Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2023

Investigation of Spray Characteristics in TOSQAN-101 Experiment Using OpenFOAM CFD Simulation

Keun Sang Choi [*], Sangmin Kim, Jongtae Kim, Jaehoon Jung
Intelligent Accident Mitigation Research Division, Korea Atomic Energy Research Institute,
Daejeon, 34057, Rep. of Korea
[*] *Corresponding author: ksc307@kaeri.re.kr*

1. Introduction

In the event of an accident resulting in core damage, the containment building becomes pressurized by steam emitted from the reactor coolant system (RCS). Containment spray serves two primary functions by directly injecting coolant into the containment building: cooling and depressurizing the containment building, as well as increasing the mixing of stratified hydrogen. Furthermore, in the case of severe accidents, the spray also eliminates fission products in aerosol form. However, the accumulation of spray cooling water may cause pressurization of the containment building by reducing its free volume or by increasing the hydrogen concentration due to the condensation of water vapor. As a result, spray operation significantly influences the thermal-hydraulic behavior of the containment building [1,2]. To investigate the thermal-hydraulic phenomena during spray injections, TOSQAN experiments were conducted at the IRSN [3,4].

This study analyzes the thermal-hydraulic behavior and spray characteristics of the TOSQAN-101 experiment using computational fluid dynamics (CFD) simulation in the OpenFOAM-v2112 environment. The Euler-Eulerian framework is used, and a custom turbulent dispersion model is applied.

2. TOSQAN-101 Experiment

The TOSQAN program, undertaken by the IRSN, aims to identify thermal-hydraulic phenomena in containment systems. Various spray tests were conducted at high temperatures to analyze the heat and mass transfer between spray droplets and gas mixtures. The TOSQAN facility comprises a closed cylindrical vessel with a volume of 7 m^3, a height of 4 m, and an internal diameter of 1.5 m. Steam is injected from the bottom of the vessel, and the vessel walls are thermostatically controlled by heated oil circulation to regulate the gas temperature inside the vessel. The spray nozzle is located 0.65 m below the top of the vessel, and a drain pipe is installed to remove accumulating spray water [3]. The benchmark simulation target of this study is the TOSQAN-101 spray experiment, which examined the interaction of the spray in reducing steam partial pressure under air-steam mixture conditions [4]. The geometry of the TOSQAN test vessel is illustrated in Fig. 1, and the initial conditions of the experiment for for thermal-hydraulic and spray characteristics are outlined in Table I-III.

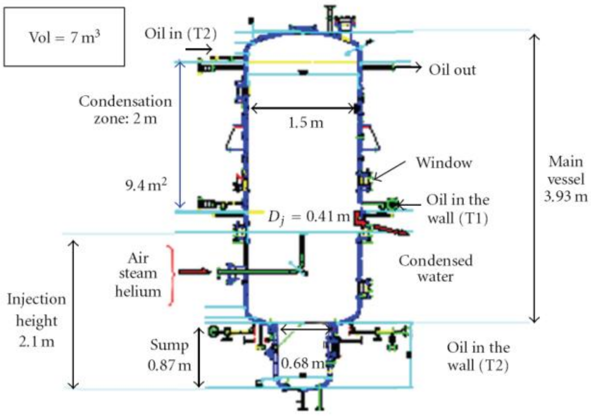

Fig. 1. The geometry of TOSQAN vessel [5]

Table I: Experimental gas initial conditions just before spray injection (t = 0 s)

Mean gas temperature out of the spray zone [°C]	131.1
Mean gas temperature in the spray zone [°C]	131.0
Total pressure [bar]	2.5
Initial gas composition (from mass balance) [mol]	Steam: 308 Air: 213

Table II: Measured mean wall temperature

Interval	Upper [°C]	Middle [°C]	Lower [°C]
Just before injection	121.8	122.3	121.7
0 – 102 s	121.4	121.6	121.3
107 s – 300 s	120.8	120.4	120.3
306 s – 601 s	120.3	120.0	119.4
End of the test	119.3	120.1	115.4

Table III: Experimental spray characteristics

Flow rate [g/s]	Angle [˚]	Droplet size [μm]	Droplet Temperature [°C]
29.96	55	145	t=0 s: 119.1 t=311 s: 22.1 t=1000 s: 27.7

3. Simulation Set-up and Results

3.1 Simulation Set-up

The computational mesh, consisting mostly of hexahedral cells, was created using the open-source software SALOME, with a total of 223,056 cells. The mesh area was divided based on the mean temperature measurement points in Table II, as shown in Fig. 2. Wall temperature boundary conditions were determined based on these regions.

The numerical solver used in this study is reactingTwoPhaseEulerFoam, an Euler-Eulerian two-phase flow solver that can consider multiple species. For the interfacial momentum transfer terms (M_d^d for drag, M_d^{td} for turbulent dispersion) of the momentum equation, the Schiller-Naumann model [6] was used for interfacial drag, and a custom turbulent dispersion model was applied, which will be discussed in the next section.

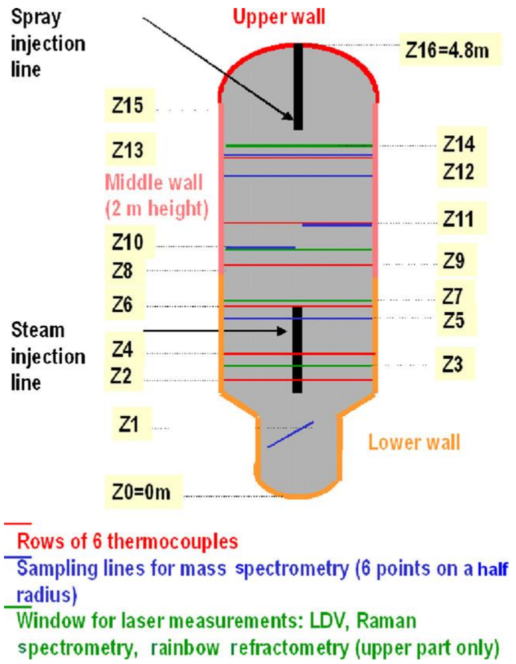

Fig. 2. Measurement points of TOSQAN vessel [3]

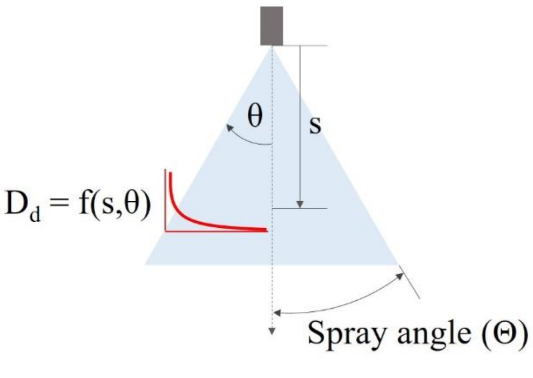

Fig. 3. Schematic expression of custom turbulent dispersion model

3.2 Custom turbulent dispersion model

The turbulent dispersion model developed in this study focuses on simulating the spray and its spreading. It assumes that the spray is injected downwards and that droplets close to the nozzle and at the center are greatly influenced by the spray droplet field, while droplets farther away from the nozzle and center are more influenced by the flow of the surrounding atmosphere. In this study, the term D_d is modeled as a power function of the distance from the origin of spray injection and the spraying angle. The model is schematically described in Fig. 3.

$$M_d^{td}=-D_d\times\nabla\alpha_d \tag{1}$$

$$D_d=C_{td}\times\left(\frac{\theta}{\Theta}\right)^n \tag{2}$$

3.3 Simulation Results

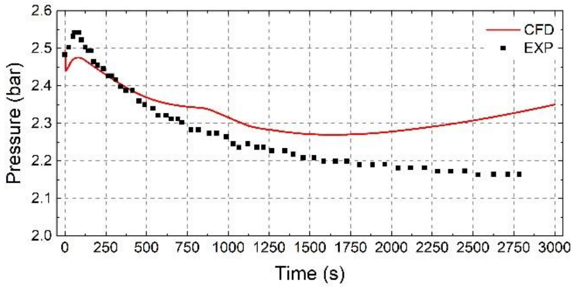

Fig. 4. Comparison of CFD and experimental data (Pressure of vessel)

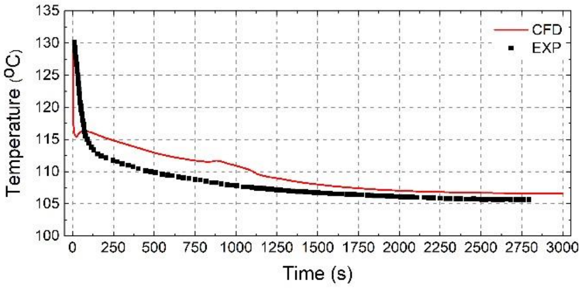

Fig. 5. Comparison of CFD and experimental data (Mean temperature of vessel)

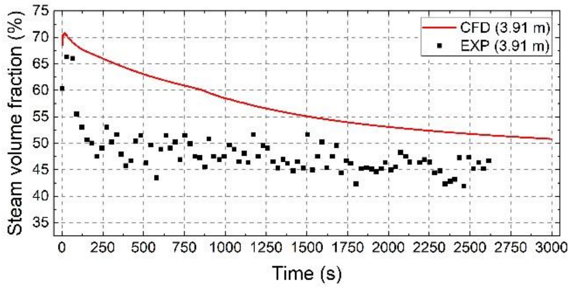

Fig. 6. Comparison of CFD and experimental data (Steam volume fraction, 3.91 m)

Fig. 4-6 present simulation results compared with experimental results. The temperature behavior of the containment vessel in Fig. 5 indicates that CFD generally approximates the experimental results well. However, as shown in Fig. 4, the degree of convergence decreases with time when looking at the pressure behavior. Comparing the CFD and experimental results of the steam volume fraction in Fig 6, a rapid decrease in steam volume fraction occurred in the experimental results initially. This means that condensation of steam initially present near the injector occurred despite the high temperature of the spray water at the start of the experiment. In contrast, in CFD, the steam volume fraction increased slightly and then decreased gradually, indicating that the condensation of steam due to spray injection was not predicted properly. This can also be seen through the pressure behavior in Fig 4. The slope of pressure decrease in the experiment and CFD is different, and even after 1500 seconds in CFD results, pressure can be seen to increase again. It is judged that there was a problem in calculating the composition at the interface between phases in predicting the condensation phenomenon of steam excessively, and in the future, we will review the spray model and perform calculations using other saturation pressure models.

4. Conclusions

In this study, we conducted a CFD analysis of the TOSQAN-101 experiment using OpenFOAM's reactingTwoPhaseEulerFoam solver and a custom turbulent dispersion model. Simulation results indicated the steam condensation was not properly predicted in CFD. To address this issue, future analysis using other saturation models and review of spray model will be performed.

ACKNOWLEDGEMENT

This research was supported by the National Research Foundation of Korea (NRF) funded by the Ministry of Science and ICT.(RS-2022-00144236)

REFERENCES

[1] J. Jung, J. Kim, G. Kim, Methodology Development for 3D Analysis of Spray Cooling in a NPP containment using OPENFOAM, in: Trans. Korean Nucl. Soc. Spring Meet., 2021: pp. 13–15.
[2] J. Kim, U. Lee, S.W. Hong, S.B. Kim, H.D. Kim, Spray effect on the behavior of hydrogen during severe accidents by a loss-of-coolant in the APR1400 containment, Int. Commun. Heat Mass Transf. 33 (2006) 1207–1216.
[3] E. Porcheron, P. Lemaitre, A. Nuboer, V. Rochas, J. Vendel, Experimental investigation in the TOSQAN facility of heat and mass transfers in a spray for containment application, Nucl. Eng. Des. 237 (2007) 1862–1871.
[4] J. Malet, P. Lemaitre, E. Porcheron, J. Vendel, A. Bentaib, W. Plumecocq, F. Dumay, Modelling of Sprays in Containment Applications : Results of the TOSQAN Spray Benchmark (Test 101), First Eur. Rev. Meet. Sev. Accid. Res. (2005) 1–12.
[5] S. Mimouni, N. Mechitoua, A. Foissac, M. Hassanaly, M. Ouraou, CFD modeling of wall steam condensation: Two-phase flow approach versus homogeneous flow approach, Sci. Technol. Nucl. Install. 2011 (2011).
[6] L. Schiller, Z. Naumann, A drag coefficient correlation, 1935.