Transactions of the Korean Nuclear Society Autumn Meeting
Busan, Korea, October 27-28, 2005

Study on the Grid Sensitivity for the CFD Analysis of the Thermal Mixing Test

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

A CFD benchmark calculation for the experiment of the thermal mixing phenomena in the subcooled water tank when the steam is discharged into the tank through the sparger has been performed to develop the methodology of a numerical analysis for a thermal mixing and to apply it to the APR1400[1,2]. The comparison of the CFD results with the test data, 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 of the upper region beside the sparger in the tank. This difference may have arisen from the fact that a sufficient mesh distribution to resolve the flow field of the jet flow at the analogous elevation to the sparger discharge head isn't generated in the grid model. The unique difference between CFD results and test data at the upper region is that the start time when the fluid temperature increases. Therefore, we decided to investigate the effect of the grid sensitivity in the CFD calculations by changing the mesh distribution in the grid model.

2. Thermal Mixing Test[3]

The thermal mixing test has been performed by changing the steam mass flux and the tank water temperature in the transient and the quasi steady state. 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. In the case of the high steam mass flux, the thermal mixing phenomena in the tank shows an almost axis symmetric pattern.

3. CFD Analysis

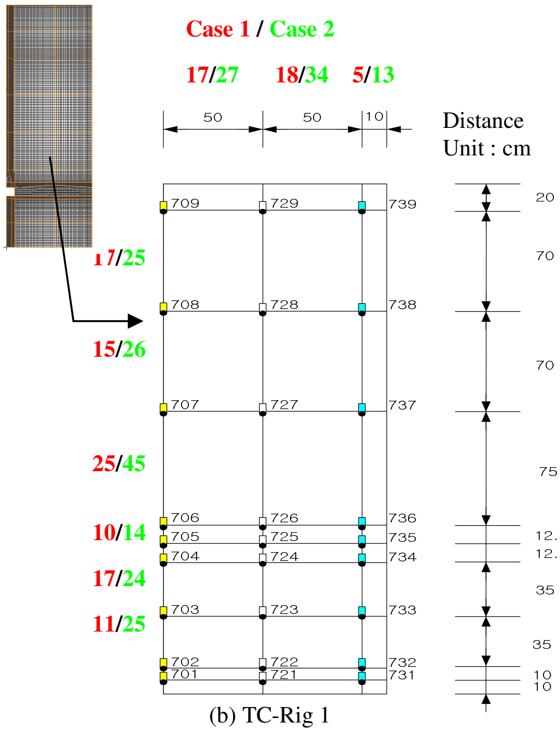
3.1 Flow Field Models and Boundary Conditions

In the test, the discharged steam from the sparger flows into the water 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[2]. Thermal mixing phenomenon in the 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]. 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.

3.2 Grid Models for the Sensitivity Study

A multi-grid with an axis symmetric condition simulating the sparger and the subcooled water tank for the CFD calculation is generated as in Fig. 1, (a). The axis symmetric model is introduced because the flow pattern in the tank was estimated as varying a little in the circumferential direction. As the first case for the sensitivity study, 9,588 cells are generated in the grid model by considering the computation time of the transient calculation for about 30 seconds. The meshes are more densely distributed around the condensation region and the initial air/water interface region than the other regions to accommodate the expected high velocity and temperature gradients. The number of cells between the locations of the thermocouples in the TC Rig 1 is represented in Fig. 1, (b). The first grid from the right wall is located at the position of 100 order of y+. In the second case for the sensitivity study, the mesh distribution is rearranged based on the comparison results between the CFD data of case 1 and the test data. 23,835 cells are generated to predict the temperature close to the test data even though the computation time has to be longer than that of case 1. Especially, more meshes are distributed at around the jet flow and a region near to the wall. The increased cell number is also shown in Fig. 1, (b). The y+ value around the right wall is decreased by 10 orders of a magnitude when compared to case 1.

(a) Grid model



(b) TC-Rig 1

Fig. 1. Grid model and the mesh distribution

3.4 Discussion on the CFX Results

Fig. 2 shows the temperature distribution at the upper region of the jet flow(TC706), the upper region near to the right wall(TC738) and the upper region beside the sparger(TC709, TC729) as time passes. The temperature comparison results between the CFD data and the test data at TC706 show that the two CFD results cases predict well the test data, but the results of case 2 are close to the test data than that of case 1. In case 2, the value of a peaking at about 5 seconds and the trend of the temperature increase are very similar to the test data. In the comparison results at TC738, both the CFD results also predict well the temperature trend of the test data, but the CFD results couldn't simulate the temperature fluctuation phenomena. This may be because we used the axis symmetric condition so that temperature of the condensed water in the circumferential direction is constant. Therefore, the CFD results can not simulate the local thermal mixing inside the upwarding flow along the right wall. The comparison results at TC709 and TC729 show that the start time of the temperature increase in the CFD results is faster than that of the test data. This means that the condensed water in the CFD analysis arrives at this region more quickly than that of the test. Especially, the temperature in case 1 starts to increase quickly at about 13 seconds. From the comparison work of the temperature contour around the jet flow near to the right wall, we can see that the numerical diffusion of the temperature and the velocity field is developed at this region. Therefore, the velocity and the temperature of the upward flow along the right wall becomes higher than that of the test data.

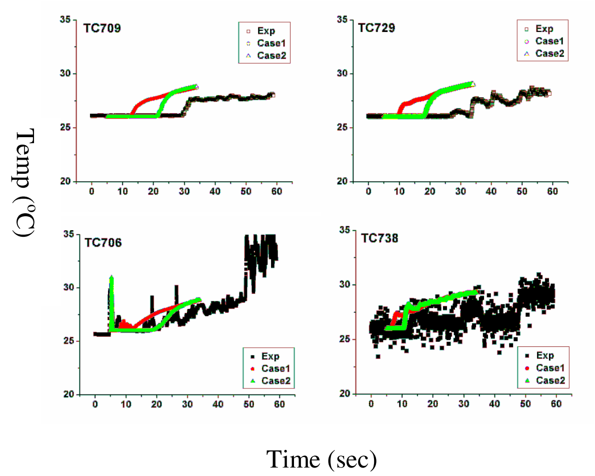


Fig. 2. Temp. distribution of the CFD and Test results

4. Conclusion

The grid sensitivity study of the CFD benchmark calculation for the thermal mixing test shows that a proper mesh distribution should be developed to predict well the test data. A sufficient mesh distribution in the turbulent jet boundary layer and a 10 orders of a magnitude y+ value is necessary for the more accurate simulation. And also, the comparison of the CFD results with test data can provide the exact criteria for the CFD simulation.

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