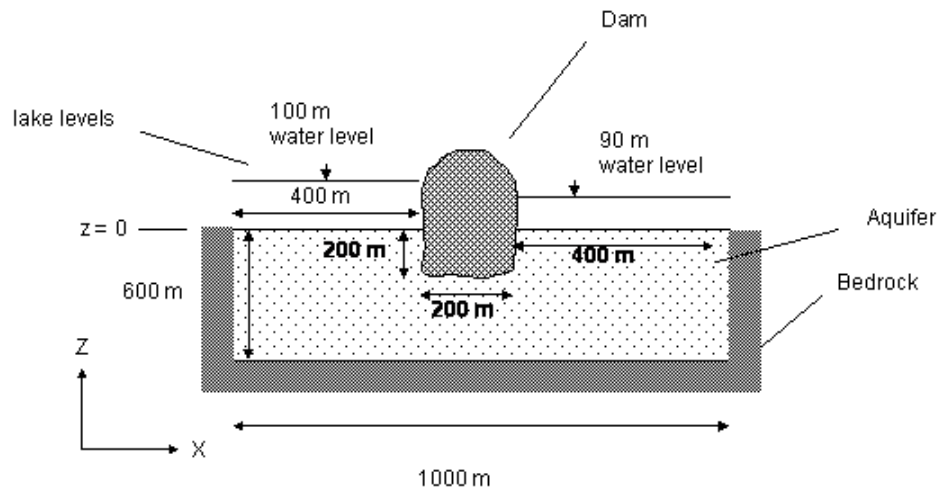


Groundwater homework: Finite Difference Modeling

Finite-difference schemes are one of the most common method for modeling saturated groundwater flow. It is easy to code and understand. One problem with it is that it requires a rectangular grid and modeling complex shapes, particularly moving boundaries is more difficult.

Consider the following situation.



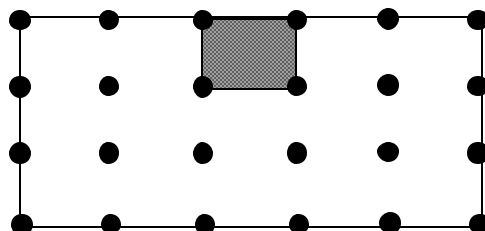
The goal is to determine the steady-state distribution of pressure in terms of water head and discharge.

1. (30 pts) Numerically simulate the flow. Using the discrete form of Laplace's equation, that we developed in class,

$$(h_1 + h_3 + h_4 + h_5)/4 = h_2$$

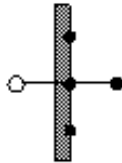
The diagram shows a central node labeled 2, with four nodes labeled 1, 3, 4, and 5 surrounding it. Node 1 is to the left, node 3 is to the right, node 4 is above, and node 5 is below node 2.

to calculate the hydraulic heads. Use the following node structure.



Define your boundary conditions. Constant pressure along the top boundary and no-flow boundaries elsewhere. To take into account, numerically, the no-flow boundaries use the idea of the image point.

no-flow boundary



The open circle is the image point on the other side of the no-flow boundary. To achieve no-flow, the head across the boundary has to be the same. Using the numbering system above, $h_1 = h_3$

For points along the no-flow boundary the equation is $(2h_3 + h_4 + h_5)/4 = h_2$

The corners are a bit different, but you can figure that out.

Start the calculation with the constant head boundaries defined and put in zeros for everything else. Then start the iteration and put the new values into a new array. Fix the decimal places to 2 (0.00). Continue the iteration with new arrays until the difference between the last iteration and the previous is zero for each cell. Zero is defined here as rounding to two decimal places (0.00).

The best way to solve this problem is to use a spreadsheet. The current version of EXCEL is most excellent for this purpose. I recommend making two parallel arrays. The first array has your calculated numbers and the parallel array has the difference from the last iteration. Fixing both arrays to two significant figures will save time. Copying the arrays and pasting them into the next set of rows with a blank row in-between is a rapid method to iterate the calculations.

- (15 pts) Print out your arrays for both differences and heads. This will include the initial head array and the first two time steps (3 head arrays each subsequent iteration with a difference array), and the final 3 head arrays (with differences) for a total of 6 head and difference arrays printed. For the difference arrays there should be 5 total, one for the 1st and 2nd iterations and 3 for the final 3 iterations. Also note how many iterations were required before you reached a zero difference (convergence).

Print out the head and difference arrays side by side noting the iteration number for each set. The 1st iteration of head is on the left and the first difference array is on the right. (The initial condition has no difference array associated with it.) The 2nd iteration under the first and the second difference array to the right and under the first difference array. So the print out will look like:

First	Second
Column	Column
head	difference
<u>arrays</u>	<u>arrays</u>

Initial arrays.

Final arrays

- (7 pts) Plot your results as a contour plot. Contour whole numbers of head (91, 92, ...99).
 - (8 pts) Calculate the discharge from each cell (defined by 4 nodes) at the downstream boundary (downstream of the dam) using the values found in the last two rows. Calculate the total discharge rate per unit width (1 m) using a value for the hydraulic conductivity of 10^{-6} m/s, and express your answer in $\text{m}^3 \text{day}^{-1}$.
2. (8 pts) Using the notes handed out in class. Write out the anisotropic homogeneous steady-state flow equation in two dimensions. Start with the form of that equation in the notes handed out in class and finish it off to its final mathematical form. This results in two lines: first line is the equation from the notes in 2-D the second line is the final mathematical form. THEN convert it to the discrete form (e.g. Δx , Δh) in the x, z plane (K_x , K_z). Show your work. The discrete form is what you would use to model the flow in (3). That is, continue the derivation until you solve for h_2 the central node.
3. Graduate Students (10 pts): Revise the numerical model in (1) using the results of (2) and recalculate the heads for an anisotropic situation. Show your initial and final arrays and plot the results for the final array. Use $K_x = 10^{-6}$ m/s and $K_z = 1.5 \times 10^{-6}$ m/s. Discuss the results and how they differ from problem (1).