Computers and Fluids 124 (2016) 12–29

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

A coupled volume-of-fluid/level-set method for simulation of two-phase flows on unstructured meshes

Néstor Balcázar [a,*], Oriol Lehmkuhl [a,b], Lluís Jofre [a], Joaquim Rigola [a], Assensi Oliva [a,*]

[a] Heat and Mass Transfer Technological Center (CTTC), Universitat Politècnica de Catalunya – BarcelonaTech (UPC), ETSEIAT, Colom 11, 08222 Terrassa, Barcelona, Spain
[b] Termo Fluids, S.L., Avda Jacquard 97 1-E, 08222 Terrassa, Barcelona, Spain

ARTICLE INFO

Article history:
Received 5 December 2014
Revised 17 August 2015
Accepted 7 October 2015
Available online 22 October 2015

Keywords:
Unstructured meshes
Level-set method
Volume-of-fluid method
Finite-volume method
Two-phase flow

ABSTRACT

This paper presents a methodology for simulation of two-phase flows with surface tension in the framework of unstructured meshes, which combines volume-of-fluid with level-set methods. While the volume-of-fluid transport relies on a robust and accurate polyhedral library for interface advection, surface tension force is calculated by using a level-set function reconstructed by means of a geometrical procedure. Moreover the solution of the fluid flow equations is performed through the fractional step method, using a finite-volume discretization on a collocated grid arrangement. The numerical method is validated against two- and three-dimensional test cases well established in the literature. Conservation properties of this method are shown to be excellent, while geometrical accuracy remains satisfactory even for the most complex flows.

1. Introduction

Numerical simulation of two-phase flows is vital for many engineering and scientific applications, such as combustion, bubbly flow, boiling heat transfer, unit operations in chemical engineering, cooling of nuclear reactors among others. The accurate modeling of interfacial flows is challenging because of the discontinuity in material properties (e.g. density, viscosity), the necessity to account for surface tension force, and due to the fact that the geometry of the interface is not known a priory. For this kind of flows it is critical a precise computation of interfacial quantities such as curvature and normal, which are used to evaluate the surface tension. Errors in the calculated surface tension force will induce non-physical velocities, commonly known as spurious or parasitic currents [30], which can grow with time and so significantly degrade simulation results. Moreover, most of industrial applications are characterized by complex domains; therefore the use of unstructured meshes is advantageous.

In order to solve the aforementioned issues many numerical methods have been developed in the past decades. For instance: the front tracking (FT) method [42,43], level set (LS) methods [1,27,28,38], volume-of-fluid (VOF) methods [16,22,44], and hybrid VOF/LS methods [26,37,39,45]. In these methods, two-phase flow is treated as a single flow with the density and viscosity varying smoothly across the moving interface which is captured in an Eulerian framework (VOF, LS, CLSVOF, VOSET) or in a Lagrangian framework (FT). Although the idea behind these methods is similar, their numerical implementation may differ greatly. A review of advantages and disadvantages of these techniques in the context of simulation of multiphase flows with sharp interfaces is given in [44]. In the front-tracking method [42,43], a stationary Eulerian grid is used for the fluid flow and the interface is tracked explicitly by a separate Lagrangian grid. This method is extremely accurate but also rather complex to implement due to the fact that dynamic re-meshing of the Lagrangian interface mesh is required [10]. Contrary to LS and VOF method, automatic merging of interfaces does not occur, and difficulties arise when multiple interfaces interact with each other as in coalescence and breakup. In the VOF method [16,22,44], the interface is given implicitly by a color function, defined to be the fraction of volume within each cell of one of the fluids. In order to advect the VOF function, the interface needs to be reconstructed using a geometric technique [22]. An advantage of VOF method is the fact that accurate algorithms can be used to advect the interface (e.g. [22]), so that the mass is conserved, while still maintaining a sharp representation of the interfaces [39]. However a disadvantage of the VOF method is the fact that it is difficult to compute accurate curvatures from the volume fraction function used to represent the interface, because it presents a step discontinuity. In level-set (LS) methods [28,38] the interface is represented by the zero-contour of a signed distance function. The evolution of this function in space and time is governed by an advection equation, combined with a special re-distancing algorithm. One major

* Corresponding authors. Tel.: +34 93 739 81 92; fax: +34 93 739 89 20 .
E-mail addresses: nestor@cttc.upc.edu (N. Balcázar), termofluids@termofluids.com (O. Lehmkuhl), cttc@cttc.upc.edu (A. Oliva).

http://dx.doi.org/10.1016/j.compfluid.2015.10.005
0045-7930/

advantage of the LS approach is the fact that the interface curvature can be accurately computed, while a disadvantage of this method is that the discrete solution of transport equations leads to numerical error in mass conservation of the fluid-phases. Recently, [27] has introduced a conservative level-set method (CLS) where mass conservation problem is greatly reduced, while a hyperbolic tangent function is employed as the level-set function. Moreover, this approach has been generalized to unstructured meshes by [1–3], in the framework of finite-volume discretizations.

On the basis of the advantages and disadvantages of VOF and LS methods, it can be concluded that they are complementary, so it is an inevitable trend to develop new methods combining VOF and LS approaches. For instance, Sussman and Puckett [39] presented a coupled level-set/volume-of-fluid (CLSVOF) method for computing 3D and axisymmetric incompressible two-phase flows. In the CLSVOF method the curvature was obtained via finite differences of the level set function which in turn is derived from the level set function and volume-of-fluid function. Yang et al. [46] present an adaptive coupled level-set/volume-of-fluid (ACLSVOF) method for interfacial flow simulations on two-dimensional unstructured triangular grids. Another CLSVOF method was implemented by Wang et al. [45] for the numerical simulation of interfacial flows in ship hydrodynamics, where the level set function is re-distanced based on the reconstructed interface with a geometric algorithm, whereas the interface jump conditions were handled by means of a ghost fluid methodology. This method was employed to simulate a gas bubble rising in a viscous liquid and a water drop impact onto a deep water pool. Sun et al. [37] have also presented a coupled volume-of-fluid and level set (VOSET) method, where a distance function is reconstructed from an iterative algorithm and interface is advected by the VOF method. The VOSET method was validated by performing two-dimensional simulations of rising bubbles and dam-break problems.

Despite those efforts, to the best of the author's knowledge most of the aforementioned coupled VOF/LS methods have been designed for regular Cartesian meshes, so that their easiness of implementation, capability and accuracy on irregular unstructured meshes is still to be proven. Therefore, the present work is aimed at making progress in the direction of developing an accurate and robust coupled VOF/LS method for simulation of incompressible two-phase flows on two- and three dimensional unstructured meshes, including surface tension effects. Thus, unstructured meshes can be adapted to complex domains, enabling us an efficient mesh distribution in regions where interface resolution has to be maximized, which is in general hard to achieve on structured grids. In the present coupled VOF/LS method, an accurate VOF-PLIC method introduced in [22] is used to advect the interface, while the interface curvature and normals employed to evaluate the surface tension force are computed by using a level-set function. This LS function is reconstructed through a geometrical procedure, based on the computation of the minimum distances between cell centroids and the plane segments provided by the PLIC-VOF method, while the surface tension force is computed in the framework of the continuous surface force model introduced by Brackbill et al. [5]. Regarding the fluid flow, a classical fractional step method [7] is used to solve the incompressible Navier–Stokes equations, which are coupled with the VOF and LS functions. The Navier–Stokes equations have been discretized by means of the finite-volume method on a collocated unstructured grid arrangement, according to the work introduced by Balcázar et al. [1]. Numerical results are contrasted against numerical and experimental data from the literature.

The outline of this paper is as follows: A summary of the governing equations and numerical methods is given in Section 2. In Section 3 numerical experiments are presented in order to validate the coupled VOF/LS method implemented in this work. These numerical experiments include the simulation of the static droplet test case, two- and three-dimensional buoyant bubbles, co-axial coalescence of two bubbles, and deformation of a drop under shear flow. Finally, the conclusions are presented in Section 4.

2. Governing equations and discretization

2.1. Incompressible two-phase flow

The conservation of momentum and mass of two immiscible incompressible and Newtonian fluids is described by the Navier–Stokes equations defined on a spatial domain Ω with boundary $\partial\Omega$:

$$\frac{\partial}{\partial t}(\rho_k \mathbf{v}_k) + \nabla \cdot (\rho_k \mathbf{v}_k \mathbf{v}_k) = \nabla \cdot \mathbf{S}_k + \rho_k \mathbf{g} \qquad \text{in } \Omega_k \tag{1}$$

$$\mathbf{S}_k = -p_k \mathbf{I} + \mu_k \left(\nabla \mathbf{v}_k + (\nabla \mathbf{v}_k)^T \right) \tag{2}$$

$$\nabla \cdot \mathbf{v}_k = 0 \qquad \text{in } \Omega_k \tag{3}$$

Here, $\Omega = \Omega_1 \cup \Omega_2 \cup \Gamma$, $k = \{1, 2\}$ denote the subdomains associated with the two different fluid phases, $\Gamma = \partial\Omega_1 \cap \partial\Omega_2$ is the fluid interface, ρ and μ denote the density and dynamic viscosity of the fluids, $\mathbf{v}$ is the velocity field, $\mathbf{g}$ is the gravity acceleration, p is the pressure, $\mathbf{S}$ is the stress tensor and $\mathbf{I}$ is the identity tensor. Assuming no mass transfer between the fluids yields a continuous velocity condition at the interface:

$$\mathbf{v}_1 = \mathbf{v}_2 \qquad \text{in } \Gamma \tag{4}$$

The jump in normal stresses along the fluid interface is balanced by the surface tension. Neglecting the variations of the surface tension coefficient σ gives the following boundary condition for momentum conservation at the interface:

$$(\mathbf{S}_1 - \mathbf{S}_2) \cdot \mathbf{n} = \sigma \kappa \mathbf{n} \qquad \text{in } \Gamma \tag{5}$$

where $\mathbf{n}$ is the unit normal vector outward to $\partial\Omega_1$ and κ is the interface curvature. Eqs. (1)–(3) and Eqs. (4) and (5) can be combined into a set of equations for a single fluid in Ω, with a singular source term for the surface tension force at the interface Γ [5,8,29]:

$$\frac{\partial}{\partial t}(\rho \mathbf{v}) + \nabla \cdot (\rho \mathbf{v}\mathbf{v}) = -\nabla p + \nabla \cdot \mu \left(\nabla \mathbf{v} + (\nabla \mathbf{v})^T \right) + \rho \mathbf{g} + \sigma \kappa \mathbf{n} \delta_\Gamma \tag{6}$$

$$\nabla \cdot \mathbf{v} = 0 \tag{7}$$

where $\mathbf{v}$ and p denote the fluid velocity field and pressure, ρ is the fluid density, μ is the dynamic viscosity, $\mathbf{g}$ is the gravitational acceleration the super-index T represents the transpose operator, δ_Γ is a Dirac delta function concentrated at the interface Γ, σ is the surface tension coefficient, κ is the curvature of the interface and $\mathbf{n}$ denotes the unit normal vector on the interface. Physical properties change discontinuously across the interface:

$$\begin{aligned} \rho &= \rho_1 H_1 + \rho_2 (1 - H_1) \\ \mu &= \mu_1 H_1 + \mu_2 (1 - H_1) \end{aligned} \tag{8}$$

with ρ_1, ρ_2 and μ_1, μ_2 the densities and viscosities of the first and second fluids, respectively, whereas H_1 is the Heaviside step function that is one at fluid 1 and zero elsewhere. In the context of the present VOF/LS method, a volume averaged indicator function will be used in place of H_1, as is defined in Eq. (10) and Section 2.7.

2.2. Volume-of-fluid method

In the volume-of-fluid method an indicator function f is used to track the interface,

$$f(\mathbf{x}, t) = \begin{cases} 1 & \text{if } \mathbf{x} \in \Omega_1 \\ 0 & \text{if } \mathbf{x} \in \Omega_2 \end{cases} \tag{9}$$