CFD analysis of steam jet injection in a hybrid SIT

Byong Guk Jeon[a], Sung-Uk Ryu[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

A hybrid safety injection tank (hybrid SIT) is a tank to supply cooling water into the reactor vessel in an emergency of an advanced nuclear power plant, APR+ [1, 2]. In case of an accident, such as rupture of a constituent pipe, the valve at the top line is opened. By that opening, steam inside the pressurizer is transported to the nitrogen-filled SIT by pressure difference. The SIT is rapidly pressurized by the steam inflow. Once the hybrid SIT pressure increases to the level of the reactor vessel, water inside the hybrid SIT is injected by gravity of water. In operation of this hybrid SIT, the required time for the pressure rise, or the beginning time of the injection, is very important. The pressure rise time depends not only on the rates of steam inflow, but also on the condensation rate inside the hybrid SIT.

Regarding the condensation phenomena, experimental correlations, heat and mass transfer analogy, and the CFD code simulation were suggested [3-5].

In our research, to understand the thermal hydraulic behaviour of the tank under the steam jet injection, a CFD code simulation was conducted using a commercial code, STAR-CCM+ [6]. Largely, we examined the wall-contact condensation, jet development, and water-contact condensation by comparing with the dedicated experimental results.

2. Methods

2.1 Description of the experiment

Figure 1 represents the schematic for the dedicated hybrid SIT experiment performed in KAERI. A small experimental facility was manufactured to enable detailed study through visualization of thermal mixing. For the visualization, a high-speed camera and the particle image velocimetry was used. The experiment was performed at low pressure because of the low durability of visualizing windows. The effects of the steam injection rate, water temperature, and nozzle size were explored.

For the experiment, steam was generated from a boiler at 1 MPa and its flow rate was controlled by a valve. At the downstream of the valve, the temperature and pressure of the steam were measured. Then, the steam was injected into the hybrid SIT through a nozzle. The tank was initially filled with air and its pressure increased gradually. In that period, some of the injected steam was condensed in contact with the wall and the water. The experiment was performed until the pressure reached 0.6 MPa (absolute).

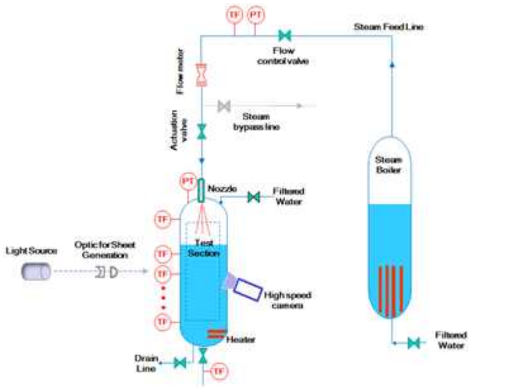

Fig. 1. Schematic of the dedicated hybrid SIT experiment

2.2 Simulation Condition

The validity of condensation and jet development models in CFD code was proved against other experiments in advance.

Figure 2 represents the computational domain for the simulation of our dedicated hybrid SIT experiment. To save the computational cost, two-dimensional calculation was conducted using the axisymmetric model. The total number of the cells was 5,300. To assure the confidence of the calculation, grid sensitivity studies were conducted.

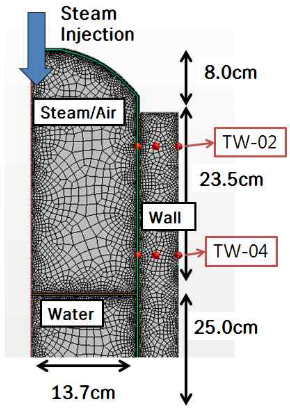

Fig. 2. Computational domain for simulation of the dedicated hybrid SIT experiment

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 was made with the time step of 0.001 s.

The domain was comprised of steam/air, water, and wall regions. Along the steam/air - wall interface and steam/air - water interface, condensation occurred. The steam/air, wall, and water domain was modeled as an ideal gas, solid, and liquid. In our calculation, the interface between gas and water was fixed. In a real situation, steam penetrates the liquid at higher velocity or at least perturbs the interface. In order to reflect the change of interface, we need to use a volume of fluid scheme. However, because of the small time step size required, we did not use that scheme. Rather, we chose the experimental case where the steam inlet velocity was low.

3. Results

The figure of merit in the dedicated hybrid SIT experiment was the pressure inside the tank. In Figure 3, pressure trend was compared between the experimental results and the CFD results. The CFD code overly predicted the pressure. It is attributed to the lower condensation.

Figure 4 represents the temperature trend inside the tank. Among many thermocouples, two data sets are provided: two wall temperatures (TW02, 04). The overall trends were well traced. In detail, the wall just above the water surface (TW04) was over-predicted while the wall at the upper part (TW02) was under-predicted. The distribution of condensation, or flow direction, needs to be further improved.

Figure 5 represents the accumulated amount of injection and condensation mass. Injection was well imposed in the simulation. The condensation amount was quite considerable: around 90 % of injection. However, that small difference led to extensive pressurization as shown in Figure 3. It means that small error in condensation models can lead to large pressure difference. Examining the condensation locations, condensation at the wall was more than two times that at the water. Because wall has higher conductivity, it played a dominant role.

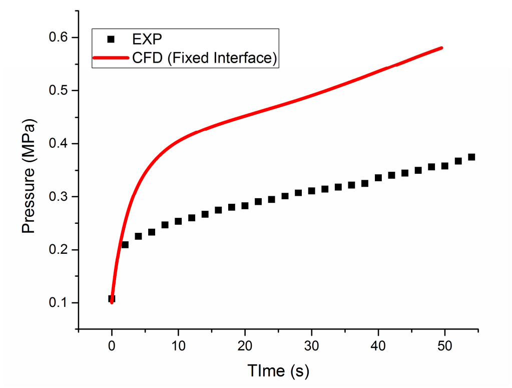

Fig. 3. Pressure trend between experiment and calculation

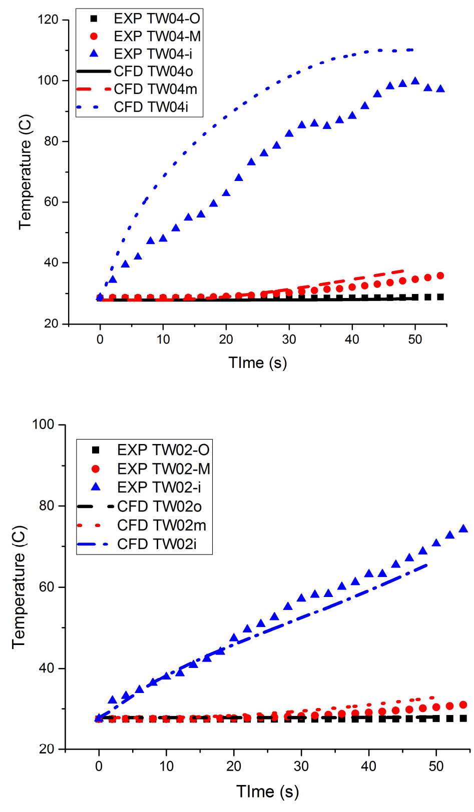

Fig. 4. Temperature trend between calculation and experiment

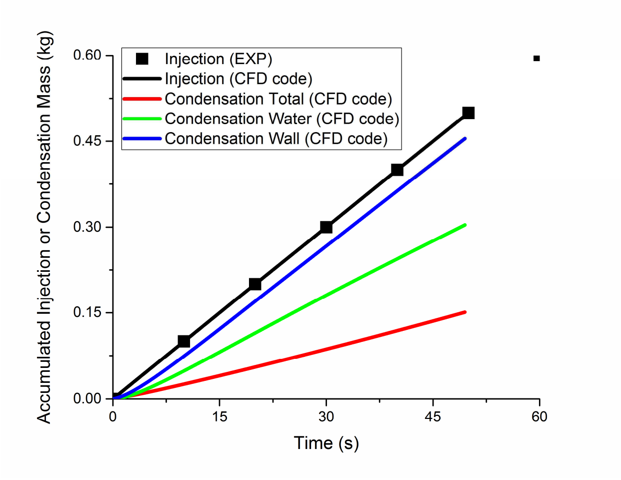

Fig. 5. Accumulated injection and condensation mass inside the hybrid SIT facility

4. Conclusions

In hybrid SIT system, the pressure rise time is important and the condensation inside the tank should be well understood. In an attempt to explore the condensation in detail, we manufactured a dedicated experimental facility for detailed measurement and visualization. In parallel, a CFD code simulation was conducted using a commercial code, STAR-CCM+. The hybrid SIT experiment was simulated by considering steam/water, wall, and water regions. From the simulation results, condensation characteristics were identified and wall temperature trends were roughly matched. However, pressure rise from the calculation was higher than the one from the experiment. Further improvement on the condensation models, especially on the water surface, is needed.

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

[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] 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

[5] B. G. Jeon, D. Y. Kim, C. W. Shin, and H. C. NO, Parametric experiments and CFD analysis on condensation heat transfer performance of externally condensing tubes. Nuclear Engineering and Design, Vol. 293, pp. 447-457, 2015

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