Journal of Computational Physics 228 (2009) 4012–4037

Contents lists available at ScienceDirect

Journal of Computational Physics

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

A front-tracking/ghost-fluid method for fluid interfaces in compressible flows

Hiroshi Terashima *, Grétar Tryggvason *

Mechanical Engineering Department, Worcester Polytechnic Institute, 100 Institute Road, Worcester, MA 01609-2280, United States

ARTICLE INFO

Article history:
Received 10 March 2008
Received in revised form 9 February 2009
Accepted 15 February 2009
Available online 1 March 2009

Keywords:
Compressible multiphase flow
Front-tracking method
Ghost-fluid method
Fluid interface
Compressible gas–water flow

ABSTRACT

A front-tracking/ghost-fluid method is introduced for simulations of fluid interfaces in compressible flows. The new method captures fluid interfaces using explicit front-tracking and defines interface conditions with the ghost-fluid method. Several examples of multiphase flow simulations, including a shock–bubble interaction, the Richtmyer–Meshkov instability, the Rayleigh–Taylor instability, the collapse of an air bubble in water and the breakup of a water drop in air, using the Euler or the Navier–Stokes equations, are performed in order to demonstrate the accuracy and capability of the new method. The computational results are compared with experiments and earlier computational studies. The results show that the new method can simulate interface dynamics accurately, including the effect of surface tension. Results for compressible gas–water systems show that the new method can be used for simulations of fluid interface with large density differences.

1. Introduction

The dynamics of interfaces separating different fluids in compressible flows is of interest in several scientific fields as diverse as astrophysics and geophysics. It is also of significant importance in many engineering applications. An in-depth understanding of multiphase flow in supersonic combustion is, for example, desirable for the development and effective operation of supersonic ramjet engine, and many other combustion systems involving fuel drops in high-speed airflow, where liquid jets must be atomized efficiently [1,2].

For computing compressible multifluids, early algorithms have treated discontinuous material interfaces with the γ (the ratio of specific heats)-model [10], the mass fraction model [12,50], or a level-set function [7] in order to identify each fluid, coupled to the Euler equations. These algorithms, usually based on classical shock-capturing methods, however, have suffered from unphysical oscillations developed at material interfaces. Abgrall [12] identified the cause of those spurious oscillations and proposed a quasi-conservative method based on the mass fraction model. This has been extended to more general equation of state [13]. Karni [10] also proposed a remedy to avoid pressure oscillations using a non-conservative scheme based on the primitive variables. Successful applications of Karni's method can be seen, for example, in computations of the interaction of a shock wave with a helium bubble. This problem has now become a classical benchmark for multidimensional compressible multifluids algorithms. Karni's approach was modified later in [11] to capture strong shocks, using the pressure evolution equation and the level-set equation.

A different approach for compressible multifluids simulations has been pursued by Glimm and co-authors [3,4], who have been using explicit marker points for capturing waves (such as contact discontinuity). The Glimm's front-tracking method, where selected waves are explicitly represented in the discrete form of the solution, Riemann solutions are constructed near

* Corresponding authors. Tel.: +1 508 831 6803 (H. Terashima), Tel.: +1 508 831 5759; fax: +1 508 831 5680 (G. Tryggvason).
E-mail addresses: htera@wpi.edu (H. Terashima), gretar@wpi.edu (G. Tryggvason).

0021-9991/$ - see front matter
doi:10.1016/j.jcp.2009.02.023

the fluid interface to propagate the solution in the normal and tangential directions using ghost nodes. For recent applications see the simulations of the three-dimensional Rayleigh–Taylor instability described in [5]. Similar idea of using Riemann solutions was used by Cocchi and Saurel [6]. Their algorithm consists of a predictor step and a corrector step. In the corrector step, a Riemann problem is solved at material interfaces in order to correct the solutions from the predictor step (where a standard Godunov type scheme is used) that generally generates numerical diffusion or spurious oscillations. More recently, following the strategy of using level-set function and adaptive mesh refinement, Nourgaliev et al. [9] have also attempted to avoid numerical oscillations near fluid interfaces by solving a suitable Riemann problem for interface cells.

Ghost nodes have more recently been used in the ghost-fluid method of Fedkiw and collaborators [8]. Fedkiw's ghost-fluid method is a level set based algorithm, which treats the interface as a moving boundary. By extrapolating the value of the discontinuous variables, such as entropy, across the fluid interface when solving the governing equations, one can reduce smearing of discontinuous variables such as density and other material properties. The method can also prevent pressure oscillations since a ghost fluid is set to be thermodynamically identical to the corresponding real fluid. Fedkiw's ghost-fluid method is easy to implement, robust, and can easily handle different fluids with different equations of state. Recently, however, it has been reported that pressure oscillations occur for flows with high density or pressure ratios and improved versions of Fedkiw's ghost-fluid method have therefore been proposed [15,24] to deal with such cases. In [15], for example, Riemann solutions are imposed for the ghost nodes in order to determine accurate interface status.

As other approach for compressible multifluids simulations, Chang and Liou [32] have recently developed a stratified flow model, which is capable of incorporating compressible gases and liquids.

So far the ghost-fluid method has only been used with the level-set methods to track the interface. It can, however, be used with other techniques for following the interface motion and in this paper we combine Fedkiw's ghost-fluid method with a front-tracking method. The front-tracking method originally developed by Tryggvason and co-workers [22,23] is used here. In this method, fluid interfaces are explicitly tracked by connected marker points. This method has been successfully applied to many multiphase flow problems, but so far all applications have been limited to incompressible flows. We therefore extend Tryggvason's front-tracking method here to solve the compressible flow equations. The concept of the ghost-fluid method, in which each fluid is solved separately, can match with the spirit of the front-tracking method, in which each fluid is distinguished sharply by explicit marker points. Another extension of Tryggvason's front-tracking method, using Fedkiw's ghost-fluid method to handle compressible flows, can be found in a recent study of Hao and Prosperetti [26]. There, the compressible fluid was limited to a simple gas model whereas here we use the full compressible Euler or Navier–Stokes equations both for the fluids.

In this paper, we limit our development and applications to two-dimensional flow fields. The proposed numerical techniques, however, can be extended to three dimensions in the same way, although the coding for restructuring front will be more complicated [23].

2. Governing equations

The governing equations for flow fields are the two-dimensional compressible Navier–Stokes equations written in Cartesian coordinate

$$\partial_t \boldsymbol{Q} + \partial_x \boldsymbol{E} + \partial_y \boldsymbol{F} = Re'^{-1}\left(\partial_x \boldsymbol{E}_v + \partial_y \boldsymbol{F}_v\right) + Fr^{2,\prime -1}\boldsymbol{G} + We'^{-1}\boldsymbol{S}. \tag{1}$$

Here, $\boldsymbol{Q}$ is the vector of conserved variables, $\boldsymbol{E}$ and $\boldsymbol{F}$ are the convective flux vectors and $\boldsymbol{E}_v$ and $\boldsymbol{F}_v$ are the flux vectors for the viscous terms:

$$\boldsymbol{Q} = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ E \end{bmatrix}, \quad \boldsymbol{E} = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho u v \\ (E+p)u \end{bmatrix}, \quad \boldsymbol{F} = \begin{bmatrix} \rho v \\ \rho v u \\ \rho v^2 + p \\ (E+p)v \end{bmatrix}, \quad \boldsymbol{E}_v = \begin{bmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ \tau_{xx}u + \tau_{xy}v - q_x \end{bmatrix}, \quad \boldsymbol{F}_v = \begin{bmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ \tau_{yx}u + \tau_{yy}v - q_y \end{bmatrix}, \tag{2}$$

where τ_{ij} are the shear stresses and q_i is the heat conduction. $\boldsymbol{G}$ is a body force (in most cases gravity). $\boldsymbol{S}$ is a source term for the surface tension, included in the momentum equations as a singular body force per unit volume:

$$\boldsymbol{S} = -\sigma \int_{\delta s} \kappa \boldsymbol{n} \delta(\boldsymbol{x} - \boldsymbol{x}_f) ds. \tag{3}$$

Here, σ is the surface tension, κ is twice the mean curvature, and $\boldsymbol{n}$ is a unit surface normal vector. The contribution of the surface tension is limited to fluid interfaces, as indicated by the delta function, δ. In the argument of δ, $\boldsymbol{x}$ is a point at which the equations are evaluated, and $\boldsymbol{x}_f$ is a point on the interfaces. Thus, in this way the surface tension effect is smeared in this study, but one may implement the surface tension effect in a sharp way using pressure difference done in [51], which will be more consistent with our method.

In addition to Eq. (1), the stiffened gas equation of state [13] is used:

$$p = (\gamma - 1)\rho e - \gamma \Pi. \tag{4}$$