Numerical Study on Single Vapor Bubble Condensation Using VOF Model

Seong-Su Jeon [a], Seong-Jin Kim, Goon-Cherl Park
[a] *Nuclear thermal hydraulics engineering laboratory, Department of Nuclear Engineering, Seoul National University, San 56-1, Sillim-dong, Kwanak-gu, Seoul, 151-742, Korea, sagu100@snu.ac.kr*

1. Introduction

Two-phase flow phenomena have been studied with theoretical and experimental methods for various systems in many industries including nuclear power plants. In particular, the precise prediction of two-phase flow variable in subcooled boiling flow is of great importance to the safety analysis of nuclear power plants and verification of thermal-hydraulic design code. However, although a comprehensive understanding of these phenomena is essential, there has been hardly any significant development for the subcooled boiling with the simultaneous bubble interactions of heat and mass transfer process. So, in this study, single vapor bubble condensation is simulated numerically using FLUENT because the bubble condensation is a pivotal parameter to describe the heat transfer phenomenon in the subcooled boiling flow. To simulate the vapor bubble condensation in subcooled boiling flow, the multiphase VOF model, which was developed to track the interface between two or more phases, and mass transfer model were used. Finally, CFD simulation results were also compared to the experimental results by image processing developed by Kim [1].

2. Numerical Methods

2.1 VOF Model

The VOF model can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain [2]. It is particularly useful because it uses a minimum of stored information, treats intersecting free boundaries automatically, and can be readily extended to three-dimensional calculation [3].

In each control volume, the volume fraction α_q of all phases sum to unity.

$$\sum_{q=1}^{n} \alpha_q = 1 \tag{1}$$

For $0 < \alpha_q < 1$, the cell contains the interface between the q^{th} fluid and one or more other fluids. Otherwise the cell is empty ($\alpha_q = 0$) or full ($\alpha_q = 1$).

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 q^{th} phase, this equation has the following form.

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q \rho_q) + \nabla(\alpha_q \rho_q \vec{v}_q) = S_{\alpha_q} + \sum_{p=1}^{n}(\dot{m}_{pq} - \dot{m}_{qp})\right] \tag{2}$$

where $\dot{m}_{pq}$ is the mass transfer rate from phase p to phase q, S_{α_q} is the mass source term.

The properties appearing in the transport equations, such as the density, are determined as follows.

$$\rho = \alpha_2 \rho_2 + (1 - \alpha_2)\rho_1 \tag{3}$$

All other properties are computed in the same manner.

A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum equation, shown below, is dependent on the volume fractions of all phases through the properties ρ and μ.

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla(\rho \vec{v}\vec{v}) = -\nabla p + \nabla\left[\mu(\nabla \vec{v} + \nabla \vec{v}^{T})\right] + \rho \vec{g} + \vec{F} \tag{4}$$

The energy equation, also shared among the phases, is shown below.

$$\frac{\partial}{\partial t}(\rho E) + \nabla(\vec{v}(\rho E + p)) = \nabla(k_{eff} \nabla T) + S_h \tag{5}$$

The VOF model treats energy, E, and temperature, T, as mass-averaged variables

$$E = \frac{\sum_{q=1}^{n} \alpha_q \rho_q E_q}{\sum_{q=1}^{n} \alpha_q \rho_q} \tag{6}$$

where E_q for each phase is based on the specific heat of that phase and the shared temperature. The ρ and k_{eff} are shared by the phases. The source term, S_h, is the volumetric heat sources term.

2.2 Mass Transfer Modeling

Because just FLUENT cannot simulate the bubble condensation, it is necessary to use the UDF, User Defined Function which is based on C language in FLUENT. Using UDF helps to model the mass transfer. Figure 1 shows the bubble condensation mechanism by mass transfer model. Mass transfer modeling procedure to condensate the vapor bubble is shown below.

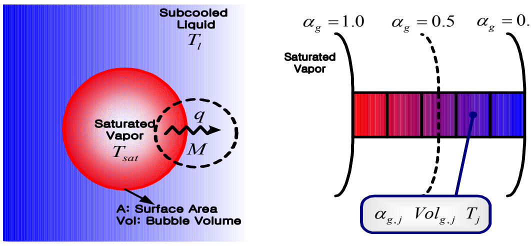

Fig. 1. Mass Transfer Modeling in the Bubble Condensation

Heat transfer rate between two phases is shown below.

$$\dot{q} = h_i(T_{sat} - T_l)A \qquad (6)$$

where h_i is interfacial heat transfer coefficient, T_{sat} , T_l are vapor and liquid temperature respectively, A is interface area between two phases.
Mass transfer rate from vapor to liquid is shown below.

$$\dot{M} = \frac{\dot{q}}{h_{fg}} = \frac{h_i(T_{sat} - T_l)A}{h_{fg}} = \sum_j \dot{m}_j Vol_j \qquad (7)$$

Thus j^{th} cell's mass transfer rate is shown below.

$$\dot{m}_j = \frac{\alpha_{g,j}\dot{M}}{\sum_j \alpha_{g,j} Vol_j} = \frac{h_i}{h_{fg}} \frac{(T_{sat} - T_{l,j})A\alpha_{g,j}}{\sum_j \alpha_{g,j} Vol_j} \qquad (8)$$

where h_{fg} is latent heat, $\dot{m}_j$ is j^{th} cell's mass transfer rate, Vol_j is j^{th} cell's volume.

By applying this equation (8) in $0 < \alpha_j < 1$, vapor bubble condensation was successfully achieved.

2.3 CFD Simulation

To simulate the bubble condensation, the actual bubble image was introduced. The bubble image was generated by image processing technique. The technique was developed by Kim to measure the interfacial heat transfer coefficient in subcooled boiling flow [1]. Therefore, by applying the bubble image in the CFD simulation, more accurate results can be obtained. For the simulation, meshes were generated in the channel by using GAMBIT. The identical conditions such as velocity and temperature distribution were implemented in the channel and actual bubble image was applied to CFD simulation. Finally, CFD calculation was performed with VOF and mass transfer model. Figure 2 shows how the actual bubble image was applied to CFD simulation.

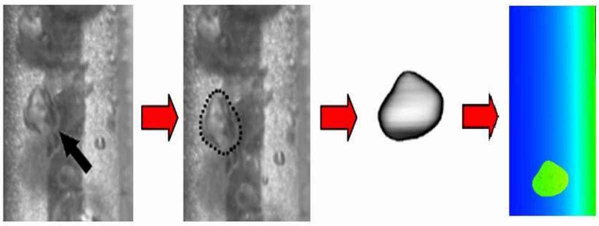

Fig. 2. Applied actual bubble image in CFD simulation

3. Results and Discussions

Figure 3 shows the bubble's behavior including the bubble's shape, velocity and volume change in the subcooled boiling flow. It is easily found that the CFD simulation results were similar to the experimental results. As the time goes by, vapor bubble rose up and condensed in the subcooled boiling flow. And as the bubble rose up, bubble agitated the water so that turbulent phenomenon was observed.

Figure 4 shows the condensing rate of the same bubble. Though the results from FLUENT do not accord with the condensing rate of the actual bubble, the trend is similar. So if the more sophisticated settings including UDF code improvement are applied, more accurate condensation results to describe the real phenomena will be obtained.

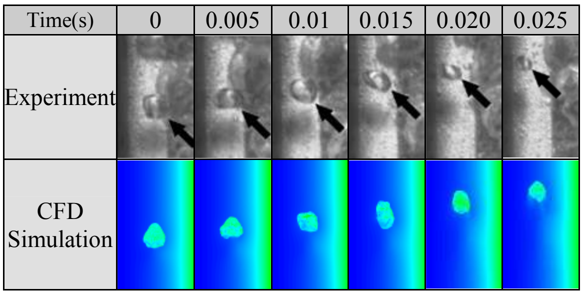

Fig. 3. CFD simulation results and experimental results on single vapor bubble condensation in subcooled boiling flow

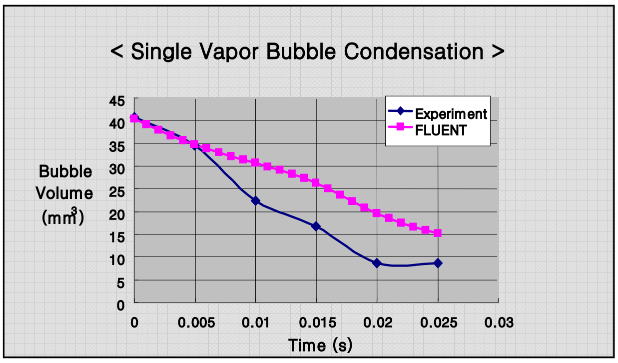

Fig. 4. CFD simulation results and experimental results on single bubble volume change in subcooled boiling flow

4. Conclusions

In this study, the actual bubble image was applied and condensation phenomenon was simulated by FLUENT. It can be concluded that the VOF model is appropriate to predict the vapor bubble condensation. Moreover, heat transfer correlations in subcooled boiling flow can be evaluated to improve thermal-hydraulic system code by the further study on the bubble condensation simulation with VOF model.

REFERENCES

[1] S. J. Kim, Experimental Study on Interfacial Heat Transfer of Bubble Condensation in Subcooled Boiling Flow, Ph. D. Thesis, Seoul National University, Korea, 2007.
[2] FLUENT 6.3 User's Guide, Fluent Inc, Lebanon, NH 03766, 2006.
[3] C. W. Hirt, B. D. Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries, Journal of Computational Physics, Vol.39, pp.201-225, 1981