Wall boiling model and wall condensation model for natural circulation simulation

Quoc-Tuan Le and Byoung Jae Kim*
Department of Mechanical Engineering, Chungnam National University, Yuseong-gu, Daejeon, South Korea
*Corresponding author: bjkim@cnu.ac.kr

***Keywords:** Natural circulation, wall boiling, wall condensation

1. Introduction

Natural circulation is a fundamental feature employed in designing small modular reactors as a passive safety system, ensuring heat removal without mechanical components. Following the accident scenario, the coolant motion in the reactor system is mainly driven by phase change processes, as liquid boils in the reactor core, and steam undergoes condensation upon contact with the containment wall. With an assumption of an ideal natural circulation loop without noncondensable gas, a wall boiling model and a wall condensation model are presented in this study to take account of phase change rates. The schematic of a natural circulation loop is presented in Figure 1. Both models are implemented in the framework of the two-fluid model in FLUENT version 2024R2 [1]. As two models show good agreement with validation cases, they can be incorporated into simulating a full circulation loop in future research.

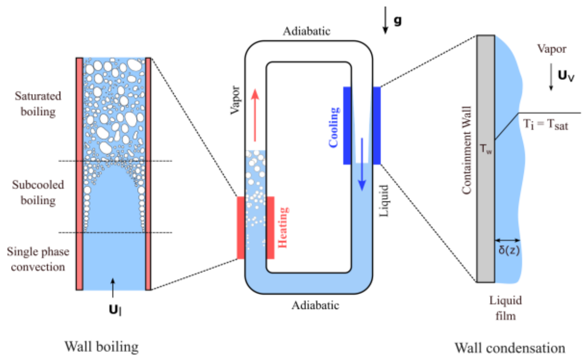

Fig. 1. Schematic of natural circulation loop

2. Wall boiling model

The RPI wall boiling model [2] is applied in the heating region where boiling occurs. Heat is transferred from the heated wall to fluid via three primary mechanisms.

$$q_c = h_c\left(1-A_b\right)\left(T_w-T_l\right) \quad (1)$$

$$q_q = h_q A_b\left(T_w-T_l\right) \quad (2)$$

$$q_e = (\pi/6)D_w^3 N_w \rho_v h_{fv} f_w \quad (3)$$

Where q_c, q_q, q_e are the convective heat flux, the quenching heat flux, and the evaporative heat flux, respectively. h_c is the liquid phase convective heat transfer coefficient calculated from the FLUENT temperature wall function. h_q is the quenching heat transfer coefficient. A_b, D_w, N_w, f_w are the bubble area of influence, bubble departure diameter, nucleate site density, and bubble departure frequency, respectively. The empirical models applied in the boiling simulation are given in Table I. The mass transfer rate in the wall adjacent cells is calculated based on evaporative heat flux. Meanwhile, vapor temperature is assumed to remain at saturation temperature in the bulk region, and the volumetric condensation rate is determined based on the thermal phase change model.

In the heating region, the initial water temperature is below saturation temperature. As the high heat flux is assigned at the wall, water is converted to vapor and moved upward in a vertical configuration. The Bartolomei and Chanturiya [3] experiment provides clear boundary conditions with the area-averaged vapor volume fraction measured with good accuracy. Therefore, it is chosen for the validation test in this study. The experimental setup includes a 2-meter vertical pipe with an inner diameter of 15.4 mm, where subcooled water was heated to induce boiling. The condition for the benchmark test is presented in Table 2. An axisymmetric boundary condition is applied as only half of the pipe is simulated. A k-omega SST turbulence model is used in the simulation. As shown in Figure 2, even though the calculated result overpredicts vapor volume fraction at the outlet of the heated pipe, the computed result shows a qualitatively similar trend with experimental data.

Table I: Empirical models in the simulation

	Model	Unit	Ref.
A_b	Del Valle & Kenning	m^2	[4]
D_w	Tolubinski & Kostanchuk	m	[5]
f_w	Cole	1/s	[6]
N_w	Lemmert – Chawla	site/m^2	[7]

Table II: Problem condition for Bartolomej test case

	Value	Unit
Pressure	4.5	MPa
Mass flow inlet	900	kg/m^2s
Heat flux	0.57	MW/m^2
Subcooling Temperature	58.2	K

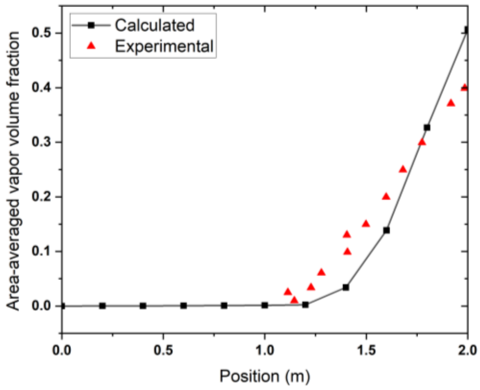

Fig. 2. Comparison of vapor volume fraction between calculated result and experimental data

3. Wall condensation model

Lee [8] proposed a subgrid film model which couples film momentum equation with two fluid equations in the CUPID code to model condensation with noncondensable gas. The subgrid film model provides wall shear stress and interfacial shear stress which can be transferred to two fluid model via wall boundary condition and source terms in momentum equations. Lee's study confirmed the validity of the wall condensation model by comparing it with COPAIN experimental data in both natural convection and forced convection regimes. The heat flux along the condensing wall in the natural heat transfer convection case was shown to be underpredicted in the fully developing region, possibly because the dropwise condensation phenomenon was not considered. Given the assumption that only filmwise condensation is present, the underprediction is within an acceptable range. In our study, the same approach is applied using the User Defined Function in FLUENT to calculate the condensation rate near the wall with a pure vapor scenario. In the case in which noncondensable is neglected, thermal resistance from the liquid film side is dominant, and the effect of vapor thermal resistance can be negligible. The energy conservation equation is calculated as

$$q = \frac{k_l}{\delta}(T_{sat} - T_w) = \Gamma_v h_{lv} \qquad (4)$$

Where k_l is the liquid film conductivity, δ is the film thickness, T_{sat} is saturation temperature, Γ_v is condensation mass flux, h_{lv} is the latent heat. With a guessed value of liquid film thickness, condensation mass flux at the wall is calculated based on equation (4)

Table III: Condition for CONAN test case

	Value	Unit
Coolant inlet temperature	343.91	K
Coolant outlet temperature	349.86	K
Vapor inlet temperature	375.2	K
Vapor inlet velocity	3.593	m/s
Pressure	1	atm

The freestream vapor velocity, which induces interfacial shear stress at the film interface, is the distinguished factor between forced and natural convection regimes. The buoyancy force dominates the inertia force for the lower vapor velocity case. Thus, the condensation phenomenon is under a natural convection regime, resulting in less influence of interfacial shear stress on the condensation rate. The CONAN experiment [9] is chosen in the cooling region to validate wall film condensation. Although the CONAN facility primarily focuses on the forced convection regime, it provides a test case with only pure vapor condensation. The measured condensation rate from the test case is used for validation in this study.

The CONAN experiment consists of a primary duct, where condensation occurs, a secondary channel with coolant water flowing in opposite directions, and an aluminum plate located in between those two. The condition of case P30 – T70 -V35 from benchmark data, which includes pure vapor, is used as the input for our simulation, as presented in Table III. Only the primary channel and solid region are considered in our computational domain. The effect of the secondary channel is applied as a convective boundary condition at the back of the aluminum plate, as shown in Figure 3. The total condensation rate in unit kg/s is compared between computed and experiment data, as shown in Figure 4. The calculated result shows an overprediction of only 3.1%, implying that our model can produce the desired phase change rate when implemented in the natural circulation simulation.

4. Conclusions

This study presents the RPI wall boiling model for boiling and the subgrid film model for pure vapor condensation. By validation with experimental data, both models show the ability to capture phase change rate without the need for fine discretization. In future research, a CFD-based model of natural circulation associated with the phase change process will be investigated using two reported models.

ACKNOWLEDGEMENTS

This work was supported by grants from the National Research Foundation of Korea, funded by the Ministry of Science and ICT (Nos. RS-2024-00393526).

REFERENCES

[1] ANSYS Inc., "Ansys Fluent Theory Guide", 2024R2
[2] N. Kurul and M.Z. Podowski, “Multidimensional effects in forced convection subcooled boiling,” Proceedings of the 9th International Heat Transfer Conference, Jerusalem, Israel, 1990, pp. 21–25.
[3] G.G. Bartolomej and V.M. Chanturiya, “Experimental Study of True Void Fraction When Boiling Subcooled Water

in Vertical Tubes," Thermal Engineering, 14, pp. 123–128 (1967)

[4] V.H. Del Valle and D.B.R. Kenning, "Subcooled Flow Boiling at High Heat Flux," Int. J. Heat Mass Transfer, 28(10), pp. 1907–1920 (1985).

[5] V.I. Tolubinski and D.M. Kostanchuk, "Vapor Bubbles Growth Rate and Heat Transfer Intensity at Subcooled Water Boiling," Proceedings of the 4th International Heat Transfer Conference, Paris, France (1970).

[6] R. Cole, "A Photographic Study of Pool Boiling in the Region of the Critical Heat Flux," AIChE J., 6, pp. 533–542 (1960).

[7] M. Lemmert and L.M. Chawla, "Influence of Flow Velocity on Surface Boiling Heat Transfer Coefficient," in Heat Transfer in Boiling, E. Hahne and U. Grigull (Eds.), Academic Press and Hemisphere, New York, NY, USA (1977).

[8] J. Lee, G.C. Park, and H.K. Cho, "Simulation of Wall Film Condensation with Non-Condensable Gases Using Wall Function Approach in Component Thermal Hydraulic Analysis Code CUPID," J. Mech. Sci. Technol., 32, pp. 1015–1023 (2018).

[9] W. Ambrosini, N. Forgione, and F. Oriolo. Experimental and CFD analysis on condensation heat transfer in a square cross section channel. In Proceeding of the NURETH 11 conference, Pope's Palace Conference Center, Avignon, France, October 2-6 2005.

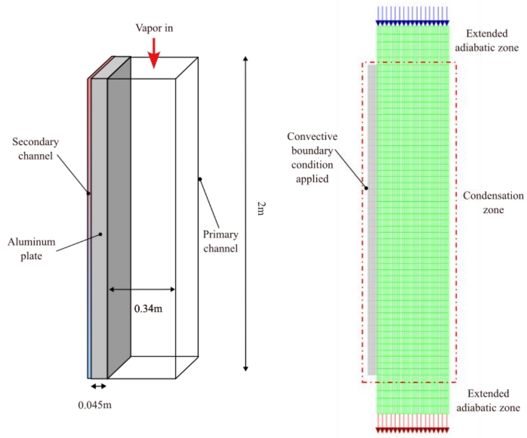

Fig. 3. Sketch of CONAN test facility and computational domain used in our simulation

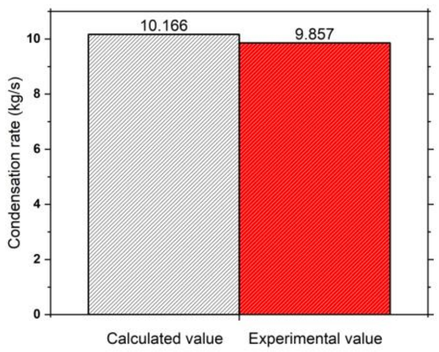

Fig. 4. Comparison of total condensation rate between calculated result and experimental data