Sensitivity Analysis for the CFD Calculation of Thermal Mixing Test

H. S. Kang,a Y. S. Kim,a H. G. Jun,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

The experimental and CFD research for an unstable steam condensation in a DCC (Direct Contact Condensation) which may happen in the IRWST (In-containment Refueling Water Storage Tank) of APR1400 were performed [1,2]. In the CFD analysis, the numerical methodology which can predict a local and global thermal mixing in the tank was developed based on the comparison of the CFD results with the test data. The steam condensation region model was developed based on the water temperature data around the steam jet to simulate the DCC [2,4]. The comparison of the CFD results with the test data of a transient discharge of the high steam mass flux showed a good agreement as a whole, but some small temperature differences between the CFD results and the test data were shown at some locations [2,3]. These differences may be caused from a insufficient mesh distribution generated in the grid model to resolve the flow field of the jet flow at the analogous or an improper selection of a numerical model. Therefore, we investigated the effect of the grid and the numerical models on the sensitivity in the CFD calculations.

2. Thermal Mixing Test [1]

The thermal mixing test was performed by changing the steam mass flux and the tank water temperature in the transient and the quasi steady states. Eight thermocouples to measure the temperature of the steam and the entrained water flowing into the steam were installed in the tank, and two measurement rigs of 27 thermocouples were installed to obtain the thermal mixing pattern. A second rig was installed to observe the extent of the thermal mixing along the circumferential direction in the tank. In the case of the high steam mass flux, the thermal mixing phenomena in the tank showed a nearly axis symmetric pattern.

3. CFD Analysis

3.1 Flow Field Models and Boundary Conditions

In the test, the discharged steam from a sparger flowed into the water as a jet flow, and then it was quickly condensed to water by the DCC [2]. The steam condensation region model for the DCC phenomenon was used [2]. Thermal mixing phenomenon in the water tank was treated as an incompressible flow, a free surface flow of air between the water, a turbulent flow, and a buoyancy flow. The governing equations used in this study were the Navier-Stokes and the energy equations with a homogenous multi-fluid model [2,3]. Turbulent flow was modeled by the standard k-ε turbulent model, and the buoyancy was modeled by the Boussinesq approximation. The inlet boundary condition was set at the end of the steam condensation region with a time dependent velocity and temperature. The pressure outlet boundary conditions were set for the tanks upper region which was extended upward by 0.5m to move the fully developed condition imposed by applying the pressure outlet condition into the downstream of the flow field. The outlet conditions for the entrained water were 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.

3.2 Grid and Numerical Models for Sensitivity Analysis

A multi-grid with an axis symmetric condition simulating the sparger and the subcooled water tank for the CFD calculation was generated (Fig. 1). The axis symmetric model was introduced because the flow pattern in the tank was estimated to vary a little in the circumferential direction and it could reduce the computational time. The meshes were more densely distributed around the condensation region and the initial air/water interface region than the other regions. The sensitivity calculation of the mesh distribution and the numerical method were performed (Table 1). Three cases only used different mesh distributions in the grid model by using the same upwind 1st method for a convection term discretization. In case 1, a total of 9,588 cells were generated, and the first grid from the right wall was located at the position of 100~300 of y+. In the second case for the sensitivity study, the mesh distribution was rearranged based on the comparison results between the CFD data of case 1 and the test data. A total of 23,835 cells and 12~50 of y+ were generated to predict the temperature close to the test data even though the computation time took longer than that of case 1. Especially, more meshes were distributed at around the jet flow and a region near to the wall. Case 3 grid model had 31,020 cells and 12~50 of y+. The meshes of case 3 were more densely located at the transition region in the upper region of the tank than those of the case 2. In case 4, the same mesh distribution of case 1 was used whereas the numerical model of the convection term discretization was changed to a QUICK scheme. Total cell number (Table 1) did not agree with the value of the horizontal times for the vertical cells because the meshes inside the sparger were generated.

Table 1. Sensitivity Calculation Conditions

	Cell No.	Horizontal × Vertical	Convection Term Discretization
Case 1	9,588	63×160	Upwind 1^{st}
Case 2	23,835	63×160	Upwind 1^{st}
Case 3	31,020	63×160	Upwind 1^{st}
Case 4	9,588	63×160	Quick

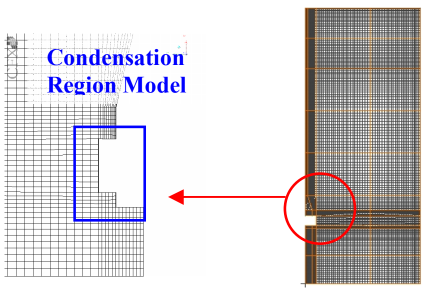

Fig. 1. Grid model and the mesh distribution

3.4 Discussion on the CFX Results

The temperature comparison test data with the CFX results for 30 seconds at 3 thermocouple locations depending on the four cases were shown in Fig. 2. The comparison of the CFD results with the test data showed a good agreement within 7~8% value [2,3]. This difference may have arisen from the fact that the temperature and the velocity of the calculated condensed water by the condensation region model were higher than the real value. Another reason may be a limitation of the condensation region model by using the area average concept. The sensitivity calculation results of CFD were very similar to each other at the region (TC706) between the sparger and the tank wall irregardless of the cases. However, the CFD sensitivity results showed a small temperature distribution difference at the upper (TC729) and lower region (TC728) where the condensed water jet arrived after colliding with the tank wall. Especially for the high upper region, case 4 using the quick scheme predicted the test data better than the other cases using the upwind scheme. And also, the phenomenon of different temperature distribution depending on the number of mesh cells was due to using the upwind 1^{st} scheme for the convection term. If the cell is not aligned to the flow field when using the upwind scheme, the results of the flow field are dependent on the mesh distribution.

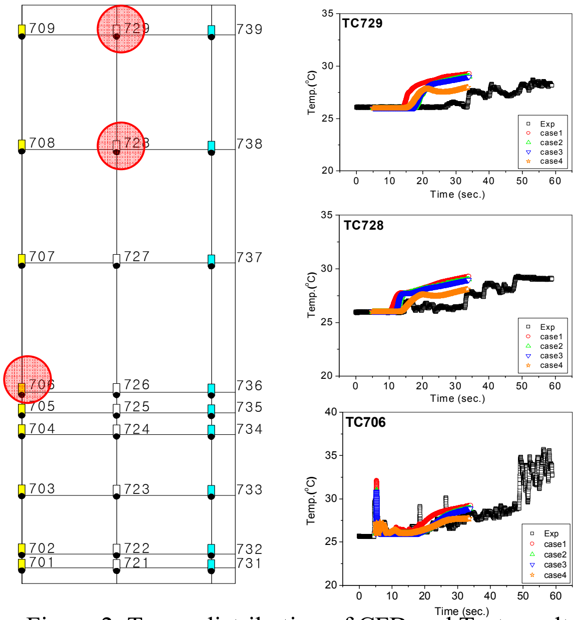

Figure 2. Temp. distribution of CFD and Test results

4. Conclusion

According to the sensitivity analysis, the numerical methods and grid meshes distribution for a thermal mixing calculation should be carefully selected. However, the commercial CFD code of CFX4.4 together with the condensation region model can simulate the thermal mixing behavior reasonably well when a sufficient number of mesh distributions and a proper numerical method are adopted.

ACKNOWLEDGEMENTS

This work was financially supported for the nuclear R&D program from the Ministry of Science and Technology of Korea.

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., “A CFD Analysis of Characteristics of the Thermal Mixing Under the Transient of the Steam Discharge in a Subcooled Water Tank”, Technical Report, KAERI/TR-3008/2005, KAERI, 2005.

[3] H. S. Kang et al., “A CFD Analysis for Thermal Mixing in a Subcooled Water Tank under Transient Steam Discharge Conditions”, J. of Computational Fluids Engineering, to be published, 2006.