# Preliminary CFD Aanalysis on Operating a Containment Spray System

Taehyub Hong [a*], Yujung Choi [a], Jong wook Go [a]
[a]*Korea Hydro & Nuclear Power Co.,LTD*
[*]*Corresponding author: hong7777@khnp.co.kr*

## 1. Introduction

The containment spray system in a nuclear power plant has roles to depressurize the containment atmosphere in a design based accident or a severe accident. The spray droplet flow cool down the containment atmosphere and condense steam. Also due to the spray droplet flow, strong convective flow can be made in the containment building, concentration of a combustible gas such as hydrogen in the atmosphere can be homogenized. Furthermore, aerosol-type fission products can be controlled through wash-down by the spray droplet flow.

In a severe accident condition, the containment spray system can influence positive effect and negative effect on mitigation of hydrogen risk. As the large amount of steam is condensed by spray droplet, the concentration of hydrogen can be increased locally. However, the spray flow can generate convective gas flow, the concentrations of the gas species can be homogenized. Because there is competition related to mitigation of hydrogen risk in the spray operation, the heat and mass transfer phenomena between the gas and the spray droplet should be modelled properly when an analysis of gas behavior in the containment building is carried out.

In this study, preliminary computational fluid dynamics simulation of gas-droplet flow in a spray injection in APR-1400 containment building is performed. The model used for the containment analysis for simulate heat and mass transfer between the gas and the droplets is validated using experimental results obtained in THAI facility. The numerical simulation has to cover the physical phenomena that occur when spray water is injected downward into stationary gas in a large closed vessel. In order to model the two phase flow of the gas and droplets, the Lagrangian approaches are applied, the gas-droplet interactions are modelled using the two-way Lagrangian particle model to simulate gas-droplet interaction, two way coupling of momentum, and the heat and mass transfer between gas phase and droplet phase. For droplet heat and mass transfer, convective heat transfer and diffusion limited mass transfer are assumed.

## 2. Methods and Results

### *2.1 Numerical model and validation*

To simulate the behavior of the gas mixture of steam and air, the momentum, heat, and mass transfer between the spray droplets and the gas was modelled using two-way coupling of the Lagrangian particles of the droplets to the continuous gas phase. For the momentum transfer the drag force and the pressure gradient force were considered. The Rans-Marshall correlation was used for the convective heat transfer and diffusion limited evaporation/condensation model for phase change at the surface of the droplets [3]. To simulate heat and mass transfer in the flow of the gas and the spray droplets, the commercial CFD code, STAR-CCM+ V12.04 by SIEMENS was used. The test area, including the THAI vessel, was modeled in three dimensions to generate a computational fluid and solid domain without geometric simplification.

The experimental result data used for numerical validation were obtain from THAI HD31-SE experiment which carried out at Becker Technologies in Eschborn, Germany. 1 kg/s of spray water is injected at 7.4 m above the bottom of the THAI vessel. the water temperature is around 20 ℃. The average droplet size is 600 μm and the spray angle is 30 degree. The initial conditions of the test are listed in Table 1 [1,2]. The spray water was injected for 166 s. After the spray injection ended (166 to 750 s), the vessel was stabilized without any gas injection.

In the numerical model for the validation, the spray injection condition was based on the experimental spray mass flow rates and spray water temperature. Fig. 1 shows the pressure change of the vessel over time. The Lagrangian spray model with phase change slightly under-predicted the pressure drop of the vessel slightly, but was in good agreement with the experimental results. Compared with the experimental results, the Lagrangian spray model did a reasonable job of predicting droplet behavior; gas flow, droplet, and heat and mass transfer between the spray droplets; and the mixture of the gas in the spray system. The numerical simulation was carried by STAR-CCM+ V12.04.

Table 1 Parameters in THAI31-SE Test

| Test Parameters | Value |
|---|---|
| Vessel Pressure | 1.495 bar |
| Gas Temperature | 89.5 ℃ |
| Steam Fraction | 25 vol% |

### *2.2 Preliminary containment analysis*

In order to test the numerical model for spray droplet flow, a preliminary analysis of an actual containment building was performed. As a geometry for the containment, the three dimensional geometry of APR-

1400 containment building was adopted. The initial gas composition was assumed to be the same the experimental condition of THAI HD31SE test. Hydrogen was assumed to be released with relatively high temperature near bottom of a steam generator in order to consider hydrogen flow out from a break position of the primary system in severe accident condition. Two calculation cases were considered depending on the containment spray system is operating or not. Injection positions of droplet particles was determined by the actual positions of the spray nozzles installed in the containment dome.

Fig. 2 depicts the gas velocity in the containment building. In the case without the spray operation, the upward velocity is generated from the release position of the hydrogen to the dome of the containment building by buoyancy force. In the contrary, in the case with spray operation, large circulation flow is generated by the interaction between the gas and the spray droplet. Gas pressure and temperature are decreased in the case with spray operation. Also the buoyancy flow of the hydrogen is suppressed by convective gas flow, and the hydrogen concentration is homogenized rather than the case without the spray operation.

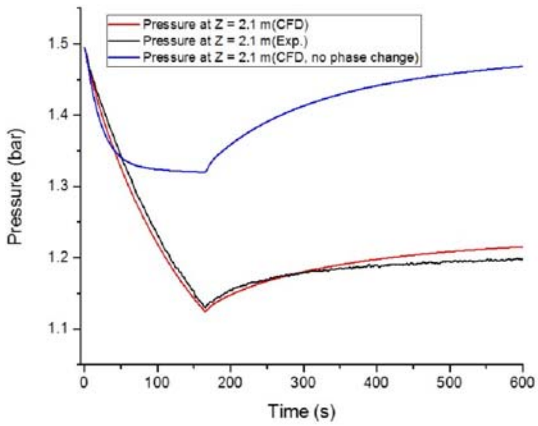



Fig. 1 Pressure Profile with Time

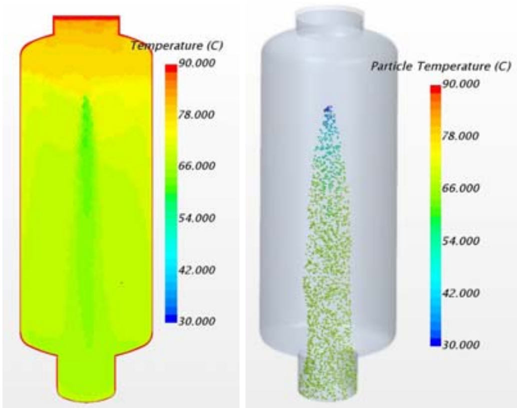
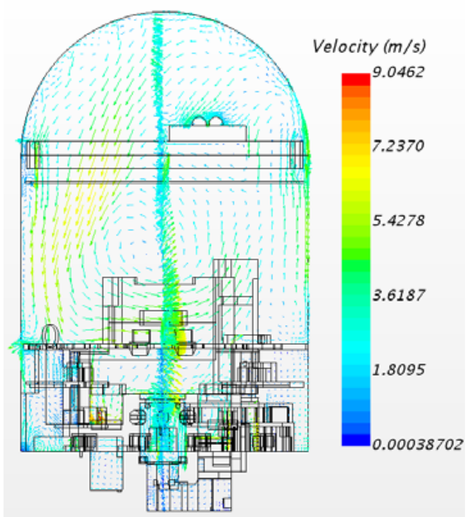


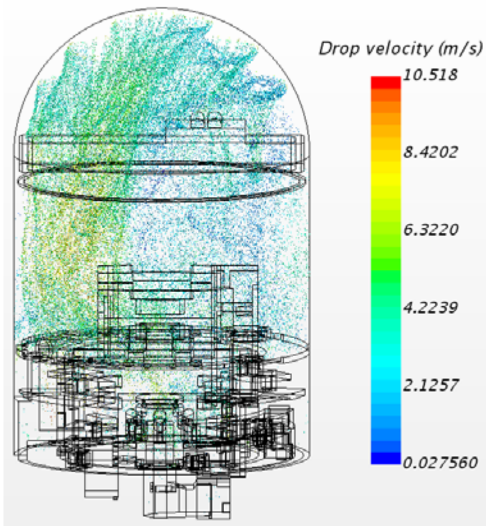

Fig. 2 Temperature of gas and droplet

### 3. Conclusions

Numerical simulation of the phenomena of heat and mass transfer between spray droplets and gas mixture was performed using CFD code. The spray model with the phase change model predicted the transient pressure. The results of the preliminary containment analysis of the spray system shows that the current CFD model is able to simulate the expected physical phenomena of gas behavior when the containment spray system is operated. If the spray model is verified for wide range of pressure and steam concentration, the CFD model in this study could be applied for simulation of containment thermal-hydraulic analysis.




(a) Gas velocity




(b) Droplet velocity

Fig. 2 Velocity of gas and droplet

### REFERENCES

[1] Gupta, S. ea al., Hydrogen combustion during spray operation - Tests HD-30, HD-31 and HD-32. OECD-NEA THAI-2 Project, Becker Technologies GmbH, Eschborn, Germany, 2014

[2] Aerosol and iodine issues, and hydrogen mitigation under accidental conditions in water cooled reactors, NEA/CSNI/R(2016)8,OECD/NEA(2017)

[3] Starccm+ V11.06 user's manual, Cd-adapco(2016)[2]