

## MAT 264: Exercise 2

### Flow in Porous Media: Single and Two-phase flow Submission Deadline: 1 May 2016

Kindly send your files to [kundan.kumar@uib.no](mailto:kundan.kumar@uib.no)

The flow in porous media has several applications. We will consider single and two phase flow in this exercise. For these flow models, the challenges are due to: non linear possibly degenerate system of PDEs, highly heterogenous coefficients, and presence of multiple scales. The design of an appropriate numerical method to simulate such problems therefore requires careful considerations. This exercise is aimed at familiarising the students with some of the basic methods.

We will be using the MATLAB Reservoir Simulation Toolbox (MRST) which has been developed by SINTEF Applied Mathematics. The code can be downloaded following the link

<http://www.sintef.no/Projectweb/MRST/Downloadable-Resources/>

Also, have a look at the tutorials: for single phase flow

<http://www.sintef.no/Projectweb/MRST/Tutorials/Flow-Solver-Tutorial/>

and

<http://www.sintef.no/Projectweb/MRST/Tutorials/Flow-Solver-with-Capillary-Pressure/>  
for two phase flow.

#### Exercise 1 – Single phase flow

Let us consider an incompressible single phase flow model. Let  $\Omega \in \mathbb{R}^d, d = 2$  or  $3$  be an open, bounded domain with smooth boundary  $\partial\Omega$ . The equation describing the flow in  $\Omega$  is given by

$$\begin{aligned}\nabla \cdot u &= f, \\ u &= -\frac{K}{\mu}(\nabla p + \rho g \nabla z).\end{aligned}$$

Here  $K$  is permeability,  $\mu$  is the viscosity of the fluid,  $\rho$  is the density of the fluid,  $g$  is the acceleration due to gravity. The unknowns are pressure  $p$  and flux  $u$ . For mathematical purposes, we will choose the coefficients  $\rho$  and  $\mu$  equal to 1. We will also ignore the gravity effects ( $\nabla z = 0$ ). The above equation needs to be complemented with the boundary condition.

- a. Take  $\Omega := (0, 1) \times (0, 1)$ ,  $f = 0$ , choose  $K = 1$  and for boundary conditions, we choose

$$\begin{aligned}p &= 1 \text{ on } x = 0, \\ p &= 0 \text{ on } x = 1, \\ u \cdot n &= 0 \text{ on } \{y = 0\} \cup \{y = 1\}.\end{aligned}$$

Choose  $h = 0.1$  and solve for pressure and flux. Can you compute the exact solution using symmetry arguments? How does the numerical solution compare with the exact solution?

- b. With the same boundary condition and  $f$  as in [a.], choose  $K = (0.1 + 0.05 \sin(2\pi x/\varepsilon))^{-1}$  with  $\varepsilon = 0.02$ . Plot the permeability field. Choose  $h = 0.1$  and solve for pressure and flux. Can you compute the exact solution? How does the numerical solution compare with the exact solution? Plot the pressure,  $\partial_x p$  and flux as a function of  $x$  for fixed  $y = 0.5$ . What happens when  $\varepsilon$  becomes smaller. Study the convergence rate. Do you see some relationship of  $\varepsilon$  and mesh size  $h$ ? Interpret the results.

- c. We will construct an analytical solution by choosing an appropriate right hand side  $f$ . Let us assume that the pressure solution is given by:

$$p(x, y) = x(1 - x)y(1 - y).$$

For the permeability tensor we choose

$$K = \begin{bmatrix} 10 & 0 \\ 0 & 1 \end{bmatrix}.$$

Now construct  $f$  such that you get the given expression of  $p$  as the exact solution. Study the convergence rates for pressure and flux.

- d. We consider  $\Omega := (0, 1) \times (0, 1)$ ,  $f = 1$  and for the boundary conditions, choose:

$$\begin{aligned} p &= 1 \text{ on } y = 0, \\ p &= 0 \text{ on } y = 1, \\ u \cdot n &= 0 \text{ on } \{x = 0\} \cup \{x = 1\}. \end{aligned}$$

With  $\varepsilon$  as a parameter, let  $\Omega_f \subset \Omega$  be defined as:

$$\Omega_f := \{(x, y) : 0 < x < 1, 1/2 - \varepsilon < y < 1/2 + \varepsilon\}.$$

The absolute permeability field is given by:

$$K(x, y) = \begin{cases} K_f, & (x, y) \in \Omega_f \\ 1, & (x, y) \in \Omega \setminus \Omega_f \end{cases}$$

where

$$K_f = \begin{bmatrix} \varepsilon^\alpha & 0 \\ 0 & \varepsilon^\beta \end{bmatrix}.$$

We take four different cases for  $\alpha$  and  $\beta$ .

- $\alpha = 1, \beta = 1$ ;
- $\alpha = -1, \beta = 1$ ;
- $\alpha = 0, \beta = -1$ ;
- $\alpha = 0, \beta = 1$

We take different values of  $\varepsilon = 0.1, 0.05, 0.025, 0.0125$ . Plot the pressure profile and discuss the results.

### Exercise 2 – Two phase flow

The governing equations for two phase flow in  $\Omega \times [0, T]$ , with  $T$  denoting the final time are:

Conservation of total phase volumes

$$\nabla \cdot \vec{u} = f_2(s). \tag{1}$$

Darcy's law for total flow

$$\vec{u} = -\lambda(s)k\nabla p. \tag{2}$$

Mass (volume) balance:

$$\frac{\partial s_w}{\partial t} + \nabla \cdot \vec{u}_w = f_1(s). \tag{3}$$

Fractional phase flux

$$\vec{u}_w = -k\lambda_0 \nabla p_c + f_w(s) \vec{u}. \quad (4)$$

Here,  $p_c$  denotes capillary pressure,  $\lambda = \lambda_w + \lambda_o$  is the total phase mobility with phase  $\alpha = \{w, o\}$  mobility  $\lambda_\alpha = \frac{k_{r\alpha}}{\mu_\alpha}$ , with  $k_{r\alpha}$  being the relative permeability for phase  $\alpha$ . The fractional flow coefficient  $f_w = \frac{\lambda_w}{\lambda_w + \lambda_o}$ .  $f_2$  and  $f_1$  are source terms. The model is a global pressure formulation of the two-phase system.

- a. Run the tutorial at the MRST webpage: Flow Solver with Capillary Pressure  
<http://www.sintef.no/projectweb/mrst/tutorials/flow-solver-with-capillary-pressure/>.  
What is the boundary condition used in the tutorial? How is the uniqueness achieved? Which models have been used for the capillary pressures and relative permeability? What are the initial conditions?
- b. Write the discrete system for the two phase flow.
- c. Change the boundary condition to Dirichlet conditions for pressure  $p$  at the left and right sides of the domain and plot the solutions. What is the impact of capillary pressure?

### Exercise 3 – Iterative IMPES solution for two phase flow

The tutorial uses an IMPES (IMplicit Pressure, EXplicit Saturation) scheme to solve this two phase flow problem where there is no iteration between the pressure solve and the transport solve. Implement an iterative IMPES scheme by making necessary changes in the code. Take the problem in exercise 2a as the test case. How many iterations are needed to converge in the beginning of the simulation, at the end of it (choose two convenient times to observe this)? What happens to the iteration count when you refine the grid? What happens to the iteration count when you increase the flow rate from the well?

### Suggested reading

Knut-Andreas Lie, An Introduction to Reservoir Simulation Using MATLAB, User Guide for the Matlab Reservoir Simulation Toolbox (MRST) available online at

<http://www.sintef.no/contentassets/8af8db2e42614f7fb94fb0c68f5bc256/mrst-book.pdf>

Jan Martin Nordbotten, Michael A. Celia, Geological Storage of CO<sub>2</sub>: Modeling Approaches for Large-Scale Simulation, Wiley Publishers