Transactions of the Korean Nuclear Society Autumn Meeting
Gyeongju, Korea, October 29-30, 2015

Numerical Analysis on Behavior of Droplet in Venturi Scrubber

W. Y. Choi [a*], D. Y. Lee [a], Y. S. Bang [a]
[a]*FNC Technology Co., Ltd.,32Fl., 13 Heungdeok 1-ro, Giheung-gu Yongin-si, Gyeonggi-do, 446-908, Korea*
[*]Corresponding author: choiwy@fnctech.com

1. Introduction

Venturi Scrubber has been well known and widely used as an efficient filter component. It consists of three part: reducer, throat and diffuser. The gas mixture containing the dust would be accelerated as flow through the reducer while the pressure would be decreased. At throat, the velocity of the gas would be at maximum and the pressure would be the lowest. Due to pressure difference between inside and outside of the throat, the liquid submerging the venture scrubber would be sucked and atomized. As the gas flow through the diffuser, the pressure would be recovered and the dust in the gas mixture would be captured by the atomized liquid droplets.

In this process of dust removal in venture scrubber, atomization (i.e. breakup of liquid droplet in the venturi scrubber) is crucial for filtering efficiency. In order to maintain the high efficiency, the injected liquid should be atomized into fine droplets and well spread. Because of its importance, the experimental study has been conducted by many researchers. However, numerical study has not been conducted extensively. As a preliminary study for estimating filtration efficiency of venturi scrubber by numerical tools, the behavior of droplet inside the venturi scrubber is simulated.

2. Methods

In this section some of the techniques used to model the venturi scrubber are described as below.

2.1 Geometry

Venturi scrubber consists of three parts, which are reducer, throat and diffuser, and there are some holes in both sides of throat segment. The cross-sectional shapes of reducer, throat and diffuser are circular, rectangular and circular form, respectively [1]. Geometry of venturi scrubber is modeled as in Fig. 1.

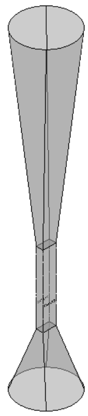
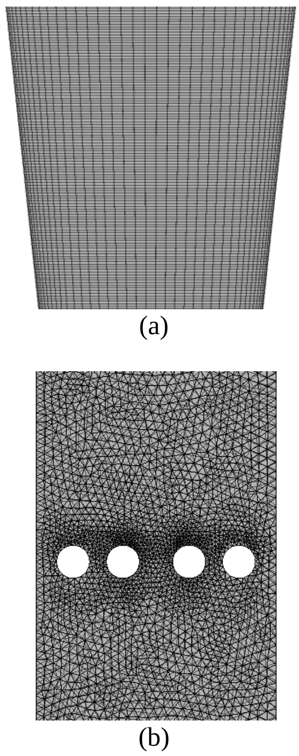


Fig 1. Geometry of venturi scrubber

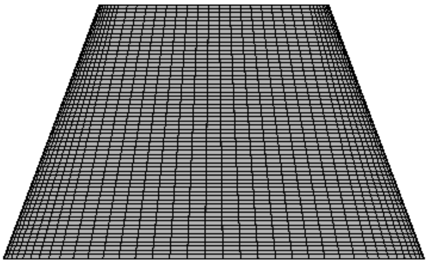
2.2Mesh

For both the reducer and the diffuser of venturi scrubber, hexahedral meshes are applied. For the throat, tetrahedral meshes are applied to model the injection through the holes in the throat which requires finer mesh. The mesh distribution is shown in Fig. 2.



(a)

(b)



(c)

Fig.2 (a) mesh at diffuser, (b) mesh at throat and (c) mesh at reducer

2.3 Mathematical modeling

2.3.1 Governing equation

The continuity equation is then:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0 \quad (1)$$

and the momentum equation becomes:

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial P'}{\partial t} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_m \quad (2)$$

where S_m is the sum of body forces, μ_{eff} is the effective viscosity accounting for turbulence, and P' is the modified pressure[2]

2.3.1 Droplet breakup model

The droplets are deformed due to turbulence of gas and external aerodynamic forces acting on the liquid. In this paper, cascade atomization and breakup model (CAB model) is used and can be expressed as

$$\ddot{y} = \frac{C_f \rho_g V^2{}_{rel}}{C_b \rho_d r_d^2} - \frac{C_k \sigma_d}{\rho_d r_d^3} - \frac{C_D \mu_d}{\rho_d r_d^2}\dot{y} \quad (3)$$

It is assumed that the rate of child droplet generation, *dn(t)/(dt)*, is proportional to the number of child droplets:

$$\frac{dn(t)}{dt} = 3K_{br} n(t) \quad (4)$$

The following equation is used to determine the child droplet size after breakup:

$$\frac{r_{d,parent}}{r_{d,child}} = e^{-K_{br}t} \quad (5)$$

The constant K_{br} depends on the breakup regime and is given as:

$$K_{br} = \begin{cases} k_1\omega & 5 < We < 80 \\ k_2\omega\sqrt{We} & 80 < We < 350 \\ k_3\omega We^{3/4} & 350 < We \end{cases} \quad (6)$$

where

$$k_2 = k_1 \frac{\sqrt{1 - \frac{1}{2}\left(\frac{C_k C_b}{C_f We_t}\right)}}{a\cos\left(1 - \frac{C_k C_b}{C_f We_f}\right)} \quad (7)$$

$$k_3 = k_2 / We_{t2}^{1/4} \quad (8)$$

2.4 Numerical methodology

2.4.1 Turbulence model

Eulerian-Lagrangian method for three phase flow is applied in this study. And steady state is employed in computation. k-e turbulence model is used for gas. The gas is applied as continuous field, but the dust particle as transport solid and the liquid as transport fluid in the domain.

2.4.2 Boundary condition

The boundary conditions for CFD model are defined as follows: the gas velocity at inlet is applied, while the dust is injected with particle transport solid with mean diameter of 0.001mm. The liquid is injected as particle transport fluid with velocity boundary condition at the throat. The pressure boundary condition is treated at the outlet.

3. Results

Pressure distribution insider the venture scrubber is presented in Fig. 3, which shows that the pressure loss at the reducer and recovery at the diffuser. Fig. 4 shows the liquid velocity distribution in the throat. Due to the pressure difference inside and outside of the throat, the liquid would be sucked and injected through the holes. The behavior that the liquid is injected through the holes, accelerated by the gas flow and atomized into small sized particles has been observed. It can be concluded that the simulation using CFD would be useful to investigate the behavior of the droplets and flow characteristics inside of the venture scrubber.

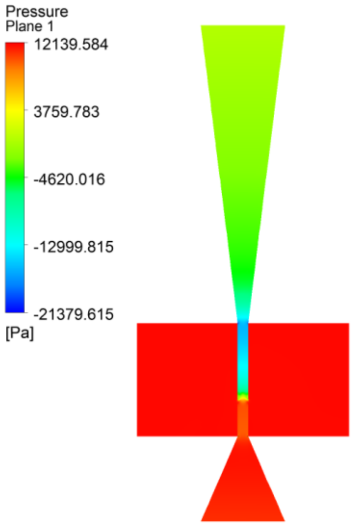


Fig. 3 Pressure Contour in Venturi Scrubber

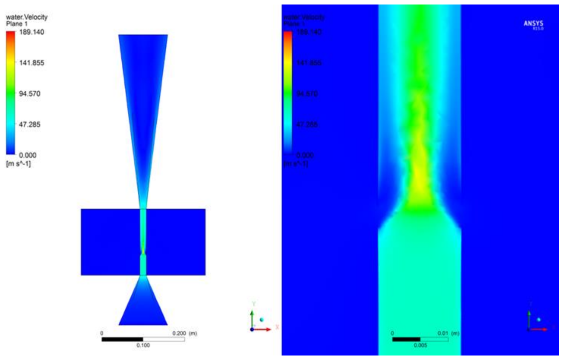


Fig. 4 Contour of Liquid Velocity

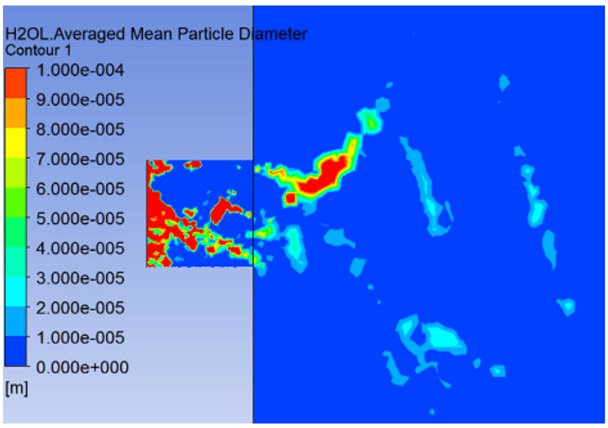


Fig. 5 Distribution of Liquid Particle Size

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. 20141510101680)

REFERENCES

[1] M. Lehner, Aerosol Separation Efficiency of a Venturi Scrubber Working in Self-Priming Mode, Aerosol Science and Technology, Vol. 28, No. 5, pp. 389~402, 1998.

[2] Ansys CFX-Solver Theory Guide. ANSYS Inc. 2013.

[3] M. Ali, C. Yan, Z. Sun, J. Wang, H. Gu., CFD Simulation of Dust Particle Removal Efficiency of a Venturi Scrubber in CFX, Nuclear Engineering and Design, Vol. 256, pp. 169-177