9 FLOW THROUGH AN EMBANKMENT

In this chapter the flow through an embankment will be considered. 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 9.1 shows the layout of the embankment problem where free surface groundwater flow occurs. 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.

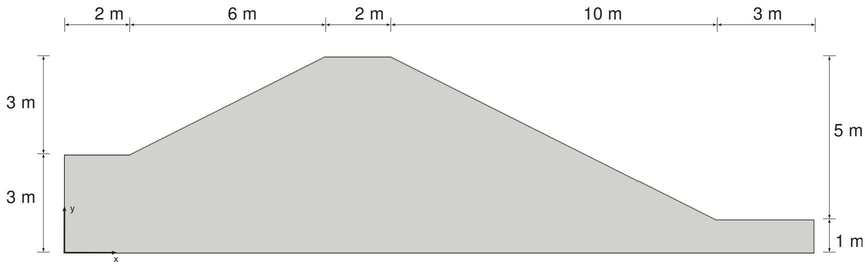

Figure 9.1 Geometry of the embankment

Objectives:

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

9.1 INPUT

To create the geometry model, follow these steps:

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 keep the default options for *Model* (*Plane strain*), and *Elements* (*15-Node*).
- Set the model dimensions to x_{min} = 0.0 m, x_{max} = 23.0 m, y_{min} = 0.0 m and y_{max} = 6.0 m.
- Keep the default values for units, constants and the general parameters and press *OK* to close the *Project properties* window.

Definition of soil stratigraphy

To define the soil stratigraphy:

Create a borehole at x = 2. The *Modify soil layers* window pops up.

- Specify the head value as 4.5.
- Add a soil layer in the borehole. Set the top level to 3. No change is required for the bottom boundary of the layer.
- Create the rest of the required boreholes according to the information given in Table 9.1.

Table 9.1 Information on the boreholes in the model

Borehole number	Location (x)	Head	Top	Bottom
1	2.0	4.5	3.0	0.0
2	8.0	4.5	6.0	0.0
3	10.0	4.0	6.0	0.0
4	20.0	1.0	1.0	0.0

Define the soil material according to the Table 9.2 and assign the material dataset to the cluster. Skip the *Interfaces* and *Initial* tabsheets as these parameters are not relevant.

Table 9.2 Properties of the embankment material (sand)

Parameter	Name	Sand	Unit
General			
Material model	*Model*	Linear elastic	-
Type of material behaviour	*Type*	Drained	-
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			
Stiffness	*E*'	$1.0 \cdot 10^4$	kN/m^2
Poisson's ratio	ν'	0.3	-
Groundwater			
Data set	-	Standard	-
Soil type	-	Medium fine	-
Use defaults	-	From data set	-
Horizontal permeability	k_x	0.02272	m/day
Vertical permeability	k_y	0.02272	m/day

- After assigning the material to the soil cluster close the *Modify soil layers* window.

9.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Select the lines as shown in Figure 9.2 and in *Selection Explorer* specify a *Coarseness factor* of 0.5.

Click the *Generate mesh* button in order to generate the mesh. The *Mesh options* window appears.

- Select the *Fine* option in the *Element distribution* drop-down menu and generate the mesh.

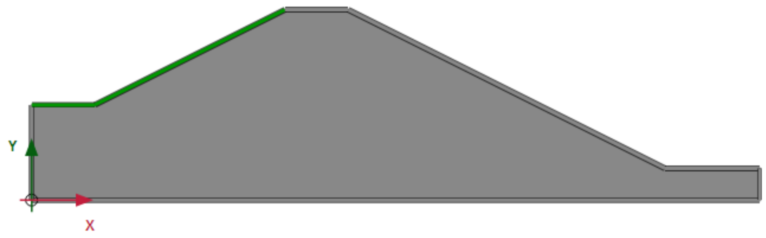

Figure 9.2 Indication of the local refinement of the mesh in the model

 View the mesh. The resulting mesh is displayed in Figure 9.3.

- Click on the *Close* tab to close the Output program.

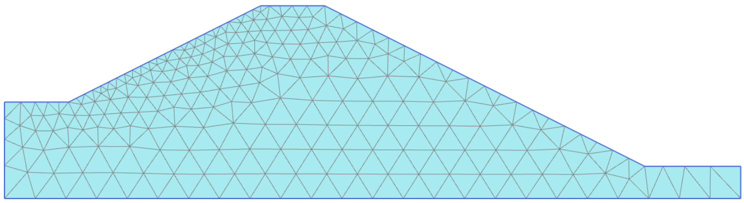

Figure 9.3 The generated mesh

9.3 CALCULATIONS

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 9.1). The model in the *Staged construction* mode is shown in Figure 9.4.

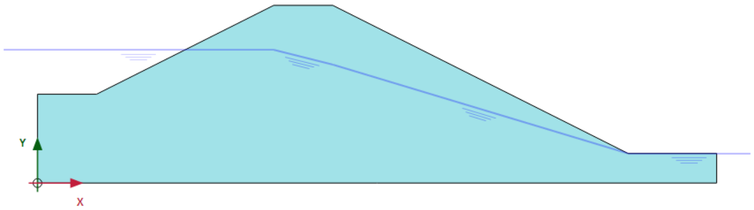

Figure 9.4 The model in the *Staged construction* mode

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

Initial phase

- Double-click the initial phase in the *Phases* explorer.
- In the *General* subtree select the *Flow only* option as the *Calculation type*.
- The default values of the remaining parameters are valid for this phase. Click *OK* to close the *Phases* window.
- In the *Model explorer* expand the *Model conditions* subtree.
- In the *Model conditions* expand the *GroundwaterFlow* subtree. The default boundary conditions (Figure 9.5) are relevant for the initial phase. Check that only the bottom boundary is closed.

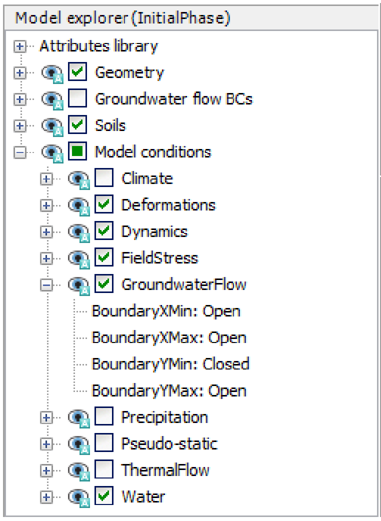

Figure 9.5 The groundwater flow boundary conditions for the initial phase

- 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*.

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

Phase 1

 Add a new calculation phase.

- In the *Phases explorer* double-click the current phase.
- In the *General* subtree select the *Transient groundwater flow* option as pore pressure calculation type.
- Set the *Time interval* to 1.0 day.

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

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

 Point to the *Select lines* option and click on the *Select water boundaries* option in the appearing menu (Figure 9.6).

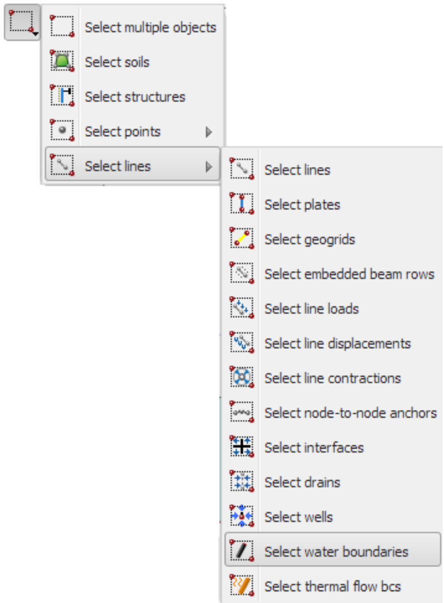

Figure 9.6 The *Select water boundaries* option in the *Select multiple objects* menu

- Select the hydraulic boundaries as shown in Figure 9.7.
- Right-click and select the *Activate* option in the appearing menu.

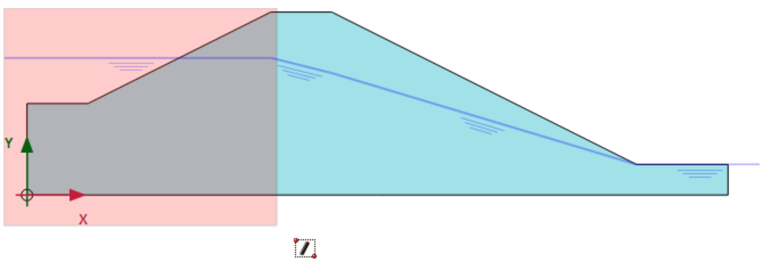

Figure 9.7 The time dependent boundaries in the model

- In the *Selection explorer* set the *Behaviour* parameter to *Head*.
- Set h_{ref} to 4.5.
- Select the *Time dependent* option in the *Time dependency* drop-down menu.
- Click on the *Head function* parameter.

Add a new head function.

- In the *Flow functions* window select the *Harmonic* option in the *Signal* drop-down menu. Set the amplitude to 1.0 m, the phase angle to 0 ° and the period to 1.0 day (Figure 9.8).

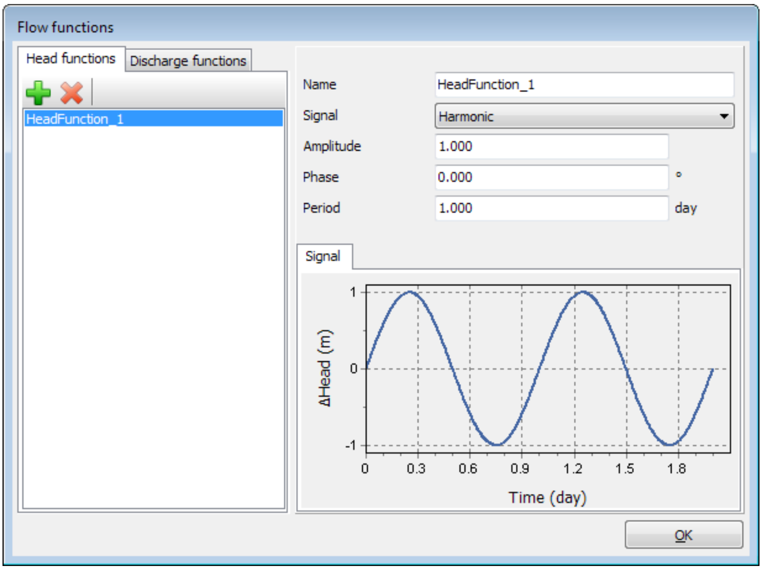

Figure 9.8 The flow function for the rapid case

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

Phase 2

Add a new calculation phase.

- In the *Phases explorer* double-click the current phase.
- In the *General* subtree select the *Initial phase* in the *Start from phase* drop-down menu.
- Select the *Transient groundwater flow* option as pore pressure calculation type.
- Set the *Time interval* to 10.0 day.
- 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.
- Click *OK* to close the *Phases* window.
- In the *Selection explorer* click on the *Head function* parameter.

Add a new head function.

- In the *Flow functions* window select the *Harmonic* option in the *Signal* drop-down menu. Set the amplitude to 1.0 m, the phase angle to 0 ° and the period to 10.0 day (Figure 9.9).
- Click *OK* to close the *Flow functions* window.

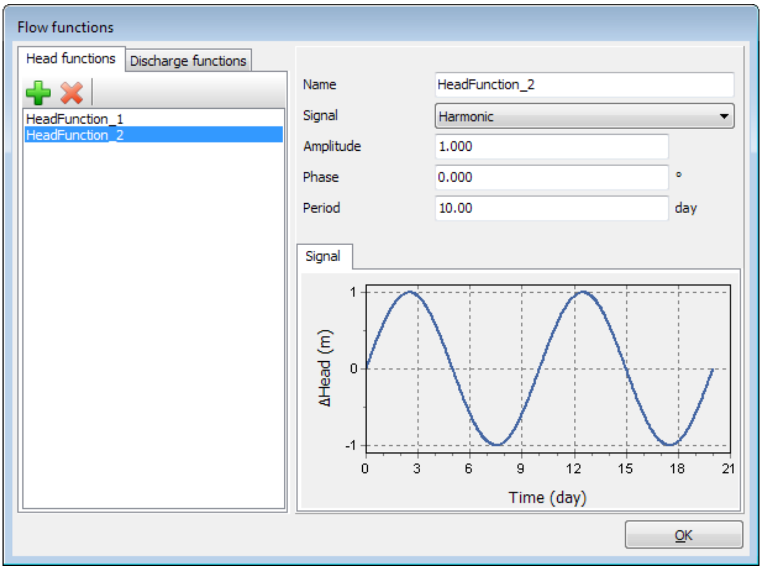

Figure 9.9 The flow function for the slow case

To select points to be considered in curves:

- 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.
- In the *Select points* window select nodes located nearest to (0.0 3.0) and (8.0 2.5) to be considered in curves.
- Click *Update* to close the output program.
- Calculate the project.
- Save the project after the calculation has finished.

9.4 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:

- In the *Stresses* menu select the *Pore pressures* → *Groundwater head*.
- Select the *Create animation* option in the *File* menu. The corresponding window pops up.
- 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.
- 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 is shown in Figure 9.10.

To view the results in a cross section:

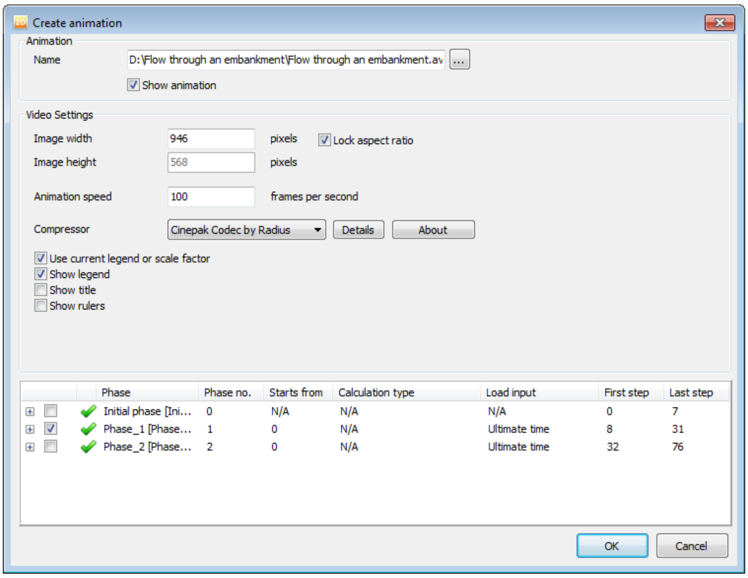

Figure 9.10 *Create animation* window

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.

- In the *Cross section* view select *Pore pressures* → p_{active} in the *Stresses* menu.
- Select the *Cross section curves* option in the *Tools* menu. The *Select steps for curves* window pops up.
- Select Phase 1. The variation of the results in the cross section is displayed in a new window.
- Do the same for Phase 2. This may take about 30 seconds,
- The variation of the results due to different time intervals in harmonic variation at a specific cross section can be compared (Figure 9.11 and Figure 9.12).

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.

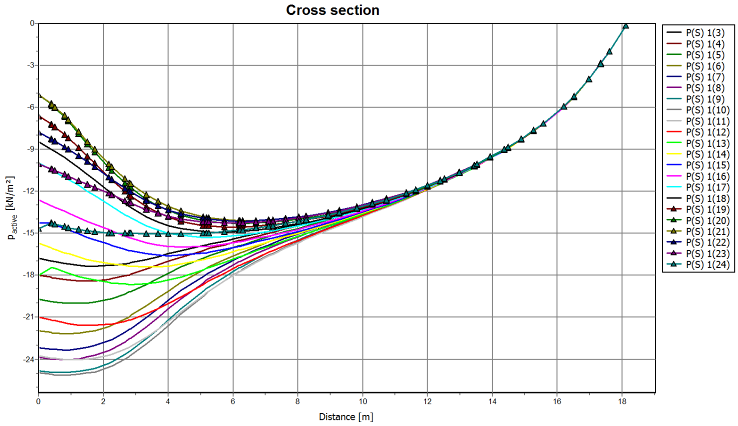

Figure 9.11 Active pore pressure variation in the cross section in Phase 1

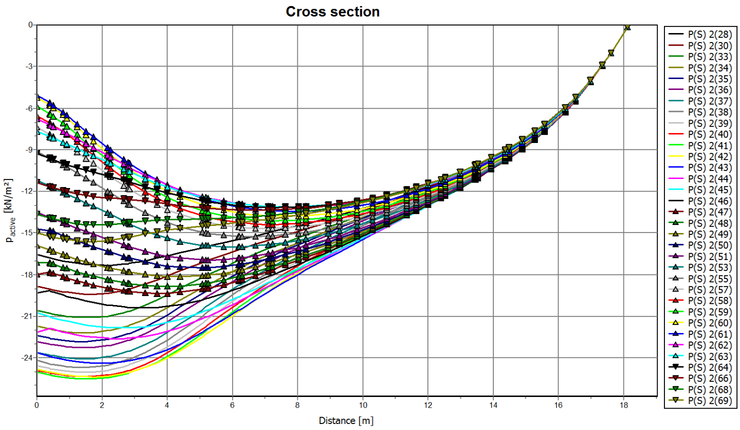

Figure 9.12 Active pore pressure variation in the cross section in Phase 2