TMA4195 MATHEMATICAL MODELLING

PROJECT 2012: AQUIFER THERMAL ENERGY STORAGE

1. Introduction

In the project we will study a so-called Aquifer Thermal Energy Storage (ATES) system with the aim of climitizing large buildings. This is a system where hot and cold water is stored in separate regions (reservoirs) in the ground – the porous soil above the bedrock. Because of the pores, this soil will contain water and is therefore called an aquifer. A minimal ATES system has two separated wells, one for the cold reservoir and one for the hot reservoir. In the summer cold water is extracted from the cold reservoir and used to cool the building. After cooling the building the water which is now warmer is injected into to warm reservoir after further heating. In the winter the process is reversed, warm water is extracted from the warm well, and after heating the building, the by now cold(er) water is injected into the cold well. At Gardermoen Airport such an ATES system is operating [1]. This system

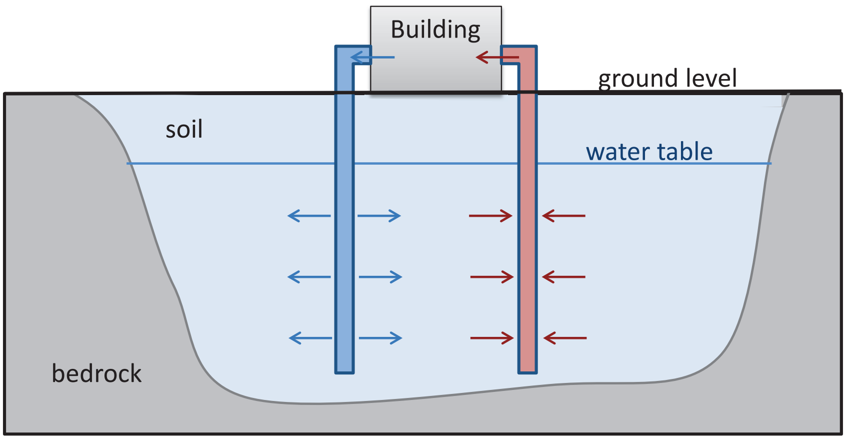


Figure 1. The ATES system in winter operation.

has one array of 9 warm wells and one array of 9 cold wells. Each well is 45 m deep, the (orthogonal) distance between the two arrays is 150 m, and in each array the wells are spaced 50 m apart. More information can be found in Table 2 and in [1]. The aquifer below Gardermoen is mainland Norways largest precipitation-fed groundwater reservoir, and it consists of different layers of clay, sand, and gravel. Some (averaged) physical parameter for the Gardermoen aquifer can be found in Table 1.

The goal of this project is to model the heat and water flow in the ATES system. Such a model can then in an other step be used to optimze the operation and design of the system.

2. About the modeling

The soil or aquifer is a porous medium meaning that it contains small volumes of open space (pores) that can be filled with air or water. The average effective *porosity* ϕ is equal to the total accessible pore volume divided by the total volume, $[\phi] = 1$ and $0 < \phi < 1$. The boundary between the regions filled with water and air is a surface called the *water table* (grunnvannspeilet). Below it, all the pore volume is filled (saturated) with water and above it the pore volume is filled with air.

Flow in porous media is pressure driven and described through the semi-empirical Darcy's law [4, 3, 2, chapter 13]:

$$\vec{q} = -K\nabla\varphi,$$

where q is the volume flux per unit area ('effective fluid velocity', $[q] = \frac{m^3}{m^2 s} = \frac{m}{s}$), $K = \frac{\rho g}{\mu}k$ the *hydraulic conductivity*,

$$\varphi = \frac{p}{\rho g} + z$$

is the *hydraulic head* or potensial, p is the pressure, and z is the vertical coordinate increasing upwards. Note that there is no flow ($q \equiv 0$) when p is equal to the *hydrostatic pressure*, i.e.

$$p(z) = p_0 + \rho g(z_0 - z)$$

where p_0/z_0 is some constant reference pressure/hight. If you think about it, this is as it must be! Furthermore, g is the gravitational acceleration, ρ fluid density, and μ the fluid dynamic viscosity. The k is the *intrinsic permeabilty* and may be a matrix. We will assume it is diagonal so that K takes the form

$$K = \begin{pmatrix} K_h & 0 & 0 \\ 0 & K_h & 0 \\ 0 & 0 & K_v \end{pmatrix}.$$

Darcy's law and flow in porous media is described in Fowler [2] chapter 13 (hand-outs), supplementary material can be found in [3, 4] and on the internet.

Transport of heat (=heat energy) in the ATES system is caused by diffusion of heat energy in the water and soil and transport/convection of heat energy due to the water flow. The flux of heat energy Φ_D^e ($\frac{energy}{area \cdot time}$) due to diffusion is described by *Fourier's law*:

$$\Phi_D^e = -\lambda\nabla T,$$

where T is the temperature and λ is the diffusion coefficient or *thermal conductivity*. The heat *energy density* e is given by

$$e = \rho c T,$$

where ρ is the mass density and c is the specific heat ($\frac{energy}{mass \cdot temperature}$). If a fluid is flowing with volume flux q, then the convective heat energy flux Φ_C^e of the fluid is given by

$$\Phi_C^e = \text{heat velocity } (= \text{water flux}) \cdot \text{heat density} = q \cdot \rho c T.$$

Realistic values for the Gardermoen aquifer of all parameters introduced above is given in Table 1.

Physical quantity	Symbol	Value	
Water density	ρ_w	1000	kg/m^3
Soil (in aquifer) density	ρ_s	1700	kg/m^3
Water table (=grunnvannspeil)	$h_{w.t.}$	13-14	m
Minimal aquifer diameter (maximal)	D	3000 (10000)	m
Aquifer depth	H	45	m
Average porosity	ϕ	0.1507	
Horizontal hydraulic conductivity	K_h	$3.8568 \cdot 10^{-5}$	m/s
Vertical hydraulic conductivity	K_v	$1.6226 \cdot 10^{-5}$	m/s
Spesific heat of water	c_w	4200	J/kgK
Spesific heat of soil	c_s	1381	J/kgK
Thermal conductivity of water	λ_w	0.6	W/mK
Thermal conductivity of soil	λ_s	1.0	W/mK

Table 1. Physical parameters for the Gardermoen aquifer.

Physical quantity	Symbol	Value	
Well depth	h_w	45	m
Well diameter	d_w	0.450	m
Hot well – cold well separation	L_x	150	m
Hot well – hot well (cold well – cold well) separation	L_y	50	m
Well production region (depth)		$17 - 42$	m
Maximal well production rate (water in/out)	$Q_{\max}$	$8.33 \cdot 10^{-3}$	m^3/s
Injection-water temperatur warm well (summer)	$T_{in,s}$	30	^{o}C
Injection-water temperatur cold well (winter)	$T_{in,w}$	4.1	^{o}C

Table 2. Physical parameters for the Gardermoen ATES system.

3. On notation and simplifying assumptions

Here is the notation for some of the relevant physical quantities of the problem:

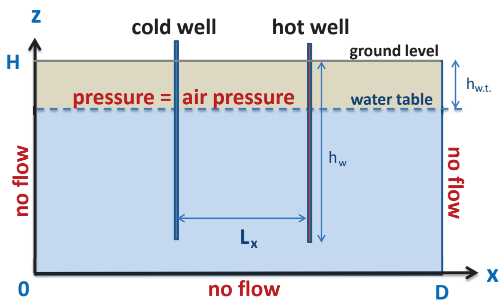
- coordinate system (x, y, z), where z is the vertical coordinate.
- g: gravitational acceleration
- p: pressure
- p_0: pressure at $z = z_0$ (reference pressure)
- φ: hydraulic head $[\varphi] = m$,
- ϕ: effective rock porosity, $[\phi] = 1$ and $0 \leq \phi \leq 1$.
- $\vec{q} = (q_1, q_2, q_3)$: volume flux of water per unit area, $[\vec{q}] = \frac{m}{s}$
- $\frac{\vec{q}}{\phi}$: effective velocity of fluid particles
- q_W, q_C: volumetric water production rate per (surface) area of warm, cold well (volume per time and area), $[q] = \frac{m^3}{m^2 s} = \frac{m}{s}$
- Q_W, Q_C: volumetric water production rate (volume/time) of warm, cold well, $[Q] = \frac{m^3}{s}$
- T: temperature

Other parameters are defined in Tables 1 and 2.

Then we give some common assumptions that you *may use or not*:

(1) The pressure at the water table equals the pressure of air at ground level (the error you make is very small). You may set it equal to 0 (only pressure differences matter in the model).

(2) Water and soil are incompressible, i.e. ρ_w and ρ_s is constant in the aquifer.
(3) Geometry: Constant water table.
- The aquifer is horizontal and box-shaped with $z = 0$ as bottom plane. Take e.g. the region between the planes $z = 0$, $x = 0$, $x = D$, $y = 0$, and $y = D$.
- Bottom and side boundaries are impermeable, there is no flow through them.
- The wells are straight lines or circular tubes parallel to the z-axis (the vertical axis). Two wells can be used for a start.



(4) Geometry: Non-constant water table.
- In a more realistic model (not for a first attempt!), the water table is not constant and can depend on x, y, t, i.e. it is a moving boundary that can be written as a graph $z = h(x, y, t)$ (this will then be the upper boundary of the aquifer).
- In this case a so-called *kinematic boundary condition* is needed:

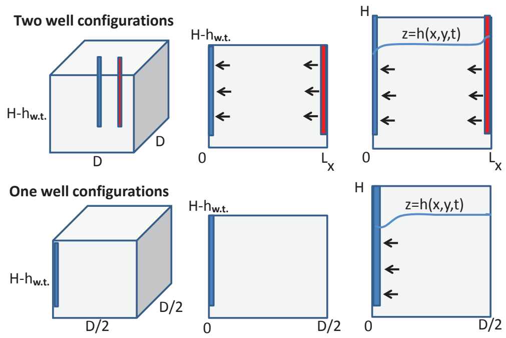
$$\phi h_t + h_x q_1 + h_y q_2 = q_3 \quad \text{at} \quad z = h,$$

in addition to the pressure condition $p = 0$ at $z = h$. The kinematic boundary condition simply enforce that any particle $(x(t), y(t), z(t))$ at the boundary remains at the boundary, i.e. that

$$\frac{d}{dt}\Big(z(t) - h\big(x(t), y(t), t\big)\Big) = 0.$$

See [2] chapter 13 (or google!) for more information about this type of boundary condition.

(5) Other interesting configurations you may consider (but motivate why). All suggestions here have vertical components, but it is equally relevant to look at horizontal cross-sections of the reservoir (e.g. a "quater five spot" case).



(6) Injection/productions rates: $q = q(z)$ could be assumed to be constant in the production region.
(7) The temperature is the same in the soil and water located at the same point.
(8) The convection of heat dominates over the diffusion of heat and the latter may be ignored (can be justified through a scaling analysis).
As a consequence, where there is no water flow there will be no heat energy flux in the water.
(9) Boundary conditions for heat flow: No flux all along the boundary, or more realistically, average air temperatures at water table, average temperatures for the 45 m depth at bottom, no flux at the sides (why?).

You could consider 3D or 2D or 1D models. In the latter cases you have to be carefull with the boundary conditions – remember to integrate up the flow rates q. Remember to argue for or discuss the choices you make.

4. Modeling projects

Every group should produce at least one scaled model, and then say something about its solutions numerically and/or analytically. Then you could try to answer e.g. between one and four out of the problems 1.1 – 1.10 or 2.1 – 2.14. There are enough challanges in problem 1, but ambitious groups may do project 2 instead. If you can find other related problems to consider that is great!

There are far too many problems for any one group to solve, and it is better to do a few problems well than many problems poorly.

Project 1: A simple model. Model and solve for the heat flow aquifer at given production rates $Q_W = -Q_C$.

a) (Medium) Model the water and heat flow. The results should be a system of PDEs, initial, and boundary conditions.
b) (Easy) Scale the model, identify small parameters.
c) (Easy – Medium) Solve the model numerically. Plot stream lines and equipotential lines in 2D or 3D models.
d) (Easy – Medium) Solve the model analytically (e.g. in 1D or in special cases).

You can use 1D, 2D, or 3D models and conservation arguments with fluxes. To solve numerically, finite difference methods, finite element methods, or finite volume methods could be used. To get exact or approximative analytical solutions, you can use separation of variables, the method of characteristics (if heat diffusion is ignored), fundamental solutions and the method of mirror sources (google "method of mirror charges" and have a look at Exercise 9, problem 2 (h)), perturbation or approximative solutions.

Note that the water flow can be solved independently of the heat flow. Consider a simplified geometry with lots of symmetries. Make reasonable simplifying assumptions. Try to reduce the number of dimensions – are all dimensions equally important? I suggest you first try to model the second two-well configuration in part (5), section 3 (e.g. a 2D model).

Here are some questions you may try to answer/discuss:

1.1 What are reasonable initial and boundary conditions? What are reasonable boundary conditions for the temperature (or the heat energy flux)?
1.2 Can you determine whether convection or diffusion dominates for the heat flow using a scaling argument?

1.3 How is the heat flow over one year with constant $Q_W = -Q_C$?
1.4 What happens under cyclic operations of the ATES system, when the summer and winter production are reversed?
1.5 Compute the warm and cold water fronts of the injected water, and the minimal distance between them. Will they overlap?
1.6 How much will the temperature drop in the warm reservoir after 6 months of winter and extraction? E.g. average or center temperatures.
1.7 What happens if the production rate $Q_W = -Q_C$ is reduced or increased? Consider e.g. the temperatur drop and the cold-warm zone separation. Any optimal values?
1.8 What happens if the distance between the cold and hot well is changed? Consider e.g. the temperatur drop and the cold-warm zone separation. Any optimal values?
1.9 Is the solution sensitive to seasonal changes in air temperatur? Compare temperature drops in warm reservoir. Use temperature boundary conditions varying with the seasons.
1.10 Is a 1D model sufficiently accurate here? Compare with 2D or 3D models.

Project 2: A more accurate model. In a more realistic model, we have to account for the fact that the water table will vary with the production rate, at least near the wells. There are at least two modeling possiblities:

(1) 2D or 3D models with free upper boundary $z = h(x, y, t)$ (the water table) and kinematic boundary conditions. See section 3 of this note and the dam-modeling problem in [2] chapter 13 for more details.

(2) 1D or 2D Dupuit-Forchheimer models. Models based on Darcy's law and the simplifying assumption that the pressure equals the hydrostatic pressure away from the wells (this is a long-dam approximation!). See the dam-modeling problem in [2] chapter 13 and my lectures for more details. Take care when deriving boundary conditions for this model.

Problems:

a) (Medium) Model the water and heat flow.

b) (Easy) Scale the model, identify small parameters.

c) (Easy – Hard) Solve the model numerically. Plot stream lines and equipotential lines in 2D or 3D models.

d) (Hard) Solve the model analytically (e.g. in 1D or in special cases).

Note that the water flow and free boundary evolution can be solved independently of the heat flow. Consider a simplified geometry with lots of symmetries. Make reasonable simplifying assumptions. Try to reduce the number of dimensions. Always motivate your assumptions. Finite difference, volume, or element methods can be used for numerical solutions:

- Naive difference methods will work, but for Model (1) it can be difficult to get a stable implementation.
- In Model (1) where you have a kinematic boundary condition, you can solve the problem in two steps: i) Solve for the boundary $z = h^{n+1}$ at time t_{n+1} using the pressure p^n at time t_n, and ii) solve for the pressure p^{n+1} at time t_{n+1} using the boundary $z = h^{n+1}$ at time t_{n+1}. Iterate.
- Finite element/volume methods are very natural in the case where there is a moving boundary (Model (1)).

- For non-linear PDEs on a stationary domain (the Dupuit models, model (2)), (monotone, conservative) finite difference methods are better. Hint: You may use that $\partial_x(hh_x) = \frac{1}{2}\partial_x^2(h^2)$.

Simple special solutions or approximate solutions can be considered e.g. for region near the well or far from the well, or when $Q_W = -Q_C$ (and $q_W = -q_C$) is small or big, or for small or large times, or big and small values of some of the other parameters. One possibility (there are others...) is to construct similarity solutions (at least for the water flow) assuming $D \gg 1$ ($D = \infty$), see e.g. section 5.5 on Similarity solutions in the lecture note of Krogstad. You could try this both for the "full" model and for the Dupuit-Forchheimer model. See also [2] for hints on the Dupuit-Forchheimer model.

In addition to the questions in problem 1, you may consider:

2.1–2.10 = Problems 1.1 – 1.10.

2.11 Compare one model from problem 2 with one model from problem 1. E.g. compare temperature drop and/or cold-hot zone separation.

2.12 What is the maximal *draw down* at the extraction well after 6 months operation? The draw down is the reduction in the hight of the water table due to extraction of water.

Related questions: How does the draw down depend on the production rate? Can the water table be so low that air will enter the production zone of the well and disturb the production? If so, what can be done to avoid this situation while still producing the same amount of warm water?

2.13 Derive the Dupuit(-Forchheimer) model from a model of type (1) via perturbation analysis. A similar case is treated in [2], try to extend the analysis to the present case.

2.14 Compare the Dupuit-Forchheimer solution to the solution of a more accurate model with free upper boundary (type (1) model). Compare the draw-downs (see 2.12)

References

[1] G. Eggen and G. Vangsnes. Heat pump for district cooling and heating at Oslo Airport, Gadermoen. Technical report, COWI, 2005. http://129.241.253.164/project/Annex29/Installasjoner/GSHP_GardermoenHP_NO[1].pdf

[2] A. C. Fowler. *Mathematical Models in the Applied Sciences.* Cambridge 1997.

[3] http://en.wikipedia.org/

[4] J. Bear. *Hydraulics of Groundwater.* Dover, 2007.