Computers and Fluids 134–135 (2016) 196–214

Contents lists available at ScienceDirect

Computers and Fluids

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

Simulations of the shock waves and cavitation bubbles during a three-dimensional high-speed droplet impingement based on a two-fluid model

Yang-Yao Niu*, Hong-Wei Wang

Department of Aerospace Engineering, Tamkang University, New Taipei City, Taiwan, R.O.C.

ARTICLE INFO

Article history:
Received 27 October 2015
Revised 16 April 2016
Accepted 20 May 2016
Available online 24 May 2016

Keywords:
Compressible liquid
Shock wave
cavitation
Two-fluid model
AUSMD

ABSTRACT

In this paper, we investigate the aerodynamic characteristics inside a droplet impingement using a compressible two-fluid model. A hybrid type Riemann solver is proposed to compute numerical fluxes across the interfaces of gas–gas, liquid–liquid and gas-liquid flows in the considered flow-fields. Here, the compressible liquid flows with high Reynolds number value allow us to use an inviscid approach and neglect the surface tension effect under the assumption of high Weber number. Numerical results demonstrate the evolution of shock-front, rarefaction, cavitation inside the droplet and the contact periphery expands very quickly and liquid compressibility plays an important role in the initial formation of flow physics inside the liquid droplet. Grid independence study is performed. A secondary cavitation zone is simulated to appear near the wall due to the expansion wave propagating downward caused by the eruption of the main cavitation bubble near the top of the liquid droplet. We also found that the growth rate of the cavitation zone is independent of the impact flow velocity. The estimated maximum wall pressure against the incoming Mach number is shown to be closer to the theoretical data than any other previous analysis.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

The thermo-fluids of high-speed liquids impact onto a rigid surface is important to the industry applications such as spray coating and cooling, steam turbine blade operation, metal cutting of materials. [1–3]. During the process of the liquid droplet impingement, we can observe some physical phenomena, such as the interaction of propagating shock, interface, rarefaction waves, the formation and collapse of cavitation bubbles and the eruption of jets. The flow conditions of the liquid droplets were with hundreds of micro characteristic sizes and the velocities of order 200–500 m/s were usually chosen as [1]. The impact velocity of a micro scale droplet over 500 m/s was also studied for solidifying metal droplets [4]. Due to the quick responding time, very high spatial and temporal resolutions are required to analyze the detailed phenomena. It is difficult to control the parameters as the impact velocities and droplet size accurately in the experiments. An initial understanding of the phenomena induced by a high-speed micro-sized droplet impact and the maximum spreading radius has been quantitated in [5,6]. Recently, comprehensive reviews of the impacts of drops on obstacles can be found in [7,8] A high-speed droplet impact on a cold surface was investigated by both experimental and numerical analysis in the works [9,11]. One of the most important parameters of the impact is the heat transferred interaction between the wall and the fluid which may cause a thermal shock on the hot solid material possibly resulting in greater erosion [12]. In other words, the impact energy could cause high transfer rate to vaporize droplet from liquid form. Once no direct contact between the liquid and the substrate occurs, the impact is said to be a film boiling impact [10–14]. As we know, there is no reliable information on the maximum pressure developed at the impact of droplet to a rigid solid surface during very early works. Most frequent estimations of the impact pressure were replied on one-dimensional model [15] which is not perfectly applicable. Heymann [16] first presented a closer approximation solution for two-dimensional case in which the conditions existing at the instantaneous edge of the contact area are treated in a quasi-steady two-dimensional manner. The pressure equals to the one-dimensional predicted pressure until the edge of contact angle reaching about half of the critical value at which the droplet breaks down and lateral outflow initiates. The contact edge pressure increases progressively with the conditions after the droplet breaks down and lateral outflow initiates, and the critical pressure value is taken as the maximum impact pressure before the droplet breaks down.

---

* Corresponding author.
E-mail address: yyniu@mail.tku.edu.tw (Y.-Y. Niu).

http://dx.doi.org/10.1016/j.compfluid.2016.05.018
0045-7930/© 2016 Elsevier Ltd. All rights reserved.

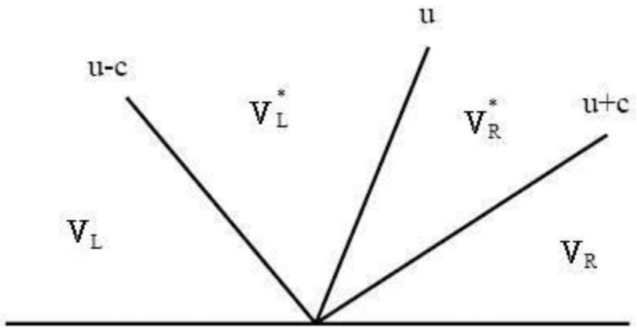

Fig. 1. The Structure of the One-dimensional Riemann problem.

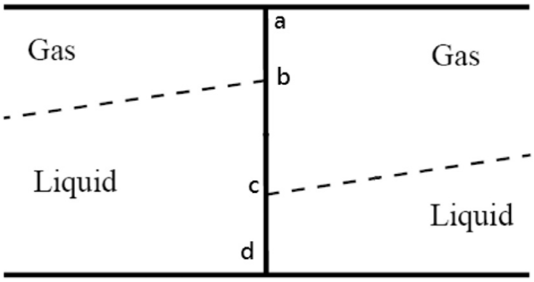

Fig. 2. Discretization based on stratified flow model.

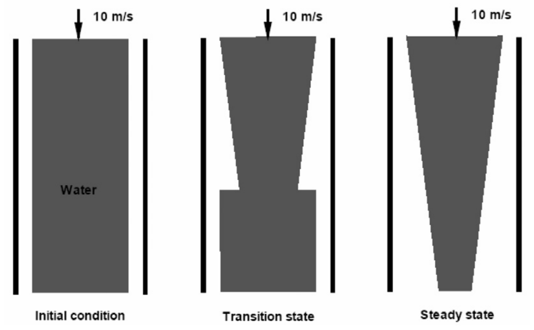

Fig. 3. Illustration of Ransom's water faucet problem.

Lesser [17] used numerical calculations to compare the pressure profile where a liquid cylinder impacts onto a rigid surface with Brunton and Rochester's exact solutions [18] and gave the solution for three-dimensional spherical drop impact. He compared some of the pressure–time histories form with the works done by Rosenblatt et al. [19] using finite-difference procedures with an artificial viscosity. Field et al. [20–23] used the high-speed photographic sequences to show that there are cases where the cavities form well away from a solid boundary. Though it is interesting to observe these cavities induced from higher velocities relevant to turbine erosion and rain erosion of aircraft components. Also, it is found that the confined shocks producing cavitation in isolated liquid volumes. Kong [23] focused the drop/wall interaction studies on diesel spray modeling applications in which it shows the wall temperature and Weber number play major influences on the outcome. Sanada et al. [24] proposed a new technique for the generation of impact by a high-speed steam–droplet spray. Through experimental data, they found that the degree of erosion is strongly dependent on temperature and the existence of cavitation inside the liquid droplet; also they measured both the droplet velocity and diameter distributions. By solving the Euler equation, they discussed the influences of cavitation bubbles on the formation of harsh erosion on a solid surface. Sanada et al. [25] used multicomponent Euler equations with the stiffened equation of state which are computed by a FV-WENO scheme [26] with an Approximated Riemann solver that accurately captures shocks and interfaces. It is found that the generated pressure depends on the droplet impact Mach number. For the low Mach number, the pressure differences at the center and the edge are minimized and the pressure is almost half of one-dimensional case. For the high Mach number, the edge pressure is almost three times greater than the center pressure.

In the development of two-phase approaches [27–30], one widely accepted approach to model fluids containing individual particles, droplets or bubbles is through the so-called two-fluid model, in which the time or space ensemble average process is applied to both continuous and disperse phases. Two sets of Navier–Stokes equations are used to describe both phases of fluids with additional inter-phasic terms for the exchange of momentum and energy between phases. Since each phase has its own velocity and temperature, the two-fluid model allows both mechanical and thermal non-equilibriums to be taken into account, and in that respect, it represents a more general model for two-phase flows. However, the two-fluid model also has some difficulties for numerical simulation. One of the issues is the non-hyperbolicity, making the system an ill-posed problem, which can be presented as numerical instability. Additional terms or assumptions, such as the interfacial pressure force [31], virtual mass force [32] and the two-pressure model [33] have to be used to make the system of equations hyperbolic. Even though the system is hyperbolic, it is difficult to derive the analytical form of its eigensystem. Hence, this disadvantage prevents it from being solved by some modern upwind schemes. However, the two-fluid model treats the interface as a weak solution in the fluid. The interface is captured by ensuring the conservation law. The problem is the common two-fluid model expressed in non-conservative form. The present of non-conservative terms in momentum equations can cause the solution to oscillate in the vicinity of interface. It requires some non-standard discretization method to ensure the exactly capturing of the interface. Niu [34] implemented the AUSMD [35] scheme to solve a seven-equation two-fluid model which involving two sets of Navier–Stokes equations and one non-conservative transport equation for volume fraction. It is found that the non-conservative transport equation can easily cause unwanted numerical instability and errors. Subsequently, the first successful work was performed by Paillere et al. [36] on the six-equation two-fluid model using modified AUSM + [37] scheme. In other respect of AUSM + in solving multi-phase flows, Edwardsand Liou [38] extended the AUSM$^+$ scheme to solve a homogeneous mixture model, which assumes all phases are in kinematics and thermodynamic equilibrium and uses one set of Navier–Stokes equations. Subsequently, Chang and Liou [39] proposed the AUSM + up scheme incorporating with the stratified flow model to discretize the system equations and used the extact Riemann solver. Their approach demonstrates robustness and high-order accuracy in their benchmark cases. However, they require the exact Riemann solver to calculate the numerical fluxes across the interfaces between different fluids. Since the exact Riemann solver needs extensive iterations to achieve the pressure, it is much more CPU time consuming than other methods, especially when stiff EOS of fluid (such as the stiffened gas model for water) is used.

In order to achieve the efficient and robust two-fluid flow calculations, we propose an approximated linearized Riemann solver instead of the exact Riemann solver to deal with the numerical flux across the liquid–gas interface. The proposed hybrid numerical flux is used to modify the previous code [40]. In numerical test cases, a one-dimensional water faucet problem and an air-water shock tube are chosen for validation. The case of a high-speed micro-droplet impact a rigid surface studied in [3,41] is chosen for two and three-dimensional numerical studies. In the present work, the impact of a spherical water droplet in a size 200 (μm)

Download English Version:

https://daneshyari.com/en/article/768069

Download Persian Version:

https://daneshyari.com/article/768069

Daneshyari.com