Hydraulic Modeling of Novel Combined Nozzle for Pool Scrubbing System

Jeong Hee Ha[a*], Woo Young Jung [a], Doo Yong Lee [a]
[a]Department of Advanced Nuclear Technology and Engineering, FNC Technology, Co., Ltd., South Korea
[*]Corresponding author: hui@fnctech.com

1. Introduction

Pool scrubbing is the one of the fundamental concept for filtration and the most common method for air cleaning [1]. Basic idea is to introduce the gas stream (e.g. air) containing the aerosol or dust with a scrubbing liquid (e.g. water). The particles are collected in the scrubbing liquid. Knowing that the particle is removed from the gas stream mainly by scrubbing phenomena, the filtration efficiency can be enhanced by enlarging the contact area between the scrubbing water and the gas/particle mixture (e.g. break the bulk of gas flow into fine bubbles) or by enlarging the contact time of them (e.g. increase the water pool depth).

In this study, the novel design of the combined nozzle has been developed to minimize the size of the filtration system and to ensure the filtration efficiency over the wide range of operating conditions. The hydraulic simulation has been conducted to investigate the flow behavior inside of the nozzle prior to evaluate the filtering efficiency. The proto-typical combined nozzle has been modeling in CFD and simulated for different conditions.

2. Novel Combined Nozzle

The schematic of the proposed combined nozzle is presented in **Figure 1**. The proto-typical design consists of a converser and a throat, a diffuser and an extra annulus diffuser. The injected gas/particle mixture would be accelerated at the throat and the scrubbing water in the bottom cover would be sucked into the throat due to pressure difference (***self-prime mode***[1]). Due to high velocity, the sucked water would break into small droplets; thus, the surface area to interact with injected particles would be enlarged. Once the mixture and the droplets are out of the diffuser, they have to flow down through the space between inside the diffuser and the annulus diffuser to be ejected to the pool. This is the one advantage of the annulus diffuser, i.e. doubles the flow path length which means increasing the residence time of the particles and bubbles in the pool. Another benefit of the annulus diffuser is that the possibility of scrubbing in the pool can be enhanced. This is because the nozzle outlet is located at the lowered position and thus, the distance from the outlet to water surface would be increased compared to the design without the annulus diffuser.

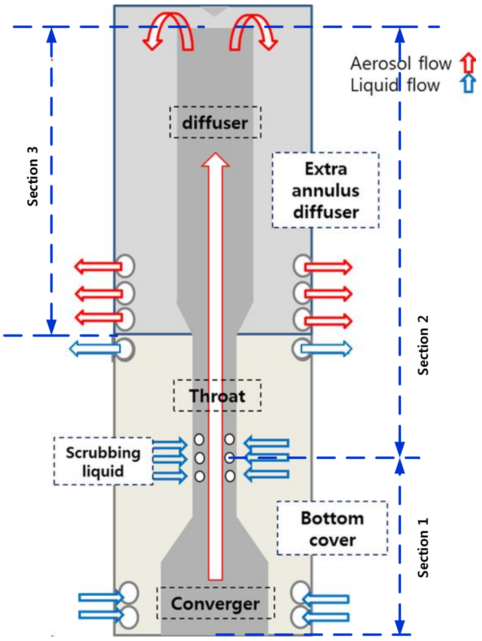

Figure 1. A schematic of the Combined Nozzle

3. CFD Modeling and Simulation

The combined nozzle is modeled by using ANSYS-CFX [2]. In order to verify the design (i.e., bottom cover) and investigate the flow behavior (i.e. pressure, velocity), the four cases in **Table I** are examined. As a reference case, Case 1 is simulated to observe the pressure and velocity changes within the combined nozzle. Focusing on the injected gas stream flow changes, the liquid suction through holes on the throat is not considered. Case 2 and Case 4 are compared to see the effect of the nozzle operation mode, i.e. active mode or self-prime mode. By comparing Case 3 to Case 4, one can examine the effects of the bottom cover.

Table I: Descriptions for the Cases of CFD simulations

Case Descriptions	
Case 1	Single phase without Bottom cover
Case 2	Two phase without Bottom cover , active mode
Case 3	Two phase with Bottom cover, self-prime mode
Case 4	Two phase without Bottom cover, self-prime mode

---

[1] Contrary to the self-prime mode, the scrubbing water can also be injected by a pump or compressor. This can be called the ***active mode***. While the self-prime mode doesn't require any electricity, the active mode has to be provided with external power.

The main fluid passing through the combined nozzle is set as a gaseous material with following properties [3].

- Molar mass : 152 g/mole
- Density : 3.33 kg/m^3
- Specific heat : 2446 J/kg/K
- Dynamic viscosity : 1.6E-05 Ns/m^2

Initial pressure of the system is 1.01325 bar for Case 1 and 2. For Case 3 and 4, water head in the pool is considered as the initial pressure. The heat transfer is not reflected. The mass flow rate of the gas stream is set to 0.4 kg/s. The atmospheric pressure boundary condition is applied for the outlet of the nozzle. The free slip boundary is imposed for the disperse phase and a no-slip boundary condition is applied for the continuous fluid at the walls. Especially, in the active mode i.e. Case 2, water is injected into the combined nozzle through the holes on the throat with the velocity of 0.2 m/s.

4. Simulation Results

The simulated results show that the combined nozzle presents a general nozzle behavior. For example, the pressure is decreased at the throat and recovered at the diffuser. **Figure 2** shows the pressure drop between the inlet and other the locations:

$$\Delta P_i = P_i - P_{inlet}$$

where P_i is the pressure at the center of the throat, the outlet of diffuser, top of the nozzle and the outlet of the nozzle. By comparing the results of Case 1 and 2, one can notice that the water injection at the throat causes the increase of the pressure drop. The comparison between the results of Case 2 and 4 presents the effects of water injected through the holes at the throat, i.e. the more the water is injected, the less the pressure drop would be.

By comparing the results of Case 3 and Case 4, the effects of the bottom cover can be investigated. As can be seen in **Figure 3** and **Figure 4**, the bottom cover forms the inside flow circulation, i.e. an upward flow is formed inside of the cover and the water is drained through the holes located at the upper part of the bottom cover. This results in increased water injection (**Table II**). On the other hand, in case of Case 4, no such circulation is observed.

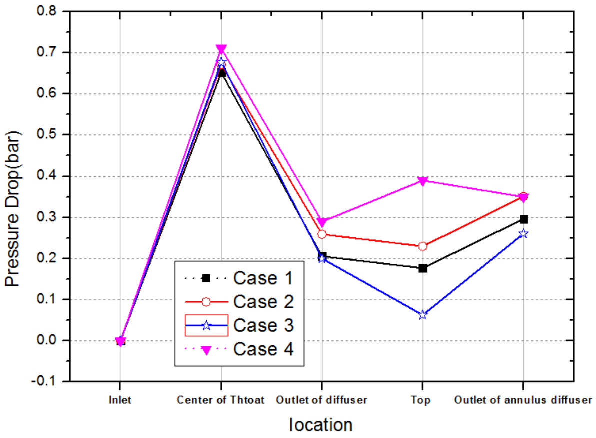

Figure 2. Pressure Drop on the Combined Nozzle

Table II: Average mass flow rates of liquid at throat

Elevation (m)	Case 3 (with bottom cover)		Case 4 (without bottom cover)	
	Mass flow (kg/s)	Velocity (m/s)	Mass flow (kg/s)	Velocity (m/s)
0.188	6.21E-04	0.09	4.03E-04	0.057
0.176	5.51E-04	0.08	3.73E-04	0.053
0.164	5.12E-04	0.07	3.30E-04	0.047
0.152	3.00E-04	0.04	1.70E-04	0.024
0.14	2.68E-04	0.04	1.39E-04	0.020

* Elevation from the inlet of the nozzle (EL. Inlet of nozzle = 0.0 m)

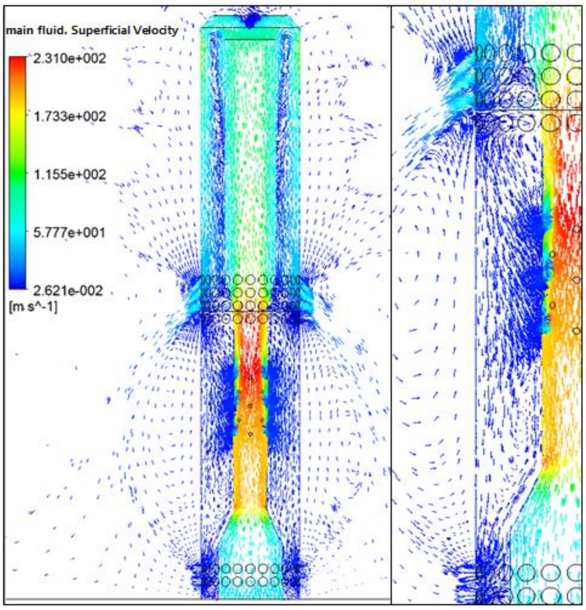

Figure 3. Velocity field of Case 3

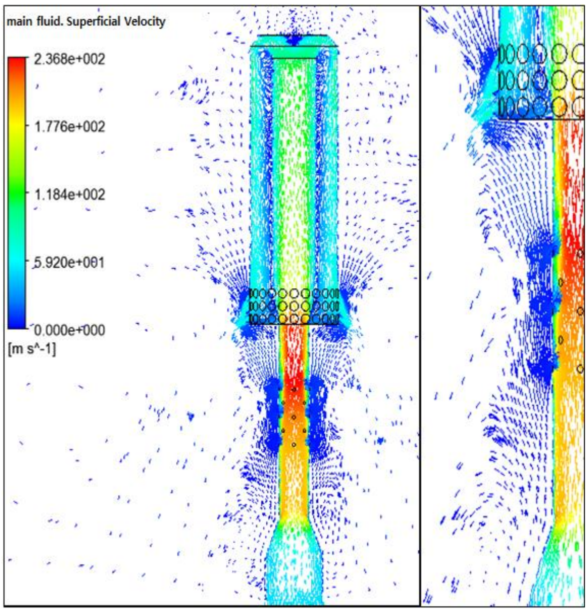

Figure 4. Velocity field of Case 4

5. Conclusions

Preliminary CFD simulation is conducted to examine the effects of key features and the flow behaviors inside the combined nozzle. It is observed that the proposed combined nozzle can be operated in the self-prime mode and bottom cover can enhance the water injection at the throat. This would be the basis for the further CFD analysis in details and experimental studies.

ACKNOWLEDGEMENT

This work was supported by the Nuclear Research & Development of the Korea Institute of Energy Technology and Planning (KETEP) grant funded by the Korea government Ministry of Trade, Industry and Energy. (No. 20131510101700)

REFERENCES

[1] William C. Hinds, Aerosol Technology – Second Edition, John Wiley & Sons, Inc., 1999
[2] ANSYS-CFX 11.0 Manual, 2007
[3] Youngsuk Bang, et al., “Severe Accident Analysis for Containment Filtered Venting System Design”, Transactions of the Korean Nuclear Society Spring Meeting, Jeju, Korea, May 29-30, 2014