International Journal of Heat and Fluid Flow 53 (2015) 98–112

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Towards numerical simulations of supersonic liquid jets using ghost fluid method

Sahand Majidi, Asghar Afshari *

School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran, Iran

ARTICLE INFO

Article history:
Received 23 August 2014
Received in revised form 13 January 2015
Accepted 13 March 2015

Keywords:
Supersonic liquid jets
Compressible multiphase flow
Ghost fluid method
Level-set

ABSTRACT

A computational tool based on the ghost fluid method (GFM) is developed to study supersonic liquid jets involving strong shocks and contact discontinuities with high density ratios. The solver utilizes constrained reinitialization method and is capable of switching between the exact and approximate Riemann solvers to increase the robustness. The numerical methodology is validated through several benchmark test problems; these include one-dimensional multiphase shock tube problem, shock–bubble interaction, air cavity collapse in water, and underwater-explosion. A comparison between our results and numerical and experimental observations indicate that the developed solver performs well investigating these problems. The code is then used to simulate the emergence of a supersonic liquid jet into a quiescent gaseous medium, which is the very first time to be studied by a ghost fluid method. The results of simulations are in good agreement with the experimental investigations. Also some of the famous flow characteristics, like the propagation of pressure-waves from the liquid jet interface and dependence of the Mach cone structure on the inlet Mach number, are reproduced numerically. The numerical simulations conducted here suggest that the ghost fluid method is an affordable and reliable scheme to study complicated interfacial evolutions in complex multiphase systems such as supersonic liquid jets.

1. Introduction

Several numerical methods are available to simulate compressible multiphase flows most of which have proven to be highly successful in simulating complex flow phenomena such as shock bubble interaction, underwater explosion, high-speed drops impacting rigid surfaces, and among others (Chang and Liou, 2007; Johnsen and Colonius, 2006; Haller et al., 2003; Marquina and Mulet, 2003).

Current approaches for modeling interfacial flows can conveniently be classified as Lagrangian and Eulerian-based methods. In Lagrangian methods, one phase is assumed to be dispersed in another phase and is modeled as Lagrangian particles in continuous fluids (Dukowicz, 1980). Therefore, interfacial evolutions are not resolved directly. In contrast, Eulerian methods consider both phases as continuum separated by a material interface (Crowe, 2005).

Eulerian approaches are generally classified into diffuse interface methods (DIM) and sharp interface methods (SIM). In diffuse interface methodology,the phase interface is described as a numerically diffused zone with smooth transition of physical quantities across the interface (Abgrall and Karni, 2001; Ansari and Daramizadeh, 2013). On the contrary, sharp interface methods try to determine the interface location as exactly as possible. Therefore, interface thickness is intended to be diminished resulting in a sharp or even discontinuous distribution of the material quantities over the interface (Gorokhovsky and Herrmann, 2008; Scardovelli and Zaleski, 1999; Tryggvason et al., 2001).

Sharp interface methods can be separated into two major categories depending on the way interface deformations are accounted for. In interface tracking, the material interface is tracked explicitly through a set of inter-connected Lagrangian marker points which form two-dimensional unstructured surface elements on an underlying Cartesian grid (Glimm et al., 1999; Unverdi and Tryggvason, 1992). Although accuracy is an advantage, topological changes such as breakup and merge are difficult to handle and the method requires frequent Lagrangian particle redistribution, surface mesh reconstruction, and adaptation. A simpler alternative is to capture the interface on a fixed grid using a phase indicator scalar variable. This approach can be subdivided into two main methods: volume of fluid method (Scardovelli and Zaleski, 1999) and level-set method (Sethian and Smereka, 2003). Unlike front tracking the

* Corresponding author.
E-mail address: afsharia@ut.ac.ir (A. Afshari).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2015.03.002
0142-727X/

interface is not explicitly tracked and some loss of small-scale flow features is therefore inevitable.

Volume of fluid (VOF) methods rely on the convection of a characteristic function as a phase indicator which is set equal to zero in one of the phases and equal to one in the other phase. Many different algorithms have been developed for reconstructing the interface at each time step using the indicator function, among which SLIC-VOF (Noh and Woodward, 1976), PLIC-VOF (Rider and Kothe, 1998), LVIRA (Puckett et al., 1997), and ELVIRA (Pilliod and Puckett, 2004) are the most famous methods. The VOF method, despite its simplicity in application, extension to three-dimensions, and intrinsic mass preservation, is notorious for having difficulties in calculating curvature and normal vectors to the interface due to its inherent discontinuity.

Level-set methods (Enright et al., 2002; Herrmann, 2008; Olsson et al., 2007; Sussman et al., 1994) on the other hand, define their characteristic variable as a signed distance function to the interface, propagating in the computational domain with the speed obtained from the fluid's velocity field. Hence, they handle interfacial evolutions automatically and interface curvature and normal are dealt with easily. Level-set method is thus chosen as the interface capturing scheme in this study. However, in contrast to the VOF methodology, level-set methods are not inherently mass preserving. In this study, the level set method coupled with constrained reinitialzation algorithm of Hartmann et al. (2010) is employed. Although, this algorithm is optimized to maintain the constancy of the interface position during the reinitialization, mass preservation is not guaranteed as will be shown later in the present paper.

Interface capturing methods are most easily applied to multiphase compressible flows by coupling them with ghost fluid methods in which each fluid zone is discretized and solved separately. The interface does not appear directly in this formulation but through the boundary conditions that are applied at the material interface. Fedkiw et al. (1999) proposed a GFM utilizing the fact that pressure and normal velocity component are continuous across the interface. Therefore, in their algorithm, pressure and velocity of ghost cells for each phase are directly replaced by the ones corresponding to the other phase at the same location. This method, although simple and easy to code, is not robust in simulating flows with strong shocks, large density ratios, and phases with different equations of state.

In order to remedy the defects of Fedkiw's GFM, Riemann solvers were applied at the interface to construct the ghost values (Hu and Khoo, 2004; Liu et al., 2005, 2003). Sambasivan and Udaykumar (2011) applied mesh refinement in their GFM to improve the quality of interface capturing and mass conservation of the scheme. Houim and Kuo (2013) extended GFM to incorporate viscous stresses and surface tension. In addition, there have been some efforts to use alternative techniques instead of level-set method to track the interface; for instance, Terashima and Tryggvason (2009) coupled Fedkiw's GFM to a front-tracking technique.

In this study, a ghost fluid method is developed based on previous numerical studies with appropriate modifications introduced in formulas and implementation in order to improve the performance of the scheme. The developed GFM is then coupled to a level-set method with a constrained re-initialization scheme which minimizes the topological changes during the re-initialization process. Several test cases are used to evaluate the performance and capability of the scheme in capturing intricate flow features. The ultimate objective of this paper is to conduct a simulation of the evolution of a supersonic liquid jet in a quiescent gaseous medium to assess the accuracy and robustness of the ghost fluid method.

Development of supersonic liquid jets have not been a subject of extensive numerical studies. Nevertheless, Zakrzewsky et al. (2004) were among the first who tried to simulate the propagation of supersonic liquid jets along with experimental study. Their simulation was carried out using the commercial CFD code FLUENT. However, their interface capturing method suffered from excessive numerical diffusion and no numerical results were presented depicting the formation of bow-shock leading the liquid jet.

Im et al. (2009) applied discrete particle technique for solving hydrodynamic equations governing the liquid jet flow. They also solved Eluer equations governing the flow induced in surrounding gas using conservation element-solution element (CESE) (Zhang et al., 2002). Anyhow, their method was not capable of visualizing some of the experimentally observed phenomena occurring at the shock- liquid jet interaction such as Mach waves radiated from the interface in the gaseous medium. This can be attributed to their modeling of liquid phase as discrete Lagrangian particles instead of using a more realistic continuum model.

To the best of our knowledge, GFM has not been previously employed to simulate the evolution of supersonic liquid jets in quiescent gaseous environment.

2. Numerical scheme

The governing equations for non-reacting compressible two phase flow without phase change are:

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = 0 \tag{1}$$

where

$$\mathbf{U} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} \rho u \\ \rho u^2 + P \\ \rho u v \\ u(\rho E + P) \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} \rho v \\ \rho u v \\ \rho v^2 + P \\ v(\rho E + P) \end{bmatrix} \tag{2}$$

Here u and v are velocity components, ρ and P represent density and thermodynamic pressure, $E = e + 0.5(u^2 + v^2)$ is the total specific energy and e is specific internal energy. Stiffened gas equation of state is used to relate pressure to specific internal energy, which describes pressure as a function of internal energy and density:

$$P = \rho e(\gamma - 1) - \gamma P_\infty \tag{3}$$

where γ is the specific heat ratio and P_∞ is a constant which accounts for the material stiffness. In order to capture interface deformations, a level-set method is employed. The level set equation is given by:

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0 \tag{4}$$

where ϕ is the level-set variable and $\mathbf{u}$ is the velocity vector.

Explicit third-order TVD Runge-Kutta algorithm is used to perform time integration. Convective fluxes are approximated by a Godunov scheme wherein local Riemann problem is solved at each cell boundary using an exact or an approximate Riemann solver in order to determine the correct upwind fluxes. Based on this scheme, explicit descretization of the one-dimensional form of (1) at each Runge-Kutta stage results in the following conservative formula:

$$\mathbf{U}_i^{n+1} = \mathbf{U}_i^n - \frac{\Delta t}{\Delta x}\left[\mathbf{F}_{i+\frac{1}{2}}(\mathbf{U}_i^n, \mathbf{U}_{i+1}^n) - \mathbf{F}_{i-\frac{1}{2}}(\mathbf{U}_i^n, \mathbf{U}_{i-1}^n)\right] \tag{5}$$

Numerical fluxes in the above formulation are determined based on the HLLC approximate Riemann solver (Toro, 2009). First the fastest left-running and right-running signal velocities are defined as:

$$S_R = \max(u_L + c_L, u_R + c_R), \quad S_L = \min(u_L - c_L, u_R - c_R) \tag{6}$$