# CFD Simulation on Single Bubble Behavior Using VOF Model

Jae-Ho Bae, Seong-Su Jeon*, Min-Seok Ko, Sang-Hun Shin
*FNC Tech., Heungdeok IT Valley, Heungdeok 1-ro, Giheung-gu, Yongin-si, Gyeonggi-do, 446-908, Korea*
**Corresponding author: ssjeon@fnctech.com*

## 1. Introduction

Boiling heat transfer is a powerful heat transfer mode used in many industrial applications such as boilers and nuclear reactors. The characteristics of boiling heat transfer are governed by bubble behavior in the system. Thus, developing predictive models and correlations for boiling heat transfer requires a sufficient understanding of parameters (bubble shape, contact angle against wall, departure diameter, lift-off diameter, bubble velocity, moving trajectory and etc.) describing bubble behavior.

There have been many experimental analyses on bubble behavior. In these experimental studies, bubble parameters were 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. Therefore, it is necessary to carry out CFD (Computational Fluid Dynamics) analysis to simulate the bubble behavior directly as a complement to experiments. Such simulations may contribute to a better physical understanding of complex phenomena regarding the bubble behavior [1].

Although many CFD studies for bubble behavior have been conducted so far, there are few researches on the bubble behavior analysis while simulating the entire bubble lifetime (bubble generation to collapse) (see Fig. 1). The entire process simulation is required to achieve complete and practical results for bubble parameters that are difficult to obtain through experiments.

In this study, CFD studies are performed to simulate the generation to collapse of a single bubble, and examine the bubble parameters by using VOF (Volume-Of-Fluid) model in the FLUENT 2019 [2]. This paper describes the main results of the study so far.

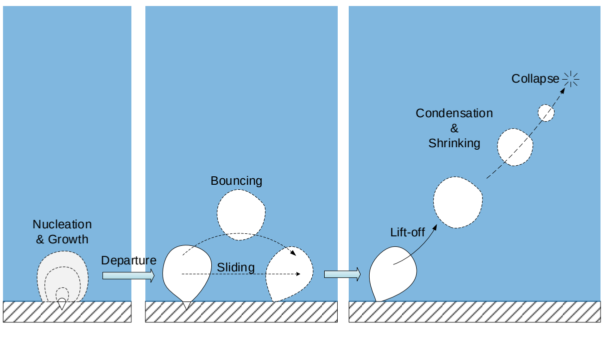


Fig. 1. Bubble behavior in subcooled boiling flow

## 2. CFD Modeling of Bubble Behavior

### *2.1 UDF Model of Interface Heat and Mass Transfer*

The bubble behavior was simulated by using the VOF model which could track the complex change of the bubble interface. The bubble growth and condensation was simulated by modeling source terms with the UDF (User-Defined Function) which could be dynamically linked with the FLUENT solver. It can be programmed by the user. The UDF model is as follows.

Lee evaporation and condensation model [3] developed to express the heat and mass transfer across bubble interface. This model is that phase change is driven primarily by deviation of interfacial temperature from saturation temperature ($T_{sat}$), and phase change rate is proportional to this deviation. Thus, the phase change happens while maintaining temperature of saturated phase and interface equal to $T_{sat}$. This model assumes that the mass transfer is occurred at constant pressure and quasi-thermo-equilibrium state as following relations:

(1) Mass transfer:

<u>Evaporation ($T > T_{sat}$)</u>

$$S_M = -r\alpha_L\rho_L\frac{T-T_{sat}}{T_{sat}}\text{ (Liquid)} \quad (1)$$

$$S_M = r\alpha_L\rho_L\frac{T-T_{sat}}{T_{sat}}\text{ (Vapor)} \quad (2)$$

<u>Condensation ($T < T_{sat}$)</u>

$$S_M = r\alpha_V\rho_V\frac{T_{sat}-T}{T_{sat}}\text{ (Liquid)} \quad (3)$$

$$S_M = -r\alpha_V\rho_V\frac{T_{sat}-T}{T_{sat}}\text{ (Vapor)} \quad (4)$$

(2) Energy transfer:

<u>Evaporation ($T > T_{sat}$)</u>

$$S_E = -r\alpha_L\rho_L\frac{T-T_{sat}}{T_{sat}}h_{fg}, \quad (5)$$

<u>Condensation ($T < T_{sat}$)</u>

$$S_E = r\alpha_V\rho_V\frac{T_{sat}-T}{T_{sat}}h_{fg}, \quad (6)$$

where $r$ is an empirical coefficient that is called as mass transfer intensity factor ($s^{-1}$), and $h_{fg}$ is the latent heat. Many research have used wide range of value of $r$, ranging from 0.1 to $10^7$. In this study, the value of $r$ is used considering numerical accuracy and calculation speed as $10^3$ $s^{-1}$.

### *2.2 CFD Simulation Procedure*

The procedure of the bubble simulation is shown in Fig. 2. Firstly, the flow channel and meshes are generated in a 2D Cartesian coordinate system. The dimensions of the test channel are 20 mm x 25 mm. The VOF model of FLUENT is used to simulate the bubble behavior. The surface tension considered important for bubble formation and behavior is 0.05891 N/m. As the initial condition, the water condition of 1 atm is considered, and the water/steam of properties derived from the steam table are used. In addition, the evaporation/condensation of bubble is simulated using developed UDF. Time step size is $10^{-5}$. The inlet and outlet faces of the channel are modeled as the velocity-inlet (0.1 m/s) and pressure-outlet (1 atm), respectively. And one wall is adiabatic wall and the other is heated wall.

### 3. Simulation Results

Unfortunately, this study could not continuously simulate whole life time of bubble from generation to collapse. Instead, in this study, the simulation was performed separately on the generation and growth of bubble, and behavior after bubble is generated. Fig. 2 shows the growth of bubble. Since the initial temperature of water is 373 K and the heating surface (383.15 K) is higher than the saturation temperature (373.15 K), small bubbles are generated and rising from the wall due to boiling. Fig. 3 and 4 show the behavior after bubble departures. It shows that bubble generated near wall lifts off, and completely condenses and disappears by sub-cooled liquid.

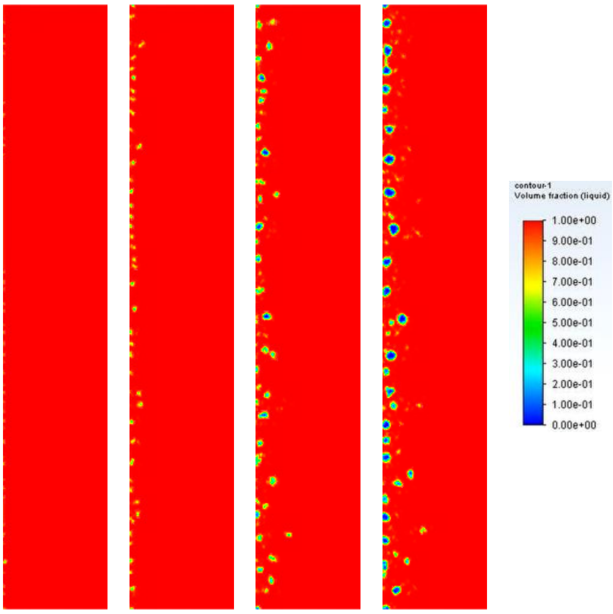


Fig. 2. Nucleation and growth of bubble
(Void Fraction: Red-Liquid, Blue-Vapor)

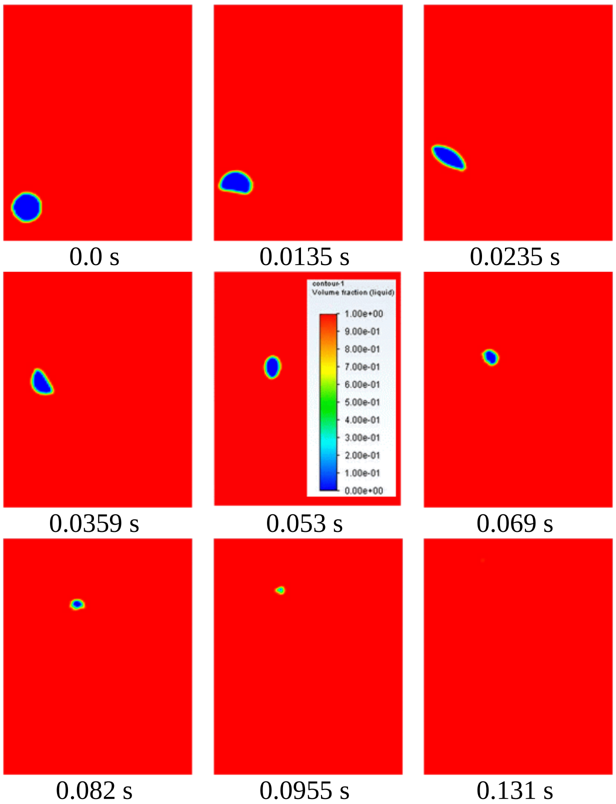


Fig. 3. Bubble behavior after departure
(Void Fraction: Red- Liquid, Blue-Vapor)

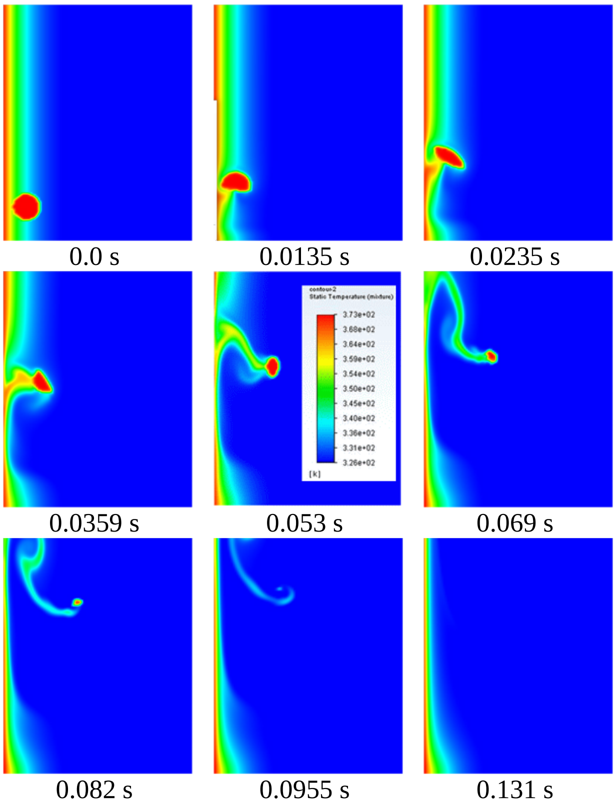


Fig. 4. Bubble behavior after departure (Temperature field)

## 4. Conclusions

This study shows the nucleation and growth of bubble, and the behavior after departure of bubble. Unfortunately, it was not succeed to see the whole life time of bubble. But, this study identifies that simulation is possible from the beginning of the bubble`s life time. Later, through research, the whole life time of bubble will be simulated with CFD code.

## Acknowledgments

This work was supported by the Korea Industrial Technology Association (KOITA) grant funded by the Korea government, MSIT (Ministry of Science and ICT). (No. KOITA-GO-2021-72)

## REFERENCES

[1] S. S. Jeon, S. J. Kim, and G. C. Park, Numerical study of condensing bubble in subcooled boiling flow using volume of fluid model, Chemical Engineering Science 66. pp. 5899–5909, 2011.
[2] ANSYS, ANSYS Fluent User`s Guide, ANSYS, Inc., 2013.
[3] Lee, W.H., A Pressure Iteration Scheme for Two-Phase Modeling. Technical Report LA-UR 79-975, Los Alamos Scientific Laboratory, Los Alamos, New Mexico, 1974.