

Preliminary Simulation of a Small-Scale Pool Experiment Using ANSYS FLUENT for Application to Large Pools of Passive Safety Systems

Jinwook Choi^a, Minseok Choi^a, Seong-Su Jeon^{a*}

^aFNC Technology Co., Ltd., Floor 32, 13 Heungdeok 1-ro, Giheung-gu, Yongin-si, Gyeonggi-do, Republic of Korea

*Corresponding author: ssjeon@fncotech.com

***Keywords :** Passive safety systems, Natural circulation, CFD, VOF(Volume of Fluid), ANSYS Fluent

1. Introduction

An innovative small modular reactor (i-SMR) is being developed in the Republic of Korea. The i-SMR adopts several passive safety systems that cool the reactor safely for a long time after an accident. Among them, the passive auxiliary feedwater system (PAFS) removes the residual heat through the steam generator, and the passive containment cooling system (PCCS) removes the heat from the containment vessel (CV) during an accident [1].

The PAFS and the PCCS use the emergency cooling tank (ECT) as a common heat sink. The passive auxiliary feedwater heat exchanger (PAFHx) of the PAFS is submerged in the ECT. During an accident, the steam that comes from the steam generator condenses inside the PAFHX, and the removed heat is transferred to the cooling water of the ECT. As a result, heating of the cooling water, natural convection and boiling occur around the PAFS heat exchanger. The heat exchanger of the PCCS is also connected to the ECT, and the heat removed from the CV is transferred to the ECT by the natural circulation flow of the PCCS.

Therefore, the temperature and flow fields formed inside the ECT can be different when the PAFS and the PCCS operate separately or at the same time. In particular, the buoyant flow around the PAFHX and the inlet and outlet flows of the PCCS interact in the same pool, and this interaction can affect the natural convection circulation, the thermal mixing, the thermal stratification and the boiling behavior in the pool. These changes can in turn affect the heat removal conditions of the PAFS and the PCCS. For this reason, it is necessary to compare the thermal-hydraulic fields inside the ECT under PAFS-only operation, PCCS-only operation and simultaneous operation of the two systems, and to evaluate whether the thermal-hydraulic interaction between the two passive safety systems increases the performance, decreases the performance, or is small enough to be neglected.

Inside the ECT, natural convection, boiling, free surface behavior and the spatial temperature distribution appear together, so a multi-dimensional analysis is required to evaluate these phenomena. In this study, the application of the commercial computational fluid dynamics (CFD) code ANSYS FLUENT is examined as a method for analyzing the multi-dimensional thermal-hydraulic behavior of the ECT in the future. ANSYS

FLUENT provides the volume of fluid (VOF) model and the Eulerian multiphase model. As the first step, the VOF model, which tracks the gas-liquid interface directly, was applied in this study.

The actual i-SMR ECT has a large analysis domain, and a long transient has to be considered during an accident. Therefore, before the large pool is analyzed directly, it is necessary to understand the basic calculation characteristics and the computational cost of the VOF model using a relatively small problem. In this study, an ANSYS FLUENT analysis model was constructed for the small-scale pool boiling experiment performed by Kim [2]. The purpose of this study is not to validate the experimental results quantitatively. Instead, it is to check preliminarily whether a VOF-based analysis model can simulate the main phenomena in the pool, such as vapor generation by heating, buoyant upward flow, natural circulation and the free surface. The study also examines the directions of model improvement and computational cost reduction that will be needed for the analysis of a large cooling pool.

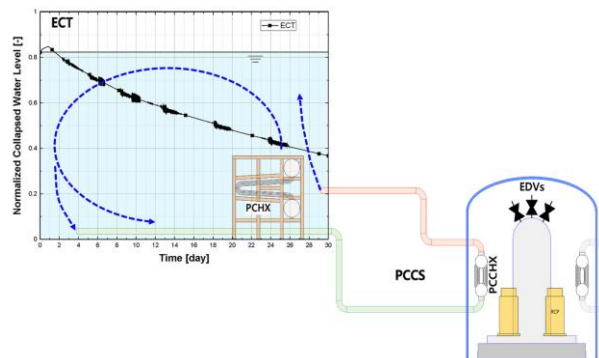


Fig. 1. Multi-dimensional thermal-hydraulic phenomena in the ECT [1].

2. ANSYS FLUENT Modeling

2.1 Geometry and Mesh

The analysis target is the small-scale pool boiling experiment performed by Kim [2]. As shown in Fig. 2, the length, the width and the height of the pool are 300 mm, 60 mm and 650 mm, respectively, and the initial water level is 400 mm. A horizontal heater rod with a diameter of 19.05 mm is installed asymmetrically at a

height of 85 mm from the bottom of the pool, and the length of the heated part is 150 mm.

In this analysis, the circular heater rod was approximated by a square cross-section inscribed in the circle in order to simplify the mesh construction. A relatively fine mesh was applied around the heater rod, and the whole computational domain consisted of about 105,000 cells. Fig. 2 shows the analysis geometry and a representative mesh.

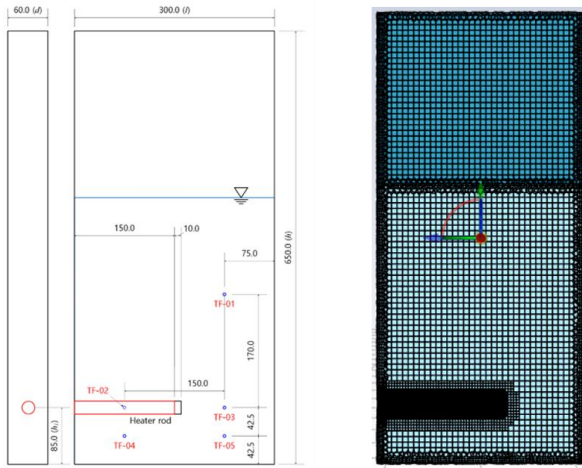


Fig. 2. Geometry [2] and Computational Mesh.

2.2 Physical Models and Boundary Conditions

The VOF model was applied to the multiphase flow analysis. To minimize the numerical diffusion of the gas-liquid interface, the explicit formulation and the sharp interface treatment were applied to the volume fraction equation, and the surface tension was taken into account using the continuum surface force model. The realizable k- ϵ model was used as the turbulence model, and the energy equation was solved together.

The working fluids were liquid water and water vapor. The phase change by evaporation and condensation was implemented with a user-defined function (UDF) based on the Lee model [3]. The main properties of the liquid water were set to change with temperature.

A pressure outlet with the atmospheric pressure condition was applied to the top of the pool, and a heat load of 600 W, which corresponds to the experimental condition, was given to the heater rod. In this step, the walls of the pool were assumed to be adiabatic, so the heat capacity of the structures of the test facility and the heat loss to the environment were not considered.

This study is not intended to reproduce the whole heat-up process up to the saturation condition, but to confirm first the VOF calculation characteristics after boiling has started. Therefore, the initial liquid temperature was set to the saturation temperature of 373.15 K. Because of these differences in the initial and boundary conditions, the present calculation was carried out as a preliminary analysis to confirm the capability of simulating the main

thermal-hydraulic phenomena, rather than as a quantitative validation of the experiment.

2.3 Numerical Conditions

The PISO algorithm was used for the pressure-velocity coupling, and a second-order spatial discretization scheme was applied to the momentum equation. The geo-reconstruct scheme was used for the calculation of the gas-liquid interface.

A time step size was applied in consideration of the numerical stability required by the explicit formulation of the VOF model. A time step size of 0.0001 s was used at the beginning of the calculation, and after that the transient calculation was carried out with the time step adjusted in the range of 0.0001 to 0.001 s. The calculation was performed using 32 CPU cores.

3. Simulation Results

Fig. 3 shows the vapor volume fraction distribution at the time when the vapor generation around the heater rod and the main flow structures in the pool have developed. As the heat is continuously supplied from the heater rod, vapor is generated in the region next to the heater rod. The generated vapor has a lower density than the liquid, so it rises toward the free surface by buoyancy. In addition, because the VOF model is applied, it can be seen that the free surface between the liquid region and the upper gas region is maintained relatively clearly.

Fig. 4 shows the velocity distribution and the velocity vectors at the same time. The upward flow that starts around the heater rod develops toward the free surface, and near the free surface the flow moves sideways, so that a recirculation flow is formed inside the pool. This shows that the phase change and the density difference caused by the heating form a buoyancy-driven flow inside the pool. In this analysis, the maximum velocity was calculated to be about 0.13 m/s. From these results, it was confirmed that the present VOF-based analysis model can calculate the basic thermal-hydraulic behavior of pool boiling, such as the vapor generation by heating, the buoyant rise of the vapor, the circulation flow of the liquid and the free surface. However, the present calculation did not reproduce the initial and heat loss conditions of the experiment. Therefore, the quantitative agreement between the calculation results and the experimental results was not evaluated in this study.

On the other hand, the VOF model required a very small time step size in order to track the gas-liquid interface directly. Even in the present analysis, which used a relatively small mesh of about 105,000 cells, about 50 h of computing time on 32 CPU cores was needed to calculate about 28 s of physical time. In an actual large cooling pool, not only the number of cells but also the required transient time increases greatly. Applying the present analysis model and numerical conditions without any change is therefore expected to cause a considerable computational burden.

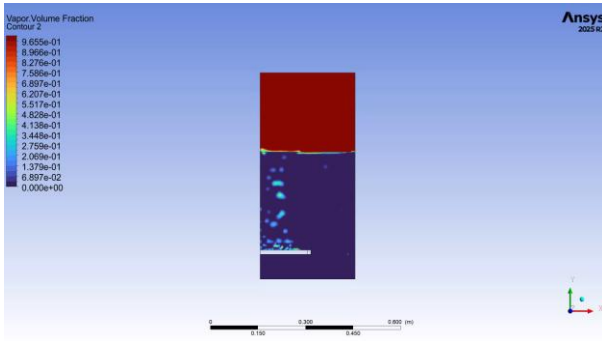


Fig. 3. Vapor volume fraction distribution after development of the two-phase flow.

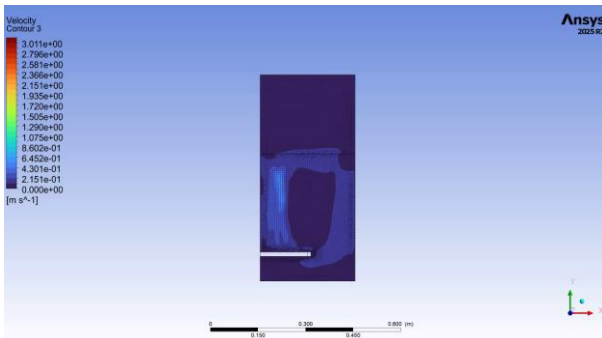


Fig. 4. Velocity magnitude and vectors after development of the two-phase flow.

Therefore, in order to apply the model to a large pool, a systematic sensitivity evaluation and optimization of the mesh system, the time step, the interface treatment method, the turbulence model, the phase change model and the related coefficients should be carried out first, rather than simply enlarging the analysis domain. In addition, the prediction characteristics and the computational cost of the VOF model and the Eulerian multiphase model need to be compared in order to select the analysis method that is more suitable for the long-time analysis of a large pool.

4. Conclusions

In this study, the VOF model was applied to a small-scale pool boiling experiment as a preliminary step toward the multi-dimensional thermal-hydraulic analysis of passive safety system cooling pools using ANSYS FLUENT. Using the present analysis model, it was confirmed that the basic two-phase thermal-hydraulic behavior, such as the vapor generation around the heater rod, the upward flow caused by buoyancy, the circulation flow inside the pool and the free surface, is calculated. However, the present calculation is not a validation analysis that reproduces all the conditions of the experiment, and the present results alone are not enough to judge the quantitative prediction capability of the VOF model or its applicability to an actual large pool. In addition, even in this small-scale analysis, a considerable computing time was required because of the small time

step size needed for the VOF interface tracking. Therefore, additional sensitivity analysis and optimization of the analysis model and the numerical conditions are necessary for the application to an actual large cooling pool.

In the future, the analysis model will be improved so that the experimental conditions are reflected more faithfully, and the effects of the main models and numerical parameters will be evaluated. The prediction characteristics and the computational cost of the VOF model will also be compared with those of other analysis methods, including the Eulerian multiphase model. Based on these studies, the multi-dimensional thermal-hydraulic fields formed in the i-SMR ECT under PAFS-only operation, PCCS-only operation and simultaneous operation of the two systems will finally be analyzed, and the thermal-hydraulic interaction between the two passive safety systems will be evaluated.

ACKNOWLEDGEMENT

This work was supported by the Innovative Small Modular Reactor Development Agency grant funded by the Korea Government(Ministry of Climate, Energy and Environment, MCEE) (No. RS-2024-00404240).

REFERENCES

- [1] S. S. Jeon, Y. J. Park, S. G. Nam, S. J. Yoon, J. S. Kim, S. H. Lee, H. S. Nam, and S. G. Lim, Design characteristics and preliminary performance analysis on passive safety systems of i-SMR, Nuclear Engineering and Technology, Vol. 58, pp. 104574, 2026.
- [2] S. Kim, S. U. Ryu, D. J. Euh, and C. H. Song, Experimental study on the thermal stratification in a pool boiling with a horizontal heat source, Annals of Nuclear Energy, Vol. 106, pp. 235-246, 2017.
- [3] W. H. Lee, A pressure iteration scheme for two-phase flow modeling, in: Multiphase Transport Fundamentals Reactor Safety, Vol. 1, pp. 407-431, 1980.