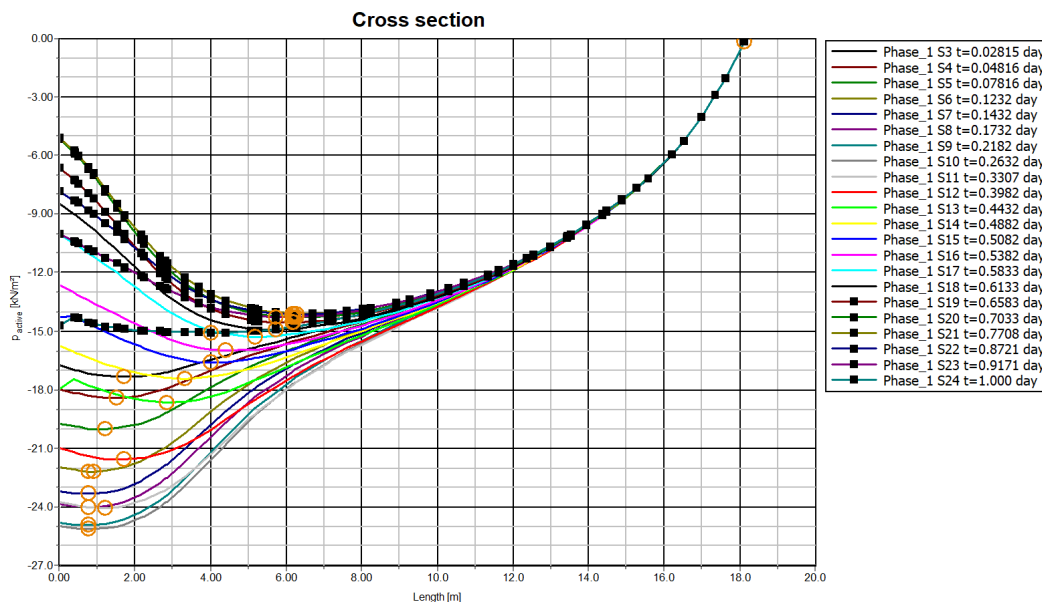

Figure 140: Active pore pressure variation in the cross section in Phase 1
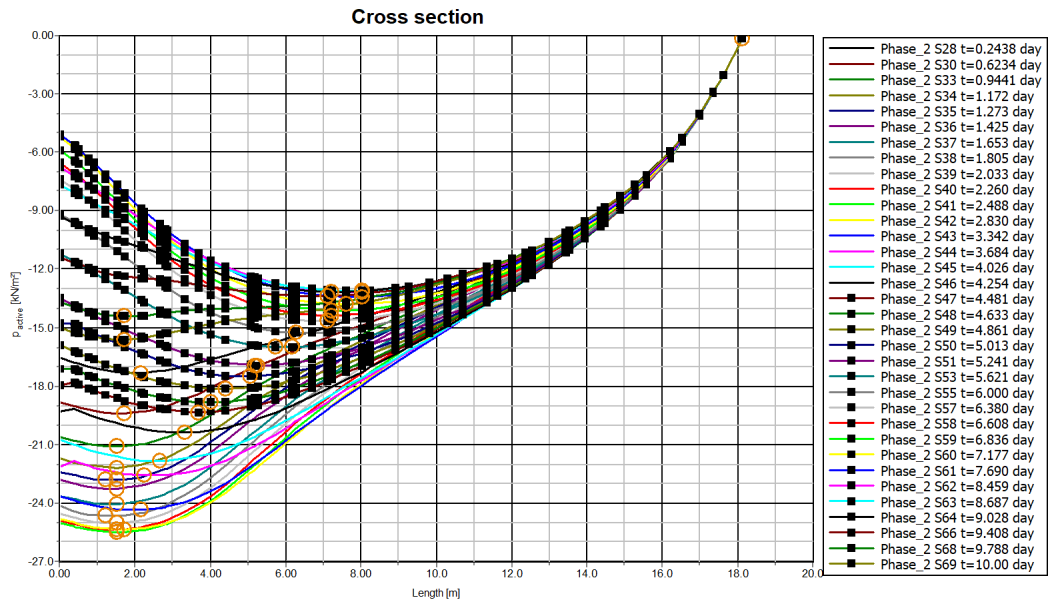
Flow through an embankment [ULT]

Results

Figure 141: Active pore pressure variation in the cross section in Phase 2