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 groundwater 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 \((x, y)\).

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 \( x = (x, y) \) 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

\[
\mathbf{u} = -\kappa \nabla (p + y),
\]

(1)

where \( \mathbf{u} = (u, v) \) (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, 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.
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 $\mathbf{u}(\mathbf{x})$ expresses an average or “effective” velocity field, where the average is taken over a region around point $\mathbf{x}$, a region that is small in comparison with the global dimension but much larger than the pore size. Thus, if $\mathbf{n} \, dS$ is a directed surface element ($\mathbf{n}$ is the surface unit normal), the flux of water in m/s through the surface element will be $\mathbf{n} \cdot \mathbf{u} \, dS$, and if $S$ is a surface inside the saturated region and $\mathbf{n}$ a unit vector field on the surface, the net flux $Q$ of water (in m\(^3\)/s) through the surface will be

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

(2)

Again, the apparent velocity $\mathbf{u}$ 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.

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)

$$h = p + y,$$  

(3)

Darcy’s law (1) takes the simple form

$$\mathbf{u} = -\kappa \nabla h,$$  

(4)

\(^2\)Or, more precisely, a symmetric second-order tensor.
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

\[
0 = \int_S \mathbf{n} \cdot \mathbf{u} \, dS = \int_V \nabla \cdot \mathbf{u} \, dV,
\]

(5)

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

\[
\nabla \cdot \mathbf{u} = 0
\]

(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 \( \Omega \) satisfies equation

\[
-\nabla \cdot (\kappa \nabla h) = 0. \quad \text{in } \Omega.
\]

(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 \( \Gamma \) into three parts, \( \Gamma_i, \Gamma_r, \) and \( \Gamma_l \) (figure 3). There is no flow through impervious boundaries, so \( \mathbf{n} \cdot \mathbf{u} = 0 \) at each point on \( \Gamma_i \), which by Darcy’s law (4) implies that

\[
\kappa \frac{\partial h}{\partial n} = 0 \quad \text{on } \Gamma_i.
\]

(8)
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 $\Gamma_r$, which means that $\mathbf{n} \cdot \mathbf{u} = -r$ on $\Gamma_r$. (The minus sign is because $\Gamma_r$’s outward-directed normal points upward but a positive $r$ means downward-directed supply.) Together with by Darcy’s law (4), we find that

$$\kappa \frac{\partial h}{\partial n} = r \quad \text{on } \Gamma_r.$$  \hfill (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 $\Gamma_l$, 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

$$h = y_l \quad \text{on } \Gamma_l.$$  \hfill (10)

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

$$\begin{align*}
-\nabla \cdot (\kappa \nabla h) &= 0 \quad \text{in } \Omega, \\
\kappa \frac{\partial h}{\partial n} &= 0 \quad \text{on } \Gamma_i, \\
\kappa \frac{\partial h}{\partial n} &= r \quad \text{on } \Gamma_r, \\
h &= y_l \quad \text{on } \Gamma_l.
\end{align*}$$  \hfill (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 horizontal 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 \( x_i = (x_i, y_i) \), \( i = 0, \ldots, I \).]

3. In the **Model Builder**, right-click **Global Definitions** and select **Parameters**.
4. In the field **Name**, write \( x_0 \), and in the field **Expression**, write \( 0 \). Continue similarly with parameters \( x_1, \ldots, x_5 \) (or whatever your last point is), that is, the \( x \)-coordinates \( x_1, \ldots, x_5 \), 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 \( y_0, \ldots, y_5 \) (or whatever your last point is), that is, the \( y \)-coordinates \( y_1, \ldots, y_5 \). These should represent a reasonable starting guess for the shape of the water table, for instance the horizontal shape \( y_0 = \cdots = y_5 = 6 \text{ m} \).

   [Now we will draw the boundary of domain \( \Omega \).]

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 \( x_5, x_4 \), and so on, at the appropriate places). Similarly, in the \( y \) field, enter \( 0, 0, 3, 0 \), and so on (use \( y_5, y_4 \), 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!):

\[
\begin{align*}
\kappa &= 5 \times 10^{-6} \text{ m/s} \quad \text{(hydraulic conductivity, fine sand)} \\
y_l &= 5 \text{ m} \quad \text{(lake water level)} \\
r &= 600 \text{ mm/year} \approx 1.9 \times 10^{-8} \text{ m/s} \quad \text{(rainfall)}
\end{align*}
\]

**Studies**

Results that are of interest to visualize are arrows and streamlines of the apparent velocity field \( \mathbf{u} \), in order to see water transport ways, level curves of the pore gauge pressure \( p \), and plots of the pressure on \( \Gamma_r \), 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 Γr
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 Γr.
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 Γr versus the x coordinate.

As pointed out in the introduction, boundary Γr 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 \( x_i \) 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 Γr, 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 Γr, as defined by parameters \( y_i, i = 1, \ldots, I \), until \( |p| \leq 5 \) cm uniformly along Γr.