International Journal of Multiphase Flow 59 (2014) 1–14

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

An embedded level set method for sharp-interface multiphase simulations of Diesel injectors

M. Arienti [a,*], M. Sussman [b]

[a] Sandia National Laboratories, Livermore, CA 94550, USA
[b] Florida State University, Tallahassee, FL 32306, USA

ARTICLE INFO

Article history:
Received 17 January 2013
Received in revised form 8 October 2013
Accepted 9 October 2013
Available online 23 October 2013

Keywords:
Injector geometry
Primary atomization
Cartesian block-structured
Contact angle

ABSTRACT

We propose a comprehensive approach for treating complex wall boundaries in two-phase, free-surface flow simulations on a Cartesian adaptive grid. The external gas–liquid interface is handled by the well-known combined level-set volume-of-fluid (CLSVOF) method. The new element is the coupling with the wall boundary representation using a second level-set function. The no-slip boundary condition at the walls is enforced by properly populating the ghost cells of a narrow band inside the solid body, using a simple and numerically robust treatment of the contact line. In this framework, merging and separation of multiple solid bodies are easily accommodated. Verification tests with grid convergence analysis are presented for a stationary/oscillating body in single-phase flow and for a drop on an inclined plane. Two examples demonstrate the suitability of the proposed approach to study liquid injection. The first is a validation study with data from a scaled-up Diesel injector, to demonstrate how the seamless calculation of internal flow and jet primary atomization can be accomplished. The second is a demonstration of transient atomization response to a measured three-dimensional needle displacement of the injector.

1. Introduction

The method for time-resolved interface-capturing called CLSVOF (combined level-set volume-of-fluid) was applied, under various implementations, to model bubble and drop dynamics in viscous and viscoelastic environments (Sussman et al., 2007; Stewart et al., 2008; Sussman and Ohta, 2009), ship waves (Sussman and Dommermuth, 2001), and underwater explosions (Kadiouglu and Sussman, 2008). Validation was carried out for sprays formed from jets subject to gas crossflow (Li et al., 2010) or impinging on each other (Arienti et al., 2013), in the latter case with excellent agreement with the statistics derived from experimental measurements.

There are, however, many free-surface flows that also require the management of complex boundary walls. One example is the process of spray formation from liquid injection. The simulation from first principles of spray atomization requires a non-trivial model of the injector in order to correctly define the boundary conditions of the calculation. Under the assumption that internal flow characteristics had limited effects on primary atomization, a simple plug flow velocity profile was assigned as a boundary condition in Arienti et al. (2013). A more realistic inflow, via correlated random velocities with assumed length scale and turbulence intensity, was generated at the orifice exit of a jet injection simulation by Ménard et al. (2007). Still, very few studies have attempted to include the inflow turbulent conditions that result from the actual injector geometry. Notably, in the simulation of jet injection in gas crossflow by Herrmann (2010), the injector was modeled as a short pipe tapering into a flush orifice. A single-phase, pre-computed large eddy simulation of pipe flow was stored as a time sequence of the pre-taper portion of the injector in the subsequent two-phase simulation.

In this paper we develop a general approach within the CLSVOF framework to include the whole injector geometry in a primary atomization simulation. This is accomplished by introducing a second level set function to represent the injector's walls in addition to the level set used to capture the gas–liquid interface. This approach is a valid alternative to boundary-fitted methods, where issues of grid deformation, re-generation and interpolation at each physical timestep can become critical in the case of moving walls. The solution we propose is to let solid boundaries and phase interfaces have unrestricted motion across underlying fixed grid lines. Without the constrain of a conformal mesh, the choice of the computational grid can be optimized for free-surface flow by selecting an isotropic and equispaced (Cartesian) grid.

The most delicate and time consuming operation with Cartesian methods becomes the intersection of the solid body with the regular grid. For single phase flow, several efficient algorithms for

* Corresponding author.
E-mail addresses: marient@sandia.gov (M. Arienti), sussman@math.fsu.edu (M. Sussman).

0301-9322/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2013.10.005

handling cut cells exist (for instance, in Aftosmis et al., 1997). Since cells cut by the solid wall can be arbitrarily small, explicit update schemes become overly restrictive for time-dependent problems, requiring either to extend the difference stencil of the spatial terms (Berger and LeVeque, 1990), or to use a cell-merging approach (Bayyuk et al., 1993).

With a moving or deforming solid, the second issue for Cartesian methods is how to enforce the correct velocity boundary condition at the interface. In the immersed boundary method originated by Peskin (1972), the flow velocity is obtained from a set of body forces distributed over the nearby field. This approach is well established for single-phase flow, where a high order of accuracy can be achieved. Fadlun et al. (2000) developed a second-order accurate immersed boundary method, on a regular grid, for unsteady three-dimensional flows in complex geometries; velocity interpolation algorithms were developed for prescribing the feedback forcing at the first gridpoint outside the boundary. To determine the velocity and pressure values of the computational cells emerging from solid body motion, Yang and Balaras (2006) introduced a field-extension strategy that recovers a sharp boundary wall instead of a smeared interface. A robust and fast method for cell tagging and shortest distance calculation with an immersed triangulation is discussed by Yang and Stern (2013).

In the development of methods for two-phase flow, an additional layer of complexity emerges because of the numerical treatment of the contact line. At the intersection of the gas–liquid interface with the solid boundary, the contact line must be allowed to move, even if such motion is a mathematical paradox because of the no-slip boundary at the solid surface. This situation in general results in a degradation of the convergence properties of single-phase algorithms for solid boundary treatment.

In the body-conformal finite element method by Baer et al. (2000), the slip is imposed at the mesh nodes forming the contact line. In the two-dimensional sharp-interface approach by Liu et al. (2005), the level set field in the vicinity of the contact line is redistanced in order to impose a specified contact angle; the slip condition is imposed on grid points in the vicinity of the contact line, while a ghost fluid method treatment is used for gas–liquid interfaces. In the level-set only method for sharp interfaces and arbitrary boundaries by Krishnan et al. (2006), a local, two-dimensional level-set field is reconstructed by fitting the interface to a parabolic curve that intersects the solid surface at exactly the given contact angle. The embedded boundary approach by Yang and Balaras (2006) was later complemented by a level-set based ghost-fluid method to treat the gas–liquid interface in the study of wake–ship interaction (Yang and Stern, 2009).

In the volume-of-fluid (VOF) continuum surface force (CSF) method by Bussmann, the contact line slip is achieved implicitly because the advection scheme for the liquid volume fraction utilizes face-centered velocities (Afkhami and Bussmann, 2009); in this way, the center of the cell is removed one half cell width away from the wall. A similar approach is used in this paper, but with the extension that a solid boundary can be arbitrarily positioned with respect to the Cartesian mesh (whereas in Afkhami and Bussmann the solid wall coincides with the boundary domain). In fuel injection simulations, a level-set based ghost-fluid method for the gas–liquid interface with sharp solid wall treatment was explored by Noël et al. (2012) and by Arienti and Sussman (2012).

We propose to augment the CLSVOF method with a second level set function to capture the motion of multiple solid boundaries of arbitrary complexity. The level set framework enables contact, merging, and separation of solid boundaries in a straightforward manner. The use of a level set function to track a moving solid boundary is not new (see, for instance, Arienti et al., 2003), but what we add in this paper is the study of the free surface interaction with it.

The methodology we propose for the treatment of the contact line is somewhat simpler than the one by Afkhami and Bussmann (2009), but it is demonstrated to be robust for an arbitrary position of the solid wall in two and three dimensions. There is also no need for an extension of the pressure field, as required in the methods of Yang and Stern (2009) and Noël et al. (2012). Given these simplifications, the outcome of the verification tests presented in this paper is rather satisfactory, opening the way to future algorithmic improvements. The validation study with an actual Diesel injector, presented later, is also relevant, since there appears to be very few studies on the effects of non-trivial orifice geometry on spray formation.

The numerical aspects of the embedded solid boundary algorithm are described first. The method is then verified with a single-phase crossflow passing over a half cylinder at low Reynolds number; the vorticity field arising from the interaction with the curved wall is compared with the results from more specialized, higher-order methods for single-phase flow, showing the lack of computational artifacts and acceptable rate of convergence. Next, the shape of a drop on a wall surface is calculated for different contact angles. The new element in these tests is that the wall is at an angle with respect to the Cartesian axes. The convergence properties of the final drop shape and the rate of volume conservation are discussed, including dynamic cases where the initial drop shape is different than the final one.

For validation purposes, we consider an early experimental study by Arcoumanis et al. (1998) where the flow velocity inside a scaled-up Bosch six-hole transparent Diesel injector was measured with laser Doppler velocimetry for evaluating average and fluctuating components. The asymmetric spray obtained from the simulation is briefly discussed to point out the relevance of an injector simulation that can capture both internal and external flow.

The last example concerns a transient injection where the injector's needle moves relative to its cap. The geometry and motion data belong to a real injector, reported by Kastengren et al. (forthcoming) and made available by the extensive data set of the Engine Combustion Network (ECN); the calculation demonstrates the ease of the proposed methodology in dealing with moving walls and a changing topology (merging and separation) of the solid boundary.

2. Numerical method

The Navier–Stokes equations for incompressible flow of two immiscible fluids are solved with the one-fluid approach according to the level-set equations for multiphase flow (Chang et al., 1996):

$$\rho(\phi)\frac{D\mathbf{u}}{Dt} = -\nabla p + 2\nabla(\mu(\phi)\mathbf{D}) - \sigma\kappa\nabla H(\phi), \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0, \tag{2}$$

$$H(\phi) = \begin{cases} 1 & \phi \geqslant 0 \\ 0 & \phi < 0, \end{cases} \tag{3}$$

$$\rho(\phi) = \rho_L H(\phi) + \rho_G(1 - H(\phi)), \tag{4}$$

$$\mu(\phi) = \mu_L H(\phi) + \mu_G(1 - H(\phi)), \tag{5}$$

$$\frac{D\phi}{Dt} = 0. \tag{6}$$

In the equations, $\mathbf{u}$ is the vector field, p the pressure, ϕ the level-set function, κ the interface curvature, and $\mathbf{D}$ the deformation tensor, $\mathbf{D} = (\nabla\mathbf{u} + (\nabla\mathbf{u})^T)/2$; H is the Heaviside function and D/Dt the material derivative; σ is the surface tension coefficient. The smooth