PBNC 2016
Beijing, China, April 5-9, 2016

A STUDY OF SPRAY APPLICATION BY A FIRE TRUCK FOR THE MITIGATION OF SEVERE ACCIDENT OUTSIDE THE NUCLEAR POWER PLANT

JongWook, Go
Korea Advanced Institute of Science and Technology (KAIST)
Daejeon, Republic of Korea

Irfan, Younus
Korea Advanced Institute of Science and Technology (KAIST)
Daejeon, Republic of Korea

ManSung, Yim
Korea Advanced Institute of Science and Technology (KAIST)
Daejeon, Republic of Korea

KEYWORDS: Spray, Fire truck, Severe Accident, CFD, Nuclear power plant, Source term reduction

ABSTRACT: When a severe accident occurs, large amounts of radioactive materials may be released. This possibility also increases the public's fear of nuclear power plants (NPPs). Also, the economic impact of a severe accident can be huge as shown in the Fukushima accidents. Given these considerations, technologies to prevent the dispersal of these materials are desirable. Technologies to capture the radioactive materials have been studied. One example of such technology is to use spray. Use of spray inside nuclear power plant has been developed with the use of alkaline water. In this study, we propose to use spray technology to collect the materials released outside the plant. Although there are various types of liquids that can be used with spray technology this study investigated water spray as the baseline technology and because sea and fresh water are readably available near NPPs. To deploy the spray technology, use of a fire engine specially equipped with a "boom" that can be extended to reach higher locations was envisioned in this study. The capability of the spray technology was analyzed through Computational Fluid Dynamics (CFD) analysis. By using a CFD modeling-based simulation, this study investigated the effectiveness of fire engine mounted sprays in reducing dispersion of radioactive materials in a severe accident. In particular the study examined the role of the distance of the spray nozzle from the containment and nozzle angle, to improve the effectiveness of the spray.

1. INTRODUCTION

The Fukushima Daiichi nuclear disaster has eroded the public's confidence in the use of nuclear power as a safe source of energy. This is especially true for residents who live near nuclear power plants (NPPs), because they are afraid of being exposed to radiation released during a severe accident. As a result, public acceptance has decreased, and the regulation of nuclear power has become more restrictive. To improve the safety of NPPs, technologies that filter radioactive aerosols, such as Filtered Vented Containment System (FVCS), have been developed and are currently being installed at NPPs. Use of spray technology outside NPPs to prevent dispersion of radioactive substances in a severe accident situation is also proposed [1].

Spray equipment can be suitable for capturing radioactive aerosols because it has been successfully applied in a variety of industrial applications. An important requirement for the successful use of spray technology is the ability to implement the technology at a NPP site in a short timeframe. An added benefit of the quick application of this technology is the positive effect it can have on the public by providing a method to reduce the impact of airborne releases from a NPP. This study is to develop a numerical methodology to investigate the use of fire truck-mounted and its performance in capturing the released radioactive materials. The investigation in this study is limited to a lab-scale environment. The role of the distance of the spray nozzle from the containment and nozzle angle was examined in this study to improve the effectiveness of the spray. To support the study, the ANSYS CFX code was used and the particle source term was modeled to analyze the effectiveness of spray technology in capturing solid particles. This modeled source term was based on the numerical modeling work for the analysis of venturi scrubber performance [2,3].

2. Mathematical modeling

Generally the Eulerian method is used to solve behavior of fluid. However, if particles, such as droplets or solid particles in fluid flow, are considered, it is better to model the behavior of these particles using the Lagrangian method.. If the Eulerian and Lagrangian methods are coupled, particles and fluid flow can both be analyzed at the same time. As analyzing the behavior of sprayed water droplets in fluid flow is important in this study coupling the Lagrangian and Euler method was undertaken.

2.1 Conservation equations

To calculate flow behavior of fluids, the Eulerian method is commonly used. In this case, conservation equations (1) and (2) have to be solved. However, solving these equations is difficult because of the turbulence effect. Therefore, the Reynolds Averaged Navier-Stokes (RANS) equations are proposed to simplify the turbulence problem.

Equation (1) is the continuity equation relative to mass conservation in incompressible flow and at steady state.

$$\nabla \cdot \boldsymbol{U} = 0 \tag{1}$$

To solve the momentum conservation equation in turbulence flow, the following equation (2), which called RANS equation, is used.

$$\frac{\partial U}{\partial t} + \nabla \cdot (\rho \boldsymbol{UU}) = -\nabla P + \rho \boldsymbol{g} + \nabla \cdot \left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \rho U_i' U_j'\right] \tag{2}$$

This research also requires the analysis of the, behavior of air and dust particles. In this study, the particles were assumed to be titanium dioxide (TiO_2) dust in connection with future experiments. It was further assumed that the dust behaves as if it were part of the air. Therefore, the behaviors of air and TiO_2 dust can be solved by using the above equations.

2.2 Turbulence modeling

Turbulent viscosity can be derived using the k-ε model. The model calculates turbulence kinetic energy (k) and dissipation (e). These calculations are based on turbulence kinetic energy and dissipation differential equations (4) and (5) respectively. In essence, these two equations make up the k-ε model. [4]

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\varepsilon + P_{kb} \tag{4}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \varepsilon) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \frac{\varepsilon}{k}\left(C_{\varepsilon 1} P_k - C_{\varepsilon 2 \rho\varepsilon} + C_{\varepsilon 1} P_{\varepsilon b}\right) \tag{5}$$

2.3 Particle transport equation

In the CFD, behavior of sprayed droplets can be calculated by the following equation (6) using the Lagrangian method. The forces influencing the behavior of a particle are drag and buoyancy. The following equation represents these forces on a single droplet. [5]

$$m_P \frac{dU_P}{dt} = \frac{1}{2} C_D \rho_F A_F |\boldsymbol{U_F} - \boldsymbol{U_P}| (\boldsymbol{U_F} - \boldsymbol{U_P}) + \frac{\pi}{6} d_P^3 (\rho_P - \rho_F) \boldsymbol{g} \tag{6}$$

where m_p is mass of a water particle, $\mathbf{U_P}$ is velocity of a water particle, C_D is drag coefficient, ρ_F is density of fluid around a particle, A_F is projection area of a water particle, $\mathbf{U_F}$ is velocity of fluid around a particle, is d_P is diameter of a water particle, ρ_P is density of water, and $\mathbf{g}$ is gravity acceleration.

2.4 Droplet breakup model

Droplets can be deformed and broken easily because droplets are a kind of fluid. To treat deformation and breakup, mathematical models are needed. The Taylor Analogy Breakup (TAB) model was proposed by O'Rourke and Amsden to treat deformation of a sprayed droplet mathematically. The TAB model uses a one-dimensional equation (7) to derive this fluid particle distortion. [6]

$$\ddot{y} = \frac{2\rho_F |\boldsymbol{U_F} - \boldsymbol{U_P}|^2}{3\rho_p r^2} - \frac{5\mu_p}{\rho_p r^2}\dot{y} - \frac{8\sigma}{\rho_p r^3} y \tag{7}$$

where μ_p is the viscosity of a droplet, r is the radius of a droplet, and σ is surface tension of a droplet.

Because one particle (a parent particle) can break up into much smaller particles (child droplets), the Cascade Atomization and Breakup (CAB) model was developed from the TAB model, and used to determine the size of child droplets. The following CAB equation is used to calculate size of child droplets after breakup. [7]

$$\frac{r_{P,Child}}{r_{P,Parent}} = e^{-K_{br} t} \tag{8}$$

where,

$$K_{br} = \begin{cases} k_1 \omega \\ k_2 \omega \sqrt{We} \\ k_3 \omega We^{3/4} \end{cases} \tag{9}$$

$$k_3 = k_2 / We_{t2}^{1/4} \tag{10}$$

$$k_2 = k_1 \frac{\sqrt{1 - C_k C_b / 2C_f We_t}}{\mathrm{acos}(1 - C_k C_b / C_f We_t)} \tag{11}$$

2.5 Collection efficiency

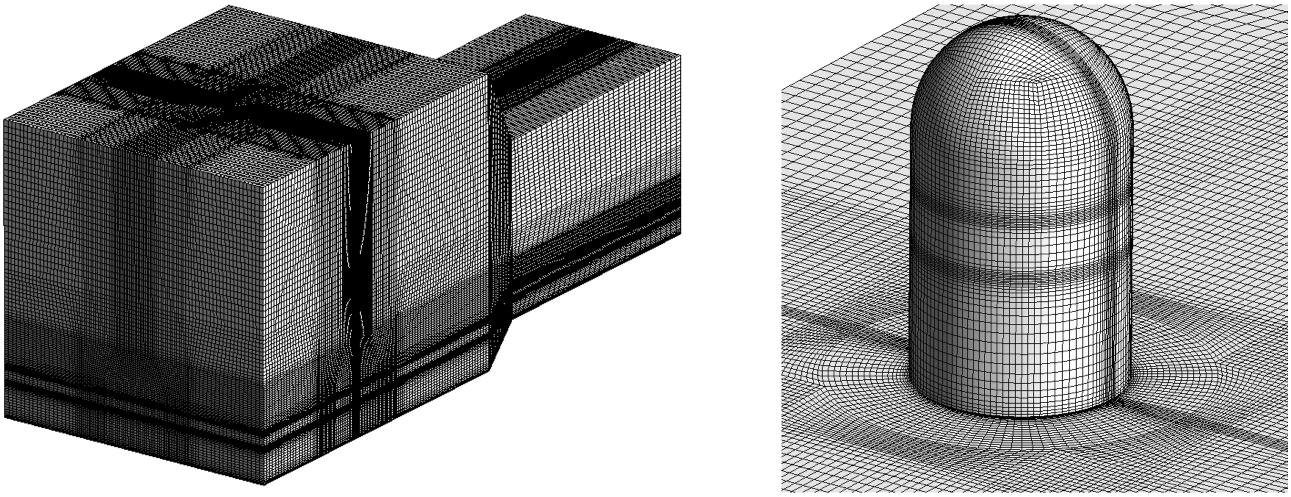
In this study, a key consideration is the collection of solid particles. Therefore, it is required to model the capture of particles. There are three mechanisms to capture solid particles when using droplets: diffusion, interception, and impaction. If the size of a solid particle is larger than 5.0 μm, the effect of diffusion and interception can be neglected because the dominant mechanism for particle capture is impaction. [8] In this study, it is assumed that dust particles are 10 μm. Therefore, the solid particle collection efficiency of a single droplet can be calculated using the following equation (12) for the process of impaction proposed by Calvert. [9]

$$\eta_s = \left(\frac{\psi}{\psi + 0.7}\right)^2 \tag{12}$$

ψ is an inertial impaction parameter defined by equation (13).

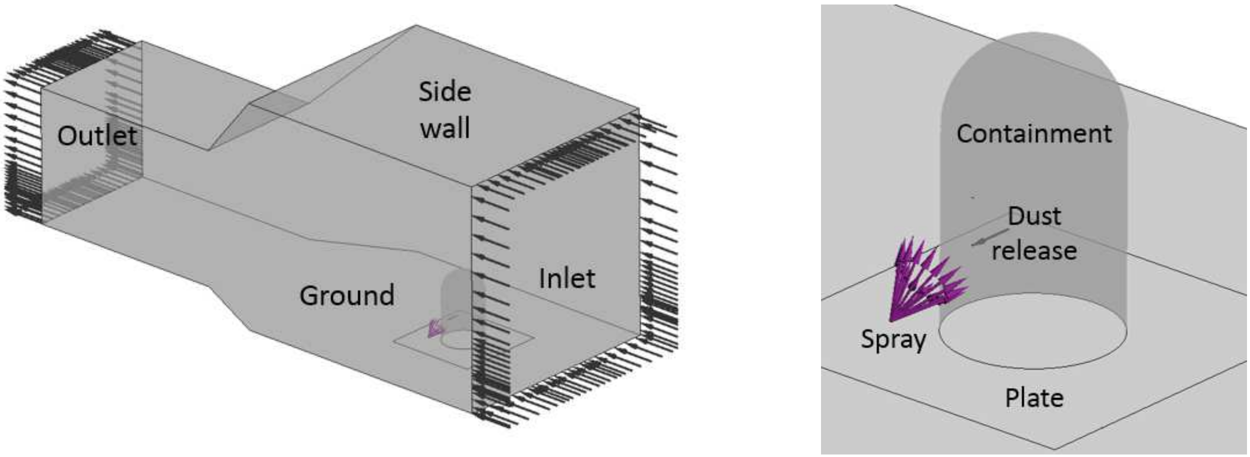
$$\psi = \frac{\rho_P d_S^2 |\boldsymbol{U_F} - \boldsymbol{U_P}|}{9 \mu d_P} \tag{13}$$

where d_s is the diameter of a solid particle and μ is the



(a) **(b)**

Fig 1. Mesh for numerical analysis



(a) **(b)**

Fig 2. Boundary conditions

viscosity of the fluid around that particle.

A single droplet can capture multiple solid particles. This relationship can be calculated by the following equation (14).

$$N_c = \eta_s \frac{\pi d_p^2}{4} |\boldsymbol{U_S} - \boldsymbol{U_P}| \frac{N_s}{dV} \tag{14}$$

where U_s is the velocity of a solid particle, N_s is the number of solid particles, and dV is the element volume.

Particle source term was modeled using the above equations (12), (13) and (14) to simulate the quantity of solid particles captured. And the total collection efficiency was calculated using the following equation (15).

$$\eta_{total} = 1 - \frac{\dot{m}_{out,solid}}{\dot{m}_{in,solid}} \tag{15}$$

where $\dot{m}_{out,solid}$ is mass flow rate to leave the domain, and $\dot{m}_{in,solid}$ is mass flow rate of inflow into the domain.

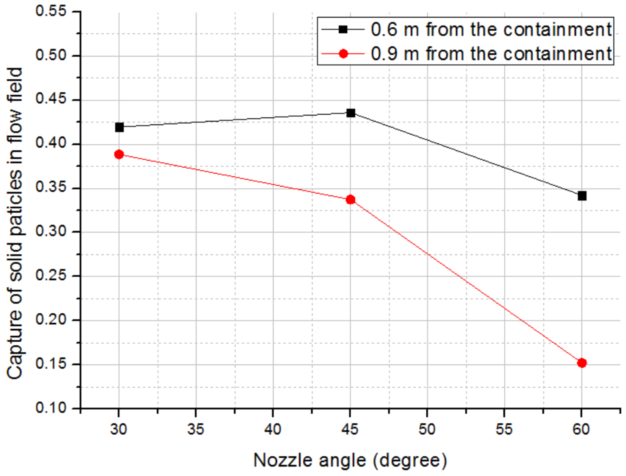


Fig 3. Capture efficiency of TiO_2 in flow field

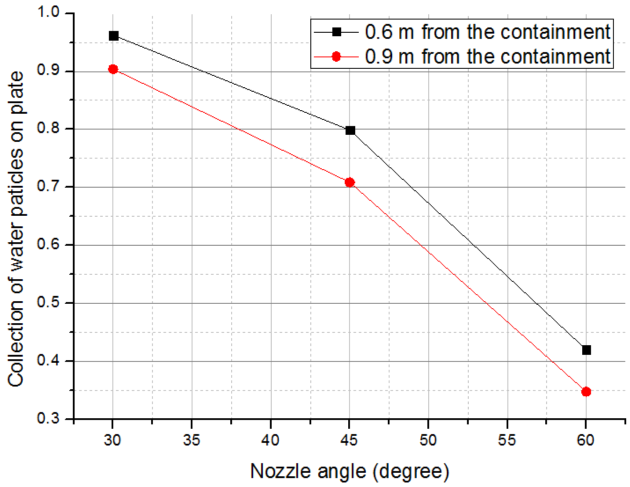


Fig 4. Collection efficiency of sprayed water particles on plate

3. Numerical simulation

3.1 Flow state

To consider the effect of turbulence, the widely used k-ε model was selected. This analysis was performed on an incompressible flow at steady state. Air and water properties were assumed to be constant at 25 °C. By coupling the Eulerian and Lagrangian approaches, we can analyze three phase flow. Water particles were analyzed using the Lagrangian approach, while the Eulerian approach was applied to TiO_2 dust and air. While TiO_2 dust was assumed as part of the air like fluid, the TiO_2 particle diameter was assumed to be 10 µm to solve equation (13) and (14).

3.2 Mesh

Fig 1 shows a mesh for this CFD analysis. To construct an unstructured mesh, ICEM CFD, a software program specifically designed to construct meshes, was used. This domain has 2.19 million hexahedron elements and 2.11 million nodes. To efficiently conduct our analysis a smaller mesh size was used around containment, while a slightly larger mesh size was used in other areas.

3.3 Boundary Conditions

Fig 2 shows the boundary conditions in this simulation. For the inlet boundary, the velocity of air was assumed to be 0.5 m/s. On the outlet boundary, the gauge pressure was set at 0 Pa. On the ground, a no slip condition was applied and a free slip wall condition was applied to the side walls because the viscous effect is not important on side walls. For the dust release region, it was assumed that dust is released at 0.001 kg/s and release velocity is 10 m/s. To simulate water spray, a particle injection model was used. Nozzle angles from the ground were assumed at 30, 45, and 60^{o} and spray distances were assumed at 60 and 90 cm from the containment. The water flow rate was set at 6 liter/min and the sprayed angle from the nozzle was assumed at 55^{o} at the spray point. Fig 2 (b) identifies the spray plate. On the plate, all of the water particles can be absorbed. On the containment wall, it is assumed that water particles can lose all of their momentum due to inelastic collisions.

3.4 Convergence criteria

To decide convergence finish, the RMS residuals and imbalances were considered. If the RMS residuals are lower than 10^{-4}, and if imbalances are lower than 5%, it was assumed that converge was achieved.

4. Results and discussion

4.1 Dust removal efficiency

In a severe accident situation, to minimize damage from radioactive substances, the aerosols have to be captured. Therefore, dust removal efficiency has to be evaluated in this study.

Fig 3 shows the efficiency of TiO_2 removal by water particles in this analysis. At 60 cm from the containment and 30° of nozzle angle, removal efficiency is about 40%. And, at 45° of nozzle angle, the efficiency improved to ~44%. At 60° of nozzle angle, the efficiency dropped to ~35%.

At 90 cm from the containment and 30° of nozzle angle, the result was a little better than the one estimated at 60 cm and 30°. However, the efficiency rapidly decreased with the increase of angle giving much efficiency than at 60 cm. And, at 90 cm and 30°, the efficiency dropped to a value around 15%.

Results indicate that if spray is close to the containment, the removal efficiency is larger in general.

4.2 Collection of water particles

If water particles can successfully capture the radioactive materials released, the spray technology can be considered effective. However, if some portion of the particles is dispersed by the wind, it could be said that the technology has failed to perform the mission. Thus, how well water particles are collected by the spray should be analyzed.

Fig 4 shows the collection efficiency of water particles on plate. At 60 cm from the containment, collection efficiencies are little higher than at 90 cm. And collections efficiencies decreased with the increase of angle. This is because many of the water particles go over the top of the containment and are blown by freestream with increase of nozzle angle.

5. Summary

In this study, the numerical analysis methodology used in evaluating a venturi scrubber was implemented to evaluate the use of spray technology for capturing contaminans released external to the reactor containment. To support the numerical simulation, ANSYS CFX was selected and the mesh was constructed by ICEM CFD. By using the proposed numerical methodology, the efficiency of dust removal and capturing water particles were calculated and evaluated. Summary of the findings is listed below:

1) Capture efficiency of TiO_2 dust by water particles is the highest at 60 cm from the containment and 45º of nozzle angle because many of the points have water particles and TiO_2 dust. Also, the velocity of water particles is higher than other cases.
2) Collection efficiency of water particles on plate is the highest at 60 cm from the containment and 30º of nozzle angle. This is because most of the water particles collide the containment surface.
3) If the capture efficiency of TiO_2 dust and the collection efficiency of water particles are considered together, the best case is at 60 cm from the containment and 30º of nozzle angle.
4) In this study, the numerical modeling, used in analyzing venturi scrubber, was applied for calculating dust removal efficiency in an external flow. The approach seems appropriate to analyze the performance of a spray system installed on a fire engine.

Future work is needed to validate the results based on experimental investigations Additional evaluations need to perform to analyze the effect of water film when modeling the behavior of water particles on the wall. Once the validation work is successful, further analysis is needed to optimize the use of spray technology mounted on a specialized fire truck in a severe accident situation.

ACKNOWLEDGMENTS

This work was supported by the Nuclear Power Core Technology Development of the Korea Institute of Energy Technology Evaluation and Planning (KETEP) under the Ministry of Trade, Industry & Energy, Republic of Korea (No. 20131510400050).

REFERENCES

[1] Ali, M., Yan, C., Sun, Z., Wang, J., & Gu, H. (2013). CFD simulation of dust particle removal efficiency of a venturi scrubber in CFX. Nuclear Engineering and Design, 256, 169-177.

[2] Pak, S. I., & Chang, K. S. (2006). Performance estimation of a Venturi scrubber using a computational model for capturing dust particles with liquid spray. Journal of hazardous materials, 138(3), 560-573.

[3] Younus, I., & Yim, M. S. (2015). Out-containment mitigation of gaseous iodine by alkaline spray in severe accident situation. Progress in Nuclear Energy, 83, 167-176.

[4] Launder, B. E., & Spalding, D. B. (1972). Lectures in mathematical models of turbulence.

[5] Haider, A., & Levenspiel, O. (1989). Drag coefficient and terminal velocity of spherical and nonspherical particles. Powder technology, 58(1), 63-70.

[6] O'Rourke, P. J., & Amsden, A. A. (1987). The TAB method for numerical calculation of spray droplet breakup (No. LA-UR-87-2105-Rev.; CONF-871142-1-Rev.). Los Alamos National Lab., NM (USA).

[7] Tanner, F. X. (2004). Development and validation of a cascade atomization and drop breakup model for high-velocity dense sprays. Atomization and sprays, 14(3).

[8] Kim, H. T., Jung, C. H., Oh, S. N., & Lee, K. W. (2001). Particle removal efficiency of gravitational wet scrubber considering diffusion, interception, and impaction. Environmental Engineering Science, 18(2), 125-136.

[9] Calvert, S. (1970). Venturi and other atomizing scrubbers efficiency and pressure drop. AICHE journal, 16(3), 392-396.

AUTHOR’S INFORMATION

JongWook Go, Korea Advanced Institute of Science and Technology (KAIST), Nuclear Engineering, +8245-350-5876, gowe1234@kaist.ac.kr

Irfan Younus, Korea Advanced Institute of Science and Technology (KAIST), Nuclear Engineering, +8245-350-5876, irfan.pnra@kaist.ac.kr

ManSung Yim, Korea Advanced Institute of Science and Technology (KAIST), Nuclear Engineering, +8245-350-3836, msyim@kaist.ac.kr