Modeling Fluid and Energy Flow in Liquid Dominated Hydrothermal Systems

Larus Thorvaldsson
Halldor Palsson

October 2, 2012

Abstract

A new modeling approach of fluid flow in geothermal reservoirs is presented in the paper. Two models are presented, one which is applicable for groundwater hydrology and another more complex for hydrothermal systems. The performance of the groundwater model is then compared with the well known Theis solution.

Both problems are formulated and solved by using a highly customizable set of C++ libraries and tools, collectively named OpenFOAM, along with polynomial interpolation for the physical properties of water as functions of temperature.

Preliminary results from a more general modeling work on hydrothermal systems are presented in simple case studies, showing the basic abilites of the programming platform to solve problems for flow in porous media. It is concluded that the modeling framework is both flexible and efficient, and an added benefit is that it is under constant improvement by a large group of developers and incorporates cutting edge technologies in numerical analysis for mathematical modeling.

1 Introduction

Using numerical methods to solve non-linear partial differential equations (PDE) first became feasible in the late 1960's with the advent of digital computers. These methods were first applied to problems involving groundwater as well as oil and gas reservoirs, while the modelling of geothermal reservoirs lagged behind [1]. This was mostly due to the fact that the modelling of geothermal reservoirs is considerably more complicated where the equations are typically of the advection-diffusion type, describing conservation of mass, momentum and thermal energy. These equations are furthermore coupled with each other and are frequently non-linear, which adds considerably to the complexity of their solutions.

The earliest efforts to apply numerical models to geothermal reservoirs were made in the early 1970's, while the usefulness of numerical modelling did not begin to gain acceptance by the geothermal industry until after the 1980 Code Comparison Study [2]. Since that study was performed, the experiences gained in carrying out site-specific studies as well as generic reservoir modelling studies have led to a constant improvement in the capabilities of numerical reservoir models.

Over the last 20 years computer modeling of geothermal reservoirs using finite volume methods has become a standard practice. The most common approach is to apply the TOUGH2 code, developed by the Earth Sciences Division of Lawrence Berkeley National Laboratory in the 1980's. TOUGH2 is a general numerical simulation code for multi-dimensional coupled fluid and heat flows of multiphase multicomponent fluid mixtures in porous and fractured media [3].

An alternative modeling work is presented here, based on OpenFOAM, which is a library of highly efficient codes developed for the solution of general PDE's. The object orientation and operator overloading of C++ has enabled the developers of OpenFOAM to build a framework for computational fluid dynamics that enables modelers to work at a very high level of abstraction [4]. This makes it possible to manipulate the set of partial differential equations that describe the problem and customize the solver itself for each class of cases that needs to be solved. This is the main motivation for using OpenFOAM, as an alternative to currently existing models, such as TOUGH2.

2 Methods and Materials

In this section the governing equations for two phase flow in porous media are presented in the form they are implemented in a numerical model. This involves the equations themselves, fluid properties, boundary conditions and then the programming implementation itself.

2.1 Solver for groundwater systems

Implementation of new models in OpenFOAM is in most cases relatively simple. Low level operations regarding individual computational cells or the solution of linear systems do not need to be addressed in most cases, and the programming framework is designed with customization in mind.

As an example of this, one can take the basic equation that describes hydraulic head in a homogenous aquifer over time

$$\frac{\partial p}{\partial t} = \frac{T}{S}\nabla^2 p \tag{1}$$

where T is the transmissivity and S is the storativity of the aquifer.

This equation can easily be implemented in OpenFOAM by the following lines

```
fvm::ddt(p) - fvm::laplacian(T/S, p)
```

where the transmissivity and storativity have been defined. Nevertheless more coding is needed, such as defining the variables as field functions, but the developer does not need to become familiar with the inner workings of the numerics. A good example are the functions `fvm::ddt` and `fvm::laplacian` shown above, which will automatically result in a construction of a linear system for an implicit solution of an unsteady diffusion equation.

In order to verify the solver, it is possible to compare it with the well known similarity solution to equation 1 that Theis gave in 1935 for a homogenous confined aquifer [5]. By using the similarity transform

$$u = \frac{r^2 S}{4Tt} \tag{2}$$

where r is the radial distance from the well and t is time, Theis showed that the drawdown s could be expressed in terms of the well function W such that

$$s = \frac{Q}{4\pi T} W(u). \tag{3}$$

Here Q is the volumetric extraction of water from the well and W is the well function, which is known as the exponential integral outside of hydrogeology literature and is defined as

$$E_1(u) = W(u) = -\gamma - \ln u + \sum_{k=1}^{\infty} \frac{(-1)^{k+1} u^k}{k \cdot k!} \tag{4}$$

where $\gamma \approx 0.57721$ is the Euler-Mascheroni constant.

Since an analytical solution exists for this case it is ideal for testing purposes of the code. However there are some differences that should be addressed, for example the well has to have some finite surface area, in order to define the boundary conditions of the aquifer, and the size of the aquifer has to be finite.

2.2 Solver for a hydrothermal system

In order to model two phase flow in porous media it must be assumed that mass and energy are conserved. The continuity equation describes mass conservation and is given such that

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \vec{u}) = 0$$

where ρ is the average density of the phases and $\vec{u}$ is the superficial velocity. This eqation is solved for every time step in order to ensure continuity.

The solver then applies a PIMPLE pressure-velocity corrector loop, where SIMPLE and PISO algorithms have been merged for a more robust pressure-velocity coupling. This makes it possible to solve stiff differential equations by coupling a SIMPLE outer corrector loop with a PISO inner corrector loop, while also achieving more stability for larger time-steps compared to PISO [6].

In this case the problem at hand involves laminar flow, where inertial forces are negligable Darcy's law can be applied to the velocity equation, giving

$$\frac{\partial}{\partial t}(\rho \vec{u}) + \frac{\mu}{\kappa} \vec{u} = 0 \tag{5}$$

instead of the full Navier-Stokes equations.

The energy equation is then given as

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\rho \vec{u} h) = \nabla \cdot (\alpha \nabla h) \tag{6}$$

where h is the enthalpy of the water and alpha is the effective thermal diffusivity of the water. Note that the porosity is disregarded here, but without the loss of generality.

The physical properties of density ρ, viscosity μ and heat capacity c_p are all assumed to be seventh degree polynomial fuctions of temperature, such that

$$f(T) = \sum_{i=0}^{7} a_i T^i \tag{7}$$

where f is the physical property and a_i are the respective coefficients given in Table 1.

Table 1: Coefficients for physical properties

	$\rho(T)$	$\mu(T)$	$c_p(T)$
a_0	-4.844433993781387e+04	27.732508110282716	1.869467009204107e+06
a_1	1.004443612129644e+03	-0.578169744144130	-3.863103544592875e+04
a_2	-8.796904074130724	0.005172930468241	3.434316124950962e+02
a_3	0.043005953646541	-2.573544345985454e-05	-1.698437537901375
a_4	-1.265747239408929e-04	7.686291653398755e-08	0.005045265306273
a_5	2.240156150413993e-07	-1.377781528721409e-10	-9.000139889194412e-06
a_6	-2.205980438513678e-10	1.372177963166062e-13	8.925534508747834e-09
a_7	9.319510246689768e-14	-5.856417743042797e-17	-3.795296368702309e-12

3 Results

3.1 Groundwater system

A one dimensional axi-symmetric mesh was generated for simulating groundwater flow around a well where water was being extracted. The mesh was divided into 100 cells.

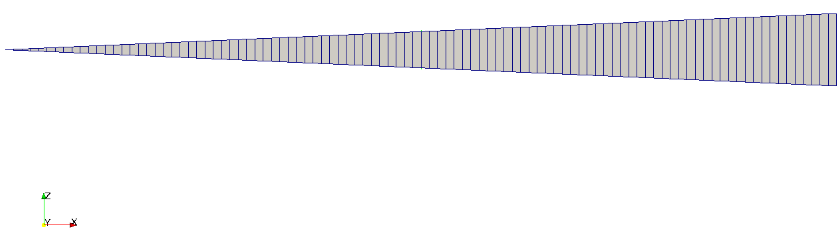

Figure 1: A top-down view of the axi-symmetrical one-dimensional mesh.

The mesh can be seen in Figure 1, its radius extends to 1 km, but the well is assumed to have a radius of 10 cm.

Figure 2 shows the numerical solution and the Theis solution at time $t = 10^5$. It can be seen that there is good agreement between the solution, despite the approximations in the numerical solution, such as the aquifer only extending to a finite radius from the well, and the requirement on the well to have some finite surface area.

Figure 3 shows the relative difference between the two solutions. It can be seen that they agree very well, where the maximum difference between the two

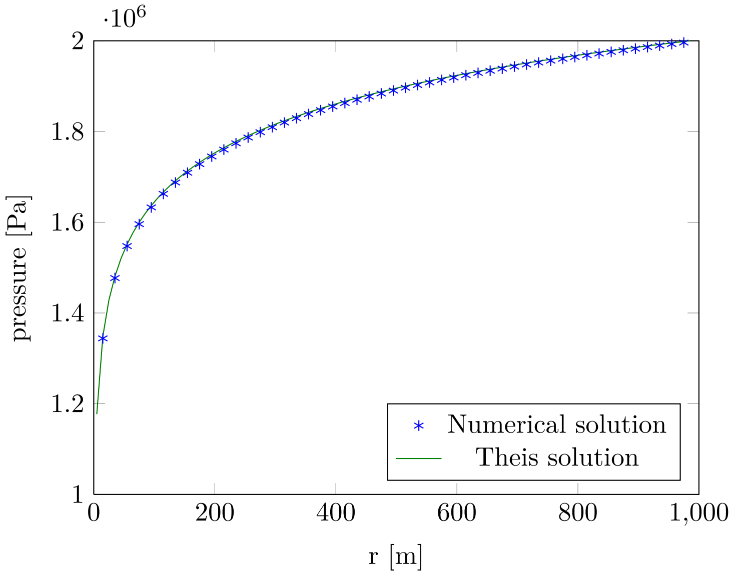

Figure 2: A comparison of the Theis solution and the numerical solution for a confined aquifer up to 1 km radius

is below 0.35%.

3.2 Natural convection in a ideal reservoir

The case study for hydrothermal systems is more complex than the previous one. The problem is modeled as two dimensional geothermal system, with constant temperature at its roots and at its surface. The system is assumed to have homogenous permeability of $\kappa = 10^{-10}$ and a temperture of 280 K at the bottom and 380 K at the top.

Figure 4 shows the temperature distribution in the reservoir resulting from the temperature difference between the roots of the system and the top. The figure shows irregular behaviour of upwards flowing regions and downwards flowing regions. This is even indicated more clearly if the velocity vectors are considered where hot plumes rise from the bottom to the top and drop again once they are cooled.

Figure 5 shows the density distribution, which drives the flow. Since it is a function of temperature it follows figure 4 closely. Lighter hotter plumes can be seen rising, while colder denser plumes fall back down.

4 Discussion

This paper illustrates the applicability of the OpenFOAM platform to take on current problems in geothermal reservoir modeling as well as flow in porous media in general. Because of the structure of the OpenFOAM libraries, the partial differential equations which describe the problems and the models themselves can be implemented in a consistent manner with minimal work.

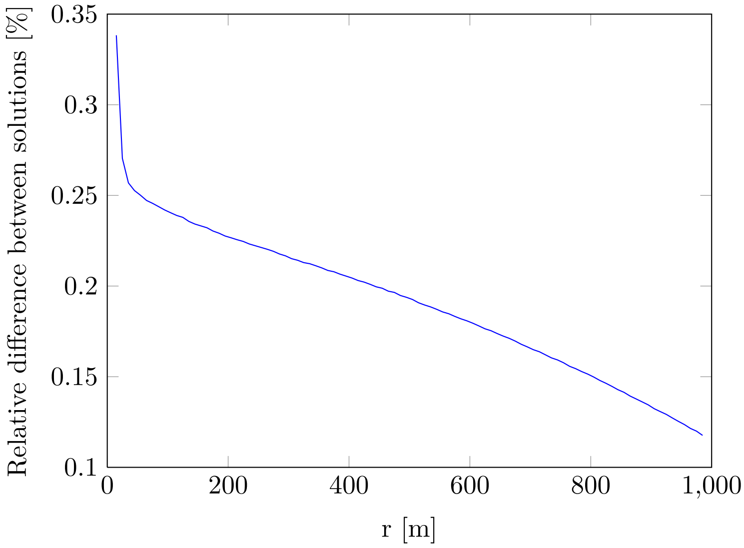

Figure 3: The relatice difference between the Theis solution and the numerical solution up to 1 km radius

However this work is still in progress, so there are many factors still unaccounted for. The hydrothermal model is for example not compressible, which makes it lack some features of the groundwater model. The next step in the project is to combine those two models, where the physical properties are also functions of pressure by applying a thermodynamic formulation such as IAPWS-IF97.

Currently the main focus of the research is to include phase changes in the model and account for a two phase mixture within some regions of the reservoir. The main challenge in this work is to ensure a stable solution despite the discontinuities in physical properties that arise as a result of phase changes. This has still not been resolved adequately and some instabilities are seen in two phase solutions, hence no results are shown here for such computations.

Despite those current issues, it can be proposed that the OpenFOAM platform is very promising for geothermal reservoir modeling. However, such further research and modeling work will always require comparison work, especially with well known and mature reservoir models.

On a whole, this approach in modeling geothermal reservoirs has several advantages over present methods. Since the libraries are highly customizable, wellbore-reservoir interaction can for example be modeled in a flexible way and adjusted to represent known data from measurements. Furthermore, standards such as IAPWS-IF97 for fluid properties can be implemented directly into the code, giving a more accurate description of hyrdothermal systems.

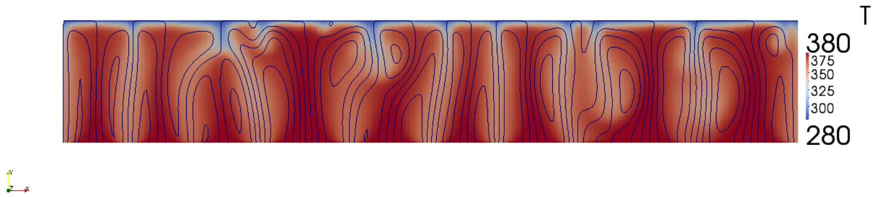

Figure 4: The temperature distribution and streamlines of a hydrothermal system

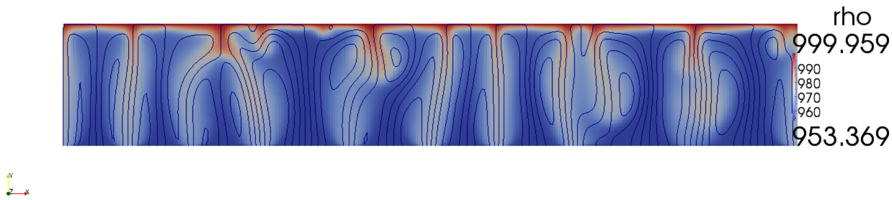

Figure 5: The density distribution and streamlines of a hydrothermal system

References

[1] Michael J. O'Sullivan. State of the art of geothermal reservoir simulation. *Geothermics*, 30(4):395–429, August 2001.

[2] Stanford Geothermal Program. Proceedings of the Special Panel on Geothermal Model Intercomparison Study. In *Report SGP-TR-42*, Stanford, CA, 1980.

[3] Karsten Pruess. TOUGH2 — A General Purpose Numerical Simulator for Multiphase Fluid and Heat Flow. Technical Report May, Lawrence Berkeley Laboratory, Berkeley, California, 1991.

[4] H G Weller, G Tabor, Hrvoje Jasak, and C Fureby. A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in Physics*, 12(6), 1998.

[5] CV Theis. The relation between the lowering of the piezometric surface and the rate and duration of discharge of a well using groundwater storage. *Transactions - American Geophysical Union*, 16:519–524, 1935.

[6] Miguel a. Rodrigues, Luis Padrela, Vitor Geraldes, José Santos, Henrique a. Matos, and Edmundo Gomes Azevedo. Theophylline polymorphs by atomization of supercritical antisolvent induced suspensions. *The Journal of Supercritical Fluids*, 58(2):303–312, September 2011.