Numerical Simulations of the Transient Behavior of an Advanced Safety Injection Tank

H. Y. Yoon[a], J. J. Jeong[b*], H. K. Cho[c], Y.S. Bang[d], K.W. Seul[d]
[a]Korea Atomic Energy Research Institute (KAERI), Daejeon, Korea
[b]School of Mechanical Engineering, Pusan National university, Busan, Korea
[c]Department of Nuclear Engineering, Seoul National University, Seoul, South Korea
[d]Korea Institute of Nuclear Safety, Daejeon, South Korea
[*]*Corresponding author: hyyoon@kaeri.re.kr*

1. Introduction

The advanced power reactor 1400 (APR1400), adopts a new design of safety injection system, consisting of four independent trains. Each train has a safety injection pump and an advanced safety injection tank (SIT, or called accumulator) equipped with a passive flow controller, named fluidic device (Chu et al., 2008).

Because of the complicated flow structure in the advanced SIT, a full-transient analysis using a CFD code is practically impossible (Lim et al., 2010). This paper deals with multi-scale numerical simulations of the advanced SIT of APR1400 using a component-scale two-phase flow code, CUPID, and a system-scale code, MARS. In this study, we mainly focus on the prediction of discharge flow behaviors.

2. The Advanced SIT of APR1400

The advanced SIT of the APR1400 is schematically shown in Fig.1. When the water level in the SIT is higher than the top of the standpipe, the water in the tank is delivered into the vortex chamber through both the supply and control nozzles as shown in Fig. 1(b). In this case, the two opposite-direction nozzles are designed to minimize the swirling flow effect and the merged flow moves directly to the exit nozzle at the center. This flow mode does not generate a vortex flow and the pressure loss in the vortex chamber becomes relatively low. Due to the low flow resistance, the SIT provides a high discharge flow. This is called a high flow mode.

Meanwhile, when the water level is lower than the top of the stand pipe, the discharge flow through the supply nozzle via the standpipe disappears and the water is delivered only through the control nozzle, as shown in Fig. 1(c). The control nozzle is designed so that the flow from the control nozzle is tangential to the vortex chamber, resulting in a strong swirling flow inside the vortex chamber. This leads to a greater flow resistance in the vortex chamber and, thus, a low flow rate. This is called a low flow mode.

As a result, the SIT passively controls the discharge flow rate of the ECC water without any moving part or any operator action.

To evaluate these features, i.e., the flow controlling performance of the advanced SIT, a prototypical full-scale test facility, VAPER, has been established.

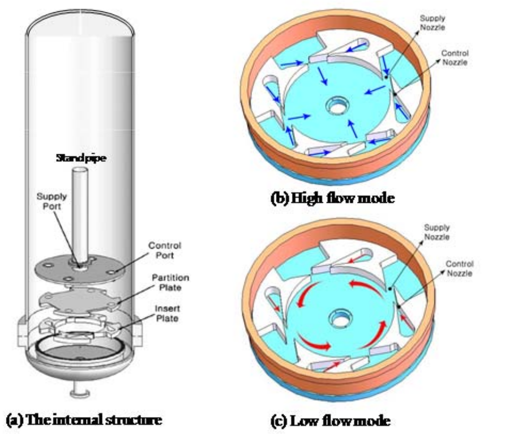


Fig. 1. The advanced SIT installed in the APR1400.

3. Numerical Simulations

3.1 The CUPID Simulations

In this work, the transient discharge behavior of the advanced SIT is analyzed using the CUPID code in three steps.

<u>Local pressure drop calculation using a CFD-scale analysis</u>

To evaluate the pressure drop through the control and supply nozzles, a CFD-scale analysis has been carried out first. For example, a two-dimensional computational domain is defined to include the control nozzle. To simulate the low flow mode design condition, an inlet flow boundary condition of 40.3 kg/s is assigned and an outlet pressure boundary condition of 0.1 MPa are set. Then, the pressure drop through the control nozzle is calculated to be 21.6 kPa. This result is retrieved in the next step to establish a flow resistance model in a component-scale analysis.

<u>Flow resistance model for a component-scale analysis</u>

Instead of a fine scale grid, a component-scale coarse grid for the advanced SIT is developed. Because the fluid dynamics near the cylindrical wall of the advance SIT hardly affect the discharge transient, a three-dimensional rectangular column is used to model the cylindrical SIT. Total number of computational grid is 17,304. The cross section area is the same as that of VAPER. To simulate the pressure drop at the supply nozzle, the control nozzle, and the vortex chamber, three different regions for flow resistance are assigned.

In the component-scale analysis of the advanced SIT, the pressure drops through the supply/control nozzle and the vortex chamber are calculated using a flow resistance model in the *k*-phase momentum equation as:

$$\frac{\partial}{\partial t}(\alpha_k\rho_k\vec{u}_k)+\nabla\cdot(\alpha_k\rho_k\vec{u}_k\vec{u}_k)=-\alpha_k\nabla P+\nabla\cdot\alpha_k\mu_k\nabla\vec{u}_k+\alpha_k\rho_k\vec{g}+\vec{F}_k^{int}+R|\vec{u}_k|\vec{u}_k$$

where k is liquid or gas, $\vec{F}_k^{int}$ is interfacial momentum transfer, and R is a flow resistance coefficient. In this step, the flow resistance coefficients for the three regions are obtained using the CFD-scale calculations and the resulting coefficients are implemented into the component-scale coarse grid.

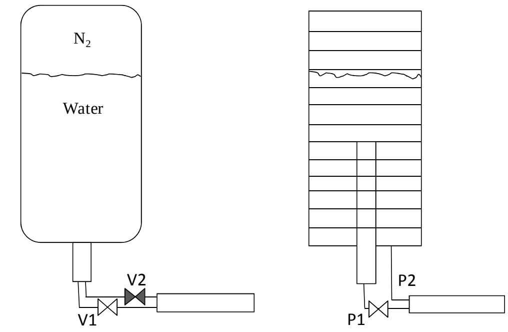
<u>Global transient analysis using the component-scale model</u>

Two cases of the VAPER experiments have been analyzed using the CUPID with the aforementioned component-scale models. Initial SIT pressures are 2.1 and 4.1 MPa, respectively. Initial water levels are the same for both cases as 89% of total height of the SIT. The calculated SIT pressure and water level of the two experiments are compared to the measured data and those from the MARS calculation (See Fig. 3). In general, the results show excellent agreements with the measured data. This indicates that the component-scale coarse grid with the flow resistance model can represent the complicated flow structure in the advanced SIT.

3.2. The MARS Simulations

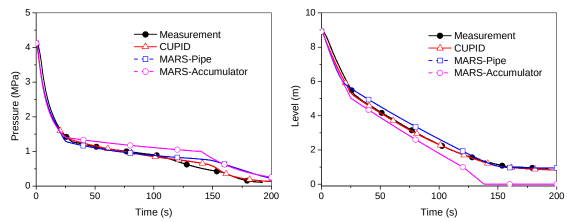
For the MARS simulations, two approaches were adopted:
(i) Using the “accumulator” component: The lumped-parameter “accumulator” component in the MARS code is adopted, where the SIT is considered as a single volume. It adopts the assumption of an ideal gas for the upper gas volume in the SIT. Two junctions (V1 and V2) in Fig. 2(a) stand for the flow paths for the high and low flow modes, respectively. The form loss factors of V1 and V2 are obtained from the experiments. Only one of the two valves is open.
(ii) Using the “pipe” component: A “pipe” component with 13 “volumes” is used to represent the SIT. Another “pipe” component is used to model the standpipe. Two junctions (P1 and P2) in Fig. 2(b) stand for the flow paths through the supply and control nozzles, respectively. When the water level in the SIT is lower than the top of the standpipe, the flow path through the supply nozzle (P1) is assumed to be closed.

Figure 3 shows the comparison of the MARS and CUPID results with the measured data. The results of CUPID are very close to the experimental data, and the MARS “pipe” model is better next. The MARS “accumulator” model is less accurate and physically inconsistent, yielding the slow depressurization and the rapid depletion of water inventory.

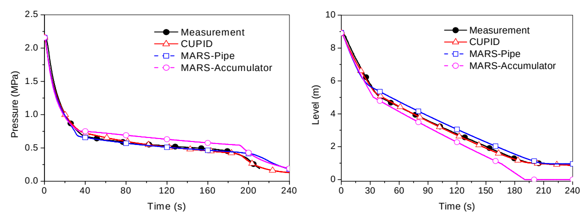


(a) “Accumulator” model (b) “Pipe” model

Fig. 2. The MARS models for the VAPER experiment.



(a) Pressure: 4.1 MPa (b) Water level: 4.1 MPa



(c) Pressure: 2.1 MPa (d) Water level: 2.1 MPa

Fig. 3. Comparison of the CUPID and MARS results with the experimental data.

4. Conclusions

By using three different length-scale (CFD, component, and system scales) analyses, the transient behavior of an advanced SIT has been successfully simulated. The result of the fine scale analysis was fed into the coarse scale analysis, leading to accurate results with less computational cost.

The comparison of all the results showed that the CUPID code predicted the experiments very well, the MARS “pipe” model was better next, and the MARS “accumulator” model was less accurate and physically inconsistent.

REFERENCES

Chu, I.C., Song, C.H., Cho, B.H., Park J.K., 2008. *Nucl. Eng. Des.* 238, 200-206.
Lim, S.G., Lee, S.H., Kim, H.G., 2010. *Nucl. Eng. Des.* 240, 1139-1150.