Transient Groundwater Analysis

1. Introduction

A transient groundwater analysis may be important when there is a time-dependent change in pore pressure. This will occur when groundwater boundary conditions change and the permeability of the material is low. In this case, it will take a finite amount of time to reach steady-state flow conditions. The transient pore pressures may have a large effect on slope stability.

This tutorial describes how to perform a transient groundwater analysis in **Slide2** using finite elements. A subsequent tutorial will describe how this affects slope stability calculations.

The finished product of this tutorial can be found by opening *Tutorial 18 Transient Groundwater.slmd* from **File > Recent Folders > Tutorials Folder**.

2. Model

To start the tutorial go to **File > Recent Folders > Tutorials Folder** and open the *Tutorial 18 Transient Groundwater initial.slmd* file. This file contains the geometry.

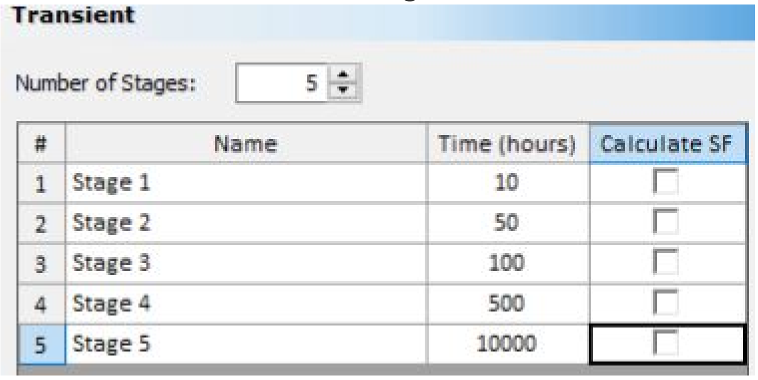
2.1 PROJECT SETTINGS

1. Open the **Project Settings** dialog from the **Analysis** menu.
2. Set the **Stress Units** to **Metric**, set the **Time Units** to **Hours** and the **Permeability Units** to **meters/second**.
3. Click on the **Groundwater** link on the left side.
4. Leave the **Method** as **Water Surfaces**.
5. Select the **Advanced** check box and choose **Transient Groundwater**.

   The **Method** refers to the method used to obtain the initial state for the transient groundwater analysis. In this tutorial, we will simply specify an initial water table but it is also possible to specify a grid of pore pressures or even to perform a steady-state finite element analysis to get the initial state. This is discussed further in subsequent tutorials.

6. Click on the **Transient** link on the left.
   Here we need to specify the times at which we wish to observe pore pressure results.
7. Change the **Number of Stages** to **5**.

8. Enter the times for each stage as shown in the table and dialog below.


Transient

Number of Stages: 5

#	Name	Time (hours)	Calculate SF
1	Stage 1	10	☐
2	Stage 2	50	☐
3	Stage 3	100	☐
4	Stage 4	500	☐
5	Stage 5	10000	☐

Name	Time (hours)
Stage 1	10
Stage 2	50
Stage 3	100
Stage 4	500
Stage 5	10000

9. Do not check the boxes to **Calculate SF** (safety factor). In this tutorial, we will only look at the groundwater modelling. A subsequent tutorial will combine this with slope stability calculation.
10. Click **OK** to close the **Project Settings** dialog.

You will now see two tabs at the bottom – one for **Slope Stability** and one for **Transient Groundwater**.

2.2 INITIAL WATER TABLE

In the Slope Stability view, you can specify the initial water table that exists prior to the transient groundwater analysis. For this tutorial we will assume that the initial water table is at the base of the dam, so we don't need to specify it. If no water table is specified then all points will have an initial pore pressure of 0.

3. Groundwater

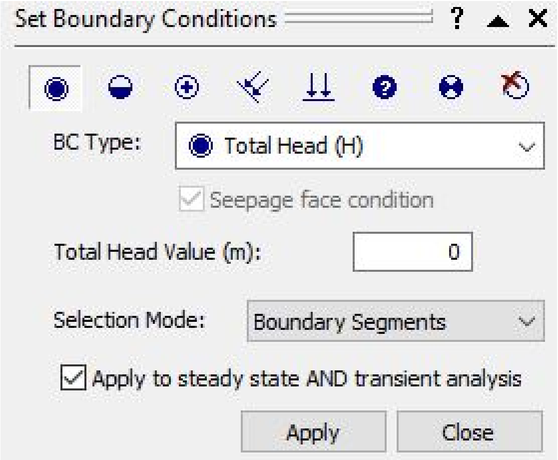
Now it is time to set up the finite element model for the calculation of transient groundwater behaviour, so

1. Select the tab for **Transient Groundwater** at the bottom of the screen.
   Before we can set up the boundary conditions we need to create a finite element mesh.
2. Select **Mesh > Discretize and Mesh**.

3.1 BOUNDARY CONDITIONS

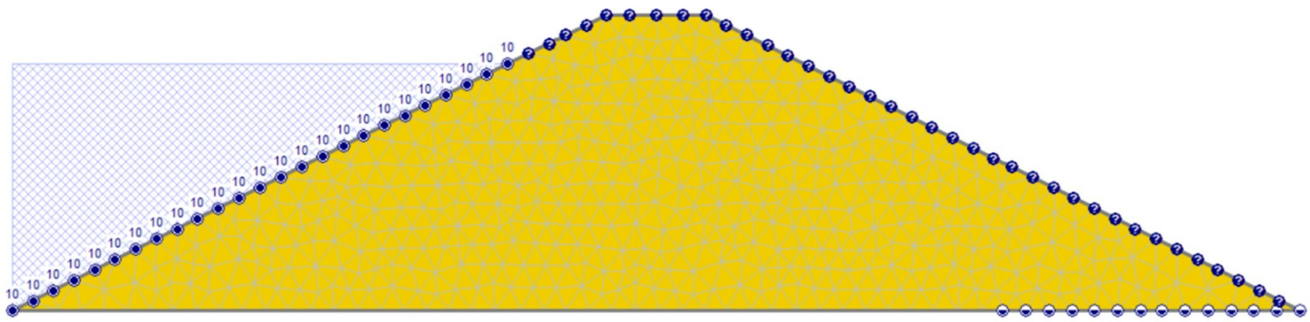
We will set up boundary conditions to simulate rapid filling of the dam on the left side and drainage from the toe drain at the bottom right.

1. Select **Mesh → Set Boundary Conditions**.
2. For **BC Type** choose **Total Head** and set the **Total Head Value** to **10 m**.



3. Click on the left side of the slope near the bottom and click **Apply**.
4. Now for **BC Type** choose **Zero Pressure**.
5. Click on the horizontal section at the bottom right of the model (the toe drain).
6. Click **Apply**.
7. Click **Close**.

The model should look like this:



3.2 MATERIAL PROPERTIES (GROUNDWATER)

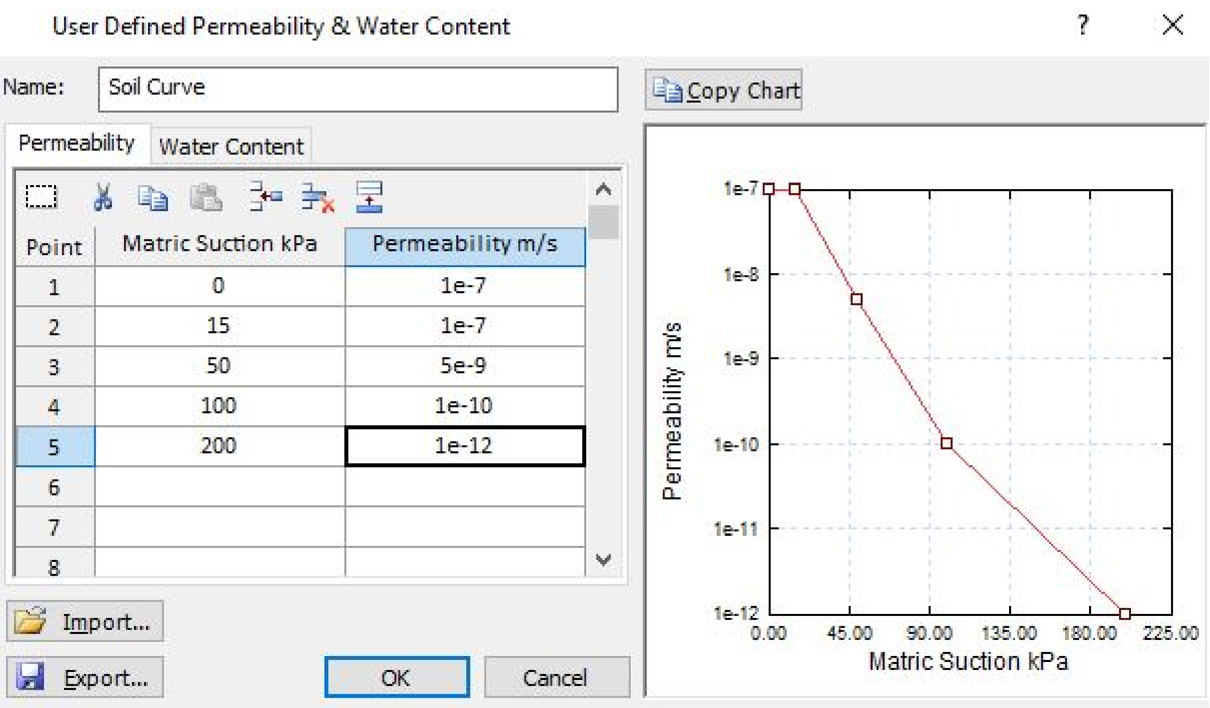
- Select **Properties > Define Hydraulic Properties**.

The hydraulic properties required for a transient analysis are the same as those for a steady-state analysis except that a water content curve (WC) must now be specified. Here you can choose from a number of possible curves that relate permeability and water content to negative pore pressure (suction). For this tutorial, we will define our own relationship.

1. Click the **New** button to define a new function.
2. Change the **Name** to **Soil Curve**.

3. Ensure that the **Permeability** tab is selected and enter the values as shown.

Point	Metric Suction (kPa)	Permeability (m/s)
1	0	1e-7
2	15	1e-7
3	50	5e-9
4	100	1e-10
5	200	1e-12



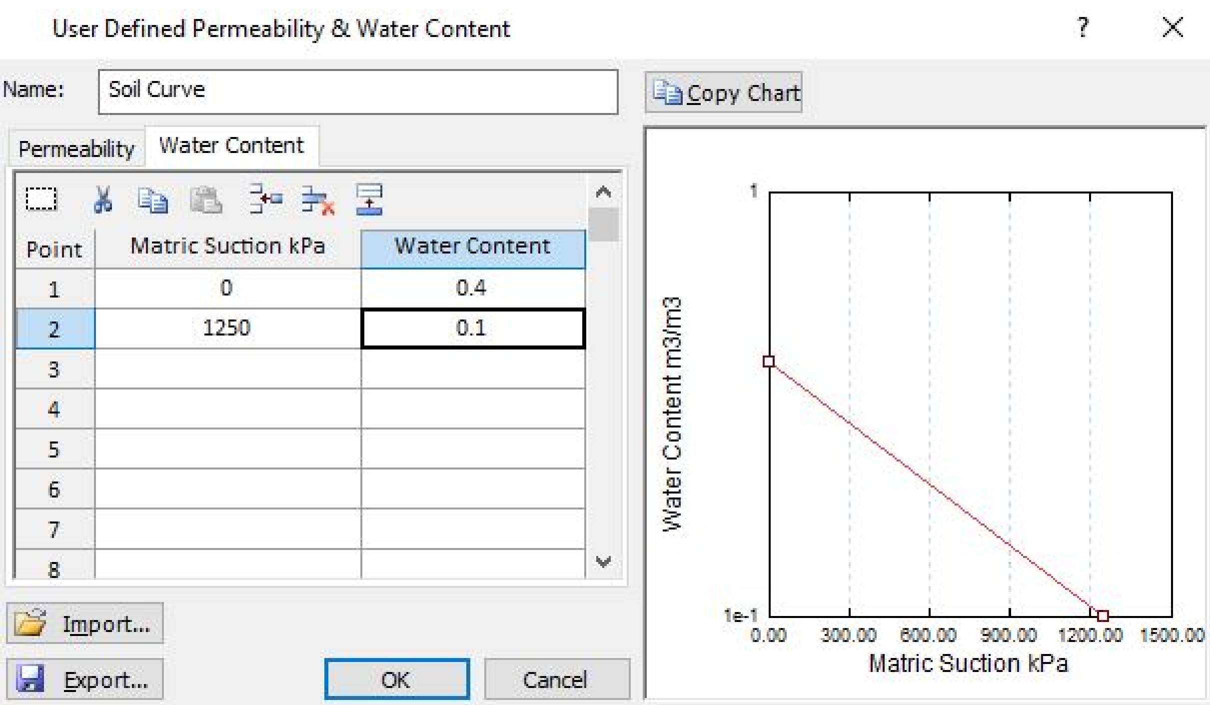
You can see the permeability will be 1e-7 m/s until the suction exceeds 15 kPa, at which time the permeability decreases with increasing suction.

We now need to define the relationship between water content and suction. The volumetric water content (θ) is the volume of water as a proportion of the total volume (θ = Vw/VT). You can also think of it as the porosity times the degree of saturation (θ = φSw) where the degree of saturation ranges from 0 (dry) to 1 (saturated).

1. Click the **Water Content** tab.

2. Enter the values as shown.

Point	Metric Suction (kPa)	Water Content
1	0	0.4
2	1250	0.1

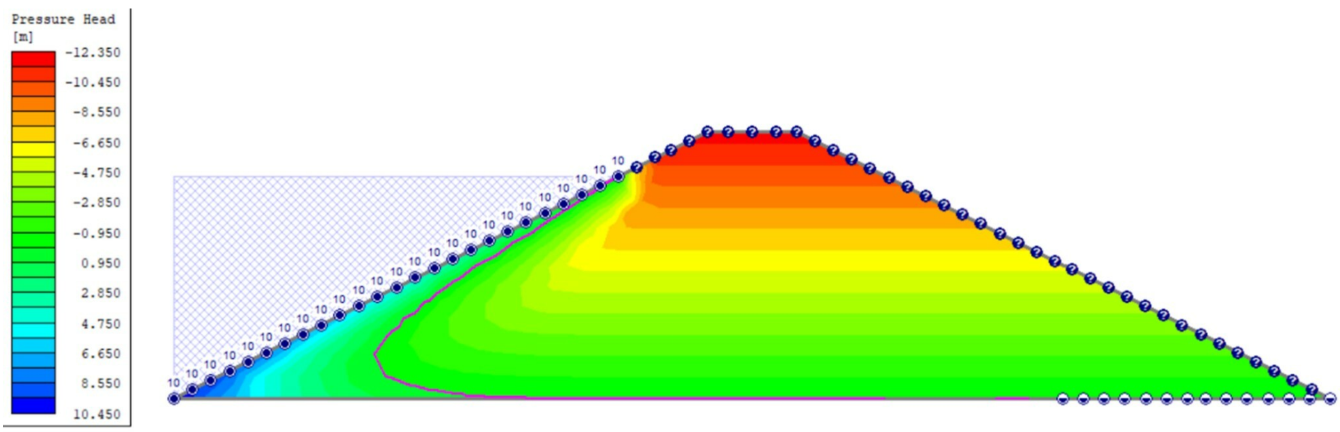


You can see that at 0 matric suction, the water content is 0.4. If we assume the material is 100% saturated at this pressure, this suggests a porosity of 0.4. As the matric suction increases, the water content decreases, suggesting a decrease in saturation.

1. Click **OK** to close the dialog.
2. In the **Define Hydraulic Properties** dialog, ensure that **Soil Curve** is the chosen **Model**. Leave all other values as default.
3. Click **OK** to close the dialog.
4. Save the file and select **Analysis > Compute (groundwater)**

3.3 INTERPRET

You will first see the initial state. All pore pressures are 0. Click on the tab for Stage 1 (10 hours). You will now see the pressure head for Stage 1 (10 hours).

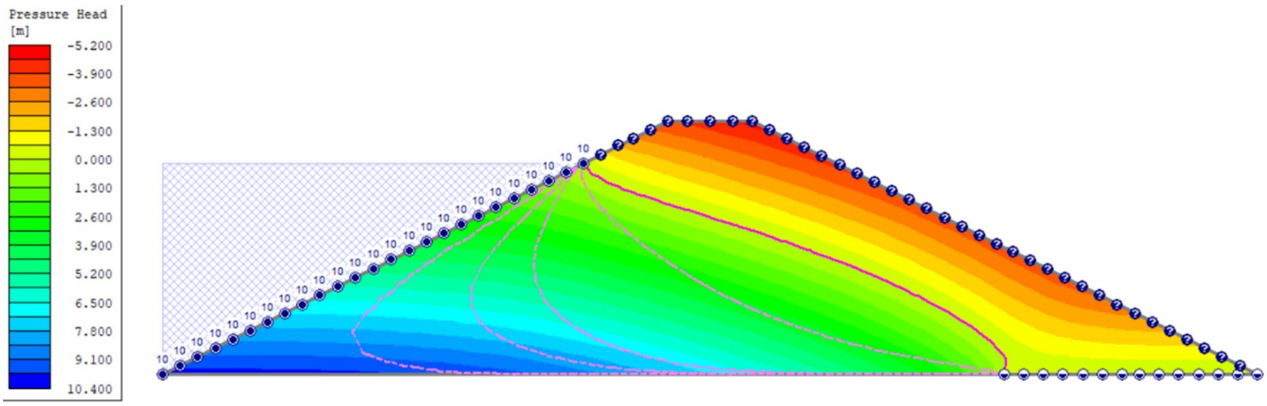


Pressure head at 10 hours

You can see how the rapid rise in water level at the left edge has induced high pore pressures along the left flank but the water has not yet flowed through the dam to increase pressures elsewhere.

1. Click through the other stages using the tabs at the bottom.
   You will see the progression of the groundwater with time and in particular the changing of the water table (pink line).
2. Click on the tab for **Stage 5 (10000 hours)**.
   This essentially represents the steady-state. You can show the progression of the water table with time by going to **View > Display Options**.
3. Select the **Groundwater** tab and, under **FEA water**, select **All Stages**.
4. Click **Done**.

   The plot will now look like this:

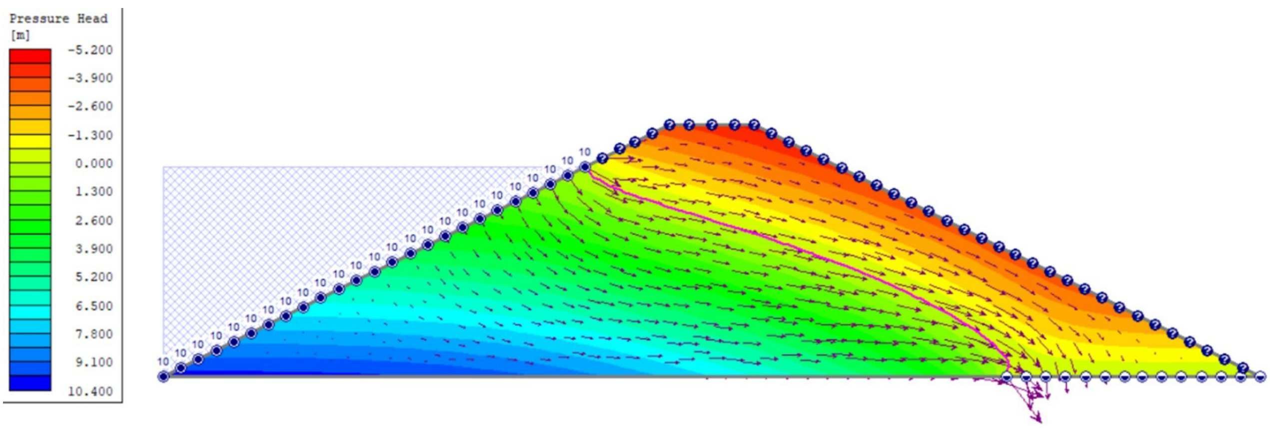


Progression of the water table with time

You can see that the solid pink line represents the water table at 10,000 hours and the dashed pink lines represent the water table at other stages.

5. Go back to the **Display Options** and turn off the water tables.
6. Show the flow vectors by clicking on the **Show Flow Vectors** button in the toolbar.

   The plot will look like this.

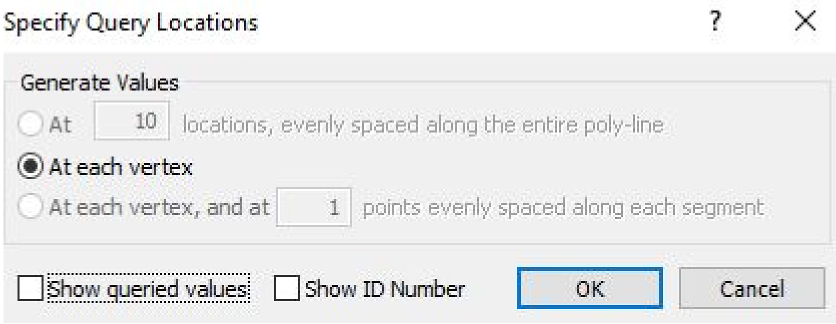


Flow vectors after 10000 hours

You can see the flow of water towards the toe drain on the bottom right. If you click through the stage tabs again, you will see the progression of flow through the dam with time.

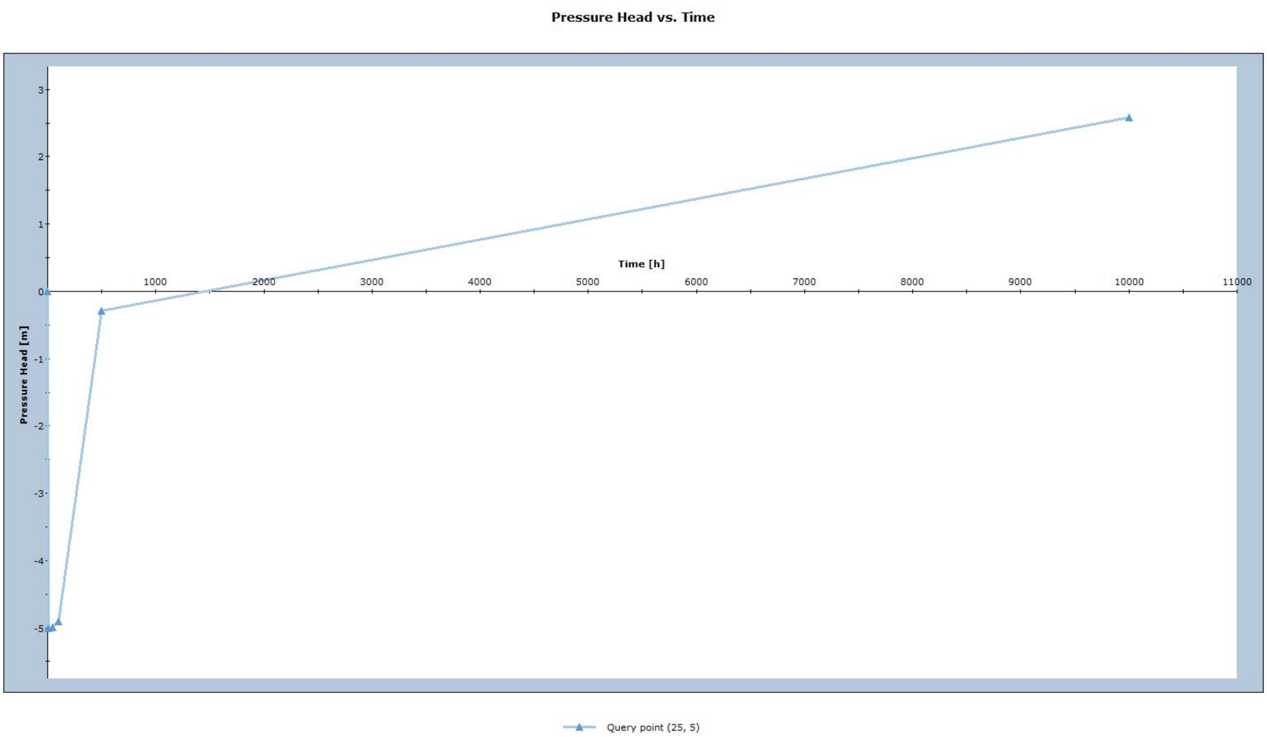
We can look at the change in pressure with time in more detail by using a query.

7. Go to **Groundwater > Query > Add Material Query**.
8. We want to examine a single point near the middle of the dam, so enter the coordinates **(25, 5)**
9. Hit Enter to finish entering points.
10. Uncheck **Show Queried Values** as shown.



11. Click **OK**.

You will now see **a +** near the middle of the dam, indicating the query point (you may need to turn off the flow vectors to see it). Right-click on the + and select **Graph Data vs. Time**. You will now see a plot of pressure head versus time at that location.

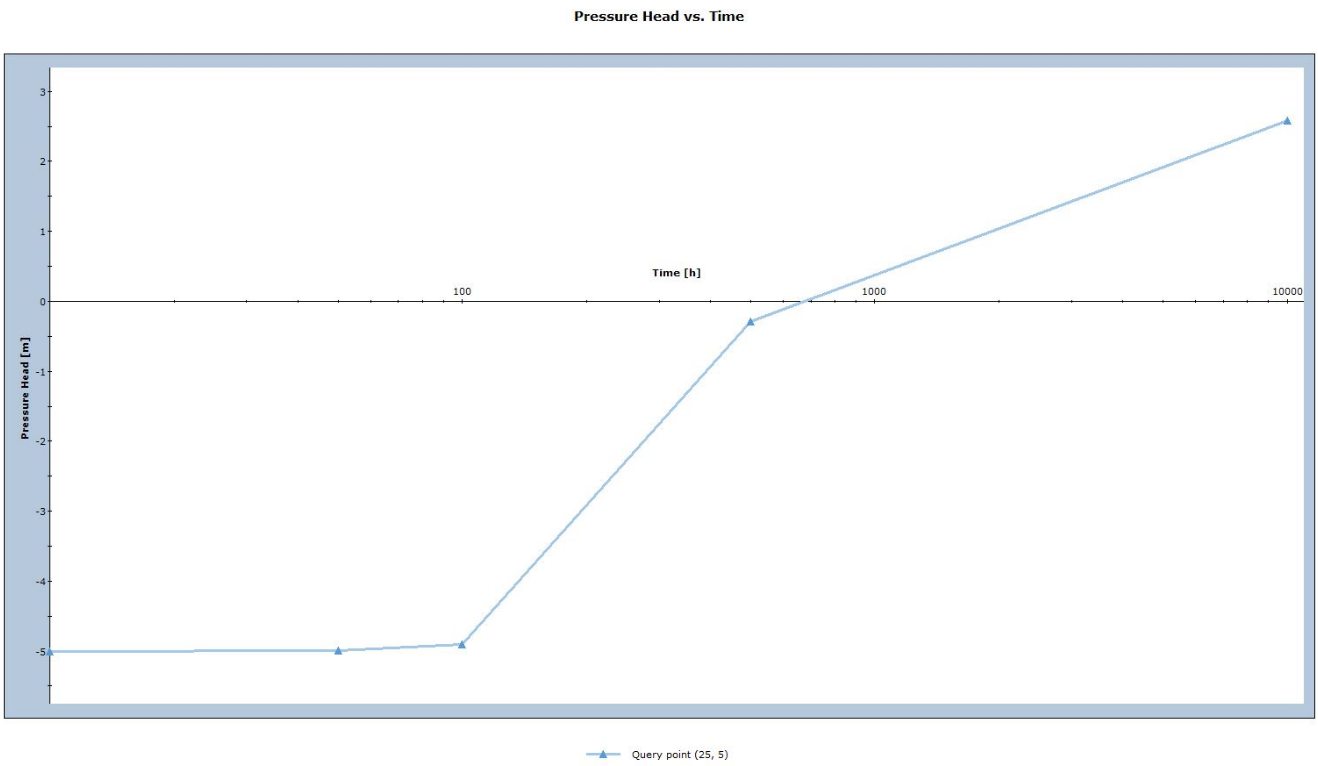


Pressure Head (m) vs. Time (h) at point (25,5)

It is often more informative to use the logarithm of time. To do this:

12. Select **Edit Properties** on the left sidebar.
13. Expand the **Axes** item and change **Logarithmic Horizontal Scale** to **Yes**.
14. Click **Close**.

The chart will now look like this:



Pressure Head (m) vs. Time (h) at point (25,5) – Logarithmic scale

You can see how the pressure front didn't reach the point until about 100 hours after the raising of the water level, and that after 100 hours the pressure head climbed to its steady-state value of ~2.4 m at 10,000 hours.

This concludes the tutorial.

4. Additional Exercise

You may want to see if 10,000 hours represents the true steady state. You can do this in two ways:

1. Add another stage at some later time (say 20,000 hours) and see if there is any change

2. Run a steady state (non-transient) analysis and see if you get the same result.