2-2012

The immersed interface method for two-fluid problems

Miguel Uh
Southern Methodist University, muh@mail.smu.edu

Sheng Xu
Southern Methodist University, sxu@mail.smu.edu

The immersed interface method for two-fluid problems

Miguel Uh & Sheng Xu
Southern Methodist University

Department of Mathematics
Southern Methodist University
Dallas, Texas, USA
email: muh@smu.edu

Abstract

In this poster, we will present the implementation of the immersed interface method (IIM) for a two-fluid problem. In the IIM, a two-fluid flow is formulated as one set of governing equations and simulated on a fixed Cartesian grid. The effect of the two-fluid interface enters the formulation as a singular force and a numerical scheme as jump conditions. In our previous work, we simulated the two-fluid circular Couette flow and a static bubble problem and demonstrate the second-order accuracy and volume conservation of the method. In the present work, we simulate the rising of a single bubble to further validate the method. This bubble rising problem is characterized by the Reynolds number, the Bond number, the density ratio, and the viscosity ratio. The effects of these parameters on the bubble rising are compared with others studies.

Goal: Develop a numerical method to simulate the two-fluid problem in an accurate and efficient way.

Idea: Account for the interfacial effects using a fixed grid immersed interface method.

Introduction

The flow of two immiscible fluids is used in many technological applications [1]. The direct numerical simulations of two-fluid problems have a potentially huge domain for increased understanding. Different numerical methods have been developed for simulating two-fluid problems with various degree of success. Because of possible breakup/coalescence in a two-fluid flow, it is generally difficult and inefficient to simulate the dynamics of each fluid separately in its own domain using an interface-fitted grid method and couple the dynamics of the two fluids through interfacial conditions.

We use the IIM to solve two-fluid problems. The IIM was originally proposed by LeVeque and Li [2] for solving elliptic equations, and later extended to other type of equations. In the IIM, a fixed grid is used for the whole domain; the interface moves through the grid; and jump conditions at the interface are incorporated into numerical schemes only at grid points near or on the interface.

The mathematical formulation

Considering a two-fluid flow with constant density and viscosity in each fluid, the single set of non-dimensional Navier-Stokes equations governing the whole system reads [3],

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{uu}) = -\frac{1}{\rho}\nabla p + \frac{1}{Re}\Delta \mathbf{u} + \frac{1}{\rho}\mathbf{F} + \mathbf{G},$$
$$\nabla \cdot \mathbf{u} = 0,$$

where

- $\mathbf{u}$ is the velocity and p is the pressure;
- the density and the viscosity are

$$\rho = \rho_1 H + \rho_2(1-H), \qquad \mu = \mu_1 H + \mu_2(1-H);$$

- $Re = \frac{\rho \mathsf{LU}}{\mu}$ is the Reynolds number, where L and U are some representative length and velocity scales, respectively;
- $\mathbf{G}$ is a body force and $\mathbf{F}$ is the singular force.

The two-fluid interface is denoted as S, and its Cartesian coordinates are denoted as $\vec{X}$, see Figure 1. The interface moves with the local fluid velocity $\vec{U}$

$$\frac{d\vec{X}}{dt} = \vec{U}.$$

In general, the interface effect is represented as the singular force as

$$\mathbf{F} = \int_S \mathbf{f}\delta(\mathbf{x} - \vec{X})d\vec{X}.$$

For the effect of the surface tension, $\mathbf{f}$ takes the following form

$$\mathbf{f} = \gamma\kappa\mathbf{n}$$

where γ is the surface tension coefficient, $\mathbf{n}$ denotes the normal to the interface, and κ is the local curvature of the interface.

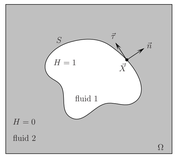

Figure 1. Schematic of the two-fluid system.

The IIM

The fundamental idea of the IIM in a flow simulation is to incorporate the effect of an interface by adding to a numerical scheme the additional terms due to jump conditions across the interface. To determine the additional terms, we follow the generalized Taylor expansion [4]. For example, a standard center differences scheme can be modified as

$$\frac{du(x_i^-)}{dx} = \frac{u(x_{i+1}^-) - u(x_{i-1}^+)}{2h} + \frac{1}{2h}\sum_{n=0}^{2}\frac{-[u^{(n)}(\xi)]}{n!}(x_{i-1}-\xi)^n + O(h^2),$$

where $[\cdot] = (\cdot)_{\xi^+} - (\cdot)_{\xi^-}$ denotes a jump and $u(x)$ is smooth except at the point ξ ($x_{i-1} < \xi < x_i$).

If we have jump conditions $[g]$, $\left[\frac{\partial g}{\partial \mathbf{n}}\right]$ and $[\Delta g]$ across the interface for a function g, then we can derive all necessary jump conditions along Cartesian coordinates to achieve second-order accuracy in the numerical scheme [4]. For a two-fluid flow, we have the following principal jump conditions [5]:

Velocity:		**Pressure:**	
$[\mathbf{u}]$,	$\left[\mu\frac{\partial \mathbf{u}}{\partial \mathbf{n}}\right]$,	$[p]$,	$\left[\frac{1}{\rho}\frac{\partial p}{\partial \mathbf{n}}\right]$, $[\Delta p]$

The difficulties in implementing these jump conditions include that

- the principal jump conditions involve the interface velocity $\vec{U}$,
- the principal jump conditions are not in the desired form because of the discontinuous fluid properties.

We interpolate the interface velocity from the current velocity field with the incorporation of the principal jump conditions. We obtain the desired jump conditions $\left[\frac{\partial \mathbf{u}}{\partial \mathbf{n}}\right]$ and $\left[\frac{\partial p}{\partial \mathbf{n}}\right]$ from those in the undesired form with the use of one-sided differences, and the augmented approach [5], respectively. The background numerical scheme in our code is the MAC scheme. The velocity field is advanced in time in an explicit manner, which greatly simplifies the application of these numerical approximations.

A bubble rising in a liquid

To validate our method, we simulate a bubble rising in viscous liquid. This problem is characterized by four non-dimensional parameters, namely, the density ratio ρ_2/ρ_1, the viscosity ratio μ_2/μ_1, the Reynolds number $Re = \frac{\rho_2 \mathsf{LU}}{\mu_2}$ (based on the liquid), and the Bond number $Bo = \frac{\rho_1 \mathsf{LU}}{\gamma}$.

The bubble is initially circular; and the initial velocity is zero everywhere in the domain. We compare our simulation results with those by Hysing et al. [7], where they use the finite element method and the level set method. Figure 2 shows the comparison for bubble evolution indicating very good match.

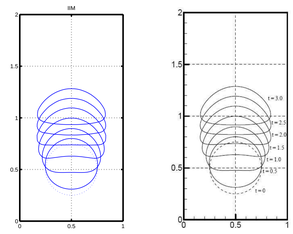

Figure 2. Interface evolution at $Bo = 10$, $Re = 35$, $\rho_2/\rho_1 = 10$, $\mu_2/\mu_1 = 10$, obtained with the IIM (left) and Finite Element Method (right) used in [7].

In Figure 3, we show the temporal evolution of velocity and pressure field.

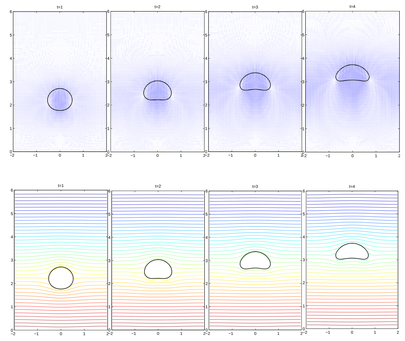

Figure 3. Temporal evolution of velocity field and pressure field (contour plots) at $Bo = 50$, $Re = 10$, $\rho_2/\rho_1 = 1000$ and $\mu_2/\mu_1 = 100$.

The bubble shapes vary as a function of the non-dimensional parameters. The effect of density ratio is shown in the Figure 4. The effect of density ratio on bubble rising velocity is more significant than on terminal bubble shape. Our results agree qualitatively with those by Jinsong et al. [8].

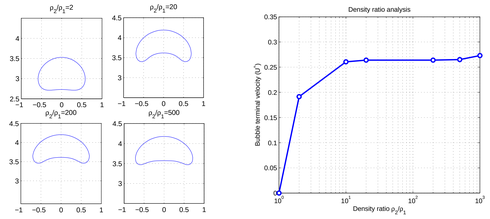

Figure 4. The predicted terminal bubble shape and terminal bubble velocity under different density ratios and $Bo = 50$, $Re = 10$, $\mu_2/\mu_1 = 100$.

The terminal shapes of single rising bubble under a range of Reynolds and Bound numbers are shown in Figure 5. The simulations show that higher Reynolds number and Bond number induce larger deformation of the bubble shape, which also agree qualitatively with those by Jinsong et al. [8].

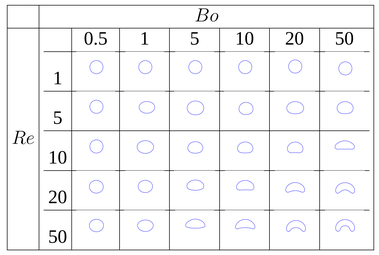

Figure 5. The predicted bubble shapes as a function of Reynolds and Bond numbers using $\rho_2/\rho_1 = 1000$, $\mu_2/\mu_1 = 100$.

Conclusions

We implement the IIM for computing incompressible immiscible two-fluid flows with different densities and viscosities. In previous work, we have shown that for fixed interfaces, the solutions were computed with a second-order accuracy and the method conserved volume in non-penetration closed interfaces. To validate the IIM for a moving interface, we simulate the rising of a single bubble in viscous liquid. The simulations results demonstrate that the method works well with large ranges of density ratio (up to 1000) and viscosity ratio (up to 500). We are currently improving the robustness of the method for large range of Reynolds number and Bond number.

References

[1] D. Joseph and Y. Renardy, *Fundamentals of two-fluid dynamics, Part I: mathematical theory and applications*, 1993.

[2] R. J. LeVeque and Z. Li, *The immersed interface method for elliptic equations with discontinuous coefficients and singular forces*, 1999.

[3] G. Tryggvason et al, *A front-tracking method for computations of multiphase flows*, 2001.

[4] S. Xu, *Derivation of principal jump conditions for the immersed interface method in two-fluid flow simulation*,2009.

[5] S. Xu and Z. Wang *Systematic derivation of jump conditions for the immersed interface method in 3-D Flow Simulation*, 2006.

[6] Z. Li and K. Ito, *An augmented approach for the pressure boundary condition in a Stokes flow*, 2006.

[7] S. Hysing and J. Lou, *Numerical Simulation of Immiscible Fluids with FEM Level Set Techniques*, 2007.

[8] J. Hua and J. Lou, *Numerical simulation of bubble rising in viscous liquid*, 2006.