Chemical Engineering Science 66 (2011) 5899–5909

Contents lists available at SciVerse ScienceDirect

Chemical Engineering Science

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

Numerical study of condensing bubble in subcooled boiling flow using volume of fluid model

Seong-Su Jeon [a], Seong-Jin Kim [b], Goon-Cherl Park [a,*]

[a] Department of Nuclear Engineering, Seoul National Univ., San 56-1, Sillim-dong, Gwanak-gu, Seoul 151-744, Republic of Korea
[b] Korea Atomic Energy Research Institute, 1045 Daedeokdaero, Yuseong-gu, Daejeon 305-600, Republic of Korea

ARTICLE INFO

Article history:
Received 3 March 2011
Received in revised form
31 July 2011
Accepted 6 August 2011
Available online 12 August 2011

Keywords:
Bubble behavior
Condensation
Heat and mass transfer
Two-phase flow
Numerical analysis
VOF model

ABSTRACT

In this numerical study, the behavior of condensing bubble was investigated using the volume of fluid (VOF) model in the FLUENT code. In order to simulate the condensing bubble with the FLUENT code, the bubble condensation was modeled using the user-defined function (UDF). For the validation of the UDF of bubble condensation, the results of CFD simulation were compared with the results of a bubble condensation experiment performed in Seoul National University (SNU). Simulation results showed good agreements with the experimental data. Moreover, the fundamental behavior of the condensing bubble was investigated in various conditions. The effects of condensation on bubble behavior were analyzed by comparing the behavior of condensing bubbles with that of adiabatic bubbles. It was found that the behavior of the condensing bubble was different from that of the adiabatic bubble in many respects including the bubble shape, velocity, rise distance and moving trajectory.

1. Introduction

Subcooled boiling flows are encountered in many industrial applications such as boilers, nuclear reactors, and the new generation of electronic and computer systems. There have been many efforts to understand and model subcooled boiling flow. In the field of nuclear engineering, subcooled boiling flows have been studied as an important issue in the optimum design and safety of nuclear systems because the presence of vapor bubbles has a large effect on the heat transfer characteristics of a nuclear system as well as pressure drops, flow instability, etc. However, subcooled boiling flow is very difficult to understand due to the complex behavior of bubbles with heat and mass transfer through the bubble interface. Therefore, in order to understand subcooled boiling flow, which is a challenging problem, it is essential to acquire thorough knowledge on condensing bubbles.

There have been many experimental analyses on bubble behavior. In these experimental studies, bubble behavior regarding the bubble size, shape, velocity and moving trajectory was investigated in various conditions. However, it has been impossible to obtain complete information about the bubble behavior due to the existence of the bubble interface between the vapor and liquid phases. The shape and the area of the varying interface, which governs the behavior of each phase, are very complex and thus difficult to measure. Moreover, in subcooled boiling flow, bubble condensation significantly affects the change of interface so this complicates the analysis of the behavior of condensing bubbles even more. Therefore, it is necessary to carry out numerical simulations for bubble behavior as a complement to experiments. Such numerical simulations may contribute to a better physical understanding of complex phenomena regarding the behavior of condensing bubbles.

One of the numerical methods for bubble simulation is the volume of fluid (VOF) model proposed by Hirt and Nichols (1981). It can deal with immiscible fluids with clearly defined interface; the direct simulation of the varying interface is possible. Many numerical studies have analyzed bubble behavior in terms of the bubble size, bubble shape, rise trajectory and bubble velocity using the VOF model. Tomiyama et al. (1991a), Wachem and Schouten (2002), Lörstad and Fuchs (2004), and Gopala and Wachem (2008) examined the applicability of the VOF model to the analysis of a single rising bubble in liquid. These examinations confirmed that the VOF model can yield good predictions of the bubble shape and terminal velocity. The VOF model also has been used to analyze the following: the effect of a channel wall on bubble shapes and terminal velocities (Tomiyama et al., 1991b); the velocity distribution and the distribution of the local wall shear stress in slug flow (Taha and Cui, 2006); the effect of the liquid velocity field on bubble motion in a linear shear flow (Tomiyama et al. 1993); the trajectories of bubbles with various

* Corresponding author. Tel.: +82 2 880 7210; fax: +82 2 889 2688.
E-mail address: parkgc@snu.ac.kr (G.-C. Park).

0009-2509/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ces.2011.08.011

initial diameters that rise in water (Krishna and Baten, 1999); the coalescence of two gas bubbles (Chen and Li, 1998; Annaland et al. 2005); the effect of the bubble size/shape and neighboring bubbles on the magnitude and direction of the lift force (Rabha and Buwa, 2009); etc. The simulation results showed good agreements with the experimental data. However, most bubble simulations were limited to cases of adiabatic systems such as air bubble-water flow, where the heat and mass transfer between each phase were not considered. Those simulation results for bubble behavior are only valid for analyzing bubble behavior in adiabatic systems.

However, in subcooled boiling flow, bubble condensation is the key parameter to describe heat and mass transfer phenomena. It significantly affects the shape and the area of the varying interface thus the behavior of the condensing bubble becomes different from that of the adiabatic bubble. Therefore, in order to understand bubble behavior in subcooled boiling flow, a numerical study of condensing bubbles considering heat and mass transfer through the bubble interface is required.

This study focused on how to simulate bubble condensation with a CFD code. Moreover, using the VOF model, the behavior of condensing bubbles in subcooled boiling flow was investigated. In order to simulate the condensing bubble with the FLUENT code, the bubble condensation was modeled using the user-defined function (UDF). For the validation of the UDF of bubble condensation, the results of CFD simulation were compared with the SNU experimental results. Through the VOF model coupled with the UDF of bubble condensation, the fundamental behavior of condensing bubble in terms of the bubble velocity, rise distance and moving trajectory was investigated under various conditions. The effects of condensation on bubble behavior were analyzed by comparing the behavior of condensing bubbles with that of adiabatic bubbles.

2. VOF model

In this study, the volume of fluid (VOF) model in a commercial CFD code FLUENT 6.2.16 (2001) is used to simulate condensing bubbles in subcooled boiling flow. The gas and liquid phases are considered as incompressible fluids and the flow is assumed to be laminar.

In the VOF model, the governing equations are solved using the volume fraction in each cell. A particular phase, '*k*', is identified by its volume fraction, α_k, in a computational cell. For $\alpha_k=1.0$, the cell is full of the secondary phase. For $\alpha_k=0.0$, the cell is full of the primary phase. For $0.0<\alpha_k<1.0$, the cell contains the interface between the primary and secondary phases. In each control volume, the volume fractions of all phases, i.e., α_k for all k, sum to unity

$$\sum_{k=1}^{n}\alpha_k=1 \tag{1}$$

Fig. 1 shows the sample calculation of volume fraction in each cell in this bubble simulation. For $\alpha_g=1.0$ and $\alpha_g=0.0$, the cell represents the vapor and liquid regions, respectively. For $0.0<\alpha_g<1.0$, the cell represents the interface region. A free-surface of the bubble is in the interface region. In this study, a bubble is defined as an aggregate composed of cells where volume fractions are in the range of $0.0<\alpha_g\leq 1.0$.

The tracking of the interface between the phases is accomplished by the solution of a continuity equation for the volume fraction of any of the phases. For the kth phase, the volume fraction equation has the following form:

$$\frac{\partial\alpha_k}{\partial t}+\vec{v}\nabla\alpha_k=\frac{S_{\alpha_k}}{\rho_k} \tag{2}$$

where S_{α_k} is the mass source term. In this study, this mass source term is modeled to simulate mass transfer between the phases during condensation.

The fields for all variables and properties are shared by the phases. Based on the local volume fraction, α_k, the appropriate properties and variables will be assigned to each control volume within the domain. In a two-phase system, if the phases are represented by the subscripts, 1 and 2, and if the volume fraction of the second of these is being tracked, the density in each cell is

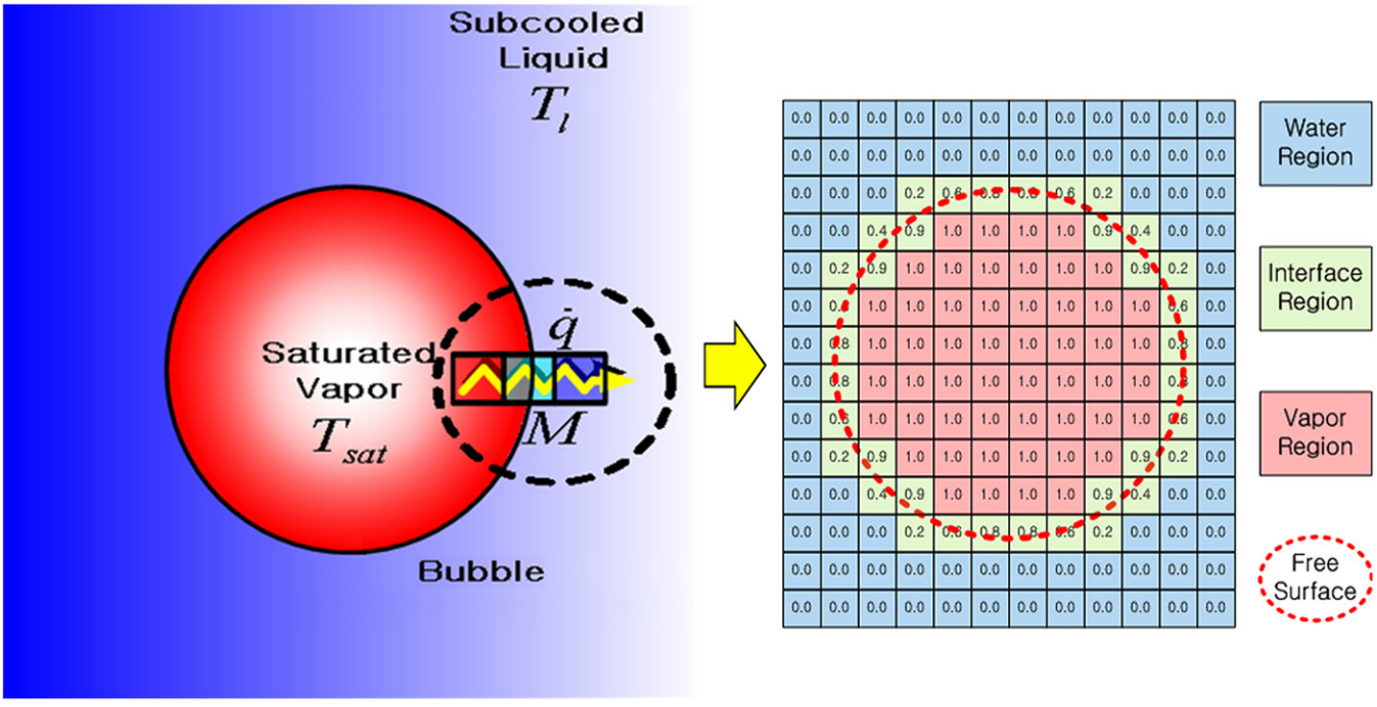

Fig. 1. A sample calculation of volume fraction in each cell using VOF model.