International Journal of Heat and Mass Transfer 127 (2018) 116–125

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical simulation of bubble collapse between two parallel walls and saturated film boiling on a sphere

Thanh-Hoang Phan, Cong-Tu Ha, Warn-Gyu Park *

School of Mechanical Engineering, Pusan National University, Busan 609-735, South Korea

ARTICLE INFO

Article history:
Received 22 December 2017
Received in revised form 26 March 2018
Accepted 29 June 2018

Keywords:
Multiphase flows
Compressible interface flows
Bubble collapse
Phase change
Film boiling

ABSTRACT

The present study aims at validating a computer simulation code for compressible interface flows with and without mass transfers. The numerical method was based on a dual-time fully compressible multiphase homogeneous mixture flow model. The interfaces with a large jump in density, pressure, velocity, and temperature were captured using a compressive high-resolution interface-capturing scheme. Lee's boiling model was adopted for implementing the mass and energy transfers via interfaces. The bubble collapse between two parallel walls was first examined. The bubble profile during the collapse process was predicted and compared with experimental images. The numerical results showed good agreement. To evaluate the present model for multiphase flows, including the effects of heat and mass transfers, computations were performed for saturated film boiling on a sphere at atmospheric pressure. The evolution of the wavy liquid–vapor interface around the sphere during the film boiling process was observed. The time-averaged heat fluxes obtained from the numerical simulation were compared with those predicted from numerical data and analytical correlations. The effects of the superheated wall on heat fluxes were also investigated.

1. Introduction

Compressible multi-fluid interface flows occur in a broad range of engineering and natural processes such as free surface flows, bubble collapses, cavitation, boiling, and condensation problems. Such flows are usually characterized by large density ratios, jump pressures, velocities, and temperatures. To deal with these flows is still a significant and challenging issue in computational fluid dynamics. This is even more challenging for flows with mass transfers across the interfaces. During the phase change process, large amounts of mass and energy are released or absorbed from one phase to another via the interfaces; making it difficult to accurately simulate the shape of the interfaces between two phases.

The dynamics of collapse of a cavitation bubble near solid walls is a major problem in industrial systems, and has been extensively studied [1–5]. During the bubble collapse, a shockwave and micro-jet can be generated towards the solid structure, causing serious structural damage in hydraulic devices. Experimental observations of the behavior of the cavitation bubble near a solid wall or between two parallel walls were conducted using high-speed movie technology [6–9]. For the numerical study, various methods were adopted to capture the bubble profiles during the growing and collapsing processes, such as the volume-of-fluid method (VOF) [10,11] and level-set method (LS) [12,13].

Film boiling is an important heat transfer phenomenon, which is characterized by the formation of a continuous vapor film on a heated wall. The vapor phase is usually generated in the thin film region on the lower portion of the surface and is removed upward through the formation and release of bubbles. In the literature, many methods have been developed to study film boiling flows, such as theoretical models, experiment correlations, and numerical simulations. To evaluate the heat transfer process, different theoretical models for film boiling have been developed since 1950. Bromley et al. [14] predicted the Nusselt number around a horizontal cylinder using a boundary layer approximation. Frederking and Clark [15] presented a model for analysis of the averaged surface heat flux in film boiling on spheres. This model assumed that the film flow is laminar, and it is valid only when the film thickness is much smaller than the radius of the sphere. Grigoriev et al. [16] proposed a more comprehensive correlation, which takes into account the effects of the diameter and turbulence in the film boiling around the spheres. An experimental and theoretical study of the natural and forced subcooled film boiling flows on spheres were performed by Dhir and Purohit [17]. They found that the heat transfer coefficient is strongly dependent on the subcooling

* Corresponding author.
E-mail addresses: phanhoang90@pusan.ac.kr (T.-H. Phan), hacongtu@pusan.ac.kr (C.-T. Ha), wgpark@pusan.ac.kr (W.-G. Park).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.06.156
0017-9310/

temperature and velocity around the sphere. Liu and Theofanous [18] conducted experiments on film boiling on spheres with single- and two-phase flows, and investigated the effects of liquid subcooling, liquid velocity, sphere superheat, sphere diameter, and sphere material on the heat transfer process. They also developed a general film boiling heat transfer correlation based on the obtained experimental data.

The various theoretical models and experimental correlations presented above use an empirical or semi-empirical coefficient for the prediction of film boiling. Therefore, these models and correlations are limited to narrow ranges of the geometrical and flow conditions. The recent use of numerical simulations to study the phase-change process is promising, because it provides detailed spatial and temporal distributions of the phase velocities, temperatures, and void fraction. Various numerical studies on saturated film boiling have been published [19–24]. Son and Dhir [25] simulated a three-dimensional film boiling on a horizontal cylinder using the LS method. Yuan et al. [26] developed a modified VOF method to simulate natural and forced film boiling on a sphere at saturated conditions. They showed a smooth liquid–vapor interface and parabolic velocity in the thin vapor film around the sphere. Moreover, the convective heat transfer coefficients were consistent with both correlations and experimental data. Arevalo et al. [27] performed an analysis of the pool film boiling heat transfer on a sphere using the VOF method. The effects of radiation on the convection mechanisms were presented. They found that the effect of radiation on heat fluxes is smaller than that of convection. Das and Das [28] developed a new technique to simulate film boiling around solid spheres with and without the effect of gravity. Their numerical results showed good agreement with the reported literature. A detailed review of the computational studies on the boiling phenomenon can be found in [29].

Among the various studies on numerical simulation of compressible multiphase flows, a fully compressible homogeneous mixture flow model based on a dual-time preconditioned technique was developed and successfully validated [30–35]. In the homogeneous approach, all the phases are assumed to be in mechanical and thermodynamic equilibrium, and non-slip velocity exists among phases. Based on this method, Ha and Park [36] presented accurate, efficient, and robust computations for compressible cavitating and ventilated flows without dependence on the Mach number. In their simulations, the obtained results were in good agreement with exact solutions and experimental data. Jin et al. [37] successfully implemented and validated a computer simulation code based on the compressible homogeneous mixture model for cavitating and flashing flows. They simulated the cavitation dynamics around a hemispherical head form and a Clark-Y hydrofoil; in addition, it showed good agreement with experimental measurements and observations. More recently, Ha et al. [38] presented the same method coupled with a compressive interface-capturing scheme for the computation of multi-fluid flows with high-density ratios and pressure variations.

This study is aimed at validating a computer simulation code for compressible interface flows with and without mass transfer. The bubble collapse with a large jump in density, pressure, temperature, and velocity is first examined. The time evolution of the bubble shape during collapsing is captured and compared to experimental images. To validate multiphase flows, including effects of heat and mass transfer, computations of a saturated film boiling on a sphere are then performed. Lee's boiling model [39] is adopted for implementing the mass and energy transfer via interfaces. The obtained results, including shape interface and time-averaged heat fluxes, are compared with published data. The superheated wall effects on the heat flux profile are also investigated.

2. Governing equations

In the present study, dual-time preconditioned fully compressible multi-phase mixture Navier–Stokes equations coupled with an interface advection equation are adopted to solve the compressible two-phase flows with and without mass transfer. The equations indicate mass conservation for the individual phase, and momentum and energy conservation for the homogeneous mixture. A dual-time procedure is applied to compute unsteady multi-phase flows. The method introduces a pseudo-time derivative in addition to the physical-time derivative. The pseudo-time derivative is used to rescale the disparity between the acoustic and convective wave speeds and remove linearization and factorization errors at a given physical-time level. This effectively yields a time-accurate solution. Following the next sequence of inner iterations, the solution is advanced to the next physical-time level. More details of the dual-time preconditioned technique and its validation for different unsteady multi-phase flows are available in previous studies [36,38]. The dimensionless form in generalized curvilinear coordinates is given by the following equations [36]:

$$\Gamma_e \frac{\partial \hat{Q}}{\partial t} + \Gamma \frac{\partial \hat{Q}}{\partial \tau} + \frac{\partial (\hat{E} - \hat{E}_v)}{\partial \xi} + \frac{\partial (\hat{F} - \hat{F}_v)}{\partial \eta} = \hat{S} + P_\sigma \left(\frac{\partial \hat{H}_\xi}{\partial \xi} + \frac{\partial \hat{H}_\eta}{\partial \eta} \right) \tag{1}$$

where t and τ denote the physical time and pseudo-time, respectively; $\hat{Q}$ is the primitive variable vector; $\hat{E}$ and $\hat{F}$ are the convective flux vectors; $\hat{E}_v$ and $\hat{F}_v$ are the viscous flux vectors; $\hat{S}$ is the source term; and P_σ is the surface tension term.

$$\hat{Q} = \frac{1}{J}\begin{pmatrix} p \\ u \\ v \\ T \\ Y_v \end{pmatrix}, \hat{E} = \frac{1}{J}\begin{pmatrix} Y_l \rho_m U \\ \rho_m uU + \xi_x p \\ \rho_m vU + \xi_y p \\ \rho_m h_0 U \\ Y_v \rho_m U \end{pmatrix}, \hat{F} = \frac{1}{J}\begin{pmatrix} Y_l \rho_m V \\ \rho_m uV + \eta_x p \\ \rho_m vV + \eta_y p \\ \rho_m h_0 V \\ Y_v \rho_m V \end{pmatrix} \tag{2}$$

$$\begin{aligned} \hat{E}_v &= \frac{1}{J}\begin{pmatrix} 0 \\ \xi_x \tau_{xx} + \xi_y \tau_{xy} \\ \xi_x \tau_{yx} + \xi_y \tau_{yy} \\ \xi_x (u\tau_{xx} + v\tau_{xy} - q_x) + \xi_y (u\tau_{yx} + v\tau_{yy} - q_y) \end{pmatrix}, \\ \hat{F}_v &= \frac{1}{J}\begin{pmatrix} 0 \\ \eta_x \tau_{xx} + \eta_y \tau_{xy} \\ \eta_x \tau_{yx} + \eta_y \tau_{yy} \\ \eta_x (u\tau_{xx} + v\tau_{xy} - q_x) + \eta_y (u\tau_{yx} + v\tau_{yy} - q_y) \\ 0 \end{pmatrix} \end{aligned} \tag{3}$$

$$\begin{aligned} \hat{S} &= \frac{1}{J}\begin{pmatrix} \dot{m}_T - c_a Y_l \rho_m v/y \\ \rho_m g_x - c_a(\rho_m uv - \tau_{xy})/y \\ \rho_m g_y - c_a(\rho_m v^2 - \tau_{yy})/y \\ \dot{m}_T h_{lv} + c_a(u\tau_{xy} + v\tau_{yy} - \rho_m h_0 v - q_y)/y \\ -\dot{m}_T - c_a Y_v \rho_m v/y \end{pmatrix}, \\ \hat{H}_\xi &= \frac{1}{J}\begin{pmatrix} 0 \\ \xi_x \alpha_v \\ \xi_y \alpha_v \\ 0 \\ 0 \end{pmatrix}, \hat{H}_\eta = \frac{1}{J}\begin{pmatrix} 0 \\ \eta_x \alpha_v \\ \eta_y \alpha_v \\ 0 \\ 0 \end{pmatrix} \end{aligned} \tag{4}$$

where p, u, v, T, and Y represent pressure, velocity components, temperature, and mass fraction, respectively; αdenotes the void fraction; q is the heat flux; g indicates the acceleration owing to gravity; h_{lv} is the latent heat of evaporation; $\tau_{ij}(i = x, y, j = x, y)$ are the viscous stresses; h_0 is the total enthalpy; ρ_m is the mixture