Numerical Simulation of Interfacial Flows

Anton Smolianski

Institute of Mathematics, Zurich University,

Winterthurerstr. 190, CH-8057 Zurich, Switzerland

Abstract

The present work is devoted to the study on unsteady flows of two immiscible viscous fluids separated by free moving interface. A unified strategy for numerical modelling of two-fluid interfacial flows is elaborated, including also the flows with possible changes of interface topology (like mergers or breakups). The presented computational approach essentially relies on three basic components: finite element method for spatial approximation, operator-splitting for temporal discretization and level-set method for interface representation. This enables us to cover a wide range of interfacial flow regimes with high accuracy. The numerical simulations of bubble dynamics phenomena are presented to validate the proposed computational method.

1 Introduction

Fluid flows with free moving boundaries/interfaces present a very important class of physical problems, a computational modelling of which is an efficient way to understanding and control of many industrial processes. Bubbly flows in chemical engineering, jet propulsion in metal forming, wave motion in marine engineering constitute just a few typical examples of real-life applications for free-surface fluid flows.

The issue of numerical modelling for interfacial flows has been addressed in a vast amount of research papers (see the overviews in [2], [4]). Among the most recent works in this field we could mention the articles [1], [6].

There are several intrinsic difficulties, a correct treatment of which essentially determines the success of any numerical method for two-fluid interfacial flows: strong non-linearity related to the flow–interface coupling, large jumps of fluid density and viscosity across the interface, the influence of the surface tension force, the necessity to maintain a sharp interface resolution (including the cases of interface folding, breaking and merging), and mass conservation of the fluid. While the detailed treatment of these features is discussed in [4], the most important algorithmical issues and the general strategy are addressed below.

2 Setting of the problem

Suppose that the motion of two viscous immiscible fluids under our investigation is confined to some box (a parallelepiped in 3D, a rectangle in 2D). For the sake

of simplicity we consider only 2D case, but the proposed approach is easily extendible to 3D situation. We denote the boundary of the box by Σ, the domains occupied with the fluids by Ω_1 and Ω_2 and the interface between the fluids by Γ ($\Gamma = \partial\Omega_1 \cap \partial\Omega_2$, where $\partial\Omega_i$ is the boundary of $\Omega_i, i = 1, 2$). Let also Ω be the entire region occupied with the fluids, i.e. the interior of the box ($\Omega = \Omega_1 \cup \Omega_2 \cup \Gamma$).

The flow of both fluids is governed by the incompressible Navier-Stokes equations

$$\rho(\mathbf{x}) \left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) - \nabla \cdot (2\mu(\mathbf{x}) \underline{\underline{\mathbf{S}}}) + \nabla p = \rho(\mathbf{x}) \mathbf{g} \,, \tag{1}$$

$$\nabla \cdot \mathbf{v} = 0 \qquad \text{in } \Omega \,, \; t > 0 \tag{2}$$

supplemented with the interfacial conditions

$$[\mathbf{v}]\big|_\Gamma = 0 \,, \quad [-p\underline{\underline{\mathbf{I}}} + 2\mu \underline{\underline{\mathbf{S}}}]\big|_\Gamma \cdot \mathbf{n} = \kappa\sigma\mathbf{n} \,. \tag{3}$$

Here the density $\rho(\mathbf{x}) = \rho_1$ in Ω_1 and ρ_2 in Ω_2, the viscosity $\mu(\mathbf{x}) = \mu_1$ in Ω_1 and μ_2 in Ω_2, $\mathbf{v}(\mathbf{x}, t)$ is the fluid velocity, $p(\mathbf{x}, t)$ is the fluid pressure, $\mathbf{g}$ is the acceleration of gravitational field, $\underline{\underline{\mathbf{S}}} = \frac{1}{2}(\nabla \mathbf{v} + (\nabla \mathbf{v})^T)$ is the deformation rate tensor, $\underline{\underline{\mathbf{I}}}$ is the identity tensor, σ is the coefficient of surface tension, κ is twice the mean curvature of the interface, $\mathbf{n}$ is the unit normal to the interface, and $[...]\big|_\Gamma$ denotes a jump across the interface Γ.

The equations (1)–(2) and interfacial conditions (3) should be complemented with some boundary condition on Σ for velocity and with the initial conditions for velocity and for interface position.

The problem at hand contains three key ingredients: (i) flow equations (i.e.

the Navier-Stokes equations with discontinuous coefficients and singular capillary force), (ii) moving interface and (iii) coupling between velocity-pressure fields and the interface (through the coefficients, capillary force and interfacial advective velocity). To attack the problem numerically we advocate the operator-splitting approach; namely, at each time step, we first resolve the Navier-Stokes system with fixed known interface, then, using computed velocity field, we find the new approximation of the interface. Having found the new interface position we can calculate its normal and curvature, and, thus, evaluate the surface tension force and the density/viscosity coefficients to be used on the next time step in the Navier-Stokes equations.

3 Discretization of the Navier-Stokes equations

Using the Marchuk-Yanenko fractional-step scheme we may separate the convective non-linearity, viscous diffusion and incompressibility from one another and treat each of them with corresponding numerical technique. Thus, on each time interval $[t_n; t_{n+1}]$ the Navier-Stokes (NS) system (1)–(3) is approximated by a sequence of three subproblems:

1. NS-convection step

$$\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = 0 \qquad \text{in } \Omega \times (t_n; t_{n+1}) \,, \tag{4}$$

$$\mathbf{v}\big|_{t_n} = \mathbf{v}^n \,,$$

$$\Longrightarrow \mathbf{v}^* \,.$$

2. Viscous step

$$\rho(\mathbf{x})\frac{\partial \mathbf{v}}{\partial t} - \nabla \cdot (2\mu(\mathbf{x})\underline{\underline{\mathbf{S}}}) = \rho(\mathbf{x})\mathbf{g} \quad \text{in } \Omega \times (t_n; t_{n+1}) \,, \tag{5}$$

$$[\mathbf{v}]\big|_\Gamma = 0 \,, \quad -[2\mu\underline{\underline{\mathbf{S}}}]\big|_\Gamma \cdot \mathbf{n} = \kappa\sigma\mathbf{n} \quad \text{on } \Gamma \,,$$

$$\mathbf{v}\big|_{t=t_n} = \mathbf{v}^* \,,$$

$$\Longrightarrow \mathbf{v}^{**} \,.$$

3. Projection step

$$\frac{\partial \mathbf{v}}{\partial t} + \frac{1}{\rho(\mathbf{x})}\nabla p = 0 \,, \tag{6}$$

$$\nabla \cdot \mathbf{v} = 0 \qquad \text{in } \Omega \times (t_n; t_{n+1}) \,,$$

$$[\mathbf{v}]\big|_\Gamma \cdot \mathbf{n} = 0 \,, \quad [p]\big|_\Gamma = 0 \quad \text{on } \Gamma \,,$$

$$\mathbf{v}\big|_{t=t_n} = \mathbf{v}^{**} \,,$$

$$\Longrightarrow \mathbf{v}^{n+1} \,,\ p^{n+1} \,.$$

Each of three subproblems must be complemented by appropriate boundary conditions (see [4]). The general strategy consists in resolving the steps successively; moreover, each step can be treated with its own time discretization scheme employing a specific (variable) time-step size dictated by stability and/or accuracy reasons.

All the subproblems (4)–(6) are discretized in space by the finite element method using a single uniform triangulation for both velocity and pressure; the equal-order $\mathbb{P}_1/\mathbb{P}_1$ approximation is utilized for the velocity-pressure, which is known to be stable if the time step $\Delta t \geq Ch^2$ (h is the mesh size, C is a constant).

The details of the solution of the subproblems (4)–(6) as well as the splitting of the interfacial conditions are discussed in [4], but here we would like to make

a brief remark on the solution of the viscous step. The use of the finite element method for spatial discretization of the viscous step is very essential, since in the weak (variational) formulation, on which the method relies, the interfacial stress jump condition becomes a *natural* condition. The method does not require to approximate a delta-function as it is usually done for accounting of singular capillary force (see, e.g., [5], [7]) and, hence, precludes the interface smearing. A sharp (i.e. zero-thickness) interface may be maintained, and the error in the interface location may be expected to be of the order of the discretization error. Particularly, the second-order accuracy can be recovered with the piecewise linear interpolation of all variables (velocity, pressure, interface).

4 Approximation of the interface

We use the level-set approach (see, e.g., [3]) for the interface capturing. The approach consists in specifying a continuous "level-set function" Φ such that Φ is the signed distance to the interface Γ, $\Phi > 0$ in Ω_2 and $\Phi < 0$ in Ω_1. Obviously, we have $\Gamma = \{\mathbf{x} |\, \Phi(\mathbf{x}, t) = 0 \; \forall t \geq 0\}$ and $|\nabla \Phi| = 1$ in Ω.

Since the interface is convected with the fluid velocity, the following evolution equation for $\Phi(\mathbf{x}, t)$ can be easily derived (see, e.g., [5]):

$$\frac{\partial \Phi}{\partial t} + \mathbf{v} \cdot \nabla \Phi = 0 \quad \text{in } \Omega \,, \; t > 0 \,. \tag{7}$$

The same, continuous piecewise-linear ($\mathbb{P}_1$), approximation is used for Φ as for the velocity and pressure. Such approximation allows us to easily localize the interface at any moment of time, and, hence, to obtain *unique* piecewise-linear

representation for the interface Γ.

One of the most severe problems in numerical modelling of the interfacial flows is a poor conservation of fluid mass during the computational process. We propose a very simple and efficient approach that uses an additional, level-set correction step to explicitly enforce the mass conservation. The correction amounts to moving the whole level-set function $\Phi(\mathbf{x})$ upward or downward in Ω according to the formula $\Phi^{new} = \Phi + C_\Phi$, where the signed constant C_Φ is found from the expression $C_\Phi = (S_{exact} - S(\Omega_2))/L(\Gamma)$. Here $L(\Gamma)$ is the length of the interface Γ, S_{exact} is the exact area of the region occupied with the second fluid (S_{exact} is always known to us), $S(\Omega_2)$ is the area of Ω_2.

It is noteworthy that if $S(\Omega_2) > S_{exact}$ we have $C_\Phi < 0$, and the level-set function Φ is to be lowered, which automatically implies shrinking of the domain Ω_2; if $S(\Omega_2) < S_{exact}$ then $C_\Phi > 0$, and Ω_2 automatically expands. The justification of the level-set correction is essentially based on two facts: Φ being a distance function in some vicinity of the interface and smallness of $|C_\Phi|$ (see [4]). It is shown that the level-set correction keeps the error of mass conservation within $\mathcal{O}(h^2)$ accuracy of the interface approximation ([4]).

Having computed the piecewise-linear approximation Φ_h of the level-set function, we can find the interface normal by virtue of the formula $\mathbf{n}_h = \nabla\Phi_h/|\nabla\Phi_h|$. This results in a piecewise-constant approximation of the normal; it is shown in [4] that applying to $\mathbf{n}_h$ a standard gradient averaging procedure leads to the piecewise-linear continuous approximation $\tilde{\mathbf{n}}_h$ having $\mathcal{O}(h^2)$ accuracy.

The curvature may be computed by two methods: either directly as $\kappa_h = -\nabla \cdot \tilde{\mathbf{n}}_h$ or by using the variational formulation for the curvature equation

$$\int_\Omega \tilde{\kappa}_h q_h \, dx = \int_\Omega \tilde{\mathbf{n}}_h \cdot \nabla q_h \, dx - \int_\Sigma (\tilde{\mathbf{n}}_h \cdot \mathbf{n}_\Sigma) q_h \, d\Sigma \,, \quad \tilde{\kappa}_h \in Q^h \,, \; \forall q_h \in Q^h \,, \quad (8)$$

where $\mathbf{n}_\Sigma$ is the outward unit normal to the boundary Σ and Q^h is the space of continuous piecewise-linear functions defined on the considered uniform triangulation. Solution of (8) amounts simply to resolving the system with the consistent mass matrix. It is demonstrated in [4] that the piecewise-constant approximation κ_h has $\mathcal{O}(h)$ accuracy, while the piecewise-linear $\tilde{\kappa}_h$ attains the accuracy $\mathcal{O}(h^2)$. The latter fact is quite remarkable, since we use only *linear* interface approximation, and allows us to evaluate accurately the surface tension force.

5 Numerical results

The numerical simulations of bubble dynamics are shown on figures 1 and 2: the first test concerns the merger of two bubbles, the second illustrates the breakup of a single bubble. At the initial moment all bubbles are spherical and start rising in an initially quiescent fluid due to the effect of buoyancy. Both cases are characterized by four non-dimensional parameters: the density ratio ρ_1/ρ_2, the viscosity ratio μ_1/μ_2, the Reynolds number $Re = (2R)^{3/2}\sqrt{g}\rho_1/\mu_1$ and the Bond number $Bo = 4\rho_1 g R^2/\sigma$. Here R is the initial radius of the bubble, and index "1" corresponds to the fluid surrounding the bubble. The results are in a good agreement with [5], [6], [7].

References

[1] A. Esmaeeli and G. Tryggvason, Direct numerical simulations of bubbly flows. Part 2. Moderate Reynolds number arrays. *J. Fluid Mech.* **385** (1999) 325–358.

[2] R. Scardovelli and S. Zaleski, Direct numerical simulation of free-surface and interfacial flow. *Annu. Rev. Fluid Mech.* **31** (1999) 567–603.

[3] J.A. Sethian, *Level Set Methods and Fast Marching Methods: Evolving Interfaces in Computational Geometry, Fluid Mechanics, Computer Vision and Materials Science.* Cambridge University Press, 1999.

[4] A. Smolianski, *Numerical Modeling of Two-Fluid Interfacial Flows.* PhD thesis, University of Jyväskylä, 2001. ISBN 951-39-0929-8.

[5] M. Sussman, P. Smereka and S. Osher, A level set approach for computing solutions to incompressible two-phase flow. *J. Comput. Phys.* **114** (1994) 146–159.

[6] M. Sussman, A. Almgren, J. Bell, Ph. Colella, L. Howell and M. Welcome, An adaptive level set approach for incompressible two-phase flows. *J. Comput. Phys.* **148** (1999) 81–124.

[7] S.O. Unverdi and G. Tryggvason, A front-tracking method for viscous, incompressible, multi-fluid flows. *J. Comput. Phys.* **100** (1992) 25–37.

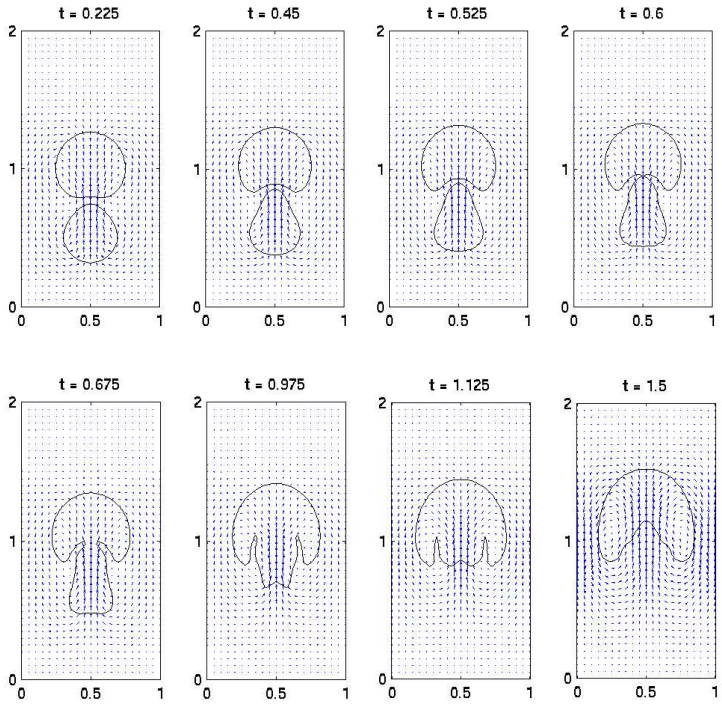

Figure 1: Merger of two rising bubbles; $Re = 35$, $Bo = 250$, $\rho_1/\rho_2 = 10^2$, $\mu_1/\mu_2 = 10$, mesh size $h = 1/40$, time step $\Delta t = h/2$.

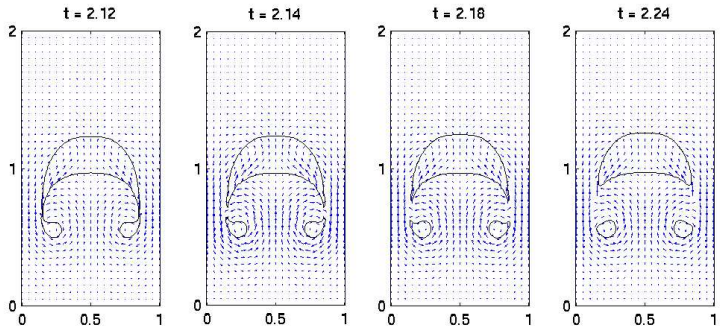

Figure 2: Breakup of rising bubble; $Re = 700$, $Bo = 500$, $\rho_1/\rho_2 = 10^3$, $\mu_1/\mu_2 = 10^2$, mesh size $h = 1/80$, time step $\Delta t = h/2$.