Excerpt from the Proceedings of the COMSOL Conference 2010 Boston

Navier-Stokes Solutions for Flow and Transport in Realistic Porous Media

N. Abdussamie
American Bureau of Shipping (ABS), Tripoli, Libya, nabdussamie@eagle.org

Abstract: Many engineering applications involve the modelling of flow and transport through porous media, such as groundwater remediation and oil recovery. This paper illustrates how to set up and solve a problem using the Navier-Stokes equations in Cartesian coordinates with the help of FEMLAB COMSOL Multiphysics software. The present pore-scale model takes one of the two-dimensional micro-media images of realistic porous media prepared previously. The constitutive relation between permeability and properties of a realistic porous medium has also been determined numerically and the final results have been compared with those obtained by CFD FUENT software.

Keywords: Porous media, Navier-Stokes solutions, Multiphysics, CFD.

1. Introduction

It is typical to represent fluid flow in the subsurface as a continuum process using average or “continuous” properties for the bulk rather than detailing the shape and orientation of each solid particle within a porous medium. Inserting the bulk properties into an equation such as Darcy’s law gives an average flow rate for the total volume. While bulk approximations typically produce excellent estimates sufficient for considering flow over large areas, they miss the between-grain nuances that a close-up Navier-Stokes analysis describes (Heinrich et al., 1996).

In this work, the flow and transport problems in porous media are studied by using pore-scale modelling approach, which allows us to investigate microscale processes and their effects on macroscale behavior (Abdussamie, 2009). This can be accomplished in the present study through computational simulations based on a detailed description of the pore space. These numerical simulations have been quite successful in predicting permeability coefficients and validating well-known relations on real porous materials.

In addition, pore-scale modeling provides opportunities to study transport phenomena in fundamental ways because detailed information is available at the microscopic pore scale (Pan, 2003). Besides, pore-scale modelling offers an important tool to develop constitutive relations that are difficult and even impossible to be obtained by lab experiments. The basic strategy is to perform numerical experiments analogous to those performed in the laboratory.

However, the pore-scale simulation provides more versatility in choice of parameters, a greater variety of quantitative data, and more importantly, easier design of numerical experiments. The dramatic evolution of computational capabilities offers to us new opportunities for simulating larger domains and modelling a wider range of processes (Abdussamie, 2009; Pan, 2003). This makes pore-scale approaches potentially attractive for field applications like petroleum reservoir simulation as measurement tools to compute transport properties, such as permeability of a particular subsurface system. Even though reservoir engineers are usually not interested in the behavior at pore scale, it is considered more interestingly scientific subject to many researchers.

Andrade et al. (1999) investigated the origin of the deviations from the classical Darcy’s law by numerical simulation of the Navier-Stokes equations in two-dimensional disordered porous media. They applied the Forchheimer equation as a phenomenological model to correlate the variations of the friction factor for different porosities and flow conditions. They also observed a transition from linear to nonlinear behaviour which is typical of experiments. It was found in their study that such a transition can be understood and statistically characterized in terms of the spatial distribution of kinetic energy in the system.

However, this paper deals with the experiment of Sirivithayapakorn and Keller

(2003). They presented an experimental model to study the effect of colloid exclusion from areas of small aperture sizes, using direct observations at the pore-scale using a realistic micromodel of porous media. Sirivithayapakorn and Keller used scanning electron microscope (SEM) to obtain realistic images of porous medium to model groundwater contamination by colloid. Although their model was designed for groundwater flow, it is also reliable for modeling the oil/gas flow, since the pore and throat dimensions are about equal to those of a petroleum reservoir. As shown in Figure 1, their lab experiments were designed on the basis of scanning electron microscope (SEM) images of thinly sliced rock. The scale at bottom indicates that pore throat and body dimensions are on the order of 1-100 μm.

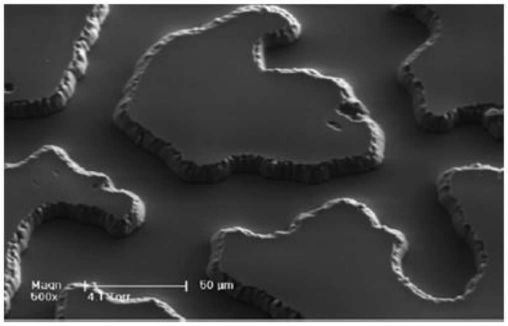

Figure 1. Scanning electron microscope (SEM) image of the repeat pattern in the silicon wafer (typical pore size 10–100 μm, pore throats 3–20 μm).

2. Governing Equations

The mathematical description for the detailed fluid mechanics in the interstitial pore space is based on the assumptions that we have steady state flow in isothermal conditions and the fluid is continuum, Newtonian, and incompressible. Thus, the continuity and Navier-Stokes equations are given as (Versteeg and Malalasekera, 1995):

$$\frac{\partial u}{\partial t} + \vec{\nabla}\cdot\left(u\vec{V}\right) = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\nabla^2 u \qquad (1)$$

$$\frac{\partial v}{\partial t} + \vec{\nabla}\cdot\left(v\vec{V}\right) = -g - \frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\nabla^2 v \qquad (2)$$

$$\vec{\nabla}\cdot\vec{V} = 0 \qquad (3)$$

where g denotes the acceleration due to the gravitational forces, ρ is the density of the fluid and ν is the kinematic viscosity. For fluid flow in a horizontal layer g can be neglected.

According to Darcy's law (Darcy, 1856) for flow in a porous medium, the absolute permeability of the present model can be numerically calculated. The seepage or superficial velocity in horizontal direction is (Fanchi, 2006),

$$U = -\frac{K}{\mu}\nabla P \qquad (4)$$

From Equation (4), K (absolute permeability) can be easily estimated.

Darcy's Law is valid for the slow flow (inertial effects can be neglected) of a Newtonian fluid through a porous medium with rigid solid matrix (Heinrich et al., 1996). Accordingly, the equations (1), (2) and (3) are to be solved numerically. Both finite volume and finite element schemes are utilized to compare their numerical solutions. Upon obtaining the numerical solutions for the system, the absolute permeability can be easily determined from formula (4) based on the approximation of ($U \approx u$).

3. Model Formulation with COMSOL Multiphysics

As mentioned above, the principle of pore-scale modelling in general is composed of two major steps. The first involves the detailed identification and specification of the porous medium morphology. Once the scanning process done, the images are transferred to Drawing Exchange Format (DXF) files, which can be easily imported into COMSOl and/or AutoCAD software, and then the model can be treated by using CFD techniques.

The present geometry of the pore scale model takes one of the 2D micro-image of a realistic porous medium and covers 640 x 320 μm. Figure 2 shows the detailed geometry of the present model and the direction of flow.

Flow and transport problems can be solved by using various solvers. COMSOL Multiphysics software has the capabilities to solve such problems through fluid mechanics module (FEMLAB 3.2, 2005). To ensure good results, the decision was also to use computational fluid dynamics techniques to solve these problems by utilizing FLUENT solver (FLUENT, Inc., 2005), in order to simulate the variety of fluid flow in multiple pores. Accordingly, the flow of a single

Newtonian fluid (water) in the void space of a porous medium is described on the microscopic level by the Navier–Stokes system of equations with appropriate boundary conditions. However, the void space configuration is usually not known in such detail to make this description feasible. Moreover, a numerical simulation on that level is beyond the capabilities of today's computers and methods.

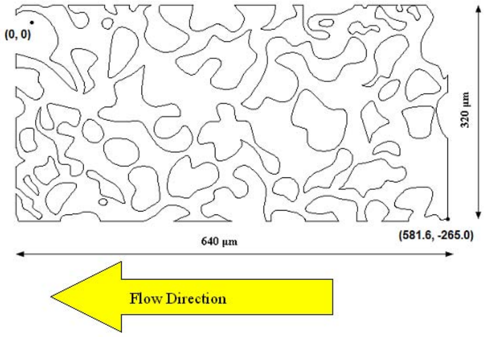

Figure 2. The details of the model geometry used.

Boundary conditions designated within the micromodel are presented Table 1. Flow rate was determined by the pressure difference between two sides of the micromodel. The kinematic viscosity and the density of water are 10^{-6} m^2/s, 1000 kg/m^3, respectively.

Table 1. Boundary conditions designated within the micromodel.

Boundary	Conditions
Inflow (Right Side)	715 Pa
Outflow (Left Side)	0 Pa
Upper and Lower Sides	Symmetry
Grain Surface	No slip

In case of using COMSOL solver, the mesh generation process is a simple function. While to create the finite-volume mesh for the FLUENT simulations, the pre-processor GAMBIT is to be utilized, which is an advanced CFD pre-processor designed for geometry description and meshing, all within a single user interface. GAMBIT imports the geometry shown above and does not mesh any unneeded regions and so does COMSOL.

The objective of this paper is to realize a good comparison between the above mentioned numerical solvers, COMSOL and FLUENT. And so our strategy is to get comparatively acceptable results using the different types of numerical mesh. GAMBIT has two types of cells in 2D (triangular and quadrilateral). Because grid type and resolution can significantly affect on both the quality of the results and the calculations speed, a short series of tests with various grid resolutions was performed for each of the simulation presented in this paper. The general mesh structure we used was generated using an automatic mesh formation in GAMBIT, with an unstructured mesh (Farrashkhalvat and Miles, 2003) in the pore body. Three cases at different number and/or type of grid cells were tested to ensure good results with those obtained by COMSOL Multiphysics. Two levels or cases have a particular number of quadrilateral cells and one level uses triangular cells.

4. Results and discussion

However, the current applications of both finite element and finite volume methods to flow through microscopic porous media have been limited to solving time-independent Navier-Stokes equations in single-phase systems. Applying the numerical method starts by discretizing the gradients of equations (1), (2) and (3) on the unstructured meshes.

Both COMSOL and FLUENT solutions predicted by a Navier-Stokes analysis for the relative velocities in the pore spaces of a micro-scale porous medium are shown as x-y plots. Velocities are higher in the narrowest pores. The fluid velocities tend to decrease in stretches where the cross-sectional area for the flow increases. The velocity and pressure distributions can also be shown as contours.

The domain plot in Figure 4 shows a comparison of the x-velocity at the outlet boundaries of the micromodel. The velocities are negative because the flow is moving in the negative x-direction. Though there is a difference in the results obtained by using finite volume solver FLUENT, the numerical model is shown to be in a good agreement to the three cases.

Even though test case one contains only 44982 quadrilateral cells, it gives nearly the same result of which test case two does, that consists of 114944 quadrilateral cells. Figure 4 reveals the highest velocities tend to occur in narrow pores with high pressure drops, as we might expect. This close-up view near the exit reveals that high velocities also develop in wide channels where pressure gradients are relatively shallow but multiple "tributaries" combine.

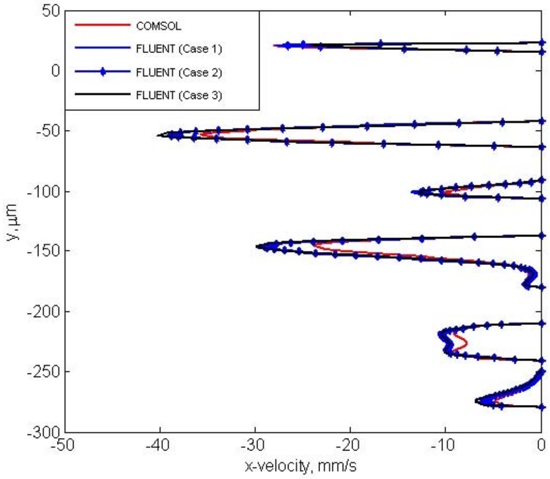

Figure 3. Comparison of x-velocities along the outlet boundaries obtained by COMSOL and FLUENT.

Additionally, to ensure a good comparison the model has been tested with the help of 2D triangular cells (case 3) consisting of 64630 cells and compared to COMSOL results. The comparison showed approximately the same result obtained by using the two previous cases.

As the absolute permeability has a significant effect on the macro-scale behaviour, it is calculated through Darcy's law based on the x-velocity along outlet boundaries obtained from Figure 3, since the x-velocity values of test case two of FLUENT solutions have been chosen to perform this step. The maximum permeability at the corresponding maximum velocity of 38.79 mm/s is found 3.47×10^{-11} m^2 and the average absolute permeability along the outlet boundaries is almost 9.1×10^{-12} m^2 which is similarly considered to the typical value obtained experimentally for consolidated rocks where the petroleum existed (Fanchi, 2006).

Eventually, another comparison can be presented regarding the absolute permeability in the horizontal direction along the outlet boundaries of the model. Figure 4 shows the absolute permeability of the pore-scale model as a function in y-coordinate, it's obvious that both numerical results are matching along the vertical coordinate.

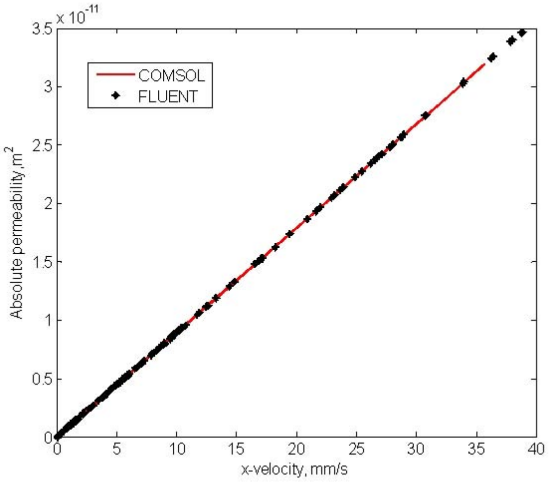

Figure 4. The absolute permeability versus the velocity in x-direction along outlet boundaries using COMSOL and FLUENT

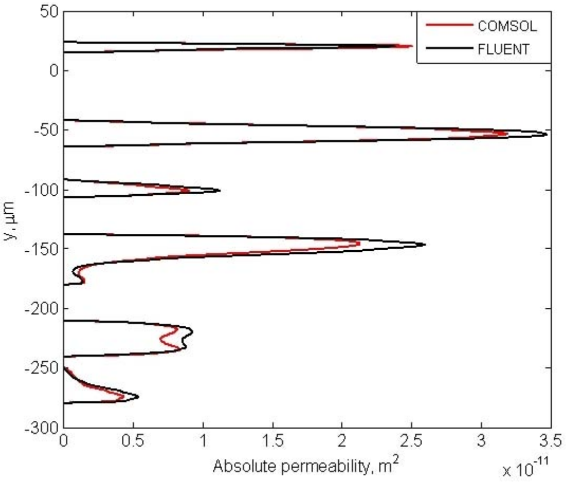

Figure 5. The absolute permeability as a function in y-direction along outlet boundaries using COMSOL and FLUENT.

5. Conclusions

The motion of the fluid has been studied and tested relying on the Navier-Stokes equations, and by implementing the model with both finite-element and finite volume techniques and numerical analysis.

From the numerical simulations of the behaviour of single phase flow, it can be stated that the obtained results using COMSOL Multiphysics solver showed a good agreement as compared to those obtained by CFD FLUENT solver. A slight difference between the two numerical methods' results was interpreted due to the discretization scheme used as well as the number and the type of cells.

The modelling predictions were also provided based on the single-phase flow. These predictions would provide the results that could be related to real-world problems.

6. References

1. Abdussamie, N., *Flow and Transport Problems in Porous Media Using CFD*, MSc Thesis, Arab Academy, Mechanical Eng., Egypt (2009).
2. Andrade, J. S. et al., *Inertial Effects on Fluid Flow through Disordered Porous Media*, Center for Polymer Studies and Physics Department, Boston University. USA (1999).
3. Darcy, H., *Les Fontaines Publiques de la Ville de Dijon, Dalmont*, Paris, France (1856).
4. Fanchi, J. R., Principles of Applied Reservoir Simulation, 3rd Edit., Elsevier, CANADA (2006).
5. Farrashkhalvat and Miles, J.P., *Basic Structured Grid Generation: with an introduction to unstructured grid generation*, Butterworth-Heinemann, Elsevier Science, London, England (2003).
6. FEMLAB 3.2 - *Fluid Dynamics Module,* COMSOL (2005).
7. FLUENT 6.2, *User's Guide*, Fluent Inc., Lebanon, NH (2005).
8. Heinrich, J.C. et al., *Mesh generation and flow calculations in highly contorted geometries*, Elsevier, Comput. Methods Appl. Mech. Engrg., 130, p.p 79-92 (1996).
9. Pan, C., *Use of Pore-scale Modeling to Understand Flow and Transport in Porous Media*, PhD Thesis, the University of North Carolina in the Department of Environmental Sciences & Engineering, USA (2003).
10. Sirivithayapakorn, S. and Keller, A., *Transport of colloids in saturated porous media: A pore-scale observation of the size exclusion effect and colloid acceleration*, Water Resour. Res., 39(4), 1109, doi: 10.1029/2002WR001583 (2003).
11. Versteeg, H. K. and Malalasekera, W., *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*, Longman Group Ltd., (1995).