Transactions of the Korean Nuclear Society Autumn Meeting
Changwon, Korea, October 24-25, 2024

CFD Modeling and Simulation of Flow Pattern Transition in Subcooled Flow Boiling under High Void Fraction Conditions

Iljin Kim, In-Yeop Kang, Gubin Lee, Hyungdae Kim *
Department of Nuclear Engineering, Kyung Hee University, Yongin-si, Republic of Korea
**Corresponding author: hdkims@khu.ac.kr*

Keywords : M-CFD, subcooled flow boiling, flow pattern, high void fraction

1. Introduction

In recent studies, Computational Fluid Dynamics (CFD) analysis have played a crucial role in the safety analysis of nuclear power plants. Accurate predictions are especially essential for the safety analysis of severe accident scenarios, as they are critical for evaluating the performance of severe accident mitigation strategies. Among these strategies, core-catcher [1] and In-Vessel Retention-External Reactor Vessel Cooling (IVR-ERVC) [2-3] systems are representative examples. In these severe accident mitigation strategies, boiling heat transfer is characterized by a combination of low and high void fractions due to the structural characteristics of the heating surface facing downward. Particularly in high void fraction regions, the behavior of bubbles becomes highly complex, making the research in this area challenging. Nevertheless, most previous studies have focused on the prediction of bubbly and dispersed flows, which are insufficient for assessing the safety of current severe accident mitigation strategies where various boiling flow regimes, such as slug flow coexist.

In high void fraction regions, various boiling flow regimes coexist, making it difficult for conventional CFD method to accurately capture these phenomena. The method is primarily designed for bubbly or dispersed flows, and thus it does not adequately reflect the formation or behavior of large bubbles. Consequently, there is a need to re-evaluate the physical modeling concepts to accurately simulate the diverse flow regimes that include high void fraction regions.

This study aims to address this gap by selecting an appropriate multiphase flow model that can accurately reflect bubble behavior under high void fraction conditions. Using this model, boiling flow analyses were conducted to compare the predictions of bubble behavior with those obtained from traditional CFD methodologies.

2. CFD Modeling

2.1 Interface capturing method

The hybrid multi-fluid solver in OpenFOAM employs an interface capturing method enhanced by Wardle and Weller's interface compression scheme [4]. This solver addresses the volume fraction transport equation of the multi-fluid model, incorporating an additional artificial compression term, as shown in Eq. 1. The artificial compression term, $\vec{u}_c \alpha_k (1-\alpha_k)$, ensures that the interface capturing method is active only at the interfaces. The constant $C_{\alpha.ki}$ determines whether the interface compression method is applied or not. When $C_{\alpha.ki} = 1$, interface compression is enabled; when $C_{\alpha.ki} = 0$, it is disabled. This constant can be set independently for each phase pair; for instance, it can be set to 0 for a dispersed gas phase-continuous liquid phase and 1 for a continuous gas phase-continuous liquid phase.

$$(1) \qquad \frac{\partial \alpha_k}{\partial t} + \vec{u}_k \cdot \nabla \alpha_k + \nabla \cdot \left(\vec{u}_c \alpha_k (1-\alpha_k)\right) = \frac{\Gamma_{ki} - \Gamma_{ik}}{\rho_k}$$

$$(2) \qquad \vec{u}_c = C_{\alpha.ki} |\vec{u}| \frac{\nabla \alpha}{|\nabla \alpha|}$$

2.2 Wall boiling model

The wall boiling model is employed to predict heat transfer between the wall and the liquid phase in a boiling system. The PRI model by Kurul and Podowski [5] is commonly applied, accounting for three heat transfer mechanisms: single-phase convection, quenching, and evaporation. The total heat flux, represented by Eq. 3, is the sum of the convective heat flux (q''_{conv}), evaporative heat flux (q''_{evap}), and quenching heat flux (q''_{quench}), with (q''_{wall}) denoting the total heat flux.

$$(3) \qquad q''_{wall} = q''_{conv} + q''_{evap} + q''_{quench}$$

$$(4) \qquad q''_{conv} = h_{conv} A_{1\phi} \left(T_{wall} - T_{liq}\right)$$

$$(5) \qquad q''_{evap} = N_a \left(\frac{\pi}{6} D_{dep}^3\right) f \rho_{vap} h_{lv}$$

$$(6) \qquad q''_{quench} = h_{quench} A_{2\phi} \left(T_{wall} - T_{liq}\right)$$

The RPI boiling model requires the use of sub-models for nucleation site density, bubble departure diameter, and bubble departure frequency. For this study, the commonly used closure models available in OpenFOAM were adopted. The nucleation site density was determined using the Lemmert-Chawla model [6], as provided in Eq. 7. The bubble departure diameter was calculated using the Tolubinsky and Kostanchuk model [7], as shown in Eq. 8. Finally, the bubble departure frequency was determined using the Cole model [8], as described in Eq. 9.

- Nucleation site density model (Lemmert and Chawla, 1997)

$$N_a = C_n N_{a.Ref} \left(\frac{T_{wall} - T_{sat}}{\Delta T_{Ref}} \right)^n \tag{7}$$

- Bubble departure diameter model (Tolubinsky and Kostanchuk, 1970)

$$D_{dep} = min\left(d_{Ref} e^{\left(-\frac{\Delta T_{sub}}{\Delta T_{ref}} \right)}, d_{max} \right) \tag{8}$$

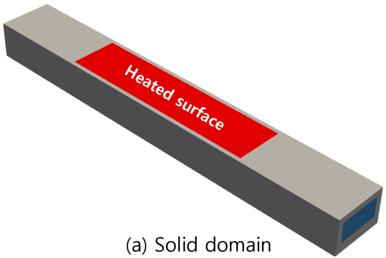
- Bubble departure frequency model (Cole, 1960)

$$f = \sqrt{\frac{4g(\rho_{liq} - \rho_{vap})}{3D_{dep}\rho_{liq}}} \tag{9}$$

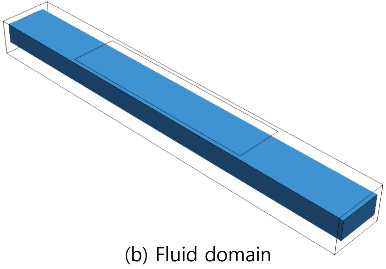
3. CFD Simulation

3.1 Simulation conditions

In this study, a flow boiling simulation was conducted within a rectangular channel, with the computational domain shown in Fig. 1. The analysis domain consists of both solid and fluid regions, and a conjugate heat transfer analysis was performed. Fig. 2 shows the configuration of the computational domain. The grid within the analysis domain was uniformly composed of square cells with a size of 0.5 mm. A 100 mm long area of interest was defined at the center of the analysis domain, with inlet and outlet regions of 50 mm each placed above and below this area. A heating surface with dimensions of 23 mm x 100 mm was located at the top of the analysis domain, where a heat flux boundary condition was applied. The cross-sectional area of the flow was set to 23 mm x 10 mm, with a velocity inlet specified as the inlet condition and a pressure outlet as the outlet condition. The working fluid used was water at a pressure of 500 kPa, with a subcooling of 20 K at the inlet. Major conditions of the simulation are presented in Table I.



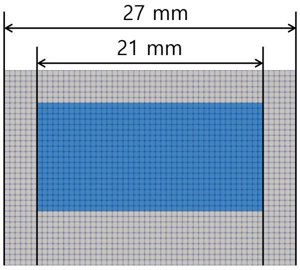
(a) Solid domain



(b) Fluid domain

Fig. 1. Schematic of the computational domain showing (a) the solid domain and (b) the fluid domain.

(a) Top view



(b) Front view

Fig. 2. Detailed dimensions of the computational domain: (a) top view, (b) front view.

Table I: Major conditions of flow boiling simulation

Variable	**Value**
P_{sys}	500 kPa
T_{in}	404.98 K
ΔT_{sub}	20 K
G	1000 kg/m^{2}s
q''_w	2000 kW/m^2

3.2 Simulation results

Firstly, Fig. 3(a) shows the results obtained using the widely applied Euler method, where the bubbles appear as continuous shapes with a uniform thickness. This method is effective for predicting the overall distribution of bubbles but has limitations in accurately capturing the detailed interfacial behavior that occurs as bubbles grow. Specifically, the Euler method simplifies the interactions between bubbles and the liquid, making it difficult to fully replicate the complex behavior observed in actual physical phenomena. This method particularly struggles with accurately depicting complex interfacial phenomena such as bubble coalescence or breakup. Consequently, while Fig. 3(a) is suitable for a general estimation of bubble distribution, it provides limited information for analyzing the detailed behavior of individual bubbles.

On the other hand, Fig. 3(b) presents the results from the Euler-VOF method, which predicts the overall bubble distribution by simultaneously displaying both dispersed gas phase and continuous gas phase. This combined view allows for a more comprehensive understanding of how different types of bubbles distribute and interact within the flow, showing both dispersed and continuous gas phase at once.

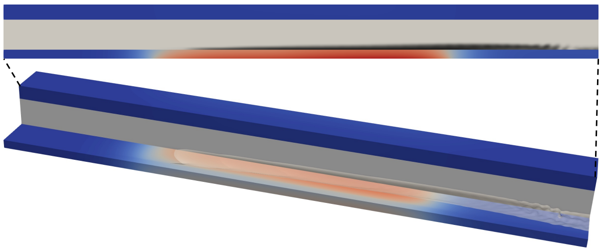
In contrast, Fig. 3(b) illustrates the behavior and interface of continuous bubbles using the VOF method.

This method allows for a detailed depiction of how large bubbles form and interact with the surrounding liquid. For example, the VOF method can precisely track the process where small bubbles coalesce into larger bubbles or where large bubbles split into smaller ones, providing a clearer understanding of the dynamic interactions between the bubbles and the liquid within the flow.

The results in Fig. 3 highlight the differences between these methods, particularly emphasizing that the Euler-VOF method can offer more precise and reliable outcomes than the Euler method when analyzing complex bubble behavior under high void fraction conditions. The VOF method is especially advantageous for accurately reflecting the complex interactions associated with the interface dynamics of continuous bubbles, enabling a more faithful reproduction of actual physical phenomena. These distinctions are critical when selecting a method for flow analysis, suggesting that the Euler-VOF method may be essential in certain scenarios.

In conclusion, Fig. 3 effectively demonstrates the performance differences between the methods used to analyze bubble shapes and behavior, indicating that the Euler-VOF method may be more effective for studies that require detailed analysis of complex flow phenomena. These findings provide valuable insights for selecting the appropriate method when accurate analysis of bubble behavior is crucial in various engineering applications.

(a) Euler method



(b) Euler-VOF method

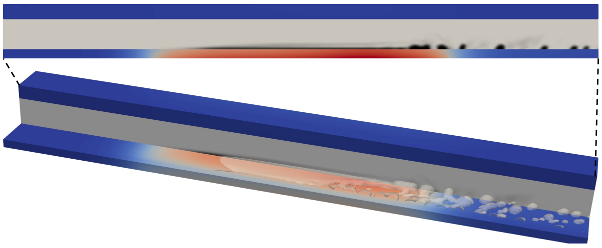


Fig. 3. Void fraction distribution comparison using different methods: (a) Euler method, (b) Euler-VOF method.

4. Conclusions

This study compared and analyzed the Euler method and the Euler-VOF method to simulate subcooled flow boiling under high void fraction conditions. The results showed that while the Euler method is suitable for predicting the overall bubble distribution, it is primarily effective in low void fraction environments and tends to form bubbles in a uniform layer on the heated wall. Due to this characteristic, the Euler method has limitations in accurately simulating the formation of large bubbles and the interactions at their interfaces under high void fraction conditions.

On the other hand, the Euler-VOF method proved to be more effective in accurately depicting the formation, growth, and interaction of bubbles in scenarios where both dispersed and continuous bubbles coexist, such as under high void fraction conditions. The VOF method was able to precisely track complex interfacial phenomena, such as the coalescence and breakup of bubbles, making it more suitable for effectively simulating the complex flow patterns that occur in high void fraction environments.

This study confirmed that the Euler-VOF method offers superior analytical capabilities compared to the Euler method for interpreting complex flow boiling phenomena under high void fraction conditions, indicating that the Euler-VOF method is a more appropriate choice when analyzing diverse bubble behaviors in such scenarios. These findings underscore the importance of selecting the appropriate method for subcooled flow boiling analysis, demonstrating that the application of the Euler-VOF method is necessary for more accurate predictions in complex flow situations, including high void fraction conditions. Future research can leverage these methods to conduct more precise analyses of bubble behavior in various engineering applications.

ACKNOLEDGEMENT

This work was supported by the Nuclear Safety Research Program through the Korea Foundation Of Nuclear Safety(KoFONS) using the financial resource granted by the Nuclear Safety and Security Commission(NSSC) of the Republic of Korea. (2106022).

REFERENCES

[1] S. M. Kim et al., Improvement of CFD code for two-phase flow with boiling and natural convection for evaluation of core-catcher cooling performance in OpenFOAM, Nuclear Engineering and Design, Vol.408, 112316, 2023.

[2] R. J. Park et al., Development of IVR-ERVC evaluation method and its application to the SMART, Annals of Nuclear Energy, Vol.161, 108463, 2021.

[3] K. S. Ha et al., Evaluations of two-phase natural circulation flow induced in the reactor vessel annular gap under ERVC conditions, Nuclear Engineering and Design, Vol.253, pp.114-124, 2012.

[4] K. E. Wardle and H. G. Weller, Hybrid Multiphase CFD Solver for Coupled Dispersed/Segregated Flows in Liquid-Liquid Extraction, International Journal of Chemical Engineering, Vol.2013, No.1, pp. 1–13, 2013.

[5] N. Kurul and M. Z. Podowski, Multidimensional effects in forced convection subcooled boiling, Proceeding of International Heat Transfer Conference 9, pp. 21–26, 1990.

[6] M. Lemmert and J. Chawla, Influence of flow velocity on surface boiling heat transfer coefficient, Heat transfer in boiling, pp. 231–247, 1974.
[7] V. I. Tolubinsky and D. M. Kostanchuk, Vapour bubbles growth rate and heat transfer intensity at subcooled water boiling, Proceeding of International Heat Transfer Conference 4, pp. 1–11, 1970.
[8] R. Cole, A Photographic Study of Pool Boiling in the Region of the Critical Heat Flux, 1960.