Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 29-30, 2009

3D Flow Field in Hold-up Volume Tank of Advanced Pressurized Reactor 1400

T. W. Kim*, Y. S. Bang, B. G. Huh, S. W. Woo
**Department of Reactor & Safety Evaluation, Korea Institute of Nuclear Safety*
19 Guseong-dong, Yuseong-gu, Deajeon, 305-338, Republic of Korea
**Corresponding author: s142ktw@kins.re.kr*

1. Introduction

Four phases after a Loss of Coolant Accident (LOCA) are progressed step by step: blow-down, refill, reflood, and long-term cooling phase. When the long-term cooling phase as the fourth step is started following the LOCA, debris occurred from Fiberglass insulation, stainless steel jack, and Epoxy coating etc. may block the sump screen and disturb long-term cooling. In special, the effect of debris can be more intensified in the Advanced Pressurized Reactor 1400 (APR 1400), since the effect of debris will be happened in the blow-down phase as the first step. In other words, the sump screen in the Emergency Core Cooling System (ECCS) can be affected by the debris of the blow-down phase [1]. Therefore, predictions of flow field and debris behavior are important problems. In addition, flow path of debris of APR 1400 includes break location, containment floor, HVT, spillway, In-containment Refueling Water Storage Tank (RWST), and then, role of HVT is an intermediate collector of coolant after LOCA.

In this work, flow field in Hold-up Volume Tank (HVT) is analyzed and effect of debris is estimated. Full transient three-dimensional flow field is calculated by commercial Computational Fluid Dynamics (CFD) code

2. Computational Analysis

In order to analyze a flow field in the HVT after the LOCA, ANSYS CFX is selected because of a sense of reliability and flexibility. Three dimensional meshes are generated by ANSYS Design-modeler. Figure 1 shows three dimensional configuration, grid shape, and information on inlet and outlet.

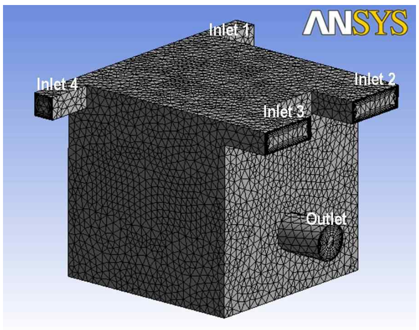


Fig 1. Three dimensional configuration of HVT designed by ANSYS design-modeler.

The inlet and the outlet described in Fig. 1 mean trench and spillway, respectively. For the simplification of simulation, two spillways are modeled by one.

2.1 Establishment of Environment

In order to install environment for simulating in the CFX, optimized conditions should be determined preferentially such as grid shape, a number of grids and time step etc. They are decided through case studies. The grid is a very important parameter for validity of results in the CFD. For the optimized grid, the total grid number was increased from 5,000 to 600,000. About 200,000 grids are determined as the optimal grid number through checking on result convergence. In general, flowing vicinity wall is very important so that inflation at the wall of inlet and outlet is applied [2].

2.2 Boundary Condition

Hydraulic features should be defined for simulating the problem, air and water are specified as working fluids, and then, "transient calculating" is adopted. The boundary conditions are based on the previous study by Bang [3, 4].

Mass flow rates which are going into the HVT are described. Figure 2 shows mass flow rate at inlet 1, 2, 3, and 4 respectively. Mass flow rate for initial conditions is determined through previous work of whole over system in LOCA of APR 1400. Atmosphere pressure is applied at inlet and outlet. Because this work focuses on just hydraulic, thermal conditions are neglected [3]. Total operating time is 100s, and time step which means time gap between motions is 0.01s. Root Mean Square (RMS) is fixed 10^{-6}. This value means that calculating value between steps has difference 10^{-6}. Namely, RMS 10^{-6} verifies a sense of reliability of computing results

2.3 Modeling

Homogeneous model and free surface model are conducted to operate the multi-phase. In this case, surface tension is established as 0.072N/m. And then, fluid buoyancy model is adopted so that buoyancy force is available. Buoyancy force based on $9.8 m/s^2$ is changed with density difference and an initial reference of buoyancy force is based on the air density as $1.185 kg/m^3$. Morphology of air and water is assumed continuous fluid. The standard $\kappa - \varepsilon$ model which is usually used is suggested for the turbulent model. In the calculating, the high resolution scheme and the second order backward Euler equations are adopted to advection and transient term, respectively.

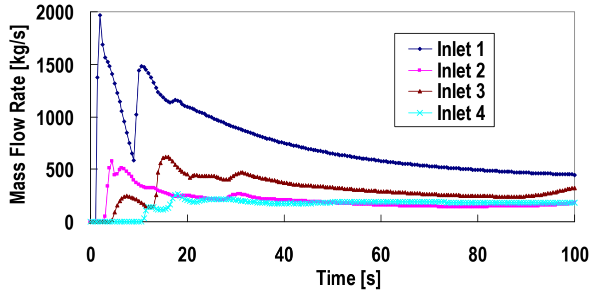


Fig. 2. Mass flow rate variation at inlet 1, 2, 3, and 4 when time goes from 0 to 100s.

2.4 Result and Discussion

Mass flow rate of initial phase (0~20s) in Fig. 2 shows variation dramatically, and then, it (20~100s) goes linear. However, tangential lines vicinity 100s goes up (inlet 1, 2, and 4) or down (inlet 3). It means that flowing is not steady state in a view of the hydraulic-dynamic.

Total mass flow rate is apt to follow the net mass flow rate because mass flow rate at outlet is not much to affect total and net. As growing of mass flow rate at outlet described in Fig. 3, however, net mass flow rate looks like steady state. Unfortunately, this point can be inflection point due to mass flow rate between total and outlet. Figure 3 make this view clear. This is reason that tangential line near 100s indicates going up rapidly. Figure 4 shows quantity of collected water and vector of velocity at 50s and 100s. Hydraulic behavior of water is wild and disorderly predicted in Fig 4 (a), and level of water at outlet fluctuates and quantity of water at outlet increases described in Fig 4 (b).

In the point of safety, consequently, hydraulic character after 100s will be important so that should be checked.

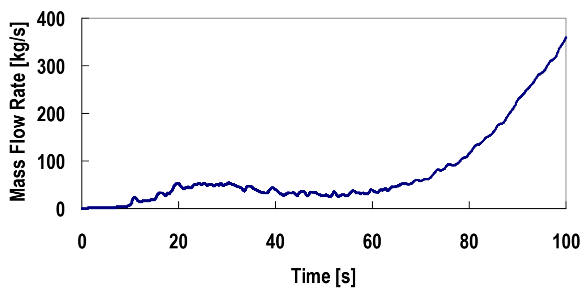
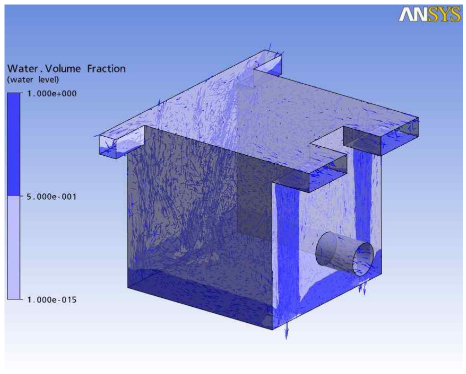
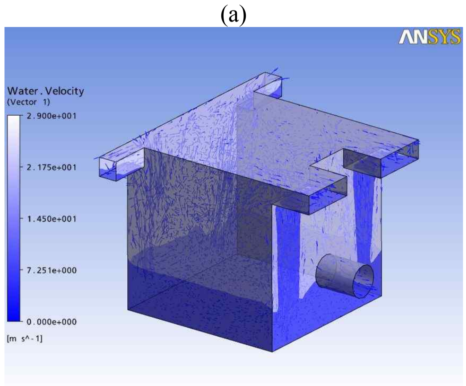


Fig. 3. Mass flow rate variation at outlet when time is going from 0 to 100s.



(a)



(b)

Fig. 4. CFX results: accumulated water capacity and vector velocity at 50s (a) and 100s (b).

3. Future Work

The precede study aim is to predict behavior of the debris and estimate effect on debris transported to IRWST. The currently analysis results on hydraulics can be used for the debris transport analysis is in HVT. For this aspect, particle tracking will be calculated as future work, so that debris's behavior will be evaluated. And then, the effect on debris against moving of coolant is estimated [5].

REFERENCES

[1] Assessment on Recirculation Sump Performance for Shin-Kori 3&4 Design, FNC Technology Co. LTD., KINS/HR-818, 2007.

[2] CFX 11.0 User Manual, 2007.

[3] Y. S. Bang, G. S. Lee, B. G. Huh, D. Y. Oh, and S. W. Woo, "Transient Flow Field in Containment Floor Following a LOCA of APR-1400," Transactions of the Korean Nuclear Society Autumn Meeting, Pyeong-chang, Korea, October 30-31, 2008.

[4] Y. S. Bang, G. S. Lee, B. G. Huh, D. Y. Oh, and S. W. Woo, "Transient Flow Field on Containment Floor Predicted by Two-dimensional Shallow Water Equation Solver," Proceedings of the 17th International Conference on Nuclear Engineering ICONE17, Brussel, Belgium, July 12-16, 2009.

[5] Y. S. Bang, G. S. Lee, B. G. Huh, D. Y. Oh, and S. W. Woo, "A Particle Tracking Model to Predict the Debris Transport on Containment Floor," Transactions of the Korean Nuclear Society Autumn Meeting, Jeju, Korea, May 22, 2009.