Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 10-11, 2007

CFD Analysis of a Thermal Mixing Test by CFX-10 Using a Parallel Computation Technique

H. S. Kang,[a] C. K. Park,[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

A CFD analysis methodology has been developed for the DCC (Direct Contact Condensation) of steam jet in a subcooled water pool based on the thermal mixing test results [1,2,3]. This methodology can be applied into the safety assessment of APR1400 IRWST (In-containment Refueling Water Storage Tank) pool. However, the assessment of the IRWST pool requires a long transient analysis of about 8000 seconds. In order to meet this requirement, a parallel computation capability should be included in the CFD methodology. Therefore, a sensitivity CFD study by CFX-10.0 using parallel computation hardware was performed, and also its result was compared with previous CFD results to establish the advantage of a parallel computation technique.

2. Thermal Mixing Test [1]

The thermal mixing test was performed at KAERI 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. In the case of the high steam mass flux, the thermal mixing phenomena in the tank showed a nearly axi-symmetric pattern.

3. CFD Analysis

3.1 Flow Field Models and Boundary Conditions

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 between the air and the water, a turbulent flow, and a buoyancy flow. The governing equations used in this study are the Navier-Stokes and the energy equations with a homogenous multi-fluid model [2,3,4]. Turbulent flow was modeled by the standard k-ε turbulent model, and a 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. The outlet conditions for the entrained water were applied to the upper and lower regions of the steam condensation region by a negative value of the velocity with the inlet condition. The same boundary conditions including the time step were applied to all the sensitivity cases.

3.2 Grid and Numerical Models for Sensitivity Analysis

A multi-grid with an axi-symmetric condition for simulating the sparger and the subcooled water tank for the CFD calculation was generated (Fig. 1). The sensitivity calculation of the mesh distribution, the numerical method and the CFD solver were performed (Table 1). The previous CFD results showed that the number of cells in the grid model and a convection term discretization method were very important for the temperature distribution results [2,3]. Three grid models (Table 1) were used for the CFD calculations. In the first grid model of 9,588 cells, the first grid from the right wall was located at the position of 100~300 of y+. As for the second grid model of 23,835 cells, 12~50 of y+ were generated to predict the temperature close to the test data. The third grid model had 31,020 cells and 12~50 of y+. In the parallel CFD calculation of case 5 and case 6, the coupled solver of CFX-10.0 was used by varying the convection term discretization method and the grid model. The High Resolution scheme is the default option in CFX-10.0 [4].

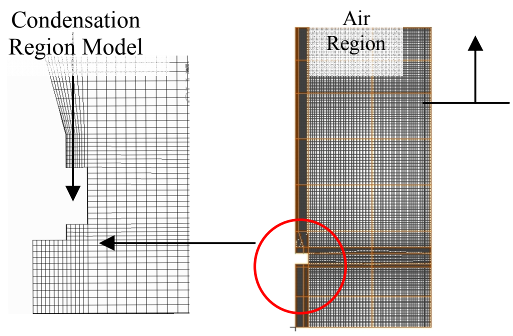

Fig. 1 Grid model and the mesh distribution

Table 1 CFD Sensitivity Calculation Condition

	Cell No.	Solver	Convection Term
Case 1	9,588	CFX-4.4	Upwind 1st
Case 2	23,835	CFX-4.4	Upwind 1st
Case 3	31,020	CFX-4.4	Upwind 1st
Case 4	9,588	CFX-4.4	QUICK
Case 5	9,588	CFX-10.0	High Resolution
Case 6	23,835	CFX-10.0	Upwind 1st

3.3 Discussion on the CFX Results

A comparison of the temperature data with the CFX results for 30 seconds at 2 thermocouple locations is shown in Fig. 2. This shows a good agreement, in general, to within in 7~8% value [2,3]. This difference (Fig. 2) 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.

The sensitivity calculation results were very similar to each other at a region of which the height is analogous to the sparger discharge hole (TC704~TC706) regardless of the cases. However, the CFD sensitivity results showed a small temperature distribution difference at a upper reason (TC729, TC728) where the condensed water jet arrived after colliding with the tank wall. Case 4 and case 6 predicted the test data better than the other cases for this upper region. Especially for TC729, Case 6 using the CFX-10.0 predicted the test data closer to test data than the other cases. This means that CFX-10.0 can be used for the CFD analysis of APR1400 IRWST thermal mixing phenomenon.

A comparison of the computation time as well as the hardware and software environments for all the cases is shown in Table 2. The CPU time of each case is normalized by that of case 1. As for the comparison of case 1 and case 5 using the first grid model of 9,588 cells, the parallel computation effect is not high. However, the parallel computation using 4 CPUs greatly reduces the computation time when compared with case 2 and case 6 which use the second grid model of 23,835 cells.

Table 2 Comparison of the CPU Time

	Hardware (Operating System)	CPU Time
Case 1	Pentium IV 3.0 GHz / 1CPU (Windows 2000)	1.00
Case 2	Pentium IV 3.0 GHz / 1CPU (Windows 2000)	4.13
Case 3	Pentium IV 3.0 GHz / 1CPU (Windows 2000)	5.46
Case 4	Pentium IV 3.4 GHz / 1CPU (Linux)	1.04
Case 5	Pentium IV 2.4 GHz / 2CPUs (Linux)	0.97
Case 6	Pentium IV 3.0 GHz / 4CPUs (Linux)	1.23

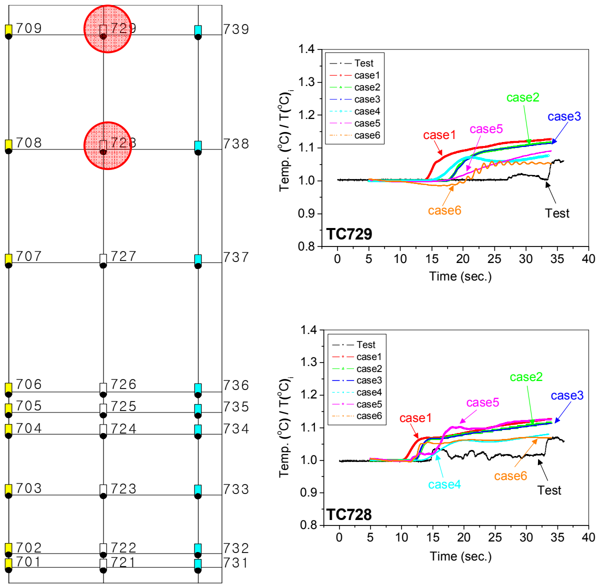

Fig. 2 Temperature distribution of CFD and Test results

4. Conclusions and Further Research

From the results of the CFD sensitivity analysis, it was found that CFX-10.0 using a coupled algorithm predicted the test data as well as CFX-4.4 using the SIMPLE algorithm. And the parallel computation capability of CFX-10.0 may be very useful for a transient calculation. Therefore, it is believed that these sensitivity calculation results may assist the establishment of a strategy for the CFD analysis of APR1400 IRWST Pool.

ACKNOWLEDGEMENTS

This work was performed under the Long-Term Nuclear R&D Program sponsored by the Ministry of Science and Technology of Korea.

REFERENCES

[1] Y.S. Kim et al., "Steam Condensation Induced Thermal Mixing Experiment Using B&C Facility", Technical Report, KAERI/TR-2933/2005, KAERI, 2005.

[2] 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*, **vol. 11**, **No. 2**, 2006.

[3] H.S. Kang et al., "CFD Analysis of a Thermal Mixing in a Subcooled Water Pool", Proc. of NTHAS5, Jeju, Korea, 2006.

[4] Ansys, Inc., "CFX-10.0 Manual", 2006.