Journal of Computational Physics 297 (2015) 611–636

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

On the combined effects of surface tension force calculation and interface advection on spurious currents within Volume of Fluid and Level Set frameworks

T. Abadie [a,b], J. Aubin [b], D. Legendre [a,*]

[a] University of Toulouse, Institut de Mécanique des Fluides de Toulouse CNRS/INPT/UPS, 1 Allée du Professeur Camille Soula, 31400 Toulouse, France

[b] University of Toulouse, Laboratoire de Génie Chimique CNRS/INPT/UPS, 4 Allée Emile Monso, BP-84234, 31030 Toulouse, France

ARTICLE INFO

Article history:
Received 9 September 2014
Received in revised form 19 March 2015
Accepted 16 April 2015
Available online 7 May 2015

Keywords:
Volume of fluid
Level set
Spurious currents
Continuum surface force
Height function
Taylor flow

ABSTRACT

This paper deals with the comparison of Eulerian methods to take into account the capillary contribution in the vicinity of a fluid–fluid interface. Eulerian methods are well-known to produce additional vorticity close to the interface that leads to non-physical spurious currents. Numerical equilibrium between pressure gradient and capillary force for the static bubble test case within a VOF framework has been reached in [35] with the height-function technique [14,35]. However, once the bubble is translated in a uniform flow, spurious currents are maintained by slight errors induced by translation schemes. In this work, two main points are investigated: the ability of Volume of Fluid and Level Set methods to accurately calculate the curvature, and the magnitude of spurious currents due to errors in the calculation of the curvature after advection in both translating and rotating flows. The spurious currents source term is expressed from the vorticity equation and used to discuss and compare the methods. Simulations of gas–liquid Taylor flow at low capillary number show that the flow structure and the bubble velocity can be significantly affected by spurious currents.

1. Introduction

Numerical simulations of industrial processes, as well as academic situations often involve two immiscible fluids. A number of computational methods have been developed over the past decade to improve the computation of multiphase flows. The numerical methods to simulate multiphase flows can be classified into two main groups: the "Lagrangian" methods and the "Eulerian" methods. In the first class of methods, the interface is generally tracked using Lagrangian markers, *e.g.* Front-Tracking [49,36] or the Point-Set method [48]. The interface is located explicitly and the calculation of geometrical properties (normal to the interface and curvature) is highly accurate. However, the implementation of such methods in 3D is not straightforward and specific algorithms are needed to deal with the distribution of markers, as well as changes in topology. The second group consists of an implicit representation of the phases in each cell with an additional scalar field. The most common approaches are the Volume of Fluid (VOF) [23,28,7,17] and the Level Set (LS) methods [45,43,22,46]. VOF methods are generally well suited to conserve the mass of the phases and appear to be a natural choice in a finite

* Corresponding author.
E-mail address: legendre@imft.fr (D. Legendre).

http://dx.doi.org/10.1016/j.jcp.2015.04.054
0021-9991/

volume framework, while LS methods are known to allow better computation of the geometrical properties of the interface. Eulerian methods have been shown to be well suited to deal with various configurations, including a single bubble rising in a liquid [7], a jet or drop breaking [35,39], coalescence [46], as well as atomization with a number of inclusions of different sizes [19].

Within this Eulerian representation of two-phase flow, great effort has been dedicated to two main features: the transport of the interface and the consideration of capillary forces. When dealing with flows where capillary forces are preponderant, such as the simulation of Taylor flow (or slug flow) in microchannels, care needs to be taken in the computation of surface tension forces. This suggests that the normal to the interface and the curvature need to be accurately estimated. This capillary force located on the interface can be represented in a spatially filtered way using an interface thickness with the Continuum Surface Force method [8] or in a sharp way using the location of the interface at a sub-cell level, *e.g.* Ghost Fluid Method (GFM) [27] and the Sharp Surface Force (SSF) [18]. Within implicit representations of the interface, many methods consider the successive derivatives of the scalar field representing the interface. More recently, [14] and [35] have shown that the construction of height functions allows a better approximation of the interface curvature. Indeed, this method consists of finding the position of the interface with a good accuracy by adding successive volume fractions in a column of fluid (see Section 3.2). Using this height function technique, [35] achieved an exact numerical balance between surface tension forces and pressure jump with the elimination of spurious currents in the case of a static interface. In the case of inviscid fluids where there is no viscous dissipation to balance the kinetic energy produced by spurious currents, [35] still observed a decrease in the intensity of spurious currents over time due to numerical dissipation. The numerical damping of spurious currents when dealing with inviscid fluids was also observed in [12] with a height-function method and a finite-volume compressible flow solver where the intensity of spurious currents decreased to zero after a certain time. This numerical dissipation was further studied in [19] and [12] with the damping of the oscillations of an inviscid droplet. A numerical viscosity was estimated by fitting the damping of the amplitude of the oscillations and as expected, the numerical dissipation decreases as the mesh is refined.

However, it was also shown in [35] that the coupling between transport schemes, surface tension force with Navier–Stokes equations and curvature estimation still needs improvements. [15] showed that the advection step has no direct influence on the spurious currents with a balanced-force surface tension algorithm when the exact curvature is imposed on both cartesian and tetrahedral meshes. However, as it will become clear in Section 4, imposing the exact curvature allows the development of spurious currents to be avoided since the spurious current source term is canceled. In cases of practical interest, the curvature cannot be imposed and the advection step introduces errors in the volume fraction field thereby leading to curvature gradients. As a consequence, the zero velocity field expected in the frame of reference moving with the bubble (like for the static case [35]) is not recovered when the interface is translated in a uniform flow [35,12]. These observations have motivated our work. Different numerical methods implemented in the same flow solver have been compared in terms of the magnitude of spurious currents and pressure jump evaluation on the basis of four test cases: the static bubble case for which a number of results are available in the literature; the translating bubble, which seems more related to physical flows; a bubble in a rotating flow; and the dynamics of Taylor bubble in a circular microchannel. The objective here is to pay attention to the coupling between the interface transport equation and the Navier–Stokes equations. Finally, the dynamics of Taylor bubbles in microchannels is considered because it appears to be representative of the ability of a particular method to deal with spurious currents since these flows are dominated by surface tension (low capillary number and low Weber number). As will be shown, the development of spurious currents in such flows can promote the development of non-physical recirculation areas and consequently, erroneous slip velocity between the bubble velocity and mean velocity in the liquid slug.

2. Numerical schemes

2.1. Spatial and temporal discretizations

The numerical code used for this study is the *JADIM* code, which has been developed to simulate dispersed two-phase flows and used to simulate various multiphase flows systems [7,17,2,29,41,42]. The interface is captured by an Eulerian description of each phase on a fixed grid with variable density and viscosity. Under the assumptions that (*i*) the fluids are Newtonian and incompressible, (*ii*) there is no mass transfer at the interface, (*iii*) the flow is isothermal and (*iv*) the surface tension is constant, the fluid flow can be described by the classical one fluid formulation of the Navier–Stokes equations:

$$\nabla \cdot \mathbf{U} = 0 \,, \tag{1}$$

$$\frac{\partial \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla)\,\mathbf{U} = -\frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla \cdot \mathbf{\Sigma} + \mathbf{g} + \mathbf{F}_{\sigma,s} \,, \tag{2}$$

where $\mathbf{\Sigma}$ is the viscous stress tensor, $\mathbf{g}$ is the acceleration due to gravity, $\mathbf{F}_{\sigma,\mathbf{s}}$ is the capillary contribution whose calculation is described in Section 3, and ρ and μ are the local density and dynamic viscosity, respectively. The density and viscosity are deduced by linear interpolation from the volume fraction C of one phase in each computational cell:

$$\rho = C\rho_1 + (1 - C)\,\rho_2 \,, \tag{3}$$

$$\mu = C\mu_1 + (1 - C)\,\mu_2 \,, \tag{4}$$