CFD Prediction for an Asymmetric Behavior of the Thermal Mixing in a Subcooled Water Tank

H. S. Kang,a Y. S. Kim,a H. G. Jun,a Y. J. Youn,a C. H. Song a
a Korea Atomic Energy Research Institute, P. O. Box 105,Yuseong,Taejon, Korea, 305-600, hskang3@kaeri.re.kr

1. Introduction

Experimental research has been performed to evaluate the steam condensation load and the thermal mixing phenomena in the subcooled water tank when the steam is discharged into the tank through the sparger [1]. The experimental results show that the thermal mixing behavior under the discharge of a high steam mass flux is almost symmetric along the circumferential direction in the tank, but the behavior is locally asymmetric due to the buoyancy force, especially around the sparger discharge hole, in case of the discharge of a low steam mass flux. A CFD benchmark calculation for the test results of a low steam mass flux has been performed to develop the methodology of a numerical analysis for a thermal mixing between the steam and subcooled water and to apply it to the APR1400. The previous CFD analysis results for the high steam mass flux with the 2-dimensional axisymmetric grid model showed a good agreement for the temperature distribution in the tank with those of the experiment [2]. However, a 3-dimensional grid model is needed to simulate a thermal mixing due to the buoyancy force, especially developed at the initial stage, in the case of the discharge of a low steam mass flux. The CFD analysis results for 20 seconds show a good agreement for the temperature in the tank with those of the experiment.

2. Thermal Mixing Test

The thermal mixing test has been performed by changing the steam mass flux and the tank water temperature [3]. The experimental facility modified from the B&C Loop consists of a pressurizer, a steam discharge line, a subcooled water tank, a steam sparger, and a steam generator. The bottom hole of a 2.5 cm diameter and the vent area of the LRR(Load Reduction Ring) are blocked in the test. These changes are intended to provide a constant steam flux for a long time in the test. In the tank, 8 thermocouples to measure in detail the temperature of the steam and the entrained water flowing into the steam are installed, and two measurement rigs of 27 thermocouples are installed to obtain the thermal mixing pattern. The second rig is installed to observe the extent of the thermal mixing along the circumferential direction in the tank. From the test results of the temperature variation of TC735 and TC653 which represent one of the comparison results for the TC-Rig 1 with TC-Rig 2, we can see that the thermal mixing in the tank shows an asymmetric pattern especially in the case of a low steam mass flux.

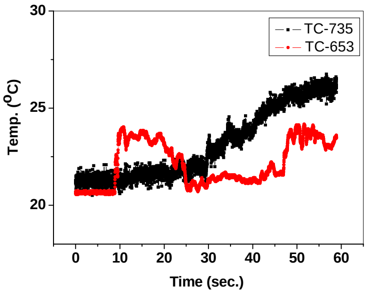


Figure 1. Experimental Results

3. Numerical Analysis

3.1 Critical Flow and Condensation Region Model

In the thermal mixing test, the saturated steam at about 5 bar inside the sparger is initially discharged into the water at 20 ^{o}C at 1 bar in the tank. A choking is likely to happen at the location of the sparger holes during this situation. The discharged steam flows into the water in the tank as a jet flow, and then quickly condenses to water in a short time and length by a direct contact condensation[3]. The numerical modeling for this condensation process is so difficult that we used the steam condensation region model in which the steam is perfectly condensed to water within the steam penetration length. The mass flow rate of the condensed water and the entrained water at the outlet of the condensation region can be calculated from the mass, momentum, and energy conservation law over the condensation region.

3.2 Grid Model and Boundary Conditions

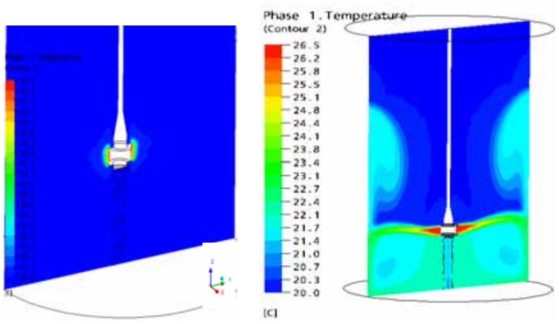
A 3-dimensional multi-block grid simulating the sparger and the subcooled water tank is generated for the CFD calculation. The 3-dimensional model is introduced because the flow pattern in the tank didn’t show a symmetric pattern in the circumferential direction (Fig. 1). The number of meshes is 180,000 cells. The inlet boundary condition, the Dirichlet condition, is set at the end of the steam condensation region with a time dependent velocity and temperature. The pressure outlet boundary conditions, the Neumann condition, are set for the tanks upper region. The outlet conditions for the entrained water are applied to the upper and lower region of the steam condensation region by the negative value of the velocity with the inlet condition in the CFX4.4.

3.3 Flow Field Models and Governing Equation

Thermal mixing phenomenon in the subcooled water tank is treated as an incompressible flow, a free surface flow of air between the water, a turbulent flow, and a buoyancy flow. Therefore, the governing equations used in this study are the Navier-Stokes and energy equations with a homogenous multi-fluid model[4]. The turbulent flow is modeled by the standard k-ε turbulent model, and the buoyancy is modeled by the Boussinesq approximation[4].

3.4 Discussion on the CFX Results

Fig. 2, (a) shows that only the temperature distribution was increased by the buoyancy force just above the sparger discharge head at 2 seconds when the very low mass flux of the hot water, from 40 ^{o}C to 60 ^{o}C, is discharged. Fig. 2, (b) shows the temperature distribution at 14 seconds governed by the inertia flow of the jet. In these figures the temperature above the sparger discharge head is decreased again to the initial value of 20 ^{o}C, which means that thermal mixing between the hot water and the entrained water actively happens. From the figures, we can expect that the temperature around the sparger discharger head increases little by little and may cause on the adverse effect to the stable steam condensation. And also, we can see that the thermal mixing pattern in the CFD calculation shows almost a symmetric pattern based on the temperature distribution. This may be because the buoyancy force developed at the initial stage is weak while the inertial force of the jet is very strong in the CFD calculation. And it is concluded that the number of the meshes in the grid system for the 3-diemensioanl CFD calculation is too small to resolve the complicated plume behavior driven by the buoyancy force. Therefore, all of them may not be able to simulate the very complicated movement of the hot water plume.



(a) 2 sec. (b) 14 sec.

Figure 2. Temperature distribution of the CFD results

4. Conclusion and Future Work

The comparison of the CFX results with the test data showed a good agreement as a whole, but some small temperature differences between the CFX results and the test data were shown at some locations of the thermocouples in the subcooled water tank. This temperature difference may have arisen from the fact that the temperature and the velocity of the calculated condensed water by the condensation region model are higher than the real value. Another reason may be the weakness of the condensation region model by using the area average concept. The assumption of a constant temperature of the condensed water at the outlet of the condensation region causes a reduction of the buoyancy force around the sparger discharge head. This also decreases the distance and the strength of the hot water plume behavior. Therefore, if we modify the grid model and the temperature profile at the inlet boundary condition, we can obtain the calculation results which are more similar to the test results. However, we can expect that the CFX4.4 with the steam condensation region model can surely simulate the thermal mixing behavior in the subcooled water tank with the minor limit. We can anticipate that the numerical model for the thermal mixing taking place for a long time and a wide range of the steam mass flux in the IRWST of the APR1400 can be developed by the use of this CFD analysis methodology.

ACKNOWLEDGEMENTS

This work was financially supported for the nuclear R&D program from the Ministry of Science and Technology of Korea. The authors are sincerely grateful for the financial support.

REFERENCES

[1] C. H. Song, W. P, Baek, M. K. Chung, and J. K. Park, Multi-Dimensional Thermal-Hydraulic Phenomena in Advanced Nuclear Reactor System: Current Status and Perspectives of the R&D Program at KAERI, *Proceeding of NURETH-10*, Seoul, Korea, 2003.

[2] H. S. Kang et al., “CFD Analysis for the Thermal Mixing Phenomena in the Subcooled Water Tank”, *Proc. of the NTHAS4 Conference*, Sapporo, Japan 2004.

[3] Y. S. Kim et al., “Experimental Study of Thermal Mixing of Steam Jet Condensation Through an I-Sparger in a Quench Tank”, *Proceedings of ’04 KNS Autumn Conference*, Yongpyoung, Korea, 2004

[4] ANSYS Inc, “CFX4.4 Manual”, 2004