Groundwater Flow in Cofferdam

1. Introduction

In this tutorial, finite element groundwater seepage analysis is used to determine the quantity of seepage entering a cofferdam. The example is based on problem 2.4 from Craig (1997). The problem is constructed and solved entirely with Slide2.

The finished product of this tutorial can be found in **File > Recent Folders > Tutorials Folder > *Tutorial 14 Cofferdam Seepage.slmd.***

2. Model
PROJECT SETTINGS

Open the **Project Settings** dialog from the **Analysis** menu and make sure the **General** tab is selected. Define the Units of Measurement as being "Metric" and Time Units as "Seconds."

Click **Groundwater** on the left. Under **Method** choose **Steady State FEA**. This enables steady-state **Finite Element Analysis** of groundwater flow. Close the Project Settings dialog by pressing the OK button.

GEOMETRY

We are going to import the geometry from a *.dxf file.

Select **File > Import > Import DXF**, and open *Tutorial 14 Cofferdam Seepage.dxf.*

Keep the default layer assignments and click **OK**.
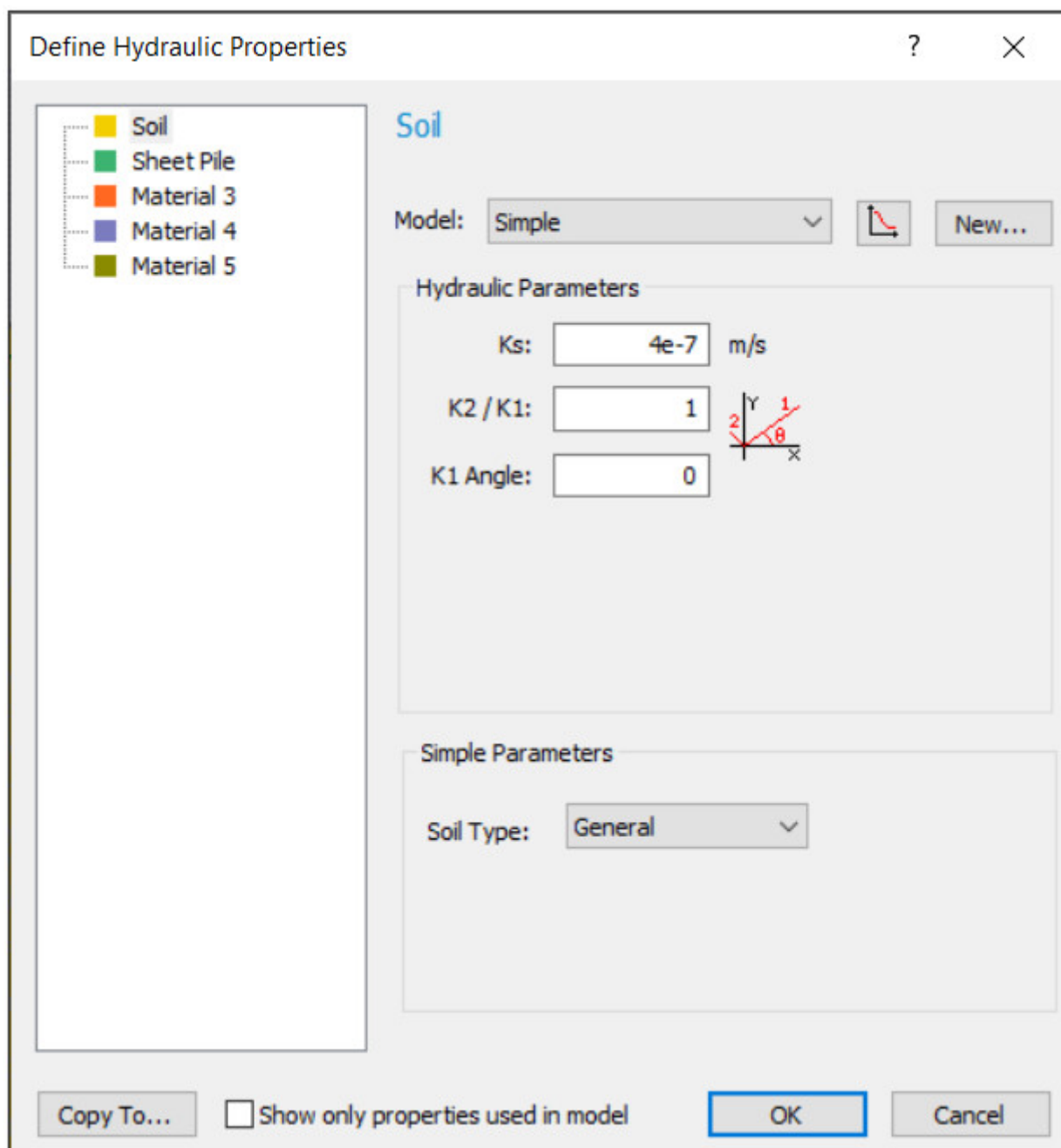
MATERIAL PROPERTIES

Select **Define Materials** from the **Properties** menu.

You will see the default material properties for Material 1. In this tutorial, we don't care about the strength of the solid material therefore leave all the default values. Change the name of Material 1 to Soil. Now click Material 2 on the left. Change the name of Material 2 to Sheet Pile. Click OK to close the dialog.

We now need to define the fluid flow properties of the soil. To do this, we first need to switch to the groundwater mode. Go to the Analysis menu and select Steady State Groundwater Mode. You can also get to groundwater mode using the Groundwater tab at the top, or by clicking on the tab at the bottom of the screen for Steady State Groundwater.

Now go to the **Properties** menu and choose ☑🖼 **Define Hydraulic Properties**. Click on the Soil on the left of the dialog. Enter 4e−7 for Ks. Ks is the saturated permeability in m/s (also called hydraulic conductivity). You may specify anisotropic permeability by specifying K2/K1 ≠ 1 and an angle to indicate the directionality. However we will assume isotropic permeability so do not change the default values.

The **Model** option at the top of the dialog refers to the function used to calculate the permeability in the unsaturated zone as a function of matric suction. Different models may be chosen, including a user-defined model. However, we will use the default Simple option. See Slide2 Help for more information on permeability models. Your dialog should now look like this.

Now select the Sheet Pile material. The sheet piling is assumed to be essentially impermeable. We wish to set the permeability to a very low value, however, we cannot choose 0 since this will lead to numerical instability. Therefore set the permeability, Ks, to 1e−20. Click OK to close the window.

ASSIGN MATERIAL PROPERTIES

By default, the entire model is assigned the properties of Soil (material 1). We wish to assign the Sheet Pile properties to the sheet pilings. From the Properties menu, select Assign Properties. Select Sheet Pile from the Assign dialog and click inside the two narrow sections representing the two sheet pilings (zoom in if necessary). Close the dialog.

MESH

Now generate the finite element mesh. Select the Mesh Setup option in the Mesh menu. Leave the default element type (3 Noded Triangles) and the number of elements (1500). Click the Discretize button followed by the Mesh button.
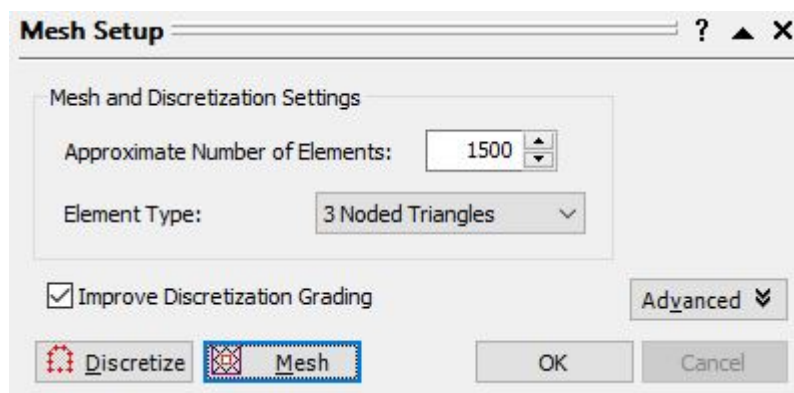

Close the Mesh Setup dialog by selecting the OK button. Your model should now appear as shown.
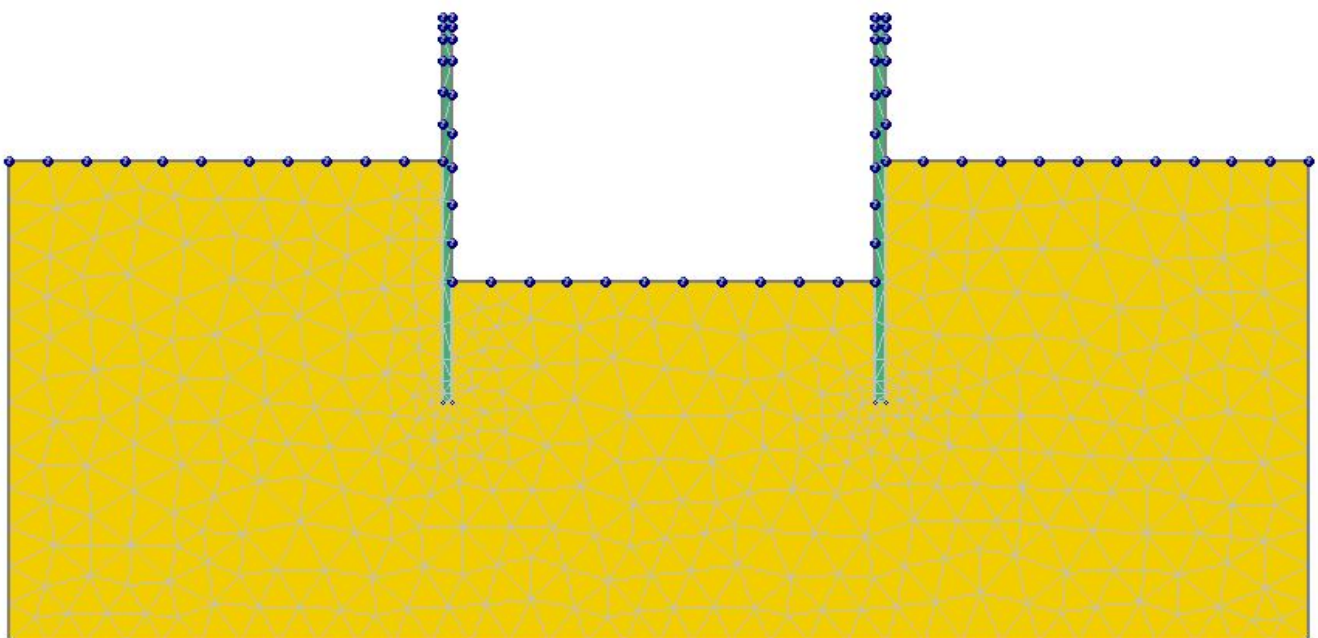
The model after meshing

BOUNDARY CONDITIONS

The model shows the default boundary conditions (no flow on the external boundaries and unknown conditions at the surface). We wish to simulate ponded water to the left and right of the sheet piling. The elevation of the top of the sheet piling is 13 m. Therefore we will set the total head for these boundaries to 13 m. To do this, choose **Set Boundary Conditions** from the **Mesh** menu. For BC Type choose Total Head. Enter a Total Head Value of 13.
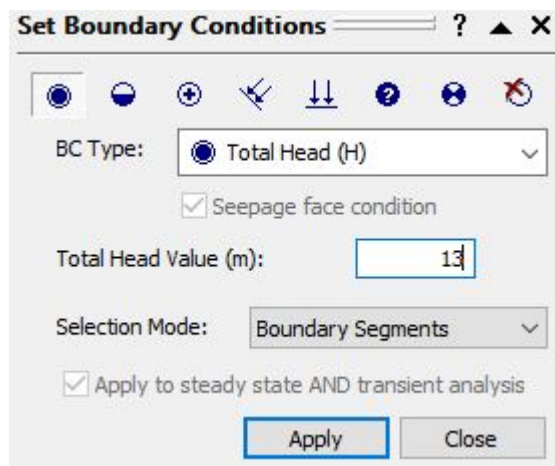
Now select the four boundary segments that enclose the ponded water:

Line 1: from (0,10) to (9,10)

Line 2: from (9,10) to (9,13)

Line 3: from (18.2,10) to (18.2,13)

Line 4: from (18.2,10) to 27,10)

Click **Apply**.

The soil surface inside the coffer dam has zero pore pressure (it is at atmospheric pressure). Therefore we need to set the pressure on this surface to zero. In the Set Boundary Condition Dialog, choose Zero Pressure for the BC Type. Click on the ground surface between the pilings and hit Enter (or right-click and choose Assign). Now close the dialog box. Your model will appear as shown.
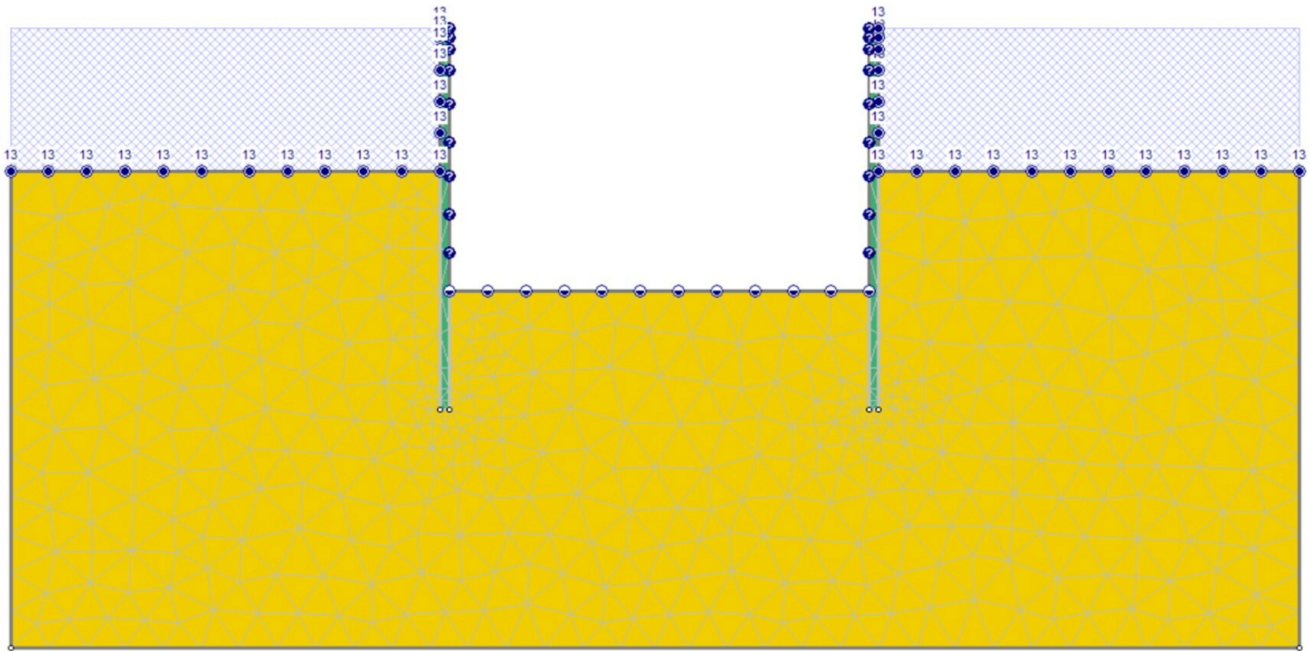
TIP: You can also right-click on a boundary to define its boundary conditions.

DISCHARGE SECTIONS

If we wish to calculate flow quantities, this is done by defining a **Discharge Section**. A Discharge Section in Slide2 is a user-defined line segment, through which the steady-state, volumetric flow rate, normal to the discharge section, will be calculated during a groundwater seepage analysis.

We wish to add horizontal discharge sections at the soil surface. To do this, choose **Add Section** from the **Discharge** menu. Enter a start point on the left boundary just below the ponded water (0,9.5). Add a finish point on the left edge of the left sheet piling just below the ponded water (9,9.5). Hit Enter to finish entering points. You can enter the coordinates using the keyboard but it is easier to just click on the model since the cursor will snap to the boundaries (if the cursor does not snap to the boundaries go to the View menu, select Snap and ensure all of the options are selected).

The discharge section is displayed as a green line, with small circles marking the endpoints as shown. The value of the flow rate across this line will be displayed in the Slide2 Interpret program when you view the analysis results.
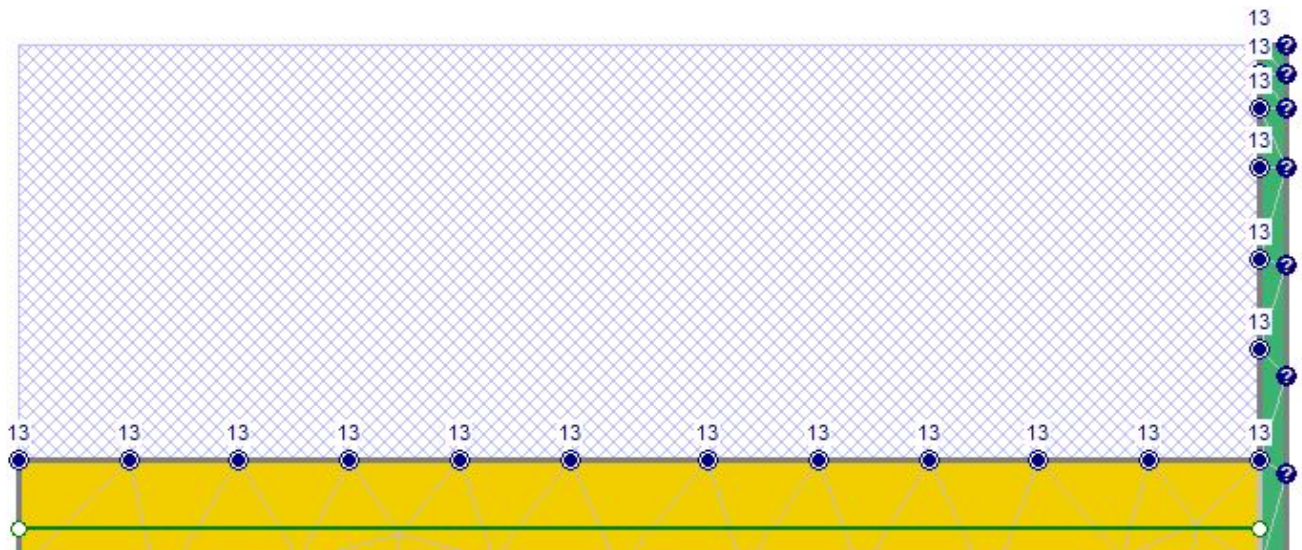
TIP: You can delete a discharge section by right-clicking on it and choosing Delete Discharge Section.

Add two more sections: one below the soil surface on the other side under the ponded water (9.2,7) (18,7) and one below the soil surface between the sheet pilings (18.2,9.5) (27,9.5).

Your final model should now appear as shown.
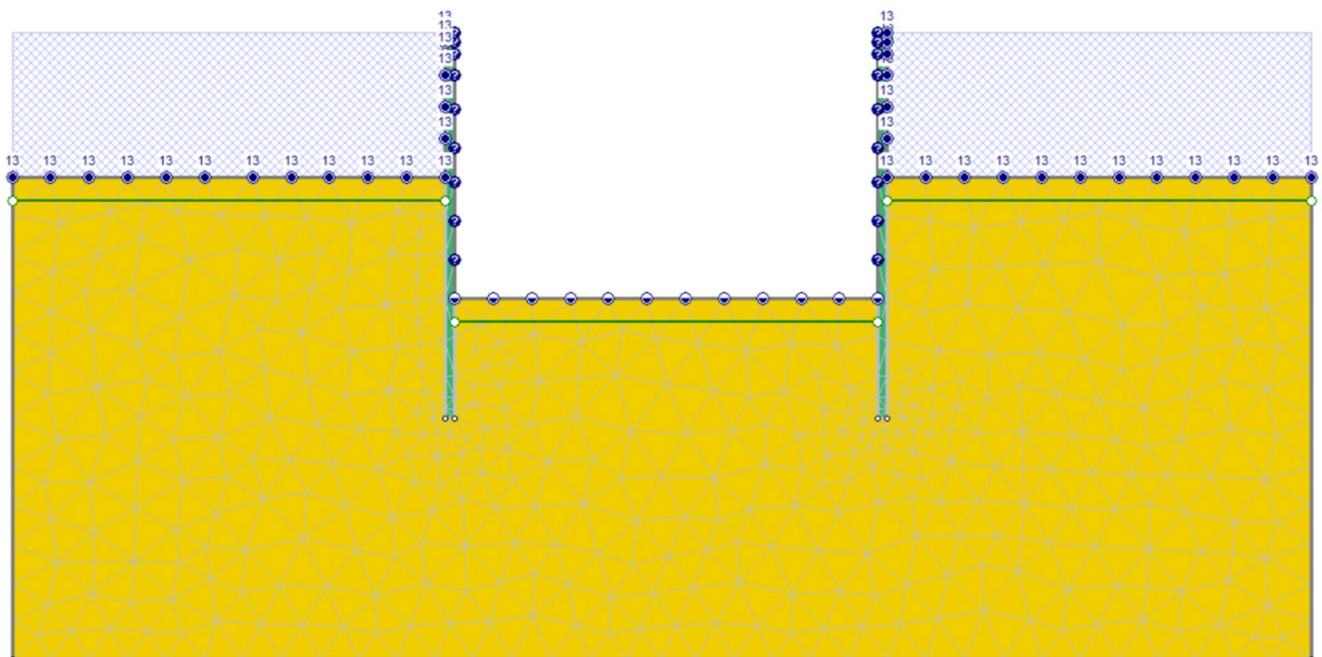
You have completed the definition of the model. Save the model.

3. Compute

Since we are only interested in the groundwater results, we only need to run the groundwater computation. Select **Compute (groundwater)** from the **Analysis** menu (or click the Compute groundwater button in the Groundwater toolbar). The analysis should take a few seconds to run.

Once the model has finished computing (Compute dialog closes), select the **Interpret (groundwater)** option in the Analysis menu to view the results.

4. Interpret

After you select the Interpret option, the Interpret program starts and reads the results of the analysis. The following screen is displayed showing the Pressure Head contours.
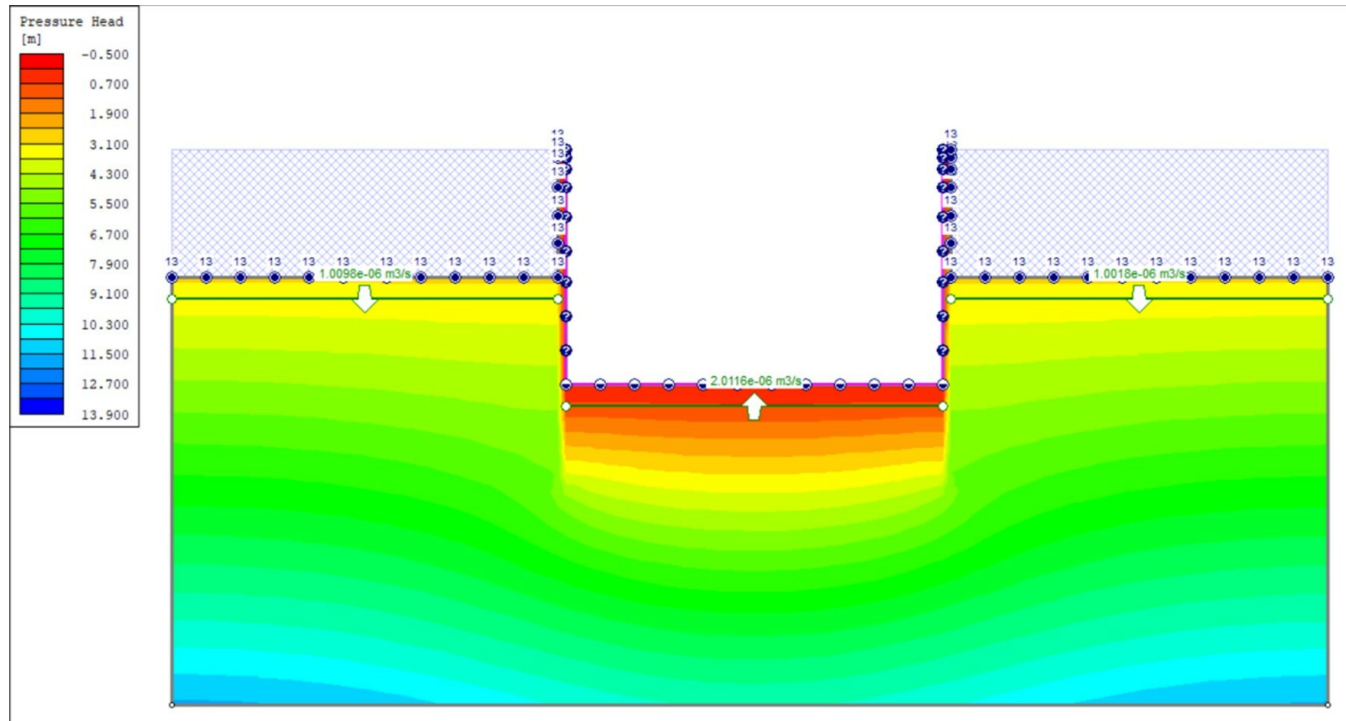
You can also see the volumetric flow rate and direction through each of the discharge sections. As you would expect, the water is flowing down from the ponded water and up into the dam. The sum of the volumetric downwards flow is equal to the volumetric upwards flow between the sheet pilings.

To see the magnitude and direction of flow throughout the model, plot the Flow Vectors by clicking the Flow Vectors button. It is clear that the groundwater is flowing around the impermeable sheet pilings with high flow rates directly below the pilings.
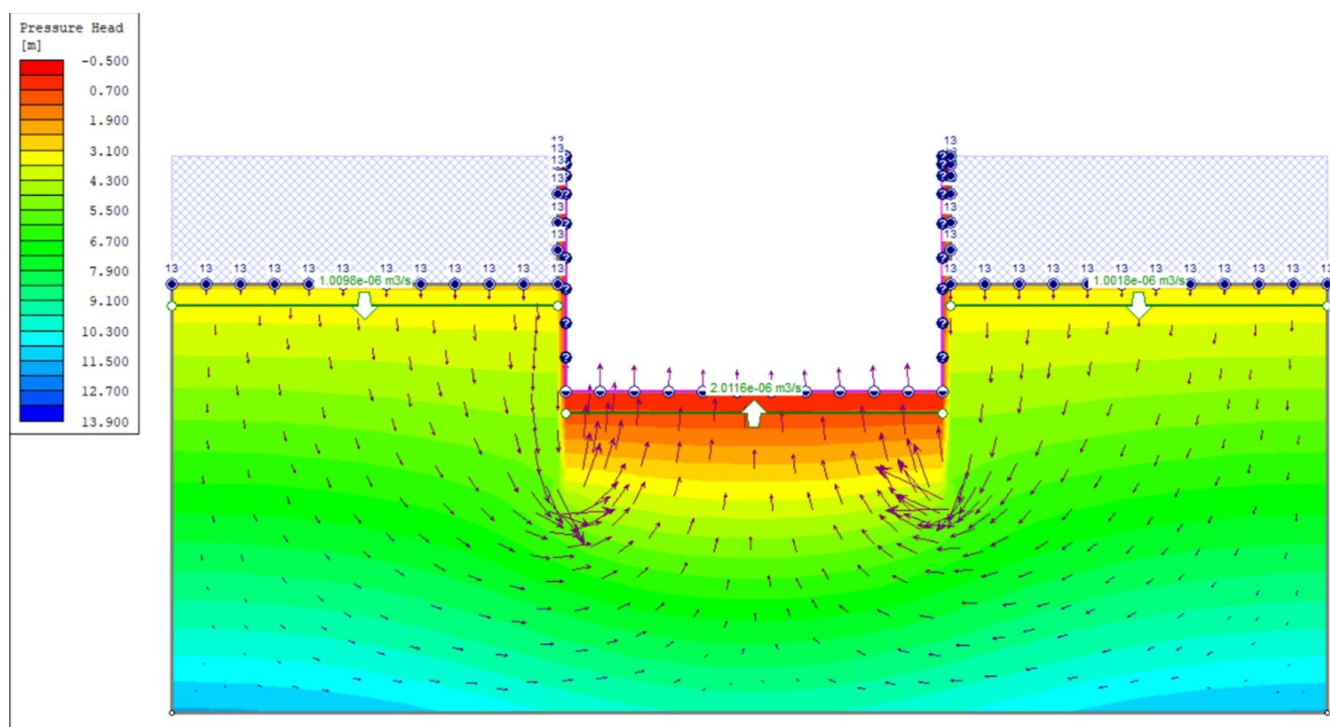
The geometry of this model corresponds to Problem 2.4 in Craig (1997). This problem asks for the quantity of seepage entering the cofferdam. From the figure above, the volumetric flow into the dam is 2.0116e-6 m3 /s. The value given in Craig (1997) is 2.0e-6 m3 /s. The model, therefore, gives the same result within the number of significant digits given. Your result may differ slightly depending on the exact position of the discharge line.

The problem also asks for a flow net to be constructed. This can also be done with Slide2, as follows. First, turn off the flow vectors by pressing the Flow Vector button again. Now change the contour data being plotted from Pressure Head to Total Head using the drop-down menu on the toolbar. Now right-click on the model and select Contour Options. Under Mode select **Filled (with lines)** and then select Done. You will now see the equipotential lines of the flownet.

To plot the flow lines, go the **Groundwater** menu and from the **Lines** sub-menu select **Add Multiple Flow Lines**. Select the top left corner of the soil as the first point (you may need to move the legend or the model prior to this). If the cursor does not snap to the node point go to the View menu, select Snap and ensure that all snap options are turned on. Now move horizontally until you intersect the sheet piling and click to establish the second point. Hit enter to finish. You will now see the Flow Line Options dialog. Here you can choose how many flow lines you wish to plot. Under Start Flow-Lines select the first option of locations evenly spaced along the polyline and leave the default value (10 locations).

Click **OK** to close the dialog. You will now see 10 flow lines plotted as shown. To complete the flownet you could repeat these steps for the right side of the model.
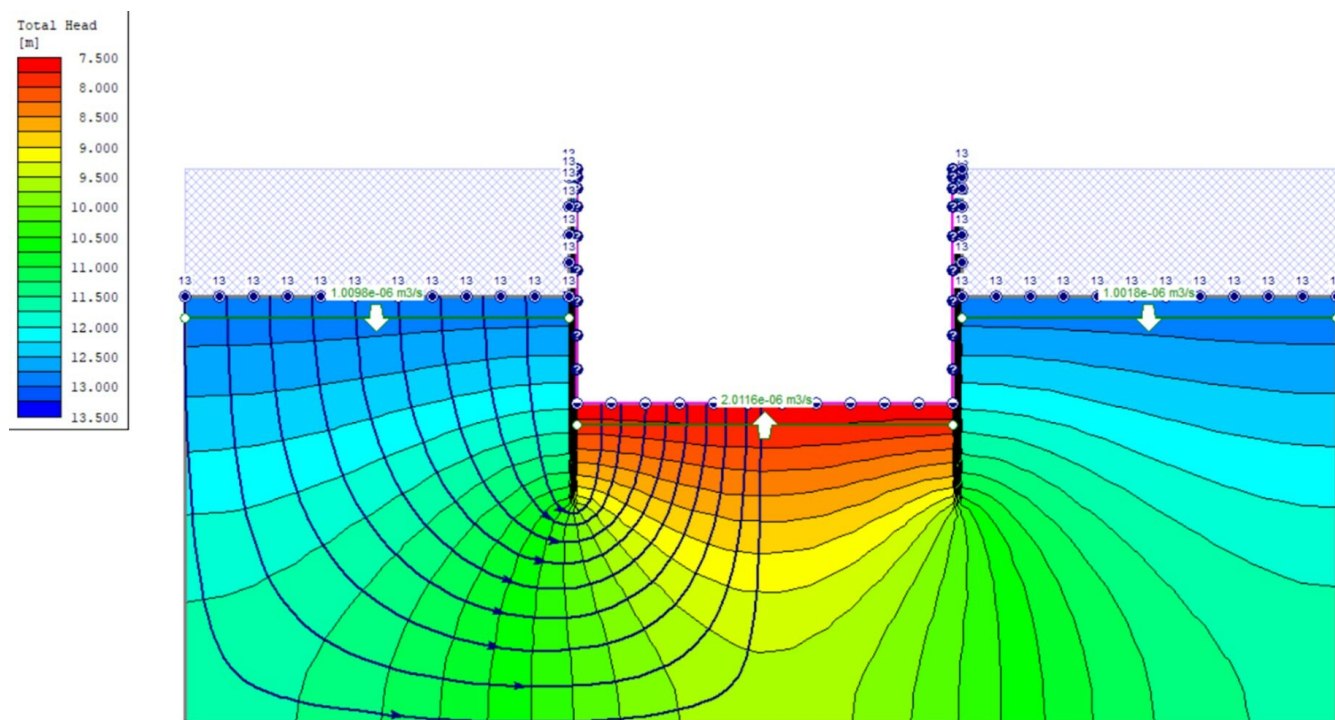

5. Additional Exercises

We can simulate pumping at the bottom of the dam by setting a value for total head less than the elevation of the surface. In the Slide2 Model program, change the boundary conditions for the bottom of the dam from zero pressure to Total Head = 7 m. Recalculate and plot the results with the Interpret program.
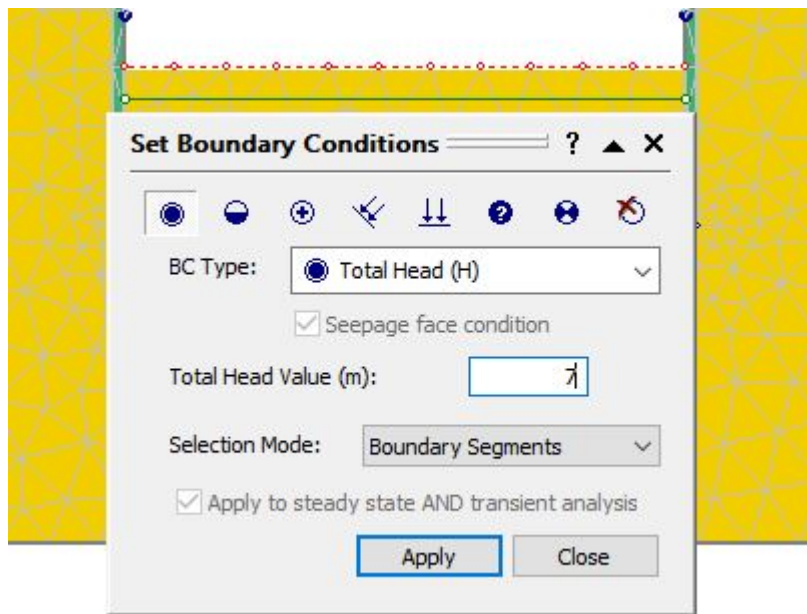
You can see that the volumetric discharge at the bottom of the dam is higher than before. You can also see that the water table has been lowered. The water table is shown as a pink line (your water table line may be obscured by the green discharge line. To hide the discharge line, right-click on it and choose Hide Discharge Sections).
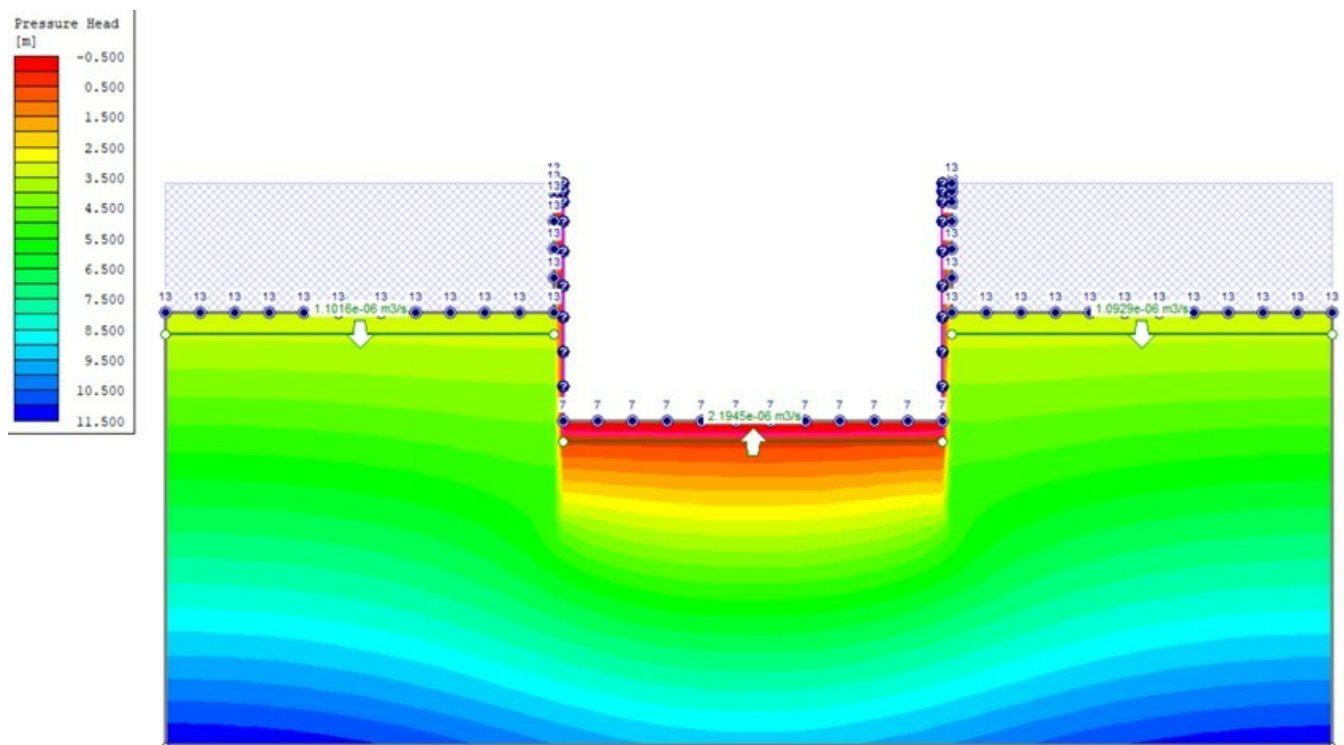
6. References

Craig, R.F., 1997. Soil Mechanics, Spon Press, London and New York, 485 pp.