Flow of Cells in Microchannels

Y.F. YAP
School of Mechanical and Aerospace, Nanyang Technological University,
Singapore 639798
pj396758@ntu.edu.sg

D. Li
Department of Mechanical, Vanderbilt University,
Nashville, TN, USA
donqing.li@vanderbilt.edu

J.C. CHAI
School of Mechanical and Aerospace, Nanyang Technological University,
Singapore 639798
mckchai@ntu.edu.sg

Abstract. This article investigates the motion of a cell in a carrier fluid flowing along a two-
dimensional channel. The cell, assumed elliptical, is modeled as a highly viscous fluid with
rigid body motion. A fixed grid approach is used. A combined formulation with one set of
governing equations to treat both phases is employed. The cell-fluid interface is tracked by a
distance function. The finite volume method is employed to solve the governing equations. For
demonstration purpose, the result for the flow of a single elliptical cell in an obstructed channel
is presented.

Keywords: cell motion

1. Introduction

In lab-on-a-chip applications, cells are often carried by an immiscible carrier fluid through
microchannels of various geometries. The carrier fluid is either pressure or electrical field driven. In
the course of their motion, these cells might agglomerate and clogging the microchannels eventually.
This is highly undesirable. Therefore, a sound understanding of the motion of these cells carried by the
carrier fluid is essential to ensure them to be transported reliably through various microchannels.

Figure 1 shows N cells flowing in a fluid in a two-dimensional channel. The cells are assumed to be
either of circular or elliptical shapes to simplify the analysis. The surfaces of the cells act as moving
boundaries for the flowing fluid. The cells undergo both translational and rotational motions under the
influence of flowing fluid. This is a transient process where the motions of the cells and the flow field
are strongly coupled. Moving mesh methods[1][3] and fixed mesh methods have been proposed for such
simulations. In the moving mesh method, the interface is sharply described by a boundary-conforming
Remeshing, a computationally expensive operation, is required frequently. In the fixed mesh methods, a fixed grid is employed for the whole computational domain for all time. Remeshing is not required. This is the main advantage of fixed mesh method. Fixed mesh methods include the immersed boundary method\(^4\), fictitious domain approach\(^5\), fictitious boundary method\(^6\) and Cartesian Grid method\(^7\).

In this article, a fixed mesh method is presented. A combined formulation with one set of governing equations to treat both phases is employed. The fluid-cell interface is identified by a distance function-like expression. The Finite Volume method\(^6\) is employed to solve the governing equations. Only 2-dimensional situations are considered in this article for demonstration purpose. Extension to 3-dimensional situation is straightforward. The remaining part of the paper is divided into three sections. The theoretical model of the present approach will first be presented. Results and discussions are presented and discussed in the subsequent section. Finally, concluding remarks are given.

2. Theoretical Model

2.1. Mathematical Formulation

The two-phase domain of interest consists of a fluid and \(N\) cells as shown in Fig. 1. The cell-fluid interfaces \(\Gamma\) are represented by a global signed distance function \(\phi\), defined as the shortest signed normal distance from the interface. \(\phi\) is defined as

\[
\phi(\vec{x}, t) = \begin{cases}
-1, & \text{if } \vec{x} \in \text{fluid region} \\
0, & \text{if } \vec{x} \in \Gamma \\
+1, & \text{if } \vec{x} \in \text{cell regions}
\end{cases}
\]  

where \(\vec{x}\) and \(t\) are the position vector and time respectively.

Each of the cell is represented by their own distance function \(\phi_i\), \(i = 1, \ldots, N\). For the \(i\)-th cell, \(\phi_i < 0\) only in that particular cell. The cell is assumed elliptical. Its location and orientation are shown in Fig. 2. Surfaces of constant \(\phi_i\) are shown as the dashed ellipses.

![Fig. 1. Flow of cells in a fluid in a two-dimensional channel.](image1)

![Figure 2: Location and orientation of the \(i\)-th elliptical cell.](image2)
The global distance function on $\phi$ is formed from all the $\phi$ as
\[ \phi = \text{MIN} \{ \phi_1, \phi_2, \ldots, \phi_N \}. \] (2)

The Navier-Stokes equations for unsteady incompressible laminar flow are
\[ \nabla \cdot \vec{u} = 0. \] (3)
\[ \frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} = - \frac{1}{\rho} \nabla p + \frac{\mu}{\rho} \nabla^2 \vec{u} + \vec{g}. \] (4)

where $\vec{u}$, $\vec{g}$, $p$, $\rho$ and $\mu$ are the velocity vector, body force per unit mass, pressure, density and viscosity respectively. The fluid properties are calculated as
\[ \alpha(\phi) = \begin{cases} \alpha_p, & \phi < 0 \\ \alpha_f, & \phi > 0 \end{cases}. \] (5)

where $\alpha$ can be density or viscosity. The subscripts $p$ and $f$ denote cells and fluid respectively. The cells are modeled as a highly viscous fluid with rigid body motion.

The position and orientation of the $i$-th cell are governed respectively by
\[ m_i \frac{d}{dt} \left( \frac{d \vec{x}_{c,i}}{dt} \right) = \vec{F}_i + \vec{G}_i. \] (6)
\[ I_i \frac{d}{dt} \left( \frac{d \theta_i}{dt} \right) = \vec{T}_i. \] (7)

where $m_i$, $I_i$, $\vec{x}_{c,i}$, $\theta_i$ are mass, moment of inertia, position vector and orientation of the $i$-th cell. $\vec{F}_i$, $\vec{G}_i$ and $\vec{T}_i$ are respectively the resultant surface force, body force and torque. The initial conditions are
\[ \vec{x}_{c,i}(t=0) = \vec{x}_{c,i0}, \quad \theta_i(t=0) = \theta_{i0}. \] (8)

The resultant surface force $\vec{F}_i$ and the torque $\vec{T}_i$ acting on the $i$-th cell are given by
\[ \vec{F}_i = \int_{\Gamma_i} \sigma \cdot \hat{n} dA, \] (9)
\[ \vec{T}_i = \int_{\Gamma_i} (\vec{r} - \vec{x}_{c,i}) \times \sigma \cdot \hat{n} dA. \] (10)

where $\Gamma_i$, $dA$, $\sigma$ and $\hat{n}$ are the interfacial area, the infinitesimal interfacial area, the stress tensor and the unit outward normal of the interface respectively.

2.2. Boundary conditions

No slip condition is applied at the channel walls. The inlet velocity $u_{in}$ is specified. The outlet velocities $u_{out}$ are calculated to ensure mass conservation. The interface $\Gamma_i$ of every cell serves as the moving boundaries for the fluid. An inner layer of the $i$-th cell, defined as $-\varepsilon \leq \phi_i \leq 0$, is set to be in rigid body motion, or mathematically as
\[ \vec{u} = \vec{U}_i + \Omega_i k \times (\vec{x} - \vec{x}_{c,i}). \] (12)

where $\vec{U}_i$ and $\Omega_i$ are respectively the translational velocity vector and angular velocity. The velocity in the remaining region of the cell defined by $\phi_i < -\varepsilon$ is forced to be zero. These are done using the internal control volume (CV) approach of Patankar.
2.3. Solution procedure

The solution procedure can be summarized as:
1. Define $\tilde{x}_{c,i0}$ and $\theta_{i0}$ for $i = 1, \ldots, N$.
2. Construct the global $\Phi$.
3. Calculate $\rho$ and $\mu$ using Eq. (5).
4. Solve Eqs. (3) and (4) for $\tilde{u}$ and $p$.
5. Calculate $\tilde{F}_i$ and $\tilde{T}_i$ for $i = 1, \ldots, N$ using Eqs. (9) and (10).
6. Calculate $\tilde{x}_{c,i}$ and $\theta_i$ for $i = 1, \ldots, N$ using Eqs. (6), (7) and (8).

Repeated Steps (2) through (6) for all time steps. In step (4), the flow field is calculated using the SIMPLER algorithm with a Finite Volume method on a staggered grid.

3. Results and Discussions

The present approach is validated against flows around a settling circular cylinder under gravity. The present solution agrees well with that of the published result. This is not presented due to limited space. Due to this constraint too, only the result for the flow of an elliptical cell in an obstructed channel is presented.

3.1. Flow of an elliptical cell in an obstructed channel

An elliptical cell of mass $m=1$ with a semi-major $a=0.2$ and semi-minor $b=0.1$ is carried by a flow in an obstructed channel. The detailed geometry of the channel is shown Fig. 3. It has a length $L=2.5$ and a height $H=1.0$. The inlet velocity $u_M$ is uniformly set to 1. The density and the viscosity of the fluid are respectively $\rho=1$ and $\mu=1$. At $t=0$, the cell is located at $(0.4, 0.35)$. Figure 3 shows locations of the cell along the channel at different $t$. Solutions for meshes of $125 \times 50$ CVs with $\Delta t = 0.0050$ and of $250 \times 100$ CVs with $\Delta t = 0.0025$ are superimposed. A mesh of $125 \times 50$ CVs with $\Delta t = 0.0050$ is sufficient to achieve spatial and time independent solution.

![Fig. 3. Motion of an elliptical cell carried by a flow in an obstructed channel.](image)

The elliptical cell undergoes both translational and rotational motions. Generally, the fluid particles impinged on the surface of the cell transfer a portion of their momentum to it, thus creating a surface force. The momentum transferred to the cell is not uniform over its surface, so is the surface force. This gives rise to a net torque acting on the cell. While the resultant surface force is responsible for the translational motion, the net torque induces rotational motion.
The trajectory and orientation of the cell in the course of its motion are shown in Fig. 4. The trajectory is the locus of the centroid. The orientation $\theta$ of the cell is described by the angle between the major axis of the cell and abscissa $x$. It is measured in terms of radian. $\theta$ is negative in this case since the cell is rotating clockwise, in accordance to the sign convention adopted in Fig. 2.

From $t=0$ to 0.3, the cell moves forward and upward. The first obstruction blocks the fluid below the cell. Therefore, the fluid below the cell is flowing slower than that above the cell. A pressure difference is thus created. This generates lift and moves the cell upward. The cell does not rotate until roughly $t=0.3$. The velocity vectors around the cell at $t=0.3$ and 0.4 are shown in Fig. 5. Only one in every three velocity vectors is shown to avoid overcrowding the figure. The fluid particles at the lower rear part of the elliptical cell (circled with thick dashed lines) flow with a larger velocity. These higher momentum fluid particles impinge on this part of the cell, transferring a larger amount of momentum to the cell. This generates a net torque that rotates the cell clockwise. When the cell approaches the second obstruction, similar phenomenon occurs. For $t=0.8$ to 0.9, faster flowing fluid particles impinge on the circled portion of the cell as shown in Fig. 11. A net clockwise torque is therefore created. The cell is further rotated clockwise.

4. Concluding Remarks
In this article, a fixed mesh approach to simulate two-phase cell-fluid flow is presented. The cells are modeled as a highly viscous fluid with rigid body motion. With the present approach validated, the motions of cells carried by an incompressible fluid in two-dimensional channel were investigated. The present approach can be extended to flow of multiple cells of different sizes and shapes in channels of different geometries.

References

[1] J. Feng, H.H. Hu and D.D. Joseph, Direct simulation of initial value problems for the motion of solid bodies in a Newtonian fluid Part 1: Sedimentation, *J. Fluid Mech.* 261 (1994) 95-134.

[2] J. Feng, H.H. Hu and D.D. Joseph, Direct simulation of initial value problems for the motion of solid bodies in a Newtonian fluid Part 2: Couette and Poiseuille flows, *J. Fluid Mech.* 277 (1994) 271-301.

[3] P. Phutthavong and I. Hassan, Transient performance of flow over a rotating object placed eccentrically inside a microchannel – numerical study, *Microfluid Nanofluid* 1 (2004) 71-85.

[4] C.S. Peskin, The immersed boundary method, Acta Numerica 11 (2002) 479-517.

[5] R. Glowinski, T.W. Pan, T.I. Hesla and D.D. Joseph, A distributed Lagrange multiplier/fictitious domain method for particulate flows, *Int. J. of Multiphase Flow* 25 (1999) 755-794.

[6] S. Turek, D.C. Wand and L.S. Rivkind, The fictitious boundary method for the implicit treatment of Dirichlet boundary conditions with applications to incompressible flow simulation, Challenges in Scientific Computing, *Lecture Notes in Computational Science and Engineering* 35 (2003) 37-68.

[7] C. Duchanoy and T.R.G. Jongen, Efficient simulation of liquid-solid flows with high solids fraction in complex geometries, *Computers & Fluids* 32 (2003) 1453-1471.

S. V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publisher, New York, 1980.