Numerical simulation of interfacial flows

A. Smolianski

Summary. The present work is devoted to the study of unsteady flows of two immiscible viscous fluids separated by a free moving interface. A unified strategy for the numerical modelling of two-fluid interfacial flows is elaborated, including flows with possible changes of interface topology (like mergers or breakups). The computational approach presented essentially relies on three basic components: the finite element method for spatial approximation, operator-splitting for temporal discretization and the 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 represent a very important class of physical problem, whose computational modelling is an efficient way to understand and control 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 number of research papers (see the overviews in [2, 4]). Among the most recent works in this field we cite 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 of maintaining 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 investigation is confined to a box (a parallelepiped in 3D, a rectangle in 2D). For the sake of simplicity we consider only the 2D case, but the proposed approach is easily extendible to the 3D situation. We denote the boundary of the box by \( \Sigma \), the domains occupied by the fluids by \( \Omega_1 \) and \( \Omega_2 \) and the interface between the fluids by \( \Gamma \) (\( \Gamma = \partial \Omega_1 \cap \partial \Omega_2 \)), where \( \partial \Omega_i \) is the boundary of \( \Omega_i \), \( i = 1, 2 \). Let \( \Omega \) be the entire region occupied by 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
\[
\rho(x) \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) - \nabla \cdot (2\mu(x)\mathbf{S}) + \nabla p = \rho(x)g, \tag{1}
\]
\[
\nabla \cdot \mathbf{v} = 0 \quad \text{in } \Omega, \quad t > 0, \tag{2}
\]
supplemented by the interfacial conditions
\[
[\mathbf{v}]_\Gamma = 0, \quad [-p\mathbf{I} + 2\mu\mathbf{S}]_\Gamma \cdot \mathbf{n} = \kappa \sigma \mathbf{n}. \tag{3}
\]
Here the density \( \rho(x) \) is \( \rho_1 \) in \( \Omega_1 \) and \( \rho_2 \) in \( \Omega_2 \), the viscosity \( \mu(x) \) is \( \mu_1 \) in \( \Omega_1 \) and \( \mu_2 \) in \( \Omega_2 \), \( \mathbf{v}(x, t) \) is the fluid velocity, \( p(x, t) \) is the fluid pressure, \( g \) is the acceleration of gravitational field, \( \mathbf{S} = \frac{1}{2}(\nabla \mathbf{v} + (\nabla \mathbf{v})^T) \) is the deformation rate tensor, \( \mathbf{I} \) is the identity tensor, \( \sigma \) is the coefficient of surface tension, \( \kappa \) is twice the mean curvature of the interface, \( \mathbf{n} \) is the unit normal to the interface, and \([\ldots]_\Gamma\) denotes a jump across the interface \( \Gamma \).

Equations (1)–(2) and interfacial conditions (3) should be complemented with a boundary condition on \( \Sigma \) 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 the 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 the 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
\[
\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} = 0, \quad \text{in } \Omega \times (t_n; t_{n+1}), \tag{4}
\]
\[
\mathbf{v} \big|_{t_n} = \mathbf{v}^n, \quad \implies \mathbf{v}^*.
\]