Numerical Analysis for Liquid Droplet Behavior in the CALIST Test Using OpenFOAM

Hyung Seok Kang[1], Jongtae Kim[1], Seokwon Whang[2], and Hyun Sun Park[2]
[1]KAERI, 111, Daedeok-daero, 989 Beon-gil, Yuseong-Gu, Daejeon, 34057, Republic of Korea
[2]POSTECH, 77 Cheongnam-Ro, Nam-Gu, Pohang, Gyeongbuk, 37673, Republic of Korea
[*]Corresponding author: hskang3@kaeri.re.kr

1. Introduction

The operation of a spray system in a nuclear power plant can decrease the temperature of the mixture gas of air-steam-hydrogen as well as affect the hydrogen concentration by condensing the steam in the containment during a severe accident [1,2]. The containment wall integrity during the severe accidents may be accurately predicted if we know the local concentration and temperature distribution of the air-steam-hydrogen mixture gas under the spray water operation in the containment. A spray analysis module based on the Lagrangian particle model in OpenFOAM was developed to accurately predict the behavior of the spray water in the containment during the severe accidents [2-4]. KAERI performed a validation analysis for the spray water behavior by using the spray analysis module against the Characterization and Application of Larger and Industrial Spray Transfer (CALIST) test on the basis of the POSTECH research results [3-6] where the validation was mainly done for the air entrainment behavior owing to the spray water discharge.

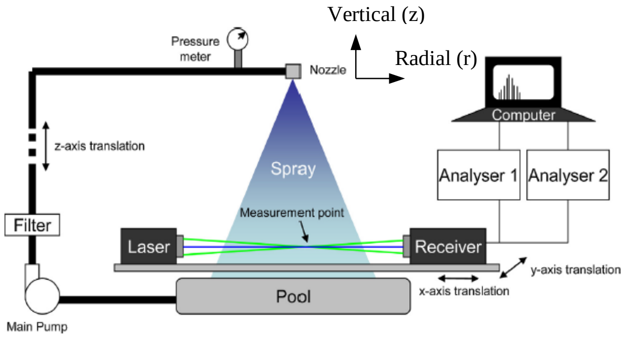
2. CALIST Test [5,6]

2.1 Test Facility

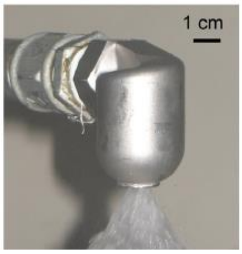
The CALIST facility consisted of a single spray nozzle with hollow cone type, water supply system using a pump, and pool which collecting the spray water (Fig. 1(a)). The water supply system circulated the water with 20 °C and 1 bar from the 5 m^3 pool to the spray nozzle (Fig. 1(b)) which discharged the water with 1 kg/s into the atmospheric environment. Distribution of velocity and diameter of the liquid droplet along the radial direction at 20 cm, 40 cm, and 60 cm from the nozzle outlet were measured by a Phease-Doppler Interferometry (PDI). Most of water was atomized to the liquid droplet in 20 cm from the nozzle outlet. Four measurement series, separated by 90°, were performed at six radial positions along the annular area such as Fig. 1(c). The spray nozzle used in the CALIST test was the same model, SPRACO 1713A, as used in French 900 MWe PWRs. Test conditions are summarized in Table 1.

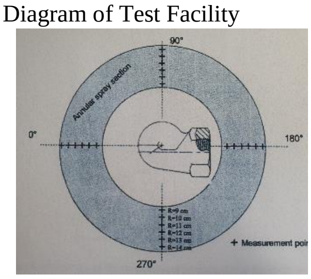
Table 1: Test Condition

Water Condition at upstream of nozzle	Spray Water Flow	Ambient Condition
3.5 bar, 20 °C	1 kg/s	1 bar, 20 °C



(a) Schematic Diagram of Test Facility



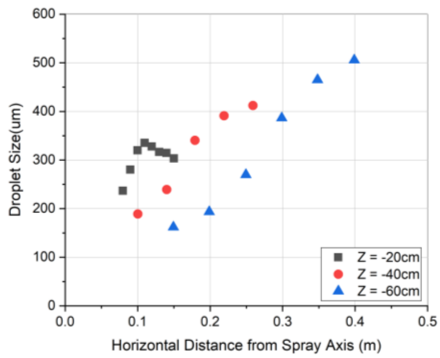


(b) Spray Nozzle (c) Measurement Locations

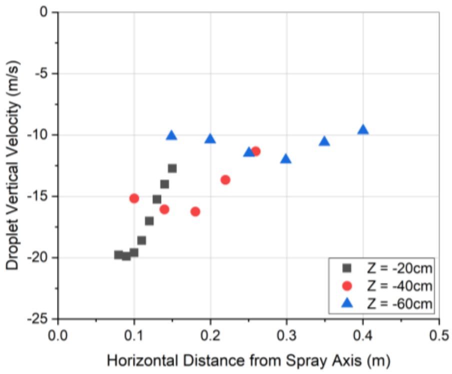
Fig. 1. CALIST Experimental Facility [5,6]

2.2 Test Data

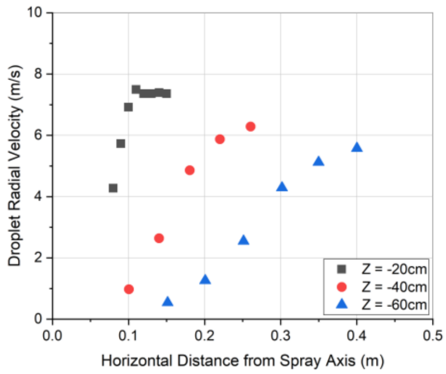
Fig. 2 shows the measured data of the geometrical mean diameters (D_{10}), vertical velocities, and radial velocities of the droplets along the radial distance from the spray axis according to the vertical lengths of 20 cm, 40 cm, and 60 cm from the nozzle outlet. Fig. 2(a) represents that larger droplet of 300 – 500 μm moves outward in the radial direction as the droplet vertically goes downward. Figs. 2(b) and (c) show that the vertical velocities of the droplets are decreased from about -20 m/s to about -10 m/s and the radial velocities are also decreased from approximately 7 to 5 m/s as the droplets move downward.



(a) Droplet Size Distribution



(b) Vertical Velocities of Droplet



(c) Radial Velocities of Droplet

Fig. 2. Measured Data in the CALIST Test [5,6]

3. CFD Analysis

3.1 Grid Model and Flow Field Models

A 3-dimensional grid model simulating the test section from the spray nozzle to the pool region including the air environment in the CALIST facility was constructed to analyze the liquid droplet behavior from the nozzle outlet such as Fig. 3 [3,4]. The spray nozzle was modeled by the hollow cone nozzle injection in the OpenFOAM-v1912 through indicating the positon in the grid model [7]. A total of about 477,015 hexahedral cells with a cell length of approximately 10 - 60 mm were generated in the grid model [3,4]. A wall condition with a constant temperature of about 20 ℃ was applied on the outer surface of the grid model [3,4].

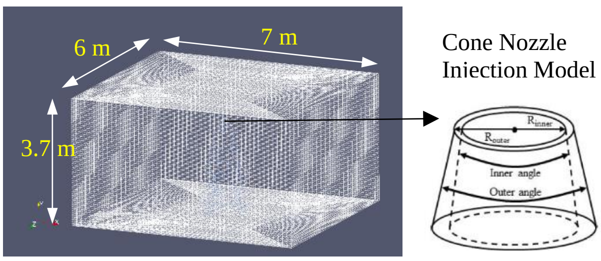


Fig. 3. Grid Model for the CALIST Test Facility [3,4]

OpenFOAM-v1912 with the Lagrangian-Eulerian model [2-4,7] was chosen for the simulation of the behavior of the liquid droplet and air entrainment in the CALIST test. The Lagrangian method using a force balance (Eqs. (1) to (4)) was used to simulate the liquid droplet injection through the hollow cone nozzle model located at 20 cm below the nozzle outlet because the measured data at 20 cm from the nozzle outlet was used as the inlet boundary condition (Fig. 3 and Table 2). Some parameters such as a velocity magnitude, nozzle angle, and shape factor in the inlet condition was modified from the POSTECH's inlet condition [3,4]. The Eulerian method using the mass conservation and Navier-Stokes momentum equations was applied to analyze the air behavior under the droplet injection. A heat transfer calculation was not performed because the experiment was conducted at the isothermal condition of 1 bar and 20 ℃ [5,6]. A turbulent flow was modeled by the standard k-ε model. The time step size in the transient calculation of 300 s was 0.1 – 2.0 ms for obtaining converged solutions.

Table 2: Parameters of the Cone Nozzle Injection Model

Spray Model (Lagrangian method)
- Diameter (d_{10}) distribution = mass Rosin Rammler model - Mass flow rate = 1 kg/s - U_{mag} = 18.02 m/s - Nozzle inner/outer diameter = 0.16 m / 0.30 m - Nozzle inner/outer angle = 21.80° / 36.87° - Parcel Per Second (PPS) = 5000 [#/s] *Inlet conditions were determined based on measured data at Z = -20 cm from the nozzle outlet.
mass Rosin Rammler model
$\mathrm{f(d, D, k)} = \frac{k}{D}\left(\frac{d}{D}\right)^{k-1} \exp\left[-\left(\frac{d}{D}\right)^{k}\right]$ - Diameter : d - Min. / Max. diameter = 30 μm / 970 μm - Mean diameter (D)= 260 μm - Shape factor (k) = 1.8

$$m_p \frac{du_p}{dt} = F_D + F_G + F_P \tag{1}$$

where,

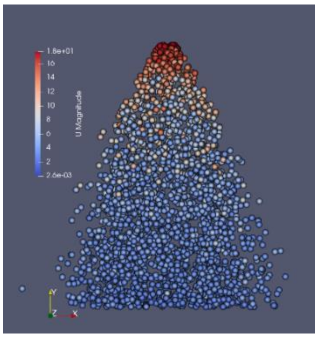
$$F_D = C_D \frac{\pi d_p^2}{8} \rho (U - u_p)\left|U - u_p\right| \tag{2}$$

$$F_G = m_p g \left(1 - \frac{\rho}{\rho_p}\right) \tag{3}$$

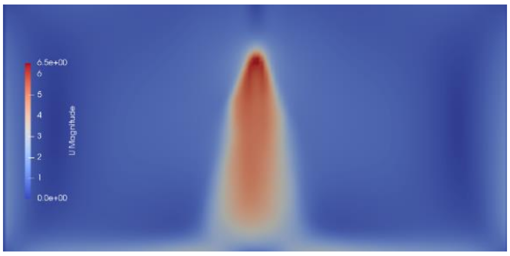
$$F_P = -\frac{\pi d_p^3}{6} \nabla p \tag{4}$$

3.2 Discussion on the CFD Analysis Results

The CFD analysis results for the droplet behavior in the CALIST test are shown in Figs. 4 and 5. Fig. 4 shows that the predicted velocity of the droplet and air by OpenFOAM can reasonably simulate the liquid droplet discharging to the downward direction from the inlet and the air flow owing to the droplet discharge. According to the comparison results between the CFD results and the test data, the CFD results only predicted the droplet existence in the range of approximately 0.15 m and 0.2 m from the spray axis at approximately 0.4 m below the nozzle outlet whereas the droplets locate in the range of approximately 0.1 m and 0.26 m in the test data. The CFD results accurately predict the test data of the droplet size, vertical velocity, and radial velocity with an error of about 20%. The discrepancy of the droplet positions between the CFD results and the test data is also shown at 0.6 m below the nozzle outlet. The CFD results only predict the droplet location from approximately 0.25 m to 0.35 m from the spray axis whereas the droplets locate in the range of approximately 0.15 m and 0.40 m in the test data. This may be explained by the fact that the uniform droplet velocity imposed at the inlet condition affect the calculated width of the droplet flow.

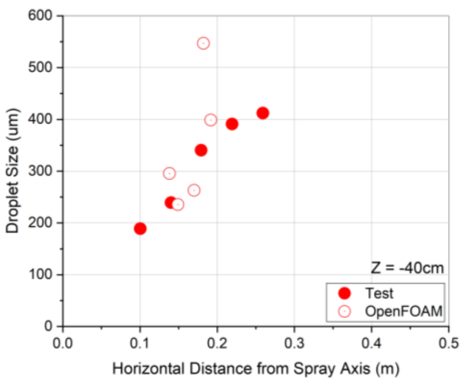


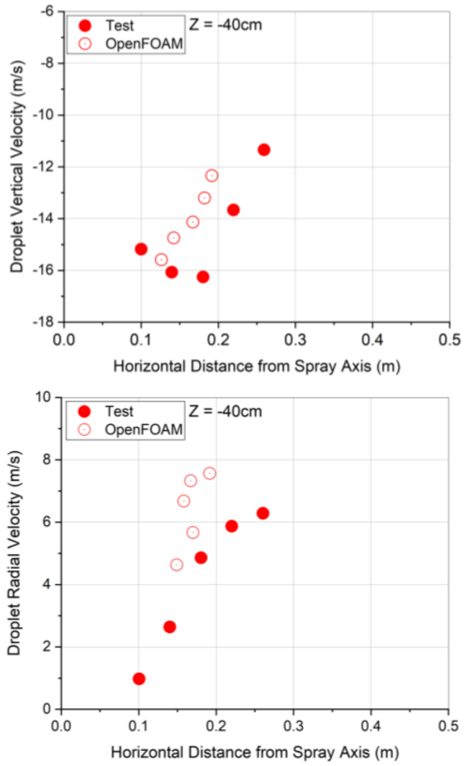
(a) Droplet Velocity Distribution (Magnitude)



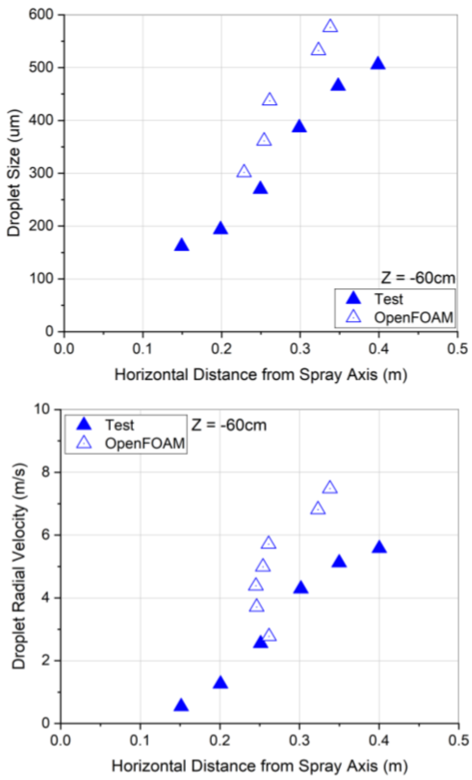
(b) Air Velocity Distribution (Magnitude)

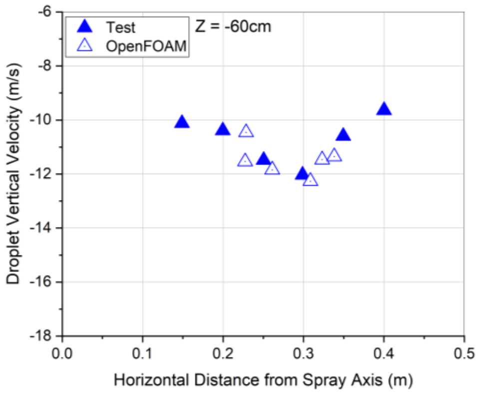
Fig. 4. Predicted Velocity Distributions of Droplet and Air





(a) Droplet Size and Vertical/Radial Velocities at Z = -40 cm





(b) Droplet Size and Vertical/Radial Velocities at Z = -60 cm

Fig. 5. Comparison of Droplet Behavior between Test Data and OpenFOAM Results

4. Conclusions and Further Work

KAERI performed the CFD calculation against the measured liquid droplet behavior in the CALIST test to validate the spray analysis module based on the Lagrangian particle model in OpenFOAM-v1912. We reasonably simulated the size, vertical velocity, and radial velocity of the droplets with an error range of approximately 20% when compared to the test data. However, the CFD results underestimated the width of the droplet flow approximately 35% when compared to the test data. To reduce this discrepancy between the CFD results and the test data, we will have to modify the inlet conditions used in the hollow cone injection model in the OpenFOAM. This validation results may decrease the uncertainty occurred when the spray analysis module is applied to a real nuclear power plant.

ACKNOWLEDGMENTS

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (Ministry of Science, ICT, and Future Planning) (No. 2017M2A8A4015277)

REFERENCES

[1] T. Kanzleiter, S. Gupta, K. Fischer, G. Ahrens, G. Langer, A. Kühnel, G. Poss, G. Langrock, F. Funke: Hydrogen and Fission Product Issues Relevant for Containment Safety Assessment under Severe Accident Conditions. OECD-NEA THAI Project Final Report No. 150 1326–FR 1, Becker Technologies GmbH, Eschborn, Germany, June 2010.

[2] Jongtae Kim, et al., Modeling for 3-D Analysis of Spray Droplet Flow in a Reactor Containment, Technical Report, KAERI/TR-7992/2019, KAERI, 2020.

[3] Seokwon Whang and Hyun Sun Park, Numerical analysis of gas mixing and stratified hydrogen behavior during spray operation based on multiphase heat and fluid flow computational model, Commission Report, KAERI/CM-2841/209, KAERI, 2020.

[4] Seokwon Whang, et al., "Numerical Study of the Break-up of the Light Gas Stratification by Spray: Gas Entrainment and Mixing", *Proc. of WORTH-9*, Chongqing, China, May 15-18, 2019.

[5] Arnaud Foissac, et al., "Droplet and Velocity Measurements at the Outlet of a Hollow Cone Spray Nozzle", *Atomization and Sprays*, Vol. 21, Number 11, pp.893-905, 2011.

[6] J. Malet, et al., "Gas Entrainment by One Single French PWR Spray, SARNET-2 Spray Benchmark", *Nuclear Eng. and Design*, Vol. 282, pp.44-53, 2015.

[7] OpenFOAM-v1912, https://www.openfoam.com