Computational Study of Flow Behavior in APR1400 Safety Injection Tank Using Multiphase Segregated Flow Model

Jai Oan Cho [a], Jeong Ik Lee [a]*, Young Seok Bang[b], Seung Hun Yoo[b]
[a]*Department of Nuclear and Quantum Engineering, Korea Advanced Institute of Science and Technology (KAIST)*
[b]*Safety Evaluation Department, Korea Institute of Nuclear Safety (KINS)*
[*]*Corresponding author: jeongiklee@kaist.ac.kr*

1. Introduction

The Advanced Power Reactor (APR) 1400 has an emergency core cooling system (ECCS). One of the most important components in the ECCS is the safety injection tank (SIT). Inside the SIT, a fluidic device (FD) is installed, which passively controls the mass flow of the safety injection of the coolant, eliminating the need for low-pressure safety injection pumps. As passive safety mechanisms are emphasized nowadays, it has become more important to model the SITs more realistically.

As shown in Fig. 1, during the high flow mode, water level is higher than the standpipe height. Hence, water flows into the vortex chamber of the FD from two ports, the supply port and the control port. Water from the two different nozzles collide and flows into the discharge pipe directly. During the low flow mode, water level is lower than the standpipe height, therefore, water can only flow into the vortex chamber through the control port. Therefore, the flow is directed to a tangential angle of the vortex chamber generating a vortex, resulting in a lower water flowrate supplied to the reactor core.

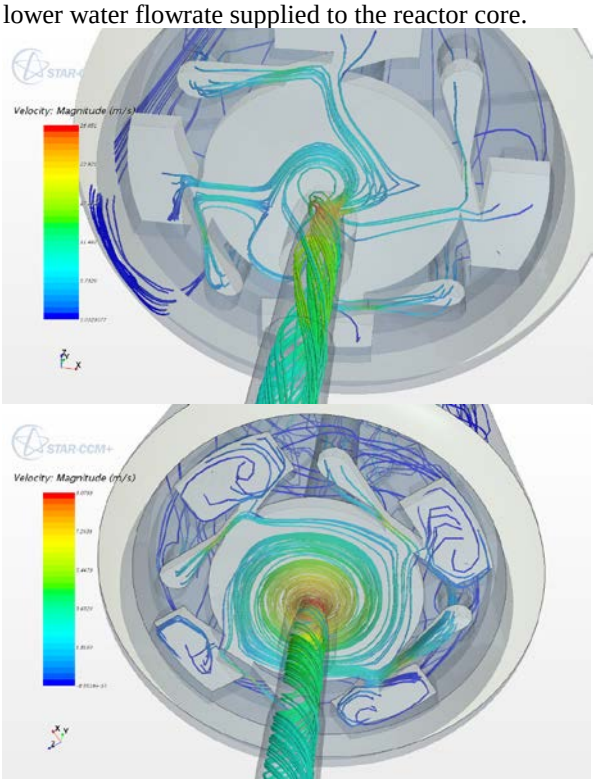

Fig. 1. Streamlines within the FD during high flow (top) and low flow (bottom) [1]

As the water level drops, nitrogen may be entrained into the discharge pipe and then into the core. This may affect the core cooling capability and threaten the fuel integrity during LOCA situations. However, information on the nitrogen flow rate during discharge is very limited due to the associated experimental measurement difficulties. Gas behavior within the tank was analyzed using commercial CFD tool.

2. Methods and Results

The problem is numerically challenging since compressible fluid and incompressible fluid coexist in the same large physical problem domain. In previous studies, the problem had been simplified in many ways to get results in a reasonable amount of time due to its complexity.

Preliminary test cases were run using the Volume Of Fluid (VOF) model. The model was selected over the other models due to its relative robustness and ease of convergence. VOF model was used to capture the free surface. Multiphase equation of state was calculated. Multiphase VOF-VOF interaction was calculated using a constant surface tension of 0.074. Segregated Flow model was selected along with the segregated multiphase temperature. The realizable K-ε model, with high y+ wall treatment was used for the unsteady computation with a time step of 1.0E-4s.

The preliminary calculation result matched the experimental results very well even with low resolution grids. [1] The mass flow rate fits the results from the experiment reasonably and the flow transition from high flow to low flow was simulated smoothly.

Although the VOF model used in the preliminary analysis was reasonably accurate in calculating mass flow rate of water, it cannot model the nitrogen entrainment accurately due to the absence of the interfacial drag. The reason is that the conservation equations are not solved for each phase. Rather, conservation equations are solved as if it were a single phase problem with the fluid property averaged depending on the volume fraction of each phase. Since only one momentum equation is solved, one velocity vector represents the velocity of both phases in a single cell. Therefore, there is no slip velocity between the different phases. With no slip velocity, there is no drag force, which can be a major cause of nitrogen entrainment. Therefore, the VOF model cannot model the nitrogen entrainment phenomena effectively.

Among the other multiphase models supported by STAR-CCM+, the Multiphase Segregated Flow (MSF) model was determined to be the optimal choice regarding the calculation characteristics. The biggest advantage over the VOF model was that conservation equations were solved for each phase. Drag force can be calculated proportional to the velocity difference between phases. Another major advantage is that dispersed and segregated two phase flows can be solved within a single framework using a large scale interface model. The flow starts with two phases separated clearly but when the flow turns from high flow to low flow, the flow within the standpipe turns into a dispersed flow. Not many models can support for such complicated conditions, but the MSF model allows such calculation in two phase conditions.

Table I: Comparison of VOF model and MSF model

Volume of Fluid (VOF)	Multiphase Segregated Flow (MSF)
Used for calculating free surface	Used for flows with stratified and dispersed flows in single framework
Conservation equations solved as if single phase	Conservation equations for each phases except for pressure
No Slip Velocity between phases	Slip Velocity between phases allows calculation of drag force
Faster Convergence Rate	Slower Convergence Rate
Easier to achieve Convergence	Harder to achieve Convergence

Two calculations were performed, one using the VOF model, the other using the MSF model. Models shared by both cases are as follows. Cases were setup using Eulerian multiphase models. Multiphase equation of state was calculated. Multiphase segregated flow was also implemented using one of the derivatives of SIMPLE. Phase coupled fluid energy was used to calculate the energy. Water density was kept constant while ideal gas laws were applied for nitrogen. SST K-Omega was used for turbulence modeling. Segregated fluid temperature was implemented. Exact Wall Distance was used. Mesh base size was set as 0.25m and volume control was applied as in the preliminary calculation. More than 180,000 cells were created to fill the domain.

In case of the VOF model, the outlet pressure boundary was set as 1bar. Pressure jump option was used with a constant pressure loss coefficient of 9.5. To avoid sudden exposure to excessive pressure difference and to model the gradual valve opening as in the experiment. The pressure was decreased from 15bar to 1bar gradually over 12s. 12s is the time it takes for the valve to open in the experiment.

Mass flow rate of nitrogen is shown in Fig 2. Both show a sudden increase and decrease during the flow transition. Nitrogen exits the tank earlier in case of VOF model calculation because the flow transition occurs faster. However, the MSF result shows higher mass flow rate. The total mass of discharged nitrogen is shown in Fig 3. The MSF model predicts a nitrogen release of 2 x 10^{-4} kg, which is a little higher than that of the VOF model. If the volume of the nitrogen in atmospheric condition (1bar 300K) were to be calculated, it would be 1.8 x 10^{-4} m^3. However, if the pressure of the reactor pressure vessel in a LOCA scenario is considered, the volume taken up by nitrogen will be even smaller.

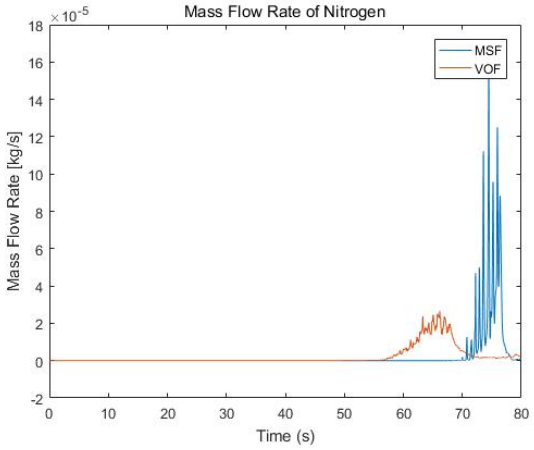

Fig. 2. Nitrogen Mass Flow Rate in MSF and VOF model

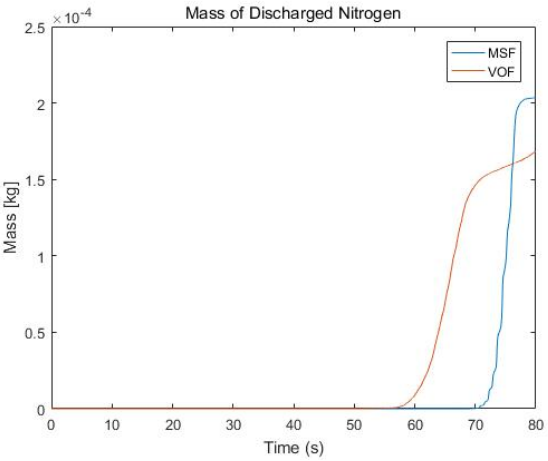

Fig. 3. Total Mass of Discharged Nitrogen in MSF and VOF model

3. Conclusions

To understand flow behavior within the SIT/FD, Computational Fluid Dynamics (CFD) analysis was used to analyze the flow behavior inside the SIT/FD. The Volume Of Fluid (VOF) model was used to model the water and nitrogen within the tank for preliminary

calculation. The preliminary results of the coarse grid calculation fit the experimental results quite well despite its coarse grid. Nonetheless, the mesh resolution was increased to capture the vortex in the fluidic device more precisely.

After checking the preliminary results, the CFD calculation was brought to another level by introducing the Multiphase Segregated Flow (MSF) model. The replacement of multiphase model was necessary in order to capture the nitrogen behavior more accurately. Although the MSF model was not the best in predicting the pressure and mass flow rate of water, the model was more suitable in calculating the nitrogen entrainment.

Quite the opposite from what was originally anticipated, CFD results on the amount of nitrogen entrained showed that discharged nitrogen mass is minimal and would not cause any significant change in the result.

ACKNOWLEDGEMENTS

Authors gratefully acknowledge that this project is funded by the KUSTAR-KAIST Institute joint research project.

REFERENCES

[1] J. Cho et al., Computational Study of the Safety Injection Tank Performance, CFD4NRS6, Sep. 13-15, 2016, MIT, Cambridge, MA, USA