Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 22-23, 2025

Numerical Modeling for Subcooled Flow Boiling Condition in a High Void Fraction Regime

Iljin Kim [a], In-Yeop Kang [b], Gubin Lee [b], Hyungdae Kim [b*]
[a] *Virtual Nuclear Reactor Research Section, Korea Atomic Energy Research Institute, Daejeon, Republic of Korea*
[b] *Department of Nuclear Engineering, Kyung Hee University, Yongin, Republic of Korea*
[*] *Corresponding author: hdkims@khu.ac.kr*

***Keywords : High void fraction, M-CFD, OpenFOAM, Subcooled flow boiling, Interface capturing method**

1. Introduction

Recent studies have demonstrated that Computational Fluid Dynamics (CFD) analysis plays a crucial role in the safety evaluation of nuclear power plants. In particular, the safety assessment of severe accident scenarios requires highly accurate predictions, as they are essential for quantitatively evaluating the performance of severe accident mitigation strategies. Representative mitigation strategies include core-catchers and In-Vessel Retention-External Reactor Vessel Cooling (IVR-ERVC) systems, where the downward-facing heating surface introduces unique flow and heat transfer characteristics compared to conventional boiling conditions.

Due to this structural configuration, vapor bubbles generated on the heating surface are less likely to detach easily, leading to coalescence with neighboring bubbles and the formation of larger vapor structures. However, most existing studies have primarily focused on nucleate boiling or dispersed bubble analyses, which are inadequate for capturing the behavior of slug bubbles, which occupy multiple computational cells. In high void fraction regions, multiple boiling flow regimes coexist, making it challenging to accurately model bubble behavior using conventional Eulerian-based CFD approaches. These methods are primarily designed for bubbly and dispersed flows and thus struggle to account for the formation and evolution of large vapor structures. Consequently, there is a need to reassess existing physical modeling approaches to ensure the accurate simulation of diverse flow regimes, particularly in high void fraction conditions.

This study addresses these limitations by selecting an advanced multiphase flow modeling approach capable of accurately representing bubble behavior under high void fraction conditions. Using this model, boiling flow analyses were performed, and the predicted bubble behavior was compared with that obtained from conventional CFD methodologies to evaluate prediction accuracy.

2. CFD Modeling

2.1 Interface capturing method

The multiphaseEulerFoam solver available in OpenFOAM, a hybrid multi-fluid solver, is utilized to model multiphase flows. This solver employs an interface capturing method that integrates Wardle and Weller's interface compression scheme [1]. In this framework, the volume fraction transport equation is enhanced by the inclusion of an additional artificial compression term, as expressed in Eq. (1). The artificial compression term, $\vec{u}_c\alpha_k(1-\alpha_k)$, ensures that interface sharpening is applied selectively at phase boundaries. The parameter $C_{\alpha.ki}$ governs the activation of the interface compression scheme; a value of 1 enables compression, while a value of 0 deactivates it. This parameter can be independently specified for each phase pair (e.g., set to 0 for a dispersed gas phase in a continuous liquid phase, and to 1 for a continuous gas phase interacting with a continuous liquid phase).

Conventionally, Euler-based analyses have interpreted all bubbles as dispersed-phase bubbles. However, the proposed method extends the capabilities of the multiphaseEulerFoam solver by incorporating an interface tracking method for continuous bubbles. This proposed method facilitates the simultaneous consideration of both dispersed and continuous bubbles. As a result, it is possible to capture complex bubble dynamics—such as coalescence, breakup, and detailed interfacial interactions—that are characteristic of high void fraction conditions. This enhancement provides a more comprehensive and accurate representation of the actual bubble behavior compared to traditional Euler-based methods.

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

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

2.2 Wall boiling model

The wall boiling model is employed to simulate heat transfer between the heated surface and the liquid phase in boiling systems. A widely adopted approach is the RPI model, proposed by Kurul and Podowski [2], which accounts for three primary heat transfer mechanisms: single-phase convection, quenching, and evaporation. The total heat flux, expressed in Eq. (3), is the sum of the convective heat flux, evaporative heat flux, and quenching heat flux.

$$q''_{wall}=q''_{conv}+q''_{evap}+q''_{quench} \tag{3}$$

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

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

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

The RPI model requires sub-models to determine nucleation site density, bubble departure diameter, and bubble departure frequency. In this study, commonly used closure models available in OpenFOAM were employed to provide these parameters. Specifically, the nucleation site density was evaluated using the Lemmert-Chawla model [3], as defined in Eq. (7). The Tolubinsky and Kostanchuk model [4] was applied to estimate the bubble departure diameter, as shown in Eq. (8). Lastly, the bubble departure frequency was calculated based on the Cole model [5], as described in Eq. (9).

- Nucleation site density model

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

- Bubble departure diameter model

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

- Bubble departure frequency model

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

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, 3000 kW/m^2
Dimension	2D, 3D

3. CFD Simulation

3.1 Simulation conditions

In this study, a flow boiling simulation was carried out within a rectangular channel, with the computational domain illustrated in Fig. 1. The analysis domain included both solid and fluid regions, and a conjugate heat transfer analysis was performed to account for heat conduction within the solid and heat transfer across the fluid-solid interface. The detailed configuration of the computational domain is presented in Fig. 2.

The computational grid was uniformly structured with square cells of 0.5 mm in size. A 100 mm-long region at the center of the domain was designated as the primary area of interest, while inlet and outlet sections, each extending 50 mm, were positioned above and below this region. The heating surface, located at the upper boundary of the computational domain, had dimensions of 23 mm × 100 mm, where a constant heat flux boundary condition was imposed. The cross-sectional dimensions of the flow channel were set to 23 mm × 10 mm. At the inlet, a velocity boundary condition was applied, while the outlet was specified as a pressure boundary condition. Water was used as the working fluid at an operating pressure of 500 kPa, with an inlet subcooling of 20 K. The key simulation conditions are summarized in Table I.

3.2 Simulation results

This study analyzed the distribution of void fraction as a key result to compare bubble behavior based on different numerical approaches. Fig. 3(a) presents the simulation results obtained using the Euler method under a heat flux condition of 2000 kW/m². In this method, bubbles near the heated wall appear as continuous structures with uniform thickness, making it effective for predicting the overall bubble distribution. However, it has limitations in accurately capturing detailed interfacial dynamics during bubble growth. Specifically, the Euler method simplifies interactions between bubbles and the liquid, making it difficult to fully replicate the complex bubble behavior observed in physical phenomena. Consequently, it struggles to accurately simulate intricate interfacial phenomena such as bubble coalescence and breakup, limiting its ability to provide detailed insights into individual bubble dynamics.

In contrast, Fig. 3(b) presents the results of a proposed methodology that selectively applies an interface tracking method to continuous phase bubbles. This approach models both the dispersed phase and continuous phase using appropriate numerical techniques, enabling a more precise prediction of the overall bubble distribution. As a result, it provides a more comprehensive understanding of how various types of bubbles are distributed and interact within the flow.

Notably, Fig. 3(b-2) illustrates the detailed interface dynamics of continuous bubbles using the VOF method. This approach accurately describes how large bubbles form and interact with the surrounding liquid. For example, the VOF method can precisely track the process of small bubbles merging into larger bubbles or large bubbles breaking into smaller ones, allowing for a clearer analysis of the dynamic interactions between bubbles and the liquid.

Fig. 4 presents the simulation results under a heat flux condition of 3000 kW/m², further demonstrating the methodological differences. The results confirmed that the proposed approach provides more reliable and accurate predictions than the conventional Euler method, particularly under high void fraction conditions. The VOF method effectively captures the complex interactions associated with the interface dynamics of continuous bubbles, making it advantageous for accurately reproducing real physical phenomena. These findings highlight the importance of selecting an appropriate numerical approach, suggesting that the proposed method may be essential in specific scenarios.

These differences become even more pronounced in three-dimensional simulations. Fig. 5 presents the three-dimensional bubble behavior obtained using each numerical method. The results from the conventional Euler method showed that, similar to the two-dimensional case, dispersed bubbles are uniformly distributed along the heated wall with a consistent thickness. In contrast, the improved approach revealed that dispersed bubbles (white) initially formed at the heated wall undergo coalescence or growth, transitioning into continuous bubbles (green) that form interfaces and exhibit distinct behaviors. Unlike the two-dimensional case, the three-dimensional simulation clearly demonstrates spatial bubble distribution and interactions.

In conclusion, this study effectively highlights the differences between numerical approaches in analyzing bubble shapes and behaviors. The results suggest that the proposed method is more suitable for detailed investigations of complex flow phenomena. These findings provide valuable insights for selecting appropriate numerical techniques in engineering applications that require precise analysis of bubble behavior.

4. Conclusions

This study effectively elucidated the performance variations between numerical approaches in analyzing bubble shapes and behaviors. By comparatively analyzing the conventional Euler method and the proposed method, the inherent strengths and weaknesses of each methodology were specifically examined. The findings indicated that while the conventional Euler method demonstrated effectiveness in predicting the overall bubble distribution, its capacity to capture detailed interfacial dynamics remained limited, particularly for continuous bubbles. Accurately modeling the interfacial behavior of continuous bubbles and the complex interactions within high-speed liquid flows proved challenging with this method.

Conversely, the proposed improved method, incorporating a selective application of an interface tracking method for continuous-phase bubbles, exhibited enhanced accuracy in predicting diverse bubble distributions and interactions. By proficiently capturing the intricate interfacial dynamics of continuous bubbles through the VOF method, the simulations presented results that more closely aligned with actual physical phenomena. The superior performance of the proposed method was particularly evident under conditions of high-speed liquid flows and elevated heat flux, underscoring the critical importance of selecting appropriate numerical techniques in specific engineering applications. Furthermore, three-dimensional simulation results more clearly differentiated these performance variations, suggesting the improved effectiveness of the proposed method for analyzing bubble behavior in realistic engineering systems.

In conclusion, this study demonstrably explained the performance variations among different numerical approaches and substantiated the improved suitability of the proposed method for the analysis of complex flow phenomena. These outcomes are anticipated to offer significant guidance for the selection of appropriate numerical techniques within engineering applications necessitating the precise analysis of bubble behavior.

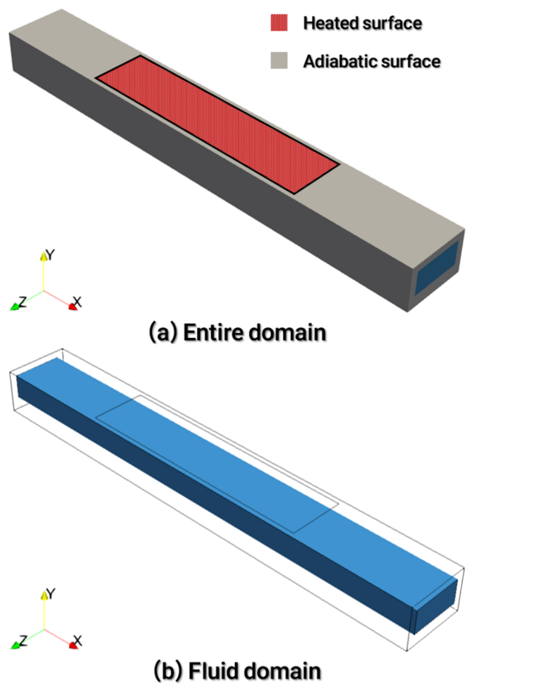

Fig. 1. Schematic of (a) entire domain and (b) fluid domain.

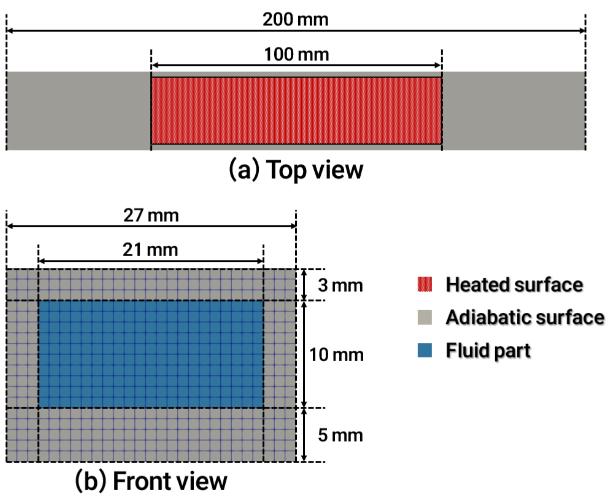

Fig. 2. Dimension of computational domains: (a) top view, (b) front view.

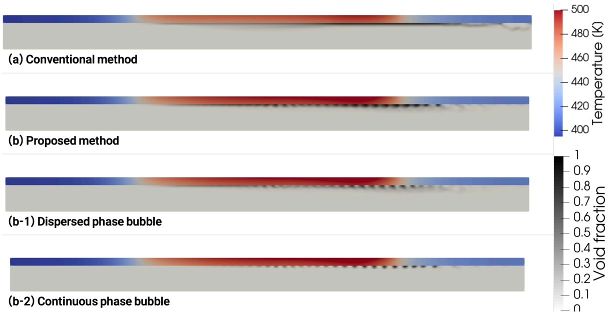

Fig. 3. Distribution of solid temperature and void fraction under low heat flux conditions (2 MW/m^2) in 2D simulation: (a) conventional method, (b) proposed method.

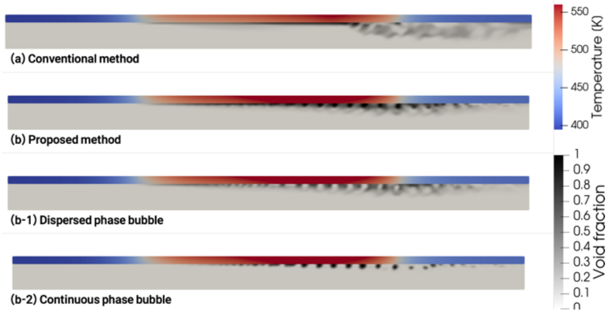

Fig. 4. Distribution of solid temperature and void fraction under high heat flux conditions (3 MW/m^2) in 2D simulation: (a) conventional method, (b) proposed method.

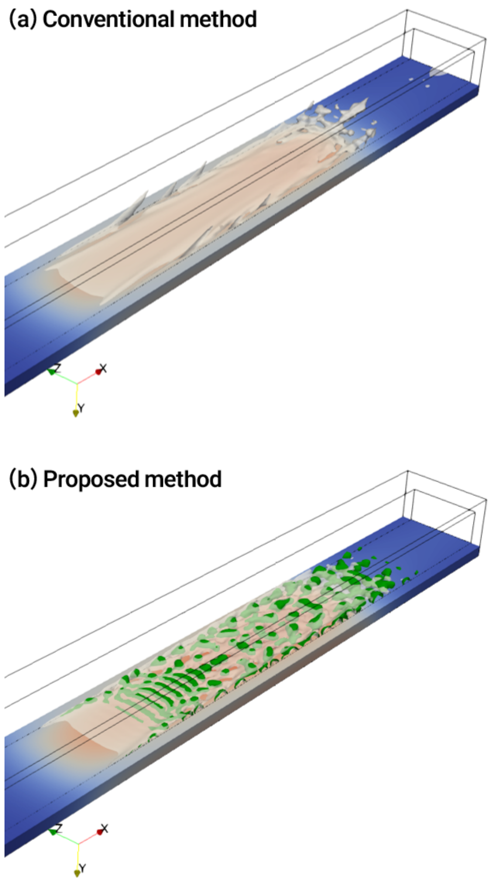

Fig. 5. Distribution of solid temperature and void fraction under high heat flux conditions (3 MW/m^2) in 3D simulation: (a) conventional method, (b) proposed method.

ACKNOLEGEDMENT

This work was supported by Korea Research Institute for defense Technology planning and advancement(KRIT) grant funded by the Korea government(DAPA(Defense Acquisition Program Administration)) (No. G08BR2200010101(KRIT-CT-22-022), Ultra-High-Flux Cooling Systems Research Laboratory, 2022). This work was supported by the Korea Atomic Energy Research Institute (KAERI) [524520-25].

REFERENCES

[1] 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.

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

[3] M. Lemmert and J. Chawla, Influence of flow velocity on surface boiling heat transfer coefficient, Heat transfer in boiling, pp. 231–247, 1974.

[4] 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.

[5] R. Cole, A Photographic Study of Pool Boiling in the Region of the Critical Heat Flux, 1960.