Numerical Modeling of Flow Boiling in a Rectangular channel using OpenFOAM

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

1. Introduction

ERVC (External Reactor Vessel Cooling) is a crucial strategy for maintaining the integrity of the reactor vessel during severe accidents. The operating conditions of ERVC, such as flow velocity and system pressure, are much lower than those in the PWR sub-channel. In addition, due to the high degree of its superheating, bubbles easily form on the heated surface. These bubbles are difficult to lift off due to the geometrical characteristics of the reactor vessel, resulting in the formation of slug bubbles through the merging of small bubbles.

When slug bubbles are generated, nucleate boiling occurs between them and the heated surface [1]. This phenomenon indicates the presence of a thin liquid film, and the behavior of this film governs the heat transfer of slug bubbles. Therefore, accurately evaluating the cooling performance of ERVC requires considering the heat transfer mechanism of slug bubbles in a high void fraction flow regime. However, the wall heat flux partitioning model used in conventional CFD software does not incorporate this mechanism. Instead, the heat transfer mechanism of slug bubbles is treated as vapor convection, which is physically unreasonable.

Various studies have been conducted to investigate the phenomena of flow boiling. One such study, conducted by Kim and Bang [1] , involved an experimental investigation of flow boiling on a downward-facing heated wall. The authors measured the length, velocity, and frequency of slug bubbles and proposed an empirical correlation of the reduction factor, which reduced the sliding bubble parameters. In another study, Muritala and Kim [2] developed a hybrid wall boiling model in the Euler-Euler model with a large-scale interface model using Star-CCM+. This model enabled a more realistic representation of the physical phenomena in flow boiling conditions.

In this study, a numerical modeling was presented for flow boiling that can cover the high void fraction regime. Using a multifield solver and a population balance model, the generated bubbles are classified into dispersed bubbles or continuous bubbles. A wall heat flux partitioning model was developed for both types of bubbles and selectively applied the appropriate heat transfer mechanism to each bubble using the bubble group information from the population balance model. To evaluate this solver, the flow boiling simulation was conducted in an inclined rectangular channel.

2. CFD methodology

2.1. Interface capturing method

The hybrid multifield solver in OpenFOAM utilizes an interface tracking method with the interface compression scheme proposed by Weller [3]. The solver solves the volume fraction transport equation of the multi-fluid model, with an additional artificial compression term, as shown in Eq. 1.

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

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

The artificial compression term, $\vec{u}_c \alpha_k (1 - \alpha_k)$, enables the interface tracking method to work only at the interfaces. The value of the constant $C_{\alpha.ki}$ determines whether the interface compression method is used or not. When $C_{\alpha.ki} = 1$, the interface compression is enabled, and when $C_{\alpha.ki} = 0$, it is disabled. The constant can be set individually for each phase. For example, the constant is set to 0 for the dispersed gas-continuous liquid interface and 1 for the continuous gas-continuous liquid interface.

2.2. Wall heat flux partitioning model

The wall heat flux partitioning model (WHFP) is used to predict the heat transfer between the wall and the liquid phase in a boiling system. The developed WHFP model is divided into two parts: one for dispersed phase bubbles and the other for continuous phase bubbles.

For dispersed phase bubbles, the PRI model of Kurul and Podowski [4] is commonly used. This model considers three heat transfer mechanisms: single-phase convection, quenching, and evaporation. The total heat flux is given by Eq. 3, where $q''_{disp.}$ is the total heat flux, q''_c is the convection heat flux, q''_e is the evaporation heat flux, and q''_q is the quenching heat flux.

- WHFP model for dispersed bubbles

$$q''_{disp.} = q''_c + q''_q + q''_e \quad (3)$$

- Convective heat transfer

$$q''_c = h_c A_{1\phi} (T_w - T_l) \quad (4)$$

- Evaporative heat transfer

$$q''_e = N_a \left(\frac{\pi}{6} D_{dep}^3 \right) f \rho_g h_{fg} \quad (5)$$

- Quenching heat transfer

$$q''_q = h_q A_{2\phi}(T_w - T_l) \quad (6)$$

For continuous phase bubbles, the liquid film beneath a slug bubble is thin enough, so it is assumed that conduction heat transfer only occurs across the liquid film. The heat flux for the continuous bubbles is given by Eq. 7, where $q''_{cont.}$ is the heat flux, k_l is the thermal conductivity of the liquid phase, δ_{film} is the thickness of the liquid film, T_w is the wall temperature, and T_{sat} is the saturation temperature.

- WHFP model for continuous bubbles

$$q''_{cont.} = \frac{k_l}{\delta_{film}}(T_w - T_{sat}) \quad (7)$$

To account for the coexistence of dispersed and slug bubbles, a hybrid wall boiling model is used, where the contribution of each WHFP model is determined by a weighting function correlated with the void fraction of slug bubbles (α_{cont}). The contribution of each model at a position is determined by Eq. 8, where α_1^f and α_2^f are the upper and lower limits of the volume fraction of slug bubbles to be applied to the weighting function, respectively. These values are typically set to 0.5 and 1.0, respectively, as suggested in previous studies.

- Blending function for WHFP model

$$H(\alpha_{cont}) = max\left(0, min\left(1, \frac{\alpha_2^f - \alpha_{cont}}{\alpha_2^f - \alpha_1^f}\right)\right) \quad (8)$$

- WHFP model for both bubbles

$$q''_w = [1 - H(\alpha_{cont})]q''_{disp} + H(\alpha_{cont})q''_{cont} \quad (9)$$

2.3 Sub-models for RPI boiling model

The RPI boiling model requires sub-models for nucleation site density, bubble departure diameter, and bubble departure frequency to be used. The commonly used closure models in OpenFOAM releases were adopted for this study. The Lemmert-Chawla model [5] was used to determine the nucleation site density as given in Eq. 10. The bubble departure frequency was calculated using the Cole model [6] as shown in Eq. 11. In this equation, The Tolubinsky and Kostanchuk model [7] was used to determine the bubble departure diameter, as shown in Eq. 12.

- Nucleation site density: Lemmert-Chawla

$$N_a = C_n N_{a.Ref}\left(\frac{\Delta T_{sup}}{\Delta T_{Ref}}\right)^n \quad (10)$$

- Bubble departure frequency: Cole

$$f = \sqrt{\frac{4g(\rho_l - \rho_g)}{3D_{dep}\rho_l}} \quad (11)$$

- Bubble departure diameter: Tolubinsky and Kostanchuk

$$D_{dep} = min\left(d_{ref}e^{\left(-\frac{\Delta T_{sub}}{\Delta T_{ref}}\right)}, d_{max}\right) \quad (12)$$

3. CFD simulation

3.1 Simulation conditions

To simulate flow boiling, a two-dimensional rectangular channel with a 30° inclined angle from the horizontal plane (as shown in Fig. 2) was used. The heated surface was located at the top of the computational domain, with a length of 150 mm. Two adiabatic areas of 50 mm were positioned on either side of the heated surface. The simulation conditions are summarized in Table I, with an outlet pressure of 100 kPa, subcooling of 5K, and a liquid velocity of 300 kg/m^{2}s. To study the effect of heat flux, the applied heat flux was varied from 100 to 300 kW/m^2, and the simulation results were compared.

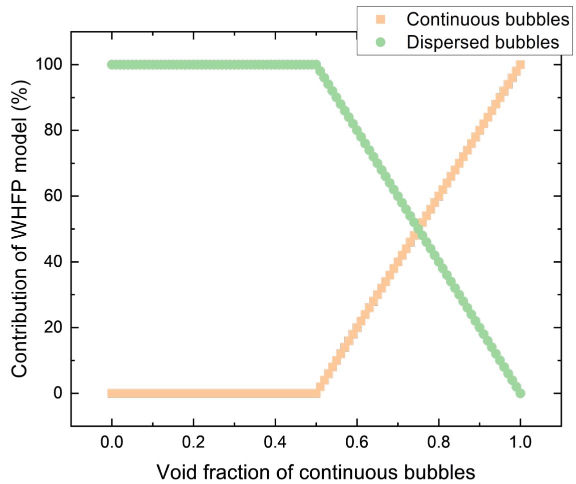

Fig. 1. Contribution of WHFP model for bubble group

Table I. Major conditions of flow boiling simulation

Variable	Value
Outlet pressure	100 kPa
Inlet subcooling	5 K
Mass flux	300 kg/m^{2}s
Applied heat flux	100, 200, 300, 400 kW/m^2
Inclined angle	30°

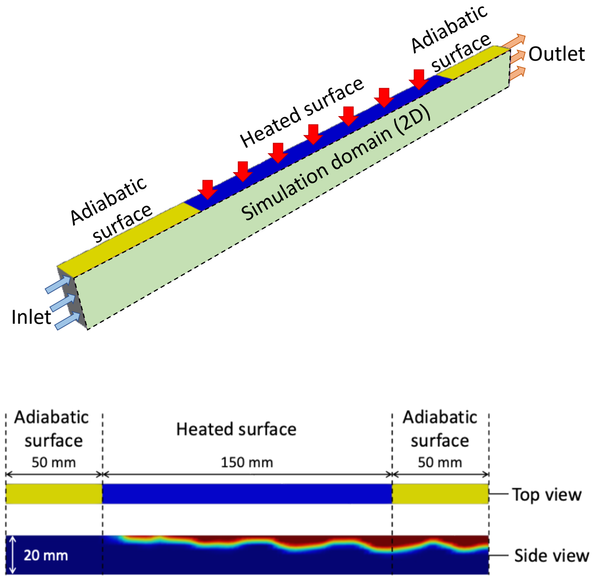

Fig. 2. Schematic of computational domain

3.2 Simulation results

Two sets of simulations were conducted to compare the effect of different numerical modeling techniques on flow boiling simulation. The reference case used conventional numerical modeling techniques such as the Eulerian approach for multifield method and the RPI model for wall heat flux partitioning. The second set used advanced numerical modeling techniques including the VOF interface capturing method to track the bubble interface and a developed wall boiling model to account for the heat transfer mechanism in the liquid film beneath the slug bubbles.

Both sets of simulations were conducted with increasing heat flux from 100 kW/m^2 to 400 kW/m^2 to observe the transition of flow regime. The VOF interface capturing method was used to track the interface of bubbles and apply the heat transfer mechanisms involved in the liquid film beneath the bubbles. The developed wall boiling model was applied to consider the heat transfer mechanisms involved in the liquid film beneath the slug bubbles.

The distribution of void fraction was set as the main comparison result, as the difference in the numerical model would have a significant influence on it. Fig. 3 shows the void fraction distribution according to the model used. In the reference case, the void fraction of bubbles behaved like a vapor film, as the dispersed bubble and continuous bubbles were mixed in the computational mesh due to the Eulerian approach. Therefore, it could not reflect the behavior of slug bubbles due to surface tension, which could have affected the wall heat flux partitioning model.

In contrast, the rounded shape of bubbles was observed in the simulation result using the developed method. In low heat flux cases, the elliptical discrete bubbles moved along the heated wall, while in high heat flux conditions, the formation of slug bubbles due to growth and merging of bubbles was observed, and the deformed interfaces were observed due to drag at the interface and surface tension. However, there was no discrete region between the slug bubble and water. This could be due to the mesh size and the momentum closure model between bubbles and water. Further research is required to resolve this issue.

4. Conclusion

This research aimed to develop numerical modeling for the high void fraction regime in flow boiling simulation on a downward heated surface. The multifield solver enabled the application of both Eulerian and VOF methods to simulate the interface of slug bubbles. The developed WHFP model reflects the contribution of each heat transfer mechanism for discrete and slug bubbles. The simulation successfully captured the slug bubble formation due to bubble merging on the downward heated surface with varying heat flux.

To further validate the accuracy of the simulation results, the shape of the slug bubble and the contribution of heat transfer mechanisms will be compared with experimental data from flow boiling on a downward-facing wall in future studies. This will provide a more comprehensive understanding of the heat transfer characteristics in flow boiling under high void fraction conditions and contribute to the development of more accurate numerical models for such systems.

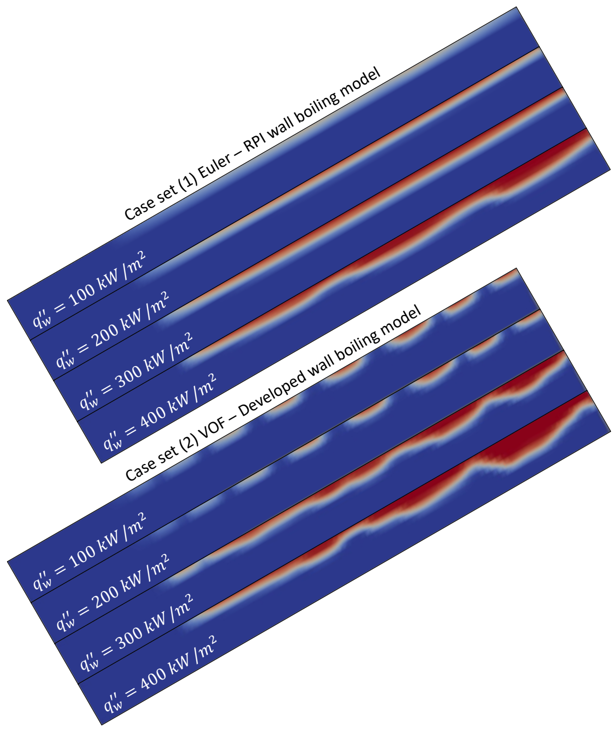

Fig. 3. Void fraction distribution of each simulation sets

ACKNOLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT: Ministry of Science and ICT) (2019M2D2A1A02059364).

REFERENCES

[1] H.-T. Kim and K.-H. Bang, "An experimental study of flow boiling from downward-facing heated wall in inclined channels," *Int J Heat Mass Transf*, vol. 133, pp. 920–929, Apr. 2019, doi: 10.1016/j.ijheatmasstransfer.2018.12.183.

[2] M. A. Amidu and H. Kim, "Modeling and simulation of flow boiling heat transfer on a downward-facing heating wall in the presence of vapor slugs," *Nuclear Engineering and Design*, vol. 351, no. May, pp. 175–188, Sep. 2019, doi: 10.1016/j.nucengdes.2019.05.032.

[3] 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, doi: 10.1155/2013/128936.

[4] N. Kurul and M. Z. Podowski, "MULTIDIMENSIONAL EFFECTS IN FORCED CONVECTION SUBCOOLED BOILING," in *Proceeding of International Heat Transfer Conference 9*, 1990, pp. 21–26. doi: 10.1615/IHTC9.40.

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

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

[7] V. I. Tolubinsky and D. M. Kostanchuk, "VAPOUR BUBBLES GROWTH RATE AND HEAT TRANSFER INTENSITY AT SUBCOOLED WATER BOILING," in *Proceeding of International Heat Transfer Conference 4*, 1970, pp. 1–11. doi: 10.1615/IHTC4.250.