Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 24-25, 2013

Effect of Water-Air Clearing on Thermal Mixing in IRWST Using Three-Dimensional CFD Analysis

Jeong Hee Ha[(a)], Doo Yong Lee[(a)*], Soon Joon Hong[(a)], Jae Sik Jeong[(b)], Man Heung Park[(b)], Young Tae Moon[(b)]
[(a)] *FNC Technology, Co. Ltd, SNU 135-308, Kwanakro 1, Kwanak-gu, Seoul, 151-742, S. Korea*
[(b)] *KEPCO E&C Co., 2354 Yonggudaero, Giheung-gu, Yongin-si, Gyeonggi-do, 446-713, S. Korea*
*Corresponding author: *dylee@fnctech.com*

1. Introduction

IRWST (in-containment refueling water storage tank) is one of the advanced design features of APR1400 (Advanced Power Reactor – 1400). Connected to the Safety Depressurization and Vent System (SDVS), IRWST is designed to absorb the high energy flow from Pilot Operated Safety and Relief Valves (POSRVs) to protect the over-pressurization of the Reactor Coolant System. Due to thermal hydraulic loads induced by discharged fluids, it is crucial to understand the phenomena occur in the IRWST and thermal mixing is one of them. It has been known that the unstable steam condensation which results in oscillations and acts as the loads on the IRWST wall and structures can occur if there is a large local temperature difference. Thus, there is a regulation related to IRWST temperature distribution (difference) to be satisfied.

To understand the phenomena and design the IRWST with sufficient safety margin, many experimental and numerical researches have been performed. The results of these researches showed that the CFD analysis predicts well the temperature distribution in the pool globally and can be a proper evaluation methodology to analyze the complex thermal mixing phenomena in the IRWST with a sufficiently fine mesh distribution and proper numerical models. But the previous studies have tended to focus the phenomenological study of steam condensation. Actually, when the POSRV is opened, the water and the air in the pipes of the SDVS are discharged firstly into the IRWST (Water/Air Clearing), which forms the fluid velocity prior to steam discharge. However, it has been observed that sole CFD analysis could not simulate the thermal mixing phenomena with those clearing.

In this study, the framework for numerical analysis of CFD has been proposed to properly simulate the thermal mixing phenomena with the water and air clearing. The boundary conditions such as the mass and energy release from the POSRV have been obtained from the one-dimensional system code, RELAP5. The boundary conditions are divided into three phases which are the water clearing, the air clearing and the steam condensation. The thermal mixing in the IRWST has been simulated using the three-dimensional CFD with three different phases of boundary condition. The computational results with the proposed approach are compared to the case without the water and air clearing.

2. Generation of Boundary Conditions

To obtain the boundary conditions for the water/air clearing, a simulation using the RELAP5 code has been conducted for an Inadvertent Operation of POSRV (IOPOSRV) accident. The nodalization of this simulation is based on the SDVS of the APR1400, especially Train B, one of two trains, which is supposed to make fluids discharge through 6 spargers.

In the calculation, the POSRV was opened at 10 seconds and the calculation was terminated at 20 seconds to simulate the water/air clearing. The pressurizer was set at 2250 psia and it is assumed that the pressure drop by discharging steam was not considered because the water/air clearing were done within 0.7 seconds after opening the POSRV. Fig.1 shows the mass flow rate of the fluids discharged into the IRWST and the mass fraction of non-condensable gas. As shown in Fig.1, the two different phases are determined as: 1) water clearing phase: 10 ~ 10.27 seconds, 2) air clearing phase: 10.27 ~ 10.63 seconds.

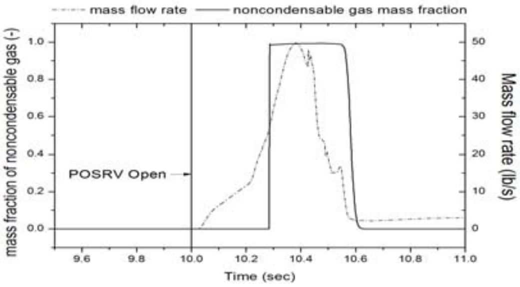

Fig.1. Mass flow rate and mass fraction of non-condensable gas through sparger hole

3. Thermal mixing in IRWST

The mass and energy release estimated by RELAP5 are used as boundary conditions for the CFD simulation of thermal mixing in the IRWST.

3.1. Geometry and mesh

For the CFD analyses of all three phases, the three-dimensional model of the IRWST and 12 spargers which are located at 90 and 180 degrees has been made as a default domain and hexahedral meshes have been generated using ANSYS CFX-11.0 with fine mesh distribution near the spargers. Table 1 and Fig.2 show the geometry dimension of the selected domain and the mesh distribution around the spargers, respectively.

Table 1. Geometry dimension and mesh

Domain Dimension[m]	External Diameter	Internal Diameter	Elevation
	43.79	32.31	3.658
Mesh Distribution	Nodes	Elements	
	1,949,477	1,816,191	

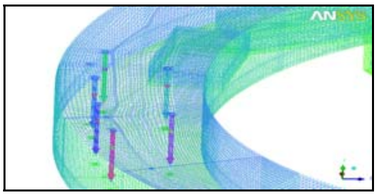

Fig.2. Mesh distribution around spargers

3.2. CFD analysis for water clearing phase

The CFD analysis for the water clearing phase is conducted by using the water discharged through the spargers from the RELAP5 calculation as inlet boundary conditions in proportion to the each area of discharge region such as, a LRR, Side and Bottom hole. The number of hole at the LRR, Side and Bottom were lumped with the equivalent area to model the inlet boundarys. A surface of the water in the IRWST was set as an outlet with an average static pressure option for the mass and momentum. And the initial pool temperature was set as 31℃. A single phase transient simulation for this model was performed during 0.27 seconds with 0.001-second time step. The turbulent flow was modeled by a Shear Stress Transport (SST) model and the buoyancy was modeled.[1] The three-dimensional flow distribution during the water clearing phase was calculated. And the flow distribution at the end of this phase was reflected as the initial condition on the air clearing phase.

3.3. CFD analysis for air clearing phase

The air clearing is simulated by the air discharge through the spargers from the RELAP5 calculation and the velocity distribution in the IRWST as the initial condition. The two phase CFD models were set with a dispersed phase for the air and a continuous phase for the water. The zero equation and SST model were used as the dispersed phase and continuous phase turbulent model, respectively. Ishii-Zuber drag force in the phase interface and Sato enhanced eddy viscosity model were applied.[1] The transient simulation was performed during 0.36 seconds with 0.0001-second time step. The flow distribution at the end of the air clearing phase was used for the initial condition on the steam condensation phase.

3.4. CFD analysis for steam condensation phase

The steam condensation phase was modeled with a Steam Condensation Region Model (SCRM) which is based on the steam condensation regions estimated through the experimental observation.[2,3] The mass and energy transfer by the steam condensation was modeled with the subdomain model in the single phase framework.[1] The flow distribution of the air clearing phase was applied as the initial condition. The SST and buoyancy model was applied as well. The transient simulation was conducted during 300 seconds with 0.5-second time step. Fig. 3 shows the comparison of temperature rise between the simulation without water-air clearing and that with water-air clearing near the sparger and at the bulk away from the sparger. It is seen that the temperature rise rate near the sparger was delayed and that at the bulk away from the sparger was faster since the effect of thermal mixing might be enhanced with the reflection of the water-air clearing.

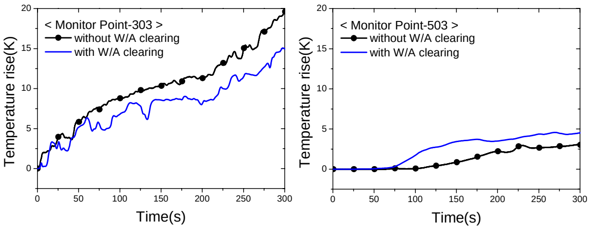

Fig.3. History of temperature rise

4. Conclusion

In this paper, the water-air clearing effects on thermal mixing in the IRWST were investigated with the CFD simulation. The boundary conditions for each discharge phase were obtained from the RELAP5 simulation. The flow distribution in the IRWST for the water clearing phase was reflected as the initial condition for the air clearing simulation. The flow distribution for the air clearing phase was applied as the initial condition for the steam condensation phase. The result of the steam condensation phase with the SCRM showed that the thermal mixing in the IRWST might be enhanced by the mixing effects of the water-air clearing before the steam discharge.

REFERENCES

[1] ANSYS Inc., 2007, CFX-11 Manual.

[2] Moon, Y. T., 2009, “CFD Simulation of Steam Thermal Mixing in Subcooled Water Pool,” Ph.D. thesis, Seoul National University, Korea.

[3] Moon, Y. T., Lee, H.D., Park, G.C., 2009, “CFD Simulation of Steam Jet-Induced Thermal Mixing in Subcooled Water Pool,” Nuclear Engineering and Design, 239, 2849-2863.