Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 27-28, 2016

CFD simulation on condensation inside a Hybrid SIT

Byong Guk Jeon[a], Sung-Uk Ryu[a], Seok Kim[a], Dong Jin Euh[a]
[a]*Korea Atomic Energy Research Institute, 989-111 Daedeokdaero, Yuseong, Daejeon, 305-353, Korea*
**Corresponding author: bejeon@kaeri.re.kr*

1. Introduction

The concept of Hybrid Safety Injection Tank system (Hybrid SIT) was proposed by Korea Atomic Energy Research Institute (KAERI) aiming at Advanced Power Reactor Plus [1]. The main advantage of the system is the ready injection of coolant into the reactor coolant system at high pressure. Opening an isolation valve, steam is delivered into the tank from a pressurizer. However, condensation inside the tank retards pressure rise and injection time. Therefore, in view of safety, it is necessary to understand the condensation phenomenon accurately. Separate effect tests at full pressure and temperature have been performed and a dedicated experiment for visualizing thermal mixing is being conducted in KAERI [2].

Because steam is spread three-dimensional way inside the tank, a CFD code becomes the adequate option over conventional system codes. CFD codes have been diversely utilized and validated for the simulation of condensation in presence of non-condensables [3]. In this paper, a CFD simulation is conducted as a preliminary study.

2. Methods

2.1 Validation of the condensation model

Before moving onto the simulation of hybrid SIT, we validated the condensation model against other experiments. The built-in condensation model of a CFD code, STAR-CCM+, was used [4]. In that model, the condensation rate was calculated from the product of the diffusion coefficient and the gradient of steam mass fraction on the cooling surface. The rate was used as the mass source term. Condensate film was considered. We selected two as references: the COPAIN experiment having a high steam velocity and the KAIST experiment having low steam mass fraction. The comparison results are given in Fig. 1 and Table 1, proving the fidelity of the model.

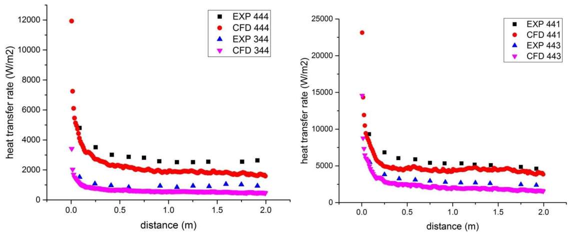

Fig. 1. Validation of the condensation model against the COPAIN experiment.

Table I: Validation of the condensation model against the KAIST experiment

Pressure	Air Mass Fraction	H_{exp} (W/m^{2}K)	H_{cal} (W/m^{2}K)
0.24	0.2	1220	1074
0.24	0.4	606	555
0.38	0.2	1437	1112
0.38	0.4	651	577

2.2 Description on the experimental facility

A small experimental facility was installed to enable detailed study through visualization of thermal mixing. At low pressure, because of the low durability of windows, the effect of steam injection rates, water temperatures, non-condensable gas, and nozzle sizes are being explored. For this facility, a preliminary simulation by a CFD code was done.

2.3 Input Preparation

A computational domain was constructed as shown in Fig. 2. To save computational costs, axi-symmetrical simulation was made. The total number of cells amounted to 5363. Mesh sensitivity studies were conducted in advance to affirm the validity of calculations. Actual geometry was reproduced even though there was a minor distortion in a circumferential direction. The domain was comprised of steam/air, water, and wall regions. Along the steam/air and wall interface, and steam/air and water interface, condensation occurred.

For physical models, a SST-k-omega turbulence model was used to resolve the boundary layer. The y+ wall treatment was also used. The implicit unsteady calculation is made with the time step of 0.001 s.

The steam/air domain was modeled as an ideal gas, whereas water and wall are modeled as solids. To be exact, water region should be treated as liquid using a volume of fluid model. However, because the calculation was so unstable, as a preliminary study, we did not consider the circulation of water. This simulation will be valid only when the thermal resistance of steam/air boundary layer is larger than that of water boundary layer.

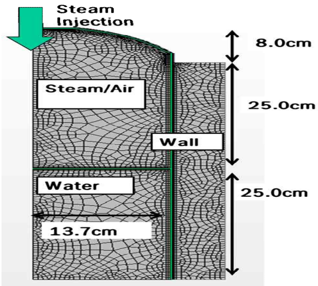

Fig. 2. Computational domain of the hybrid SIT

3. Results

The figure of merit is the pressure inside the tank. In Fig. 3, for a case of high steam injection, 0.6 kg/min, pressure trend was compared between the experimental results and the CFD results. We can see that the CFD code highly overpredicts the pressure. It is attributed to the lower condensation, maybe at the water surface. Figure 4 represents the accumulated amount of injection and condensation mass. Injection is well reproduced in simulation. We can see that the condensation amount is quite considerable: 93% of injection at 100s. However, that small difference leads to extensive pressurization. Fig. 5 represents the comparison of condensation rates between water surface and wall structure from the CFD calculation. We can see that condensation at the water surface is major only for the very early period and wall structure contributes dominantly after 5 seconds. Figure 6 shows the temperature and the velocity profiles. Wall is most heated near the water surface. Steam is injected with a high velocity, 8.0 m/s at 110 s, from the injection nozzle but is slowed down to reach much lower value at the water surface, 1.5 m/s at 110s. The steam moves over the water surface, hits the wall and rotates counterclockwise. From the visualized results, it was found that the condensation depth of water was more than 20 cm whereas the CFD code gave only 1 cm of thermal stratification because of neglect of steam penetration into the wall. The penetration of gas into water has been widely studied by both experiments and CFD codes [5-6]. We are also working on simulation of water as liquid region using a VOF model. There are still endeavors to get stable and reliable results.

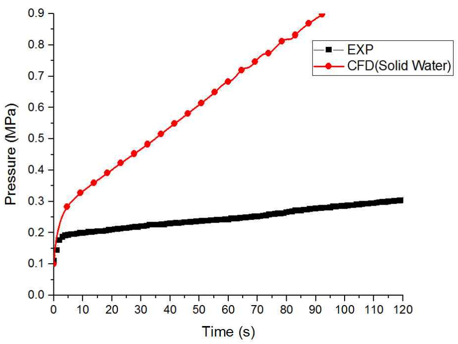

Fig. 3. Comparison of the tank pressure between the experiment and the CFD calculation.

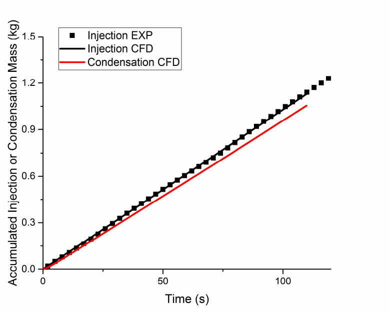

Fig. 4. Accumulated amount of steam injection and condensation

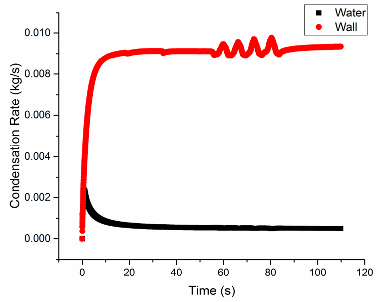

Fig. 5. Comparison of water and wall condensation rates from the CFD calculation.

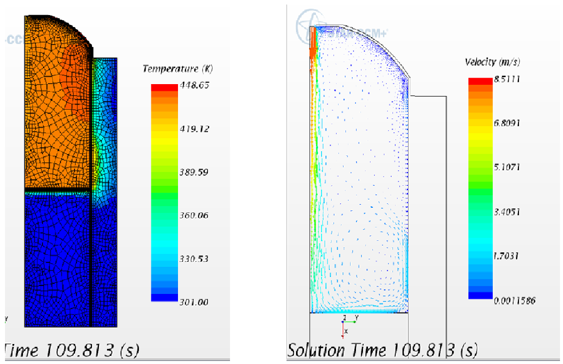

Fig. 6. Temperature and velocity profile from the CFD calculation

4. Conclusions

In Hybrid SITs, condensation inside the tank affects its pressure rise and injection time. In an attempt to explore the condensation in detail, we manufactured a dedicated experimental facility for visualization of condensation-induced thermal mixing and conducted a preliminary CFD simulation. Its condensation models were validated first and then computational domain was constructed. The water region was modeled as a solid for stable calculation. The CFD results gave less condensation and excessive pressurization because of lack of steam penetration into the water. In the future, the water region will be modeled as liquid using a VOF model.

Acknowledgments

This work is supported by the National Research Foundation of Korea (NRF- 2012M2A8A4004176) grant funded by Ministry of Science, ICT and Future Planning of the Korea government.

REFERENCES

[1] Kwon, T, Euh, D.J., Bae, J., Park, C.K., 2011. Hybrid high pressure safety injection tank
for SBO. In: Trans. of the KNS Autumn Meeting, Taebaek, Korea, May 26–27.

[2] S. U. Ryu, H. B. Ryu, H. S. Park, and S. J. Yi, An experimental study on the thermal-hydraulic phenomena in the Hybrid Safety Injection Tank using a separate effect test facility, Annals of Nuclear Energy, Vol. 92, pp. 211-227, 2016.

[3] A. Dehbi, F. Janasz, and B. Bell, Prediction of steam condensation in the presence of noncondensable gases using a CFD-based approach, Nuclear Engineering and Design, Vol. 258, pp. 199-210, 2013.

[4] CD-adapco, STARM-CCM+ Documentation - Version 11.04, 2016

[5] V. Tanskanen, CFD modeling of direct contact condensation in suppression pools by applying condensation models of separated flow, phD thesis at Lappeenranta University of Technology, 2012.

[6] D. Munoz-Esparza, J. -M. Buchlin, K. Myrillas, R. Berger, Numerical investigation of impinging gas jets onto deformable liquid layers, Applied Mathematical Modelling, Vol. 36, pp. 2687-2700, 2012.