Journal of Computational Physics 371 (2018) 67–91

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A robust and efficient finite volume method for compressible inviscid and viscous two-phase flows

Aditya K. Pandare, Hong Luo *

Department of Mechanical and Aerospace Engineering, North Carolina State University, Raleigh, NC, 27695, USA

A R T I C L E   I N F O

Article history:
Received 13 October 2017
Received in revised form 30 March 2018
Accepted 9 May 2018
Available online xxxx

Keywords:
Two-fluid model
Six equation
All-speed methods

A B S T R A C T

A robust and efficient density-based finite volume method is developed for solving the six-equation single pressure system of two-phase flows at all speeds on hybrid unstructured grids. Unlike conventional approaches where an expensive exact Riemann solver is normally required for computing numerical fluxes at the two-phase interfaces in addition to AUSM-type fluxes for single-phase interfaces in order to maintain stability and robustness in cases involving interactions of strong pressure and void-fraction discontinuities, a volume-fraction coupling term for the AUSM$^+$-up fluxes is introduced in this work to impart the required robustness without the need of the exact Riemann solver. The resulting method is significantly less expensive in regions where otherwise the Riemann solver would be invoked. A transformation from conservative variables to primitive variables is presented and the primitive variables are then solved in the implicit method in order for the current finite volume method to be able to solve, effectively and efficiently, low Mach number flows in traditional multiphase applications, which otherwise is a great challenge for the standard density-based algorithms. A number of benchmark test cases are presented to assess the performance and robustness of the developed finite volume method for both inviscid and viscous two-phase flow problems. The numerical results indicate that the current density-based method provides an attractive and viable alternative to its pressure-based counterpart for compressible two-phase flows at all speeds.

1. Introduction

Multiphase flows arise in a variety of engineering applications ranging from atomized fuel flow in internal combustion engines to nucleate boiling in water reactors. Computation of flows involving multiphase physics is thus a topic of significant research efforts and multiple approaches exist to deal with these types of problems. The two types of methods used for multiphase flows are the interface tracking and interface capturing methods. Interface tracking methods, as the name suggests, employ actual tracking of the interface between two phases (viz. bubble or droplet surfaces) and smoothing the fluid properties across this interface. The level-set, volume-of-fluid and front-tracking are some famous interface tracking methods. The Arbitrary Lagrangian–Eulerian method has also been used to capture multiphase interfaces accurately [1,2]. Interface capturing methods, on the other hand, dynamically "capture" the interfaces; just like a standard finite-volume or discontinuous Galerkin method would capture shocks and contact discontinuities without any special treatment. This means

* Corresponding author.
E-mail address: hong_luo@ncsu.edu (H. Luo).

https://doi.org/10.1016/j.jcp.2018.05.018

that each phase is treated as a separate continuum, and that there is no clear distinction between cells containing one phase or the other. Each cell contains a fraction of each phase, denoted by the volume fraction α. This approach is thus also termed as the "interpenetrating continua" approach, or the Effective Field Model (EFM). Both these approaches have their benefits and drawbacks. It is obvious that the EFM is unable to compute individual interfaces between two phases like the interface tracking methods. This may be important in applications where the interface details like bubble-dynamics are the focus. But the obvious drawback of interface tracking methods is that they are computationally prohibitive when applied to flows with large number of interfaces, such as bubbly flows through pipelines. Also, implementation of these methods can prove to be quite complicated in such situations, due to phenomena like bubble-coalescence or breakup and condensation. In situations where average-states of the multiphase flow are of interest and resolution of interfaces is not important, the EFM is used.

The EFM is derived from individual continuum equations for each phase, called the local instant formulation, and then applying an averaging procedure [3]. Assuming velocity and thermal non-equilibrium but pressure equilibrium leads to the 6-equation Wallis model [4] of two-fluid flows. Solving the 6-equation model is a challenging problem. Legacy multiphase codes use such pressure-based methods and resort to numerical diffusion and Picard-type iterations to make the system stable. Many of these methods resort to adding dissipation by the means of staggered grids and other techniques to obtain stable numerical results [5,6]. These methods are usually termed as pressure-based algorithms since the working variable is pressure. They fall under the general class of operator-splitting type of methods. Pressure-based methods are better conditioned for low *Ma* flows, since at these speeds, the density is practically constant, and small errors in density (due to spatial or temporal discretizations) translate to large errors in pressure. Also, the pressure-based methods eliminate the speed of sound by reformulating the continuity and momentum equations by assuming that the velocity field is solenoidal ($\nabla \cdot \boldsymbol{v} = 0$). This results in a CFL criterion based solely on the flow velocity, independent of the acoustic speed, and larger time-step sizes leading to faster convergence. These pressure-based algorithms have been used successfully to obtain very accurate results for low speed flows, both for single-phase flows [7,8] and two-phase flows [9,6]. However, since these methods use operator-splitting to decouple the momentum and energy equations, compressibility can not be rigorously treated. The incompressibility assumption is deeply ingrained in their formulation. These algorithms result in significant inaccuracies and instabilities when high-temperature phenomena such as boiling and other problems involving large density gradients need to be simulated.

An alternative, potentially more promising approach is to use the density-based (fully compressible) method for multiphase flows. However, there are a number of issues in solving the 6-equation model by use of a density-based method:

- Extremely low-speed flows ($Ma < 0.001$): Many multiphase flows in engineering applications are very low Mach number flows. This is known to cause difficulty in convergence. Also, the time-steps satisfying stability conditions (acoustic CFL) are too small to be feasible. The Riemann fluxes used in conventional compressible density-based methods tend to be highly diffusive in the $Ma < 0.01$ flow regime.
- High contribution of source terms: Interfacial force terms such as drag, interphasic mass transfer, etc. have such large contribution to the residual vector that the stable time-steps are further lowered.
- Non-hyperbolic nature of the standard (6-equation) two-fluid model: The two-fluid model has two possibly complex eigenvalues in its standard form. Naive discretizations employing density-based (hyperbolic) methods fail due to this.
- Non-conservative spatial and temporal derivatives: If discretized incorrectly, these can prevent preservation of stationary contact discontinuities.

One way to resolve low *Ma* convergence issue is to solve the two-fluid system in a fully-implicit way using a primitive variable transformation matrix. Nourgaliev and co-workers [10,11] have used a $[p, \boldsymbol{v}, T]$-formulation to solve near-incompressible flows using fully compressible density-based solver. Using pressure as a primitive variable makes the implicit density-based system better conditioned. Treating the source terms due to interfacial forces and phase-changes implicitly also relaxes the time-step restrictions, leading to faster convergence times.

The 6-equation model is non-hyperbolic, and cannot be directly solved by a conventional density-based method. A detailed eigenvalue analysis of the Wallis model can be found in [12]. Even so, the non-hyperbolic 6-equation model has been used to simulate inviscid two-fluid flow first by Toumi [13] and then by Chang, Liou and co-workers [14–17]. By adding an interface pressure term suggested by Stuhmiller [18], they prove that the system can be rendered hyperbolic [19]. They use the stratified-flow model to discretize the non-conservative terms in the system and an all-speed two-phase extension of the AUSM flux, called the AUSM$^+$-up, in addition to an exact Riemann solver to discretize the inviscid fluxes. The exact Riemann solver was only used in situations where the jump in the volume-fraction at the cell-faces is substantial. It has also been proven in [14] that this choice of flux, coupled with the stratified flow discretization of the pressure flux and other non-conservative terms, satisfies the pressure non-disturbance condition (also known as Abgrall's criterion [20,21] or well-balancedness [22]), which is essential for the solver to maintain a stationary contact discontinuity. However, the iterative procedure involved in the exact Riemann solver makes this method quite expensive. Kitamura and Nonomura [23] replaced the exact Riemann solver with a two-fluid HLLC flux. This resulted in a relatively inexpensive flux function. However, knowledge of the complete eigenstructure of the system is a must for such a flux function. This might not be easily available, when more complex terms such as virtual mass are included in the governing equations. Niu [24,25] has used a primitive variable solver to efficiently solve the Riemann problem, albeit using a Newton iteration in the procedure. Other approaches