Journal of Computational Physics 229 (2010) 6853–6873

Contents lists available at ScienceDirect

Journal of Computational Physics

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

Simulation of axisymmetric jets with a finite element Navier–Stokes solver and a multilevel VOF approach

A. Cervone, S. Manservisi *, R. Scardovelli

DIENCA, Montecuccolino Laboratory, University of Bologna, Via dei Colli 16, 40136 Bologna, Italy

ARTICLE INFO

Article history:
Received 7 April 2009
Received in revised form 25 February 2010
Accepted 22 May 2010
Available online 2 June 2010

Keywords:
Jet simulation
Volume-of-fluid
Finite element
Multilevel approach

ABSTRACT

A multilevel VOF approach has been coupled to an accurate finite element Navier–Stokes solver in axisymmetric geometry for the simulation of incompressible liquid jets with high density ratios. The representation of the color function over a fine grid has been introduced to reduce the discontinuity of the interface at the cell boundary. In the refined grid the automatic breakup and coalescence occur at a spatial scale much smaller than the coarse grid spacing. To reduce memory requirements, we have implemented on the fine grid a compact storage scheme which memorizes the color function data only in the mixed cells. The capillary force is computed by using the Laplace–Beltrami operator and a volumetric approach for the two principal curvatures. Several simulations of axisymmetric jets have been performed to show the accuracy and robustness of the proposed scheme.

1. Introduction

A stream of liquid matter that enters a gas region and breaks into small liquid masses is a very complex phenomenon that can occur in many natural and engineering processes. In nature, jets appear on very different spatial scales, ranging from galaxies to atoms. In industrial applications, many devices require the injection of liquid or gas phases as essential part of the combustion process or as primary coolant for heat removal. The dynamics of a liquid jet is rather sensitive to the boundary conditions, to the turbulence inside the liquid and to the surrounding gas and, in spite of large efforts, the mechanisms leading to small fluid structures and droplets are still an open question and object of active research. In recent time, many studies have investigated jets with direct numerical simulations, using the Volume-of-Fluid (VOF), level set and front tracking methods. Front tracking is based on the Lagrangian advection of marker particles that are attached to the interface, while VOF methods rely on the advection of the volumetric fraction of the reference phase in the grid cells. In the level set method the interface is given by the zero-level isosurface of a continuous function defined by the signed distance to the interface. Overviews of several front tracking and front capturing methods can be found in [1,2] for the front tracking, and in [3,4] for the VOF and level set methods, respectively. Realistic spatial simulations are those involving jets entering at one boundary of the computational domain and developing several instabilities in its interior. The simulation of a three-dimensional turbulent jet entering a gas environment, with a uniform discretization of the domain, can be found in [5], an example that illustrates the considerable power of front capturing methods. Other examples can be found in [6–10] or in the reviews [11,12] and citations therein.

One of the most popular method to track the interface is the VOF technique. In this method the interface is reconstructed from the color function data representing the fraction of each grid cell which is occupied by the reference fluid phase. The calculation of the interface geometrical properties, i.e. the local normal and curvature, cannot always be performed

* Corresponding author. Tel.: +39 051 644 1718; fax: +39 051 644 1747.
E-mail address: sandro.manservisi@mail.ing.unibo.it (S. Manservisi).

0021-9991/$ - see front matter © 2010 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2010.05.025

accurately with a fixed number of cells (a 3×3 block of cells in two dimensions and $3 \times 3 \times 3$ cells in three dimensions) by using standard methods, such as the Parker and Youngs' reconstruction or the height function method. For this reason, either smoothing strategies or more complex algorithms with a variable element cell support must be adopted [13,14]. The VOF method has the potential of an exact conservation of the mass if the advection of the color function is performed correctly and there is essentially no limit to the complexity of the interface, since breakup and merging events are handled automatically. However, this implicit treatment of the changes in interface topology does not necessarily imply that they represent the physical reality. For these reasons adaptive grid methods have also been developed, where the domain discretization is adjusted to follow in a more accurate way the different spatial scales and the temporal evolution of the flow structures [15–18].

The computation of the singular capillary force is usually a rather delicate task since the local lack of balance with the pressure gradient is at the origin of the so-called spurious currents which can affect the interface dynamics, both with the continuum surface force (CSF) method [19], and the ghost fluid method (GFM) [20], that imposes sharper boundary conditions on the interface. However, several improvements have been accomplished in the last years with finite volumes and the fractional-step pressure-correction method to integrate in time the Navier–Stokes equations. Firstly, with the incremental form of the predictor step [21], the pressure gradient and the capillary force partially balance each other, and exactly for the spatially-continuous equations in the case of Laplace's law. Secondly, the balanced-force approach has shown that spurious currents are greatly reduced if the pressure gradient and the capillary force are estimated at the cell faces and if the spatial discretization of these two terms is consistent [22,23,14]. More particularly, for a circular droplet Laplace's law is exactly satisfied if the analytical curvature is used, while the system relaxes by viscous damping towards a numerical equilibrium [14] when the curvature is estimated with the height function method. In this case the initial perturbation is determined by the numerical approximation of the curvature which is not constant along a circular interface.

In previous papers [24,25] we have shown that no spurious currents are generated also with a finite element approach if the analytical curvature is considered. In this framework Laplace's law is readily recovered and the force balance on the cell faces is implemented in a natural way with the integration by parts of the Navier–Stokes equations. In this paper we propose a multilevel VOF approach combined with a FEM flow solver that improves the representation of the interface by using more than one linear interface in each cell of the coarse grid, assuring in this way that the automatic breakup and merging events occur at spatial scales smaller than the coarse grid spacing. This paper addresses mainly situations where the spatial scales of the flow are fully resolved but the interface still requires a higher resolution. These situations are rather common, for example a circular fluid body requires a radius of curvature bigger than five grid spacings even in very simple flows, such as translations and solid-body rotations, in order not to be distorted to such an extent that the normal and curvature computations are not enough accurate. In these conditions the grid refinement necessary for the VOF algorithm would be too heavy when extended to the pressure and velocity fields. The multilevel approach reduces the interface discontinuity at the boundary of the cells of the coarse grid by increasing the actual resolution, and it computes more accurately the capillary force on the interface. As a result the balance between discontinuous forces and singular terms is more precise and spurious currents are reduced. The use of an auxiliary mesh to compute a number of geometrical and physical quantities is not new but it has been used to improve the evaluation of the normal, curvature and surface tension stresses [26,27].

The approach presented in this paper is somewhat similar to an adaptive refinement method, however it differs deeply as the mesh subdivisions are only virtual and it does not require any effort for the modification of the mesh, and any extra storage and communication of the flow variables between different levels. In some cases, when the interfacial curvature varies over orders of magnitude and all the different scales need to be resolved, this methodology becomes clearly inefficient if compared to an adaptive mesh refinement and it should be combined with some other approach. However in many cases this further refinement, with respect to the velocity and pressure spatial scales, is required more or less uniformly along the interface for a smoother representation of the interface itself [27]. Since the number of cells cut by the interface is usually very low with respect to the total number of cells of the very refined grid, we have implemented a sparse matrix approach in order to achieve a very compact memorization of the color function data at the highest resolution. In this study we consider the evolution of an axisymmetric jet in cylindrical coordinates. The principal curvature in the r, z plane is computed with the Laplace–Beltrami operator, the other curvature is computed on the interface and then extended radially for the volumetric computation of the capillary force.

In Section 2 we introduce the governing equations and discuss the variational formulation and the multilevel approach. In Section 3 we present the compact storage scheme of the color function data on the fine grid and the numerical implementation of the capillary force. In Section 4 we first validate our approach against the analytical linear dispersion relation for jets and then present the results of the simulations of an axisymmetric jet injected in a gas environment, with different frequencies and inlet velocities. Finally we present our conclusions.

2. Governing equations, variational formulation and finite element model

2.1. Governing equations

In Fig. 1 we consider a typical situation where an axisymmetric liquid jet (reference phase) is injected in a gas environment (secondary phase). Let $\Omega \subset R^3$ be a cylindrical domain with boundary Γ which contains an inlet area, $\Gamma_{in} \subset \Gamma$, and an