Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **9**, No. 5, pp. 1111-1132

DOI: 10.4208/aamm.OA-2016-0084
October 2017

A Comparison Study of Numerical Methods for Compressible Two-Phase Flows

Jianyu Lin[1], Hang Ding[1,*], Xiyun Lu[1] and Peng Wang[2]

[1] *Department of Modern Mechanics, University of Science and Technology of China, Hefei, Anhui 230026, China*
[2] *Institute of Applied Physics and Computational Mathematics, Beijing 100088, China*

Received 8 August 2016; Accepted (in revised version) 20 October 2016

Abstract. In this article a comparison study of the numerical methods for compressible two-phase flows is presented. Although many numerical methods have been developed in recent years to deal with the jump conditions at the fluid-fluid interfaces in compressible multiphase flows, there is a lack of a detailed comparison of these methods. With this regard, the transport five equation model, the modified ghost fluid method and the cut-cell method are investigated here as the typical methods in this field. A variety of numerical experiments are conducted to examine their performance in simulating inviscid compressible two-phase flows. Numerical experiments include Richtmyer-Meshkov instability, interaction between a shock and a rectangle SF_6 bubble, Rayleigh collapse of a cylindrical gas bubble in water and shock-induced bubble collapse, involving fluids with small or large density difference. Based on the numerical results, the performance of the method is assessed by the convergence order of the method with respect to interface position, mass conservation, interface resolution and computational efficiency.

AMS subject classifications: 76E17, 76L05, 76T10

Key words: Diffuse interface method, modified ghost fluid method, cut-cell method, compressible flow, two-phase flow.

1 Introduction

In past decades a variety of numerical methods have been developed for simulation of inviscid compressible two-phase flow. Since these methods allow for complicated interface deformations or topology changes of interfaces, they have been extensively used to investigate the high-speed flows involving shock-interface interactions, and therefore become

*Corresponding author.
Email: linjiany@mail.ustc.edu.cn (J. Y. Lin), hding@ustc.edu.cn (H. Ding), xlu@ustc.edu.cn (X. Y. Lu), wangpei@iapcm.ac.cn (P. Wang)

powerful numerical tools to elucidate the underlying fluid mechanisms. In these methods it is the key issue to appropriately deal with the jump conditions at the fluid-fluid interfaces. Depending on the manner of modeling the interface, the numerical methods can be generally grouped into two types: diffuse interface methods and sharp interface methods.

In the diffuse interface methods [1–7] the interface between two immiscible fluids is modeled by an interface region of finite thickness, in which the fluids are allowed to mix to some extent. For the fluid mixture in the diffuse interface region, it is very important to give consistent thermodynamic laws [3], which essentially resolve the jump conditions at the interface. The interface in the diffuse interface method can be represented by the field of different parameters, e.g., mass fraction [2], volume fraction [4] or specific heat ratio [8]. In order to reflect the discontinuous nature of the immiscible fluids, the thickness of the diffuse interface is supposed to be much smaller than the characteristic length scale of the flow. However, the non-uniform flow field would stretch or compress the diffuse interface region. How to suppress the interface diffusion remains a big challenge in the diffuse interface simulation of compressible multiphase flows, and recent efforts can be found in [9–14].

In the sharp interface methods the interface is treated as a sharp contact discontinuity. Two typical sharp interface methods on Cartesian meshes are: ghost fluid methods [15–19] and cut-cell methods [20, 21]. The ghost fluid methods generally resolve the fluid flows in the finite difference framework, in particular at the Cartesian cells that contain the interface, where the discretization of governing equation in one fluid requires the information of the flow variables at the cells in the other fluid (or *ghost cells*). It is suggested by Fedkiw et al. [15] that the pressure and velocity can be copied from the other fluid directly while the density is obtained by extrapolating entropy from the side of the bulk fluid. Liu et al. [16] proposed a modified ghost fluid method, which includes a Riemann solver in the calculation of the flow variables at the ghost cells, to provide a non-oscillatory pressure field in the presence of strong shock and detonation waves at the interface. In the cut cell methods complex interfaces are projected onto a fixed structured mesh, and for two-dimensional computations the interface can be effectively represented by a number of piecewise linear segments that split the corresponding Cartesian cells. Consequently, a set of unstructured cells are generated in the vicinity of the interface [21–23]. Therefore, the interface coincides with the cell faces of the unstructured cells, and the jump conditions across the interface can be resolved by solving a local Riemann problem at the cell faces. In order to generate unstructured interface cells and eliminate the unnecessarily small ones, cut-cell methods often involve with complicated geometrical algorithms to split Cartesian cells and merge unstructured cells [23]. For the sharp interface methods, the interface evolution can be modeled by any popular interface tracking methods such as level-set [15], front tracking [19] and volume-of-fluid [24].

Despite of their success in simulating inviscid compressible two-phase flows, these methods have not been systematically compared yet, and therefore, it is not clear for a particular method about its advantages and disadvantages relative to the other ones.