Computers & Fluids 90 (2014) 147–154

Contents lists available at ScienceDirect

Computers & Fluids

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

Accuracy improvement of axisymmetric bubble dynamics using low Mach number scaling

Yang Liu *, Mingyu Sun [1]

Institute of Fluid Science, Tohoku University, Sendai, Miyagi 980-8578, Japan

ARTICLE INFO

Article history:
Received 25 April 2013
Received in revised form 18 September 2013
Accepted 19 November 2013
Available online 3 December 2013

Keywords:
Bubble dynamics
Volume of fluid
Numerical sound speed
Low Mach number
Riemann solver

ABSTRACT

Bubble collapse and associated shock wave emission are characterized by the compressibility of both gas and liquid. The bubble motion may, however, be in the low Mach number flow regime when the bubble is near the maximum size at the early stage and at its full rebound. Although it is quite well known that a compressible flow solver encounters difficulties in the low Mach number regime, the influence of the low Mach number on the simulation of bubble collapse and rebound is not clear. In the present work, an axisymmetric compressible solver based on the acoustic Riemann solver is used to simulate the dynamics of bubble motion. The artificial viscosity terms in the Riemann solver are rescaled for the low Mach regime by following the concept of numerical sound speed, which was originally developed in the AUSM family scheme. The numerical results are compared with the solutions of the Rayleigh model for bubble collapse and the Keller–Miksis model for bubble collapse and rebound. It is found that the low Mach number scaling improves the accuracy of the size of a collapsing bubble considerably.

1. Introduction

Since the work of Lord Rayleigh who solved the problem of the collapse of an empty cavity in infinite liquid [1], the problem of bubble collapse and rebound has extensively been studied by many researchers, because of its importance in industry, the erosion damage on ship propellers for instance. An overview of the theoretical works in this field is given in the review article by Plesset and Prosperetti [2]. If the surface tension, viscosity, compressibility and thermal effect are neglected, the motion of the bubble can be described with the Rayleigh equation [1]. The initial velocity of the bubble boundary is zero and in the early stage, this velocity is quite small, so the incompressible assumption is reasonable. Nevertheless, the Rayleigh equation would predict that the velocity of the bubble interface approaches infinity as the bubble radius approaching zero, which indicates that the liquid compressibility should not be neglected when the bubble radius gets very small compared to the initial bubble size. Shock wave emission in the liquid during the bubble collapse has been observed by experiments (e.g. [3]). For numerical investigation of the late stage of the bubble collapse, the compressibility has to be taken into consideration.

Several compressible numerical investigations of bubble collapse have been published in recent years. Müller et al. [4] compared two compressible gas–liquid two-phase flow methods for bubble dynamics simulation. One is the diffuse interface two-fluid method [5] which solves the Riemann problem for each cell interface to evolve density, momentum and energy in time and solves the gas fraction equation to track the location of the interface. The other is the level set combined with ghost fluid method [6]. The numerical results were compared with the Keller–Miksis model [7]. In their work, only the quasi-one-dimensional Euler equation was solved. The bubble collapses in both the free space and near a rigid wall are simulated by Müller et al. [8]. Johnsen and Colonius [9] investigated shock-induced and Rayleigh collapse of a bubble, using a fifth order accurate finite volume weighted essentially non-oscillatory (WENO) scheme to solve the Euler equation and the γ-based model of Shyue's [10] to track the bubble interface. Lauer et al. [11] investigated symmetric and asymmetric cavitation bubble dynamics using the conservative interface method of Hu et al. [12]. The mass transfer across the vapor–liquid interface was also included.

All above-mentioned schemes are explicit. Nagrath et al. [13] developed an implicit solver that combines the ghost fluid with the level set approach to treat the air–water interface, and they further studied the implosion and rebound of a small air bubble in water and reported that the bubble may deviate from spherical symmetry at the final stage of the bubble collapse. In order to combat with the sever time step at the low Mach number regime of bubble implosion, they employ the Rayleigh equation to

* Corresponding author. Present address: Institute of Structural Mechanics, China Academy of Engineering Physics, Mianyang, Sichuan 621000, China. Tel.: +86 18281436303.

E-mail addresses: yangl5@mail.ustc.edu.cn (Y. Liu), sun@fmail.ifs.tohoku.ac.jp (M. Sun).

[1] Principal corresponding author. Tel.: +81 0222175233.

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.11.024

provide the early solution until the gas Mach number reaches approximately 0.3, which then serves as the initial condition for the fully compressible solver.

In the low Mach number regime, a compressible solver encounters a few difficulties. The most critical problem is the lose of convergence rate and accuracy. There are at least two successful strategies to solve the problem. The preconditioning method [14–16] introduces a preconditioner matrix and alters the characteristics of the governing equations, so that all waves move at the similar speeds. Another method is to directly rescale the numerical viscosity that is built in the compressible flow solvers, so that the discretized equation may converge to the incompressible limit at the low Mach regime [17–21]. The concept of numerical speed of sound employed in the AUSM family schemes [17,18] is one of the successful methods in this category.

In the present work, the method of rescaling of numerical viscosity is introduced in an implicit compressible solver for the simulation of the bubble dynamics. The flow field is assumed to be axisymmetric. The Lagrange-remap method is used to solve the axisymmetric Euler equation. In the Lagrange step, the volume of fluid method, or more precisely the volume tracking method, is used to sharply resolve the gas–liquid interface. The numerical results are compared with the theoretical solutions and the numerical results in the literature. It is found that low Mach number scaling does improve the accuracy significantly in the simulation of bubble collapse and rebound.

2. Numerical method

A two-step procedure is followed in our simulation. First is the Lagrange step, in which the volume, momentum and energy of a Lagrangian particle or a control volume moving at the speed of flow velocity is updated. Then in the remap step, the conservative quantities of the particle are remapped back to the initial fixed Eulerian grid.

Consider the mass, momentum and energy conservation laws in three dimensional Lagrangian coordinate system

$$\frac{\partial}{\partial t}\int_{V(t)}\mathbf{U}dV+\oint_{S(t)}\mathbf{n}\cdot\mathbf{F}dS=0, \tag{1}$$

where $V(t)$ is a time-dependent control volume enclosed by the boundary $S(t)$, $\mathbf{n}$ is the outward unit vector normal to the boundary of the control volume, $\mathbf{U}$ is the vector of conservative variables, and $\mathbf{F}$ is the flux vector:

$$\mathbf{U}=\begin{pmatrix}1\\ \rho\mathbf{u}\\ \rho E\end{pmatrix},\quad \mathbf{F}=\begin{pmatrix}-\mathbf{u}\\ \mathbf{I}p\\ p\mathbf{u}\end{pmatrix}, \tag{2}$$

where ρ is the density, p is the pressure, $\mathbf{u}$ is the velocity vector, $\mathbf{I}$ is the unit tensor, $E=e+\mathbf{u}\cdot\mathbf{u}/2$ is the specific total energy, and e is the specific internal energy. For axially symmetric problems in which $\mathbf{U}$ is independent from the angular coordinate θ , and if the motion in circumferential direction is neglected (the velocity at the angular direction $u_\theta=0$), Eq. (1) can be expressed only in axial and radial (denoted by x and y hereafter) directions, with all physical quantities uniformly distributed in the circumferential direction.

The mass and energy conservation equation can be expressed in the 2D control volume in x–y plane:

$$m\frac{d}{dt}\frac{1}{\rho}+\oint_L\mathbf{N}\cdot\mathbf{u}ydL=0, \tag{3}$$

$$m\frac{d}{dt}E+\oint_L\mathbf{N}\cdot\mathbf{u}pydL=0, \tag{4}$$

where L is the line along the boundary of the 2D cell $ABCD$ as shown in Fig. 1, $\mathbf{N}$ is the outward unit vector normal to the boundary of this 2D cell, m is the mass of the material in the axisymmetric domain $ABCD-A'B'C'D'$.

The area weighted formulation [22] is used for the momentum equation, which can ensure spherical symmetry for one dimensional spherical flow:

$$\frac{m}{\bar{y}}\frac{d\mathbf{u}}{dt}+\oint_L p\mathbf{N}dL=0, \tag{5}$$

where $\mathbf{u}=(u\quad v)^T, u, v$ is the velocity component in x, y direction respectively;

$$\bar{y}=\frac{\int_A ydA}{\int_A dA} \tag{6}$$

is the averaged pseudo radius; A denotes the 2D cell $ABCD$.

Although a rectangular cell is shown in Fig. 1, Eqs. (3)–(5) can be applied to cells of any shape.

In order to close the system, the equation of state (EOS) for the fluids is needed. In present simulation, ideal gas EOS is used for gas, and Tait EOS is used for liquid:

$$p=B_0\left[\left(\frac{\rho}{\rho_c}\right)^n-1\right]+A_0, \tag{7}$$

where A_0, B_0, n, and ρ_c are parameters corresponding to materials. For water, the following values can be specified: $A_0=0.1$ Mpa, $B_0=331$ Mpa, $n=7.15$, $\rho_c=10^3$ kg/m^3; these parameters are used throughout present work.

To obtain the pressure and normal velocity at the interface of the 2D control cell, the acoustic linearized Riemann solver [23] is used:

$$p^*=\frac{\rho^l a^l p^r+\rho^r a^r p^l+\rho^l a^l\rho^r a^r(u_n^l-u_n^r)}{\rho^l a^l+\rho^r a^r}, \tag{8}$$

$$u_n^*=\frac{p^l-p^r+\rho^l a^l u_n^l+\rho^r a^r u_n^r}{\rho^l a^l+\rho^r a^r}, \tag{9}$$

where the superior $*$ denotes the value at the face of the 2D control cell; the superior l/r denotes the value at the left/right side of the face; the subscript n denotes the velocity component at $\mathbf{N}$ direction, $\mathbf{N}$ is the unit vector normal to the face, pointing from left to right; a is the sound speed.

The concept of numerical speed of sound [17] is employed to the acoustic linearized Riemann solver. A scaling function f_a is used to obtain the numerical sound speed $\tilde{a}$

$$\tilde{a}=f_a(M_o)a. \tag{10}$$

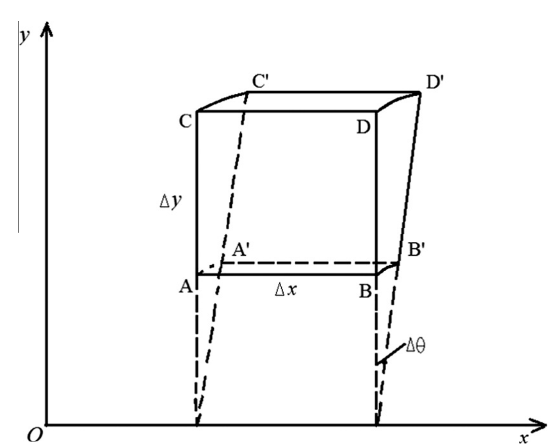

Fig. 1. Sketch of a control volume for axisymmetric problems, x, y are the axial and radial coordinate directions respectively.