Institutionen för datavetenskap
Umeå universitet
Martin Berggren

Case Study I: Groundwater flow computations

Modeling of steady groundwater flow in a aquifer

An *aquifer* (sv. *akvifär*) is an underground region that hosts significant amounts of ground water in materials such as gravel (sv. *grus*), sand, or clay. Aquifers are important sources of fresh water.

Figure 1 shows a vertical cross section of an underground region with a simple shape. The light rectangular area in the middle is made up of fine sand with homogeneous hydraulic conductivity and porosity. The dark area to the left and below the sandy rectangular area consists of an impervious (sv. *ogenomträngligt*) material such as solid rock. The region of sand borders to the upper right to a concrete structure that also is impervious and to the lower right to a lake. The sand is recharged with water (rain) from above, and water can also flow across the boundary to the lake.

We assume that there is a steady rain of r m/s from above, and that the water level in the lake is constant. The rain and the lake water will wet the sand. Since the rain is steady and the water level in the lake is constant, the conditions underground will eventually reach steady state. We may then divide the rectangular sandy region into an upper unsaturated zone (lightly tinted in figure 2) and a lower saturated zone (darkly tinted in figure 2). In the saturated zone, all available pores in the sand is filled with water, but not so in the unsaturated zone.

The interface between the saturated and unsaturated zone is called the *water table* (sv. *grundvattenyta*). (Despite its name, the water table is not in general a horizontal surface, as illustrated in figure 2!) Our task is to compute the shape of the water table and the flow pattern within the saturated zone at steady state. The flow patterns can be particularly important to know for environmental studies, since pollutants will be transported along with the groundwater flow.

We will at once simplify the situation and assume planar symmetry, that is, that the vertical cross section illustrated in figure 2 extends for a large distance (infinitely) in the direction perpendicular to the plane. This means that we may model in the two space dimensions of the plane (x and y).

Within the lower saturated layer, it is *pressure differences* and *gravity* that are the dominating mechanisms that drive the flow of water. For groundwater flow analysis, it is standard practice to measure pressure not in Pa but in meters of water level. That is, the *pore gauge pressure* (sv. *övertryck*) is p (in meters of water level) at a point $\boldsymbol{x} = (x, y)$ in the aquifer if this pressure would be able to sustain a column of water with the height p meters. The pore gauge pressure is measured relative to the atmospheric pressure, that is, $p = 0$ means that the pore gauge pressure equals the atmospheric pressure. This last condition holds at the water table.

A good model for the slow flow of water that takes place in the saturated layer is *Darcy's law*, which for groundwater flow takes the form

$$\boldsymbol{u} = -\kappa \nabla (p + y), \tag{1}$$

where $\boldsymbol{u} = (u, v)$ (in m/s) is the *apparent velocity* (explained below), $\kappa > 0$ (in m/s) is the hydraulic conductivity of the material, p (in m water level) is the pore gauge pressure, and y the vertical level (from an arbitrary but fixed *datum*, that is, a horizontal coordinate plane). Moreover, $\nabla = (\partial/\partial x, \partial/\partial y)$ is the gradient operator.

The form (1) of Darcy's law requires that the fluid flow is creepingly slow[1] and that the hydraulic conductivity of the material in the aquifer is *isotropic*, that is, that its conductivity

---

[1] More precisely, relation (1) holds for small so-called Reynolds numbers. The *Reynolds number* of a flow case expresses the quotient between inertial and viscous forces. The Reynolds number is usually small for groundwater flow since the forces on the water in the pores of the medium is dominated by adhesive and capillary effects.
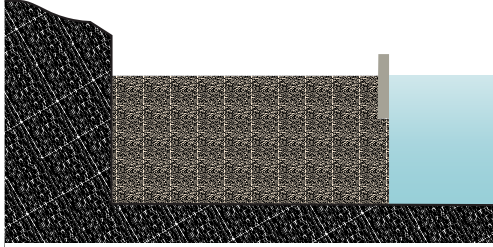
FIGURE 1. An aquifer containing fine sand (middle light rectangular region) that borders an impervious rock to the left and below, a concrete structure to the upper left, and a lake to the right.
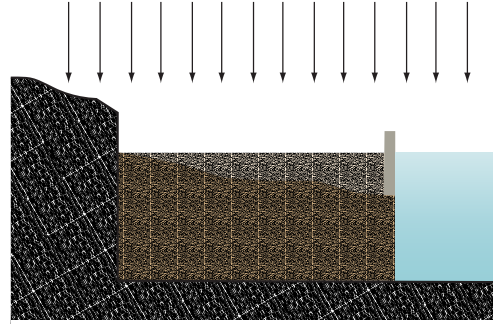

FIGURE 2. Assuming a steady recharge of rain from above and a steady lake water level, the lower, darker part of the sand will be saturated with ground water.

property is the same in every direction. Expression (1) can be generalized to the case of *anisotropic* media, such as when the ground has a layered structure, for which the conductivity is different in the directions parallel and perpendicular to the layers. In such cases, K will not be a constant but a symmetric matrix[2] with positive eigenvalues. The eigenvalues yields the conductivities in the directions of corresponding eigenvectors.

On the micro level, water is sipping through the pores in the sand in a complicated manner. In order to capture large-scale effects, however, it is not important to know the precise micro-level velocity field. The apparent velocity $u(x)$ expresses an average or "effective" velocity field, where the average is taken over a region around point x, a region that is small in comparison with the global dimension but much larger than the pore size. Thus, if $n\,dS$ is a directed surface element (n is the surface unit normal), the flux of water in m/s through the surface element will be $n \cdot u\,dS$, and if S is a surface inside the saturated region and n a unit vector field on the surface, the net flux Q of water (in m^3/s) through the surface will be

$$Q = \int_S n \cdot u \, dS. \tag{2}$$

Again, the apparent velocity u is *not* the real velocity of the fluid on the micro scale, but a quantity that indicates average effects. In particular, it is a quantity that through expression (2) can be used to calculate the flux of water through surfaces.

Note also the minus sign in Darcy's law (1). Assume, for instance, that the pressure is increasing in the positive x coordinate direction, that is, that $\partial p/\partial x > 0$. The pressure increase in the positive x direction will force the flow of water in the opposite, negative x coordinate direction. Moreover, recall that y is the vertical level, so $-\nabla y = (0,-1,0)$, which means that Darcy's law (1) tells us, reasonably, that gravity forces the water in the downward direction.

Question 1. Assume that there is no flow of water (apparent velocity zero) in a saturated aquifer. What will the pore gauge pressure be?

By introducing the *pressure head* (sv. *trycknivå*)

$$h = p + y, \tag{3}$$

Darcy's law (1) takes the simple form

$$u = -\kappa \nabla h, \tag{4}$$

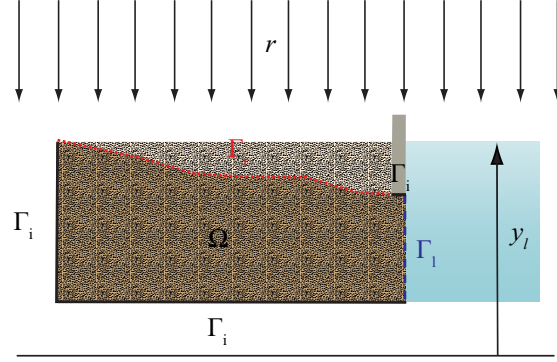
[2] Or, more precisely, a symmetric *second-order tensor*.

FIGURE 3. Definition of the parts of the boundary to the saturated region Ω. The impervious Γ_i (solid black), the recharge Γ_r (dotted red), and the lake interface Γ_l (dashed blue). Note that Γ_i consists of two disconnected parts that borders to the rock and to the concrete structure, respectively. The datum, that is, the horizontal coordinate plane $y = 0$ is indicated, and y_l is the lake water level.

which is the form which we will use from now on; in particular, the pressure head h will be the dependent variable in the boundary value problem that we will implement in Comsol.

Darcy's law (4) contains two unknowns, the pressure head and the apparent velocity. The two unknowns requires two equations, so we need another equation to close the system. That equation comes from the conservation of mass. At the pressures considered here, water can be regarded as an incompressible fluid. Let V be an arbitrary connected region inside the saturated region (a *control volume*) and let S be the boundary surface of V. Since the medium is completely saturated, the water in V has nowhere to go except through the boundary S; the pores are completely filled with water and the water cannot be compressed. Thus, "what comes in must go out": the net influx of water must be exactly balanced by the net outflux at each point in time. The net flux Q of water through S is given by expression (2), so since $Q = 0$, expression (2) together with the divergence theorem (also called Gauss' theorem) yields

$$0 = \int_S \boldsymbol{n} \cdot \boldsymbol{u}\, dS = [\text{the divergence theorem}] = \int_V \nabla \cdot \boldsymbol{u}\, dV, \tag{5}$$

where $\nabla \cdot \boldsymbol{u} = \partial u/\partial x + \partial v/\partial y$ is the *divergence* of the velocity field $\boldsymbol{u}$. Since equation (5) holds for *any* control volume in the saturated region we conclude that

$$\nabla \cdot \boldsymbol{u} = 0 \tag{6}$$

at every point in the saturated region, that is, the velocity field is *divergence free*. Equation (6) is called the *incompressibility condition* and holds in general for all velocity fields associated with constant-density incompressible fluids such as water or air at low speeds. (Darcy's law, however, is particular for the flow in porous media).

Together, Darcy's law (4) and the incompressibility condition (6) implies that the pressure head h inside the saturated region Ω satisfies equation

$$-\nabla \cdot (\kappa \nabla h) = 0. \qquad \text{in } \Omega. \tag{7}$$

To solve the partial differential equation (7), we need to supply it with suitable boundary conditions. We divide the boundary of the saturated region Ω into three parts, Γ_i, Γ_r, and Γ_l (figure 3). There is no flow through impervious boundaries, so $\boldsymbol{n} \cdot \boldsymbol{u} = 0$ at each point on Γ_i, which by Darcy's law (4) implies that

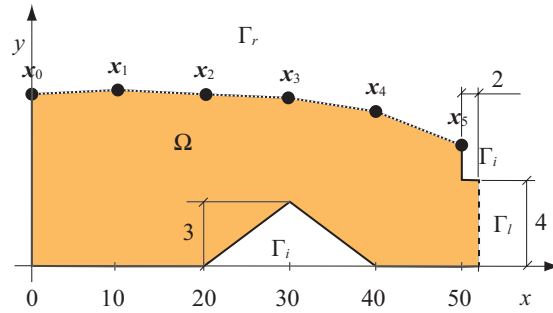$$\kappa \frac{\partial h}{\partial n} = 0 \qquad \text{on } \Gamma_i. \tag{8}$$

3

FIGURE 4. The saturated region with the shape of the water table parameterized by points $x_0, \ldots,$ x_5 (not drawn to scale).

We assume that there is a constant supply of r m/s rain from above (Göteborg?). The water is transported through the unsaturated region through gravity and capillary forces. In steady state, this supply penetrates through the boundary Γ_r, which means that $\boldsymbol{n} \cdot \boldsymbol{u} = -r$ on Γ_r. (The minus sign is because Γ_r's outward-directed normal points *upward* but a positive r means *downward*-directed supply.) Together with by Darcy's law (4), we find that

$$\kappa \frac{\partial h}{\partial n} = r \qquad \text{on } \Gamma_r. \tag{9}$$

In the lake, the pressure at vertical level y is simply $y_l - y$, the height of the column of water at this level. At the boundary Γ_l, the pressure will attain the value of the pressure in the lake at this depth, thus $p = y_l - y$, which, by the pressure head definition (3) means that

$$h = y_l \qquad \text{on } \Gamma_l. \tag{10}$$

To summarize, the boundary value problem for the pressure in the saturated region Ω is given by equations (7), (8), (9), and (10), that is,

$$\begin{aligned}
-\nabla \cdot (\kappa \nabla h) &= 0 && \text{in } \Omega, \\
\kappa \frac{\partial h}{\partial n} &= 0 && \text{on } \Gamma_i, \\
\kappa \frac{\partial h}{\partial n} &= r && \text{on } \Gamma_r, \\
h &= y_l && \text{on } \Gamma_l.
\end{aligned} \tag{11}$$

Task 1. Derive the variational form associated with boundary-value problem (11).

Calculating the water table level

Figure 4 illustrates the saturated region that will be used for computations. The shape of the water table is defined by linear interpolation of the points $x_i = (x_i, y_i)$, $i = 0, \ldots, I$. The horisontal locations of these points are $x_i = 50 \cdot i / I$ m, $i = 0, \ldots, I$, but the vertical elevations y_i are unknown, and the purpose of the exercise is to estimate these. In figure 4, $I = 5$, but you are free to choose more points if you like!

In Lab 1 of this case study, *Equation-based Modeling in Comsol Multiphysics,* we constructed the geometry using *solid modeling,* that is, the geometry was defined through set operations on

solid primitives like disks (circles) and rectangles. A conceptually different way to construct a geometry is *surface modeling*. In the present 2D case, "surface modeling" means that we define a solid by specifying a closed curve that will constitute the solid's boundary. (In 3D, we instead specify a closed *surface*, and the solid will then be defined as the interior of the surface.) For the present case, surface modeling is more convenient to use than solid modeling.

Geometry definition using surface modeling

1. Click on **New** in the **File** menu or launch Comsol in order to open the **Model Wizard**.

2. Select 2D on the Select Space Dimension Page and press the **Finish** button.

   [Next, we will parameterize the locations of the points $\boldsymbol{x}_i = (x_i, y_i)$, $i = 0, \dots, I$.]

3. In the **Model Builder**, right-click **Global Definitions** and select **Parameters**.

4. In the field **Name**, write x0, and in the field **Expression**, write 0. Continue similarly with parameters x1, …x5 (or whatever your last point is), that is, the x-coordinates $x_1, \dots, x_5$, which should have values according to figure 4. (If you use more than 6 points, you will of course need to specify a narrower spacing than the one in figure 4). Then define parameters y0, …y5 (or whatever your last point is), that is, the y-coordinates $y_1, \dots, y_5$. These should represent a reasonable starting guess for the shape of the water table, for instance the horisontal shape $y_0 = \dots = y_5 = 6$ m.

   [Now we will draw the boundary of domain Ω.]

5. In the **Model Builder**, right-click **Geometry 1** and select **Polygon**.

6. In the Polygon page that appears in the **Settings** tab, there are two fields for $x-$ and $y-$ coordinates for the locations of corners in a polygon. Enter the numbers derived from figure 4 in comma-separated lists. That is, in the **x** field, enter 0,20,30,40, and so on (use x5, x4, and so on, at the appropriate places). Similarly, in the **y** field, enter 0,0,3,0, and so on (use y5, y4, and so on, at the appropriate places).

7. Click the **Build Selected** button and the computational domain will appear in the **Graphics** window.

Implementation

Now implement the variational form you derived in Task 1 above by following the steps in Lab 1, section *Implementation in Comsol Multiphysics*, making the appropriate changes in order to comply to the present equation. Following parameter values are reasonable (define them in the Parameters page!):

$$\kappa = 5 \times 10^{-6} \text{ m/s} \qquad \text{(hydraulic conductivity, fine sand)}$$
$$y_l = 5 \text{ m} \qquad \text{(lake water level)}$$
$$r = 600 \text{ mm/year} \approx 1.9 \times 10^{-8} \text{ m/s} \quad \text{(rainfall)}$$

Studies

Results that are of interest to visualize are arrows and streamlines of the apparent velocity field $\boldsymbol{u}$, in order to see water transport ways, level curves of the pore gauge pressure p, and plots of the pressure on Γ_r, in order to find the water table location. In particular, plotting arrows is a good debugging tool, in order to confirm that the water is flowing generally in the expected direction! How to plot arrows streamlines was explained in Lab 1. How to plot level curves and 1D boundary plots of p is explained below.

Plotting level curves

1. In the **Model Builder**, right-click on **Results > 2D Plot Group 1** and select **Contour**.

2. On the **Contour** page that then appear in the Settings tab, enter in the field **Expression**, the expression for the pore gauge pressure (see definition(3)). If you click the box labeled **Level labels**, you will see labels for the level curves.

3. Since the dimensions is quite extended in the horizontal direction, it may be difficult to clearly see the level curves. They will be more visible if the plot is scaled to a square format. To do so, select on **Model 1 > Definitions > View 1 > Axis** and unselect the box labeled **Preserve aspect ratio**. Then click on **Zoom Extents** in the right upper corner of the Graphics tab in order to fill up the Graphics window. Then click on **Results > 2D Plot Group 1 > Contour 1** in order to see the contours.

Plotting pressure on Γ_r

1. Right-click on **Results** in the **Model Builder** and select **1D Plot Group**

2. Right-click on **Results > 1D Plot Group 3** and select **Line Graph**

3. In the **Line Graph** page that will appear in the **Settings tab**, add the boundary segments corresponding to Γ_r.

4. Enter the expression for the pore gauge pressure (definition (3)) in the field labeled **Expression**.

5. Click on the ▶ to the left of **x-Axis Data**. In the drop-down menu **Parameter** that then appears, choose **Expression** (instead of the default **Arc length**) and write x in the field **Expression**.

6. Click on the **Plot** button to see a plot of the pore gauge pressure on Γ_r versus the x coordinate.

As pointed out in the introduction, boundary Γ_r represents a water table if the pore gauge pressure $p = 0$. Thus, the problem of finding the shape of the water table corresponds to finding locations for the points $\boldsymbol{x}_i$ such that $p = 0$. However, when computing the solution associated with your initial guesses of elevations $y_0, \ldots, y_I$ and plotting the pressure on Γ_r, you will see that the pressure is not zero, unless you are extremely lucky with your initial guess of the elevations!

Task 2.

1. Modify the shape of Γ_r, as defined by parameters y_i, $i = 1, \ldots, I$, until $|p| \leq 5$ cm uniformly along Γ_r.