International Journal of Heat and Mass Transfer 78 (2014) 830–838

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Heat transfer with the growth and collapse of cavitation bubble between two parallel heated walls

Bin Liu [a,b], Jun Cai [a,*], Xiulan Huai [a]

[a] Institute of Engineering Thermophysics, Chinese Academy of Sciences, Beijing 100190, China
[b] University of Chinese Academy of Sciences, Beijing 100080, China

ARTICLE INFO

Article history:
Received 18 November 2013
Received in revised form 1 July 2014
Accepted 17 July 2014
Available online 8 August 2014

Keywords:
Cavitation bubble
Bubble dynamics
Heat transfer enhancement
VOF
Numerical simulation

ABSTRACT

Based on the axisymmetric Navier–Stokes equations and the volume of fluid (VOF) method, the dynamical characteristics of a single cavitation bubble between two parallel plates are numerically investigated together with the effect of bubble motion on heat transfer. By comparing the evolutions of the bubble profile with experimental photos, the validity and reliability of the present model has been verified. On this basis, the velocity fields of the micro-jet induced by the bubble motion are exhibited as well as the resultant heat transfer distribution on the solid wall. The intrinsic link between jet effect and heat transfer distribution is analyzed in details. Unlike the collapse of the bubble near a single wall, the bubble between two parallel walls displays the different changes of the profiles during the motion, and contains two collapse processes. The original big bubble is first split into two small sub-bubbles (the first collapse), and these sub-bubbles further collapse near the upper and lower walls respectively (the second collapse). It is found that the second collapse of the sub-bubble has greater impact on heat transfer than the first collapse. In addition, the effects of such parameters as the initial bubble radius, gas pressure inside the bubble and non-dimensional stand-off distance on the bubble behaviors and heat transfer are discussed in details, and some interesting phenomena and beneficial conclusions are obtained.

1. Introduction

The dynamics of cavitation bubble near a rigid wall, which is very important to reveal the mechanism of surface damages in fluid machineries, has been widely studied [1–7]. Most of these investigations are based on the assumption that the bubble is in a semi-infinite fluid domain. However, in the case of cavitating flow between two solid walls (e.g. a narrow gap), the growth and collapse of the bubble will significantly exhibit some behaviors different from those in the semi-infinite assumption. With high speed photography technology, Chahine [8] experimentally observed the behaviors of the spark-generated bubble, and found that the bubble was first constricted to a dumbbell shape at the beginning stage of the collapse and then was split into two parts by the liquid jet. Subsequently, Kucherenko and Shamko [9] further examined the characteristics of electric-explosion cavities between two parallel walls. It was found that the bubble gradually became a cone shape with decreasing the gap distance. Azam et al. [10] studied dynamics of an oscillating bubble in a narrow gap using high-speed videography. Behaviors of the bubble from the side and the front were simultaneously observed by two high-speed cameras. The oscillation of bubble surface between a convex and a concave curvature was firstly found in the case of low gap width. By means of experimental and numerical ways, Ishida et al. [11] thought that the large lateral pressure should be responsible for the split of the bubble in a narrow gap. At the same time, they developed a numerical model predicting the shape change during the bubble collapse. However, the effect of non-condensable gas was neglected in their model and the resulting liquid pressure was overestimated, which made that the split phenomenon of the bubble could not be well explained. On the basis of these studies, Mousavi and Ahmadi [12] investigated the bubble behaviors and the resultant liquid jet between two solid walls by the volume of fluid (VOF) method. They found that the bubble collapse between two parallel walls had a faster jet velocity compared with the situation near a single solid wall, and a longer distance between bubble center and secondary wall could bring a larger jet diameter. By analyzing the effect of gap height on the bubble dynamics, Gonzalez-Avila et al. [13] concluded that the collapse time increased with the decrease of gap height. Can [14] used Level-Set method for the simulation of a laser-generated vapor bubble

* Corresponding author. Tel.: +86 10 82543033.
E-mail address: caijun@mail.etp.ac.cn (J. Cai).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2014.07.050

growing and collapsing between two solid discs. A better correspondence with the experimental data was obtained for the smaller gaps. Further, based on the Green's function, Hay et al. [15] developed a model for a pulsating spherical bubble in a viscous compressible liquid between two parallel viscoelastic layers. Their simulation results demonstrate that the shear viscosity has an important impact on the dynamics between two layers. All above-mentioned investigations enable a better understanding of the bubble dynamics between two parallel solid walls.

Recently, some researchers paid their attentions to the phenomenon of cavitation enhanced heat transfer [16–18]. However, most of these investigations were conducted from a macro perspective. Brandon et al. [19,20] experimentally investigated the forced convection heat transfer with hydrodynamic cavitation in silicon channels. With deionized water as work medium, the heat transfer coefficient of the cavitating flow was higher than that of the non-cavitating flow by 67%. Even, a higher enhancement rate of heat transfer, 84%, was achieved when the fluid was R-123. For the cavitating flow in a micro-channel, the behaviors of cavitation bubble are significantly affected by the two side-walls of the channel. Due to the small scales involved, the channel can be deemed as a narrow gap with two parallel walls. As few investigations focus on the bubble dynamics between two parallel walls, the mechanism of cavitation enhanced heat transfer is still not clearly revealed for such problems. In the present work, a numerical model is developed to describe the dynamical behaviors of cavitation bubble between two parallel walls, and further predict the effect of bubble motion on the heat transfer on the solid wall. In this model, the VOF method is employed to capture the interfaces of the bubble, and the FLUENT commercial software is used to solve the governing equations in the whole calculation domain. Firstly, the changes of bubble profile, the resultant jet velocity field, and the distribution of surface heat transfer are presented. Then, the effects of such parameters as the initial bubble radius, initial gas pressure inside the bubble and non-dimensional stand-off distance on the bubble dynamics and heat transfer are discussed in details.

2. Mathematical model and numerical methods

The calculation domain is shown in Fig. 1. A spherical bubble with the radius of R_0 is initialized in a stationary fluid between two parallel plates. In the present simulation, the growth and collapse of the bubble is considered to be axisymmetric. Therefore, a two-dimensional (2D) axisymmetric coordinate system where x and r denote the axial and radial coordinates respectively is selected for the simulation. H is the height of the channel constituted by upper and lower walls, and is set as 15 mm. l_p is the distance between the bubble center and the upper wall, and the l_w is the one between the bubble center and the lower wall. The outlet, on which the pressure boundary condition is imposed, is located at the position of $r = \pm 700$ mm so as to eliminate its effect on the calculation results. In $-30\text{ mm} \leqslant r \leqslant 30\text{ mm}$ regions, meshes are refined to capture the motion of gas–liquid interface.

In the present study, the following assumptions are made to simplify the model.

(1) The liquid is considered to be incompressible water, while the vapor inside the bubble is compressible steam conforming to the ideal gas law.
(2) The liquid phase is immiscible with the vapor phase, and mass transfer between two phases is neglected.
(3) The effect of gravity is ignored.
(4) The flow pattern is considered to be laminar.

The VOF method is employed to track the interface between phases and a set of Navier–Stokes equations are solved in the whole calculation domain. The equations of continuity, momentum, energy and volume fraction can be formulated as

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot [\mu(\nabla \vec{v} + \nabla \vec{v}^T)] + \vec{F}$$

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot [\vec{v}(\rho E + p)] = \nabla \cdot (k \nabla T_f)$$

$$\frac{\partial \alpha_l}{\partial t} + \nabla \cdot (\vec{v}\alpha_l) = 0$$

where ρ, $\vec{v}$, p, and μ are the density, velocity vector, pressure and viscosity, respectively. $\vec{F}$, E, k, T_f and α_l denote the surface tension term, total energy, thermal conductivity, liquid temperature and volume fraction of water, respectively. The volume fraction of vapor (α_g) is computed based on the following equation

$$\alpha_g + \alpha_l = 1$$

The density, viscosity and total energy in each control volume are calculated according to the volume fraction values of water and vapor, and could be given as

$$\rho = \rho_l \alpha_l + \rho_g \alpha_g$$

$$\mu = \mu_l \alpha_l + \mu_g \alpha_g$$

$$E = \frac{\alpha_g \rho_g E_g + \alpha_l \rho_l E_l}{\alpha_g \rho_g + \alpha_l \rho_l}$$

where the subscripts l and g represent the liquid and vapor phases, respectively. The ideal gas law is used to calculate the vapor density

$$\rho_g = \frac{p M_w}{R T_g}$$

where M_w is the molar mass of vapor, R is the gas constant, and T_g is the temperature of gas inside the bubble.

Surface tension on the bubble interface appears in the momentum equation in the form of the volume force, $\vec{F}$, which is calculated by the continuum surface force (CSF) model.

$$\vec{F} = \sigma \frac{\rho \kappa \nabla \alpha_l}{0.5(\rho_g + \rho_l)}$$

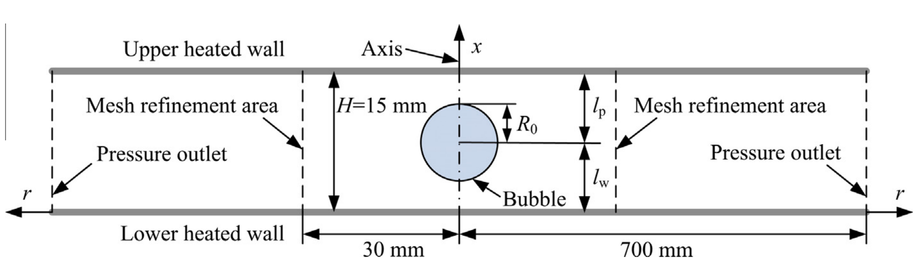

Fig. 1. Schematic diagram of the calculation domain.