Prediction of natural circulation flow in ULPU-V facility loop using the wall boiling model for inclined surface

Gi Su Lee [a], Byoung Jae Kim [a*]
[a]*School of Mechanical Engineering, Chungnam National University.*
[*]*Corresponding author: bjkim@cnu.ac.kr*

1. Introduction

The in-vessel retention (IVR) of molten corium is one of the possible serious accident management action that could be applied at partial nuclear power plants. One of the critical points required to prove the feasibility of an IVR strategy is the heat removal capability through the vessel wall by convection and boiling in the coolant.

Many numerical simulation studies for predicting natural circulation use the RPI wall boiling model. However, the RPI wall boiling model does not take into account the inclined wall effect of the heater blocks. In this study, the inclined wall effect of the heater blocks was reflected in the wall boiling model, which was used to simulate the natural circulation experiment in the ULPU-V facility.

2. ULPU-V facility

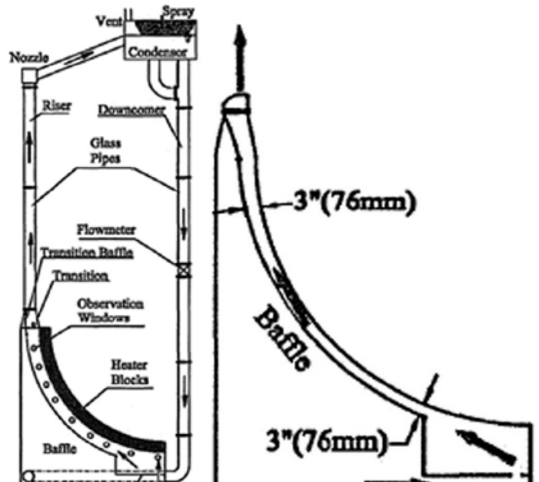

Fig. 1. Schematic of the ULPU-2400 configurations V. [1)]

ULPU is an experimental facility at the University of California Santa Barbara designed to assess the coolability limits of an IVR strategy. It was first introduced to perform a program of tests specifically devoted to the Lovissa reactors currently operating in Finland and subsequently modified to assess the Westinghouse AP600 and AP1000 reactors. The facility is a full-scale representation, with respect to height, of a reactor lower head and the whole flow path between the reactor vessel and the reflecting thermal insulation, all the way to the top venting openings. The full-scale height of the flow path (gravity head) is represented for accurate simulation. [1)]

As shown in Fig 1, we simulated equidistant flow channels with respective depths of 3" (76 mm) and the distance to the vessel is 3" at the lowest point of the curved baffle. This figure also shows the direction of coolant injection.[1)]

Some previous works investigated a wall boiling model to predict natural circulation flow rate in this experimental facility, however they does not reflect the inclined wall effect of the heater blocks. [2) 3)]

3. Wall boiling for an inclined wall

The RPI model is the most widely used wall boiling model, and the heat applied to the fluid from the heater blocks consists of single-phase convection, quenching, and evaporation. [4)]

This study deals with the heat transfer by evaporation and calculates as follows.

$$Q_E = \dot{m}_W H_{LG}$$

Where Q_E is the heat flux by evaporation, and $\dot{m}_W$ is the mass flow rate that evaporates from the wall, and H_{LG} is the latent heat associated with evaporation and condensation.

The mass flow rate produced by evaporation is given as follows.

$$\dot{m}_W = \rho_G \frac{\pi}{6} d_W^3 f N$$

Where ρ_G is the density of bubbles, and d_w is the bubble departure diameter. In addition f is the frequency of bubble departure, and N is the nucleation site density.

According to the recent experiment[5)], it was reported that the d_w and f are dependent on the heater surface inclines. Therefore this study, reflecting the results of this experiment, d_w was modified as follows in the area of the heater blocks. [5)]

$$d_{W0} = \min(0.0014, 0.0006e^{-\frac{\Delta T_{sub}}{45.0}})$$

$$\Delta T_{sub} = T_{sat} - T_l$$

$$d_W = d_{W0} \times (1.022y^3 - 1.339y^2 + 0.9117y + 1)$$

Where d_{w0} is the default bubble departure diameter for the RPI model in ANSYS FLUENT, and y is the height from the floor. [6)]

Numerical simulation was performed using ANSYS FLUENT and d_w was modified using UDF.

4. Simulation conditions

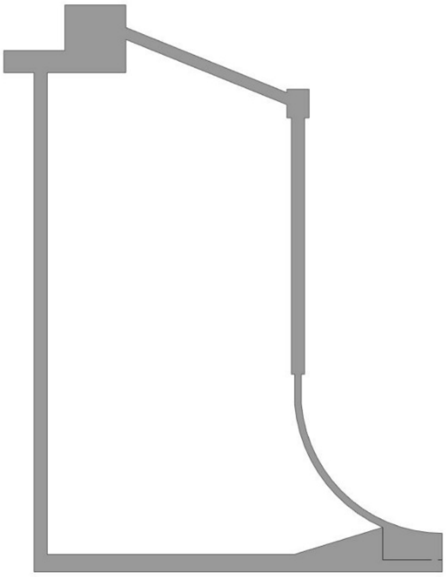

Fig. 2. Computational domain of ULPU-2400 configurations V.

Fig. 2 represents the computational domain of ULPU-2400 Configuration V for computational simulation.

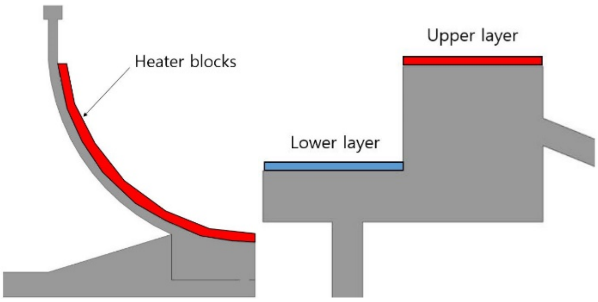

Fig. 3. Heater blocks (left), Condenser (right).

Fig. 3 is an enlarged part of Fig. 2. In the left figure, the thermal load generated inside the reactor vessel is shown as the heat flux boundary condition on the wall corresponding to the heater blocks. The upper surface of the condenser was divided into two layers to promote natural circulation under atmospheric pressure by allowing only the reentry of the vapor flow in the upper layer and the liquid flow in the lower layer.

Table I: Test conditions for ULPU-V simulated runs. [1)]

Run #	4
Water	DI
Heater treatment	Tap water boiling treatment
Baffle position	3”-3”
Power shape	T40B
CHF position (◦)	71
CHF (kW/m^2)	1782
Flow rate (m^3/min)	0.644

In this study, the SST(Shear Stress Transport) model was used for the turbulence model, and a two-dimensional abnormal simulation was performed with a time step of 0.025s. The test conditions are given in Table I.

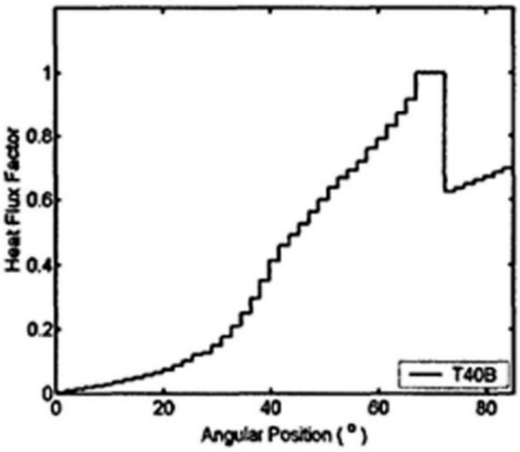

Fig. 4. Angular dimensionless heat flux profile. [7)]

The results of the ACOPO test were used for the heat flux boundary conditions corresponding to the thermal load from inside the reactor vessel. The heat flux at the wall of the reactor vessel, which is formed according to the distribution of molten corium inside the reactor vessel, consists of a deflection function form defined along the hemispherical shape starting from the center of the lower half of the reactor vessel, and provides the maximum with non-dimensionalization to 1. Fig. 4 represents the dimensionless heat flux function used in this simulation. In this study, the heat flux boundary condition of the reactor vessel for each simulation shown in Fig. 5 was composed by multiplying the non-dimensional heat flux function of Fig. 4 by the fraction of the critical heat flux to allow this simulation until the maximum heat flux reaches 1.782 MW/m^2, the critical heat flux obtained from the ULPU-V test.

4. Result and discussion

First, the liquid velocity applying UDF in the downcomer mentioned in Chapter 3 is shown in Fig. 5.

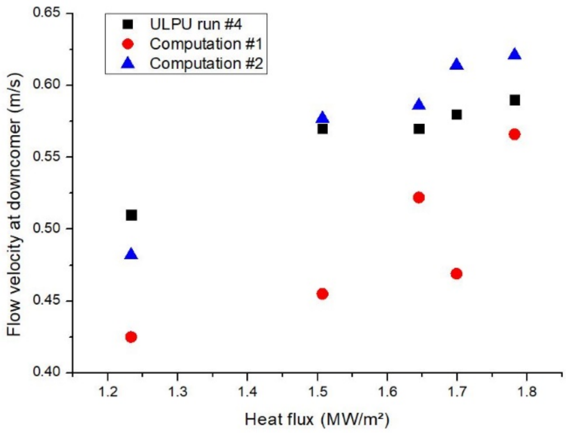

Fig. 5. Comparison of the liquid velocity applying UDF in the downcomer.

'Computation #1' is before applying UDF, and 'Computation #2' is after applying UDF. In the former case, the error range was around 13.7% compared to the ULPU test results, however in the latter case, the error range was predicted to be about 4.1%. Therefore, it can be seen that the application of UDF reflecting the inclined wall effect to the RPI wall boiling model better predicts the coolant speed at the downcomer. In addition, the error range in the previous wall boiling model is approximately 10% in CFX[2] and 3.4% in NEPTUNE CFD[3], indicating that this simulation is well predicting actual trends.

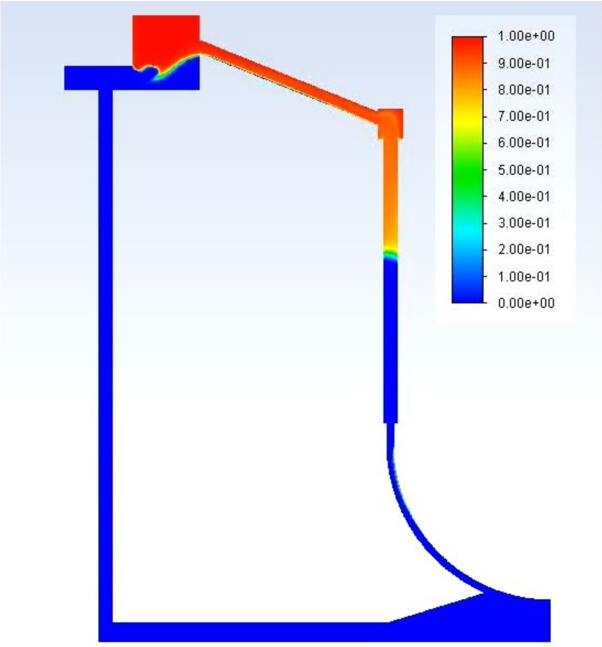

Fig. 6. Liquid volume fraction in the cooling loop; Heat flux=1.233 MW/m^2

When the maximum heat flux on the reactor wall is 1.233 MW/m^2, the liquid volume fraction in the coolant pipe is shown in Fig. 6. The bubbles generated by the subcooled boiling phenomenon on the walls of the reactor are condensed directly from the rear of the heater blocks, and the coolant rises to the single phase flow, and then the bubbles are regenerated and released out through the condenser by flashing phenomenon at the top due to reduction of saturation temperature by altitude.

5. Conclusions

Since the previous RPI wall boiling model does not take into account the inclined wall effect of the heater blocks, this study reflected the inclined wall effect of the heater blocks in the wall boiling model and used it to simulate the natural circulation experiment in the ULPU-V facility. As a result of the simulation, the average speed of the coolant at the lower part corresponding to the natural circulation flow rate of the coolant was more similar to the previous test results by applying UDF considering the inclined wall effect of the heater blocks. In addition, the results of this simulation showed better results than the previous works.

ACKNOWLEDGEMENTS

This work was supported by a National Research Foundation of Korea grant funded by the Ministry of Education (No. NRF-2020R1A2C1010460).

REFERENCES

[1] T-N. Dinh, J. P. Tu, T. Salmassi, T. G. Theofanous, "Limits of Coolability in the AP1000-Related ULPU-2400 Configuration V Facility", CRSS Technical Report 0306, 2003.
[2] Jungsoo Suh, Huiun Ha, "Numerical Simulation on the ULPU-V Experiments using RPI Model", Journal of the Korean Society of Safety, Vol. 32, pp. 147-152, 2017.
[3] M. Jamet, J. Lavieville, K. Atkhen and N. Mechitoua, "Validation of NEPTUNE CFD on ULPU-V experiments", Nuclear Engineering and Design, Vol. 293, pp. 468-475, 2015.
[4] B. Koncar, I. Kljenak and B. Mavko, "Modelling of Local Two-phase Flow Parameters in Upward Subcooled Flow Boiling at Low Pressure", International Journal of Heat and Mass Transfer, Vol. 47, pp. 1499-1513, 2004.
[5] Satbyoul Jung, Hyungdae Kim, "Effects of surface orientation on nucleate boiling heat transfer in a pool of water under atmospheric pressure", Nuclear Engineering and Design, Vol. 305, pp. 347–358, 2016.
[6] ANSYS, Inc., "ANSYS Fluent Theory Guide", Canonsburg, PA, USA, 2019.
[7] B. R. Sehgal et al., "Nuclear Safety in Light Water Reactors, Academic Press", Waltham, MA, USA, 2012.