Pore-Scale numerical simulation of two phase flow of newtonian and viscoelastic fluids

J. PINILLA a, C-H BRUNEAU a

a. IMB Université Bordeaux 1 and INRIA M C 2 Team
351 Cours de la Libération, 33405 TALENCE

Abstract :
This work is motivated by the need for better understanding the Polymer Enhanced Oil Recovery technique at the pore-scale. We consider two phase immiscible and incompressible fluids in a microchannel network. The Oldroyd-B rheological model is used to capture viscoelastic behavior. The interface between the two fluids is followed by a Level-Set method. The dynamics of the triple point is modeled by the Cox’s theory. Numerical simulations in a two dimensional microfluidic pore network are presented.

Mots clefs : Two-phase flow ; Level-set method ; Oldroyd-B model

1 Introduction

This work is motivated by the need for better understanding the polymer Enhanced Oil Recovery (EOR) technique at the pore-scale. In the polymer EOR, polymers are injected into the reservoir to modify fluid properties to make them more favorable for oil recovery.

In the oil recovery industry, it is very important to understand multiphase flow processes at the pore-scale to better describe its behavior at the macroscopic scale. Microchannels network are frequently used as a laboratory model to reconstruct the flow conditions in a porous medium at pore-scale. Experimental results on two-phase flow in microfluidic devices, composed of straight microchannels, with controlled size heterogeneities can be found in [5].

The modeling of two phase flow at microfluidic scale has already been studied in the past (for example [11], [12]). At this scale, the flow is generally laminar and the movement of the interface between the two fluids is controlled by the effect of the surface tension. The works of [11] and [12] deal with the modeling of two immiscible and newtonian fluids in microfluidic, the first one in 2D and the second one in 3D. In the context of ink jet plotters, a two phase flow with a newtonian and a viscoelastic fluid, has been modeled and compared to experiments, [13].

One of the most important aspects to consider in two phase flow simulation, is the moving contact line problem. Many attempts to simulate the dynamic of the contact line have been developed, a very good review of these methods can be found in [7]. The dynamic contact angle model used in this work
is based on the theoretical analysis of Cox [6]. Cox provides a general hydrodynamic description of a moving contact line, that links the triple point velocity to the dynamic contact angle.

Our aim is to describe a pore scale numerical model to simulate a two phase flow of Newtonian and viscoelastic fluids into a porous medium, incorporated with a dynamic contact angle model which describes the fluid-fluid-wall dynamics. Then, we perform simulations with realistic parameters to compare them with experimental results.

We consider two phase immiscible fluids in a microchannel network. The low Reynolds number due to the small dimensions and small velocity at this scale allow us to use the incompressible Stokes equations to describe the Newtonian fluid flow. The Oldroyd-B rheological model is used to capture the viscoelastic behavior [2]. In order to perform numerical simulations in a complex geometry like a microchannel network, we use a penalization method [1]. To follow the interface between the two fluids, we use the Level-Set method. In the Level-Set method, a function \( \phi \) is used to implicitly represent the interface between the two fluids [4]. In our algorithm, the interface is the zero level of \( \phi \). The dynamic contact angle model used in this work is based on the theoretical analysis of Cox [6].

2 Modeling two phase flow at pore scale

2.1 Domain description

In the Figure 1(a) we show a microchannel network which represents a porous medium. This microchannel network is formed by channels with diameter \( d_c \), separated by a distance \( d \). We are going to focus our attention to a little part of this microchannel network, for example the zone in the dotted lines. A zoom of this zone is presented in the Figure 1(b).

Let \( \Omega \) be the domain shown in the Figure 1(b), we denote by \( \Omega_f \) the fluid zone and \( \Omega_s \) the reunion of solid obstacles. The boundary of \( \Omega_f \) is \( \partial \Omega_f = \partial \Omega_s \cup \Gamma_D \cup \Gamma_P \cup \Gamma_N \). Here, \( \Gamma_D \) denotes the inlet boundary where a constant velocity is imposed, \( \Gamma_P \) is the part of the boundary where we specify periodic boundary conditions and \( \Gamma_N \) is the open boundary where we impose non reflecting boundary conditions [3].

2.2 System description

Under the hypothesis presented in Sec. 1, the governing equations are:
\begin{equation}
\begin{aligned}
\nabla \cdot \mathbf{U} &= 0 \\
-\nabla p + \nabla \cdot (2\eta(\phi)\mathbf{D}) - \nabla \cdot \mathbf{\tau}_p - \gamma\kappa n\delta_{\Sigma} - \frac{\mathbf{U}}{K} &= 0 \\
\mathbf{\tau}_p + \lambda(\phi) \left( \frac{\partial \mathbf{\tau}_p}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{\tau}_p - (\nabla \mathbf{U})\mathbf{\tau}_p - \mathbf{\tau}_p(\nabla \mathbf{U})^t \right) + \frac{\mathbf{\tau}_p}{K} &= 2\eta_p(\phi)\mathbf{D} \\
\frac{\partial \phi}{\partial t} + \tilde{\mathbf{U}} \cdot \nabla \phi &= 0
\end{aligned}
\end{equation}

In the system above, \( \mathbf{U} \) is the velocity field, \( p \) is the pressure, \( \mathbf{\tau}_p \) is the polymeric stress tensor and \( \mathbf{D} \) is the rate of deformation tensor defined by \( \mathbf{D} = \nabla \mathbf{U} + (\nabla \mathbf{U})^T \). In the Stokes equation, \( \gamma \) is the surface tension coefficient, \( \kappa = \nabla \cdot n \) is the curvature of the interface, \( n = \frac{\nabla \phi}{|\nabla \phi|} \) is the normal to the interface and \( \delta_{\Sigma} \) is the Dirac delta function which is zero everywhere except at the interface \( \Sigma \). The discontinuous fields \( \eta(\phi), \eta_p(\phi) \) and \( \lambda(\phi) \) are defined by:

\[
\eta(\phi) = \begin{cases} 
\eta_1 & \text{if } \phi < 0 \\
\eta_2 & \text{if } \phi > 0 
\end{cases} \quad \eta_p(\phi) = \begin{cases} 
\eta_{p1} & \text{if } \phi < 0 \\
0 & \text{if } \phi > 0 
\end{cases} \quad \lambda(\phi) = \begin{cases} 
\lambda_1 & \text{if } \phi < 0 \\
0 & \text{if } \phi > 0 
\end{cases}
\]

where \( \eta_1 \) is the solvent dynamic viscosity in the fluid 1, \( \eta_2 \) is the dynamic viscosity of fluid 2, \( \eta_p \) is the polymer contribution to the solution viscosity, \( \lambda_1 \) is the characteristic relaxation time and \( \phi \) is the level set function, which is advected with the flow.

The terms \( \frac{\mathbf{U}}{K} \) and \( \frac{\mathbf{\tau}_p}{K} \) added in Stokes and tensor equations, are the penalization terms. Here, \( K \) can be considered as a non-dimensional permeability coefficient. It is set to a very large value in the fluid zone (e.g. \( 10^{16} \)) and to \( 10^{-8} \) in the solid zone. Consequently we recover the original Stokes and tensor equations in the fluid, and we enforce \( \mathbf{U} \) et \( \mathbf{\tau}_p \) to vanish in the solid zone [1].

The velocity \( \tilde{\mathbf{U}} \) used in the advection equation comes from the resolution of the Stokes equations. At the wall, near the triple contact line, we modify the tangential component of \( \mathbf{U} \) according to the contact angle.

### 2.3 Contact model

At the triple point, where Fluid 1 and Fluid 2 meet at the solid wall, we adopt a contact angle model based on the work of Yu et al. [13] and the theoretical analysis of Cox [6]. First, we define \( \theta_d \), as the angle made by the interface and the solid wall, \( \theta_a \) the advancing contact angle, \( \theta_r \) the receding contact angle and \( v_b \) is the tangential velocity of the interface at the triple point. If \( \theta_d > \theta_a \) and \( v_b > 0 \), the triple point is allowed to move towards the Fluid 2. If \( \theta_d < \theta_r \) and \( v_b < 0 \), the triple point is allowed to move towards the Fluid 1. Otherwise, the triple point is not allowed to move.

Cox provides a general hydrodynamic description of a moving contact line, that relates the triple point velocity \( v_d \) to the dynamic contact angle \( \theta_d \) by:

\[
v_d = \gamma g(\theta_d) - g(\theta_o) \left/ \eta_1 \ln \left( \frac{\xi}{L} \right) \right.
\]

where \( \gamma \) is the interfacial tension, \( \theta_o \) can be \( \theta_a \) or \( \theta_r \), \( \eta_1 \) is the viscosity of Fluid 1, \( \frac{\xi}{L} \) is the relation between macroscopic and microscopic scales and

\[
g(\theta) = \int_0^\theta \frac{d\theta}{f(\theta, q)}
\]
where \( q = \eta_1 / \eta_2 \) and

\[
f(\theta, q) = \frac{2 \sin \theta \left( q^2 (\theta^2 - \sin^2 \theta) + 2q (\theta (\pi - \theta) + \sin^2 \theta) + ((\pi - \theta)^2 - \sin^2 \theta) \right)}{q (\theta^2 - \sin^2 \theta) ((\pi - \theta) + \cos \theta \sin \theta) + ((\pi - \theta)^2 - \sin^2 \theta) (\theta - \cos \theta \sin \theta)}
\]  

(4)

When the triple point is allowed to move, the velocity is calculated from the Eq. (2). This velocity is imposed on the edge of the wall in a close vicinity of the triple point.

3 \ Discretization

The Stokes equation is discretized in space on a rectangular cartesian staggered grid using a finite volume method for a mesh of MAC (Mark And Cell) type. The incompressibility constraint is treated with the Augmented Lagrangian method.

The level set equation is discretized in time by an explicit Euler method. The space discretization is done through a WENO-5 scheme on a grid that is twice finer than those used for solving Stokes equation. On this new grid, the velocity components are calculated by linear interpolation. Periodically, the level set function is re-initialized as a signed distance function by a Fast-Marching method \cite{8}. The Oldroyd-B model is solved in two steps: the first consists in solving a local problem, then we solve a convection equation.

The position of the contact line at time \( t^{n+1} \), \( X^{cl}(t^{n+1}) \), is obtained from \( X^{cl}(t^{n+1}) = X^{cl}(t^n) + v_d \Delta t \) \cite{10}. And the angle \( \theta_d \) is obtained by using the following expression:

\[
\theta_d = \pi - \arctan \left( \frac{dx}{X^* - X^{cl}} \right)
\]  

(5)

where \( X^* \) is the position of the interface two cells next to the triple point in the fluid part. Once the contact angle is known, we can calculate \( g(\theta) \) with the Eq. (3).

4 \ Results

Here is given a numerical example of the two phase flow simulation when a newtonian fluid pushes another one. The domain dimensions are \( 50 \mu m \times 50 \mu m \). The interfacial tension \( \gamma \) is such that the capillary number \( Ca = 0.01 \). The capillary number \( Ca = \eta_1 V/\sigma \), where \( \eta_1 \) is the viscosity of fluid 1, \( V \) is the velocity at inlet and \( \sigma \) is the surface tension coefficient. The advancing angle \( \theta_a = 130^\circ \). The dynamic viscosities are \( \eta_1 = 1 \times 10^{-3} \) Pa \cdot s and \( \eta_2 = 1.34 \times 10^{-3} \) Pa \cdot s.

In Figure 2 we show the fluid velocity field and the evolution of the interface at different times. The solid line in the fluid is the zero level set of \( \phi \), that is, the interface between the two fluids.

The time evolution of the contact angle from the moment that the interface touches the wall, is shown in the Figure 3(a), and the contact line velocity, in the Figure 3(b). The triple point is steady until \( \theta_d \) reaches \( \theta_a = 130^\circ \), then it moves according to the increase of the angle \( \theta_d \) that reaches an asymptotic value.

The Figure 4 illustrates a plot of the dynamic contact angle \( \theta_d \) versus log_{10}(Ca). The qualitative behavior is expected, as we increase the capillary number \( Ca \), the dynamic contact angle \( \theta_d \) will also increase.

The solid curve is drawn by using Equation (2) with \( x = 0.3 \) and \( L = 10^{-10} \) which are values presented in \cite{9}. The Figure 4 clearly shows that the numerical results verify the Cox’s law.
5 Conclusions

A numerical technique for the simulation of two fluids in a microchannel network has been developed. This technique involves a level set approach, a penalization method and a numerical contact angle model based on the Cox’s relation. The 2D simulations are performed with realistic parameters. The qualitative and quantitative behavior of contact angle $\theta_d$ are verified: $\theta_d$ increases as we increase the Capillary number Ca and $\theta_d$ satisfies the Cox’s law.
Références


