Investigation of dust particle removal efficiency of Self-priming venturi scrubber using CFD

Sarim Ahmed, Hassan Mohsin, Kamran Qureshi, Ajmal Shah, Waseem Siddique, Khalid Waheed, Naseem Irfan, Masroor Ahmad, Amjad Farooq

PII: S1738-5733(17)30032-3
DOI: 10.1016/j.net.2018.01.016
Reference: NET 502

To appear in: *Nuclear Engineering and Technology*

Received Date: 18 January 2017
Revised Date: 22 December 2017
Accepted Date: 20 January 2018

Please cite this article as: S. Ahmed, H. Mohsin, K. Qureshi, A. Shah, W. Siddique, K. Waheed, N. Irfan, M. Ahmad, A. Farooq, Investigation of dust particle removal efficiency of Self-priming venturi scrubber using CFD, *Nuclear Engineering and Technology* (2018), doi: 10.1016/j.net.2018.01.016.

INVESTIGATION OF DUST PARTICLE REMOVAL EFFICIENCY OF SELF-PRIMING VENTURI SCRUBBER USING CFD

Sarim Ahmed[1], Hassan Mohsin[1], Kamran Qureshi[1], Ajmal Shah[2], Waseem Siddique[2], Khalid Waheed[1], Naseem Irfan[1], Masroor Ahmad[1], Amjad Farooq[1]

[1] Centre for Studies on Emerging Reactor Technologies (CERT), Department of Nuclear Engineering PIEAS, P. O. Nilore, Islamabad, Pakistan

[2]Department of Mechanical Engineering PIEAS, P. O. Nilore, Islamabad, Pakistan

*Corresponding Author Email: waseem@pieas.edu.pk

Abstract: A venturi scrubber is an important element of filtered containment venting system (FCVS) for the removal of aerosols in contaminated air. The present work involves computational fluid dynamics (CFD) study of dust particle removal efficiency of a venturi scrubber operating in self-priming mode using ANSYS CFX. Titanium oxide (TiO_2) particles having sizes of 1 micron have been taken as dust particles. CFD methodology to simulate the venturi scrubber has been first developed. The Cascade atomization and breakup (CAB) model has been used to predict deformation of water droplets, while the Eulerian-Lagrangian approach has been used to handle multiphase flow involving air, dust, and water. The developed methodology has been applied to simulate venturi scrubber geometry taken from the literature. Dust particle removal efficiency has been calculated for forced feed operation of venturi scrubber and found to be in good agreement with the results available in the literature. In the second part, venturi scrubber along with tank has been modelled in CFX and transient simulations have been performed to study self-priming phenomenon. Self-priming has been observed by plotting the velocity vector fields of water. Suction of water in the venturi scrubber occurred due to the difference between static pressure in the venturi scrubber and the hydrostatic pressure of water inside the tank. Dust particle removal efficiency has been calculated for inlet air velocities of 1 m/s and 3 m/s. It has been observed that removal efficiency is higher in case of higher inlet air velocity.

Keywords: Venturi scrubber, Self-priming venturi scrubber, FCVS, Dust particles, CFD

1. Introduction

Nuclear power plants (NPPs) operating around the globe follow an advance design philosophy that provides a very low level risk to the environment and the public. The core damage frequency of a nuclear reactor is on the order of 10^{-4} to 10^{-6} per reactor year [1]. NPPs are equipped with many safety systems that are designed to cope with design basis accident (DBA) and ensure mitigation of consequences of severe accidents (SA).

Maintaining the integrity of reactor containment in cases of SA is very important. SA in nuclear reactors lead to continuous release of fission fragments, gases, and steam inside the containment. The most important fission products are Iodine-131 and Cesium Iodide (CsI). Due to continuous release of fission products and steam, the inside pressure of the containment rises continuously. Out of four physical barriers in an NPP, the reactor containment is considered to be the last barrier to prevent the release of radioactivity to the environment. The pressure rise might damage the integrity of the containment once it exceeds the design pressure limit. The Fukushima incident in Japan had a similar cause in that there were explosions due to hydrogen gas buildup inside the containment [2]. To avoid this damage and the possible release of hazardous materials to the public, many NPPs around the world are equipped with Filtered Containment Venting System (FCVS) and many countries are emphasizing the employment of such a system, especially after the Fukushima event [3]. In an FCVS, radioactive effluents (e.g. aerosols, elemental iodine, and organic iodide) contained within the air are scrubbed using a scrubbing solution in a scrubber tank that holds sodium hydroxide (NaOH) and sodium thiosulphate ($Na_2S_2O_3$); clean air is allowed to exit from a separate stack installed in the NPP at the top of scrubber tank; a metallic fiber filter is installed that removes submicron sized aerosols that are not scrubbed in the solution. Various designs of FCVS have been proposed and can be broadly categorized into devices using dry method and wet method [3].

Due to the high efficiency advantage of wet scrubbers, they are being widely used around the globe. In a wet scrubber, a venturi scrubber is an important element. It can be either partially submerged or fully submerged [2] [4]. Venturi scrubbers work on the principle of achieving high velocity at the throat section through the converging section, which has a continuously decreasing cross-section. This high velocity is then helpful in generating droplets of scrubbing solution for the efficient collection of aerosols. At the end, the diffuser section (with a gradually increasing cross-section) allows the deceleration of the fluid for pressure recovery. The venturi scrubber is very efficient in collecting fine aerosols and dust particles from the purging gas for gas cleaning purposes. The venturi scrubber can operate in two ways, either in force feed or in self-priming mode. The difference between the two can be seen in Figure 1. In the force feed scenario, pumps are used to supply the scrubbing solution to the venturi scrubber, while in the case of self-priming, the pressure difference between the hydrostatic head of the tank and the static pressure inside the throat forces the fluid to flow into the venturi scrubber. In both cases, liquid after being injected inside the domain is transformed into a stream of fine droplets that increases the removal rate of aerosols.

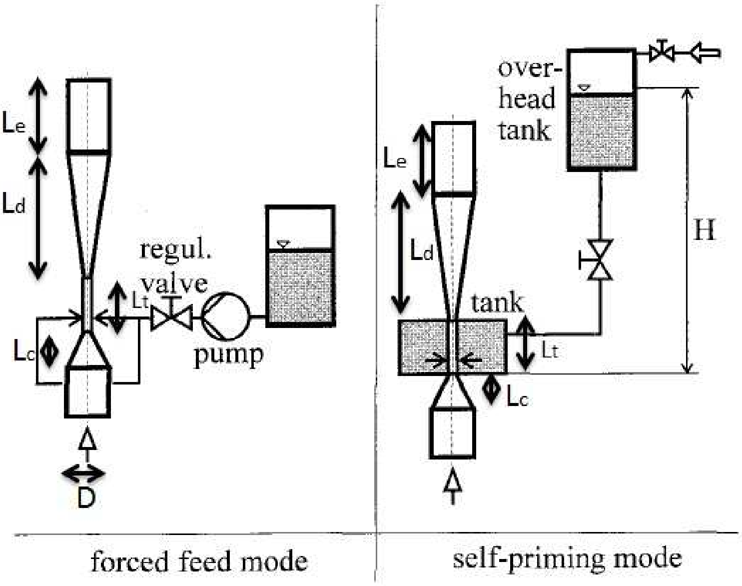

Figure 1: Working modes of venturi scrubber [5]

The present work is concerned with CFD investigation of the dust removal efficiency of the venturi scrubber. To perform simulations, CFD methodology has first been developed. TiO_2 particles are used as dust particles and water is used as the scrubbing solution. Then, the methodology is applied to calculate dust removal efficiency of the force feed venturi scrubber geometry given by Pak and Chang [6], and results are compared. At the end, methodology is applied to estimate the dust particle removal efficiency of the self-priming venturi scrubber at different inlet air velocities. In the past, to estimate the dust removal efficiency, researchers have performed CFD simulations of the force feed venturi scrubber. However, investigation of the dust removal efficiency in a self-priming venturi scrubber using CFD has in this study, to the best of authors' knowledge, been performed for the first time. The results obtained from this research work are believed to increase the knowledge of the research community working in this important area.

2. Previous Work

As far as CFD simulation of dust particle removal efficiency of the force feed venturi scrubber is concerned, a lot of work has been done in the past.

Pak and Chang [6] in 2006 investigated the removal efficiency of dust particles from air passing through a force feed venturi scrubber. A model was developed in the KIVA code. The Eulerian-Lagrangian approach was used in simulation. The Basset–Boussinesq–Oseen (B–B–O) equation was used to predict the particle motion. Dust particles were considered as spherical and drag on deformable water droplets was predicted using the correlation of Schmehl et al. [7].

Majid Ali [8] investigated the dust particle removal efficiency of a force feed venturi scrubber using ANSYS CFX. The Eulerian-Lagrangian approach was used in the simulation. The Cascade atomization and breakup model (CAB) was used to predict the liquid droplet breakup.

3. Methodology

The Eulerian-Lagrangian approach has been used to model the multiphase (air+water+dust) flow. Hydrophobic TiO_2 particles are taken as dust particles to simulate insoluble aerosols produced within the containment due to interaction of corium and concrete after a core meltdown accident. The Lagrangian approach is used to track water and dust particles, while air is modelled through the Eulerian approach and is taken as a continuous fluid in the case of the forced feed venturi scrubber. However, in the case of a self-priming venturi scrubber with a tank filled with water, water has been taken as a continuous fluid, while air has been treated as a dispersed fluid with dust particles as a dispersed solid. The RNG K- ε model has been used to capture turbulent effects. The Schiller Nauman model has been used to estimate the particle drag coefficient [9]. The Cascade atomization and breakup (CAB) model has been used to simulate the deformation of water droplets [9]. This model models the breaking of water coming into the venturi domain into small droplets due to the aerodynamic forces of air. These are the droplets that are then used to capture dust particles. The CAB model treats the aerodynamic force of air as the applied force, surface tension as the restoring force, and the viscosity of water as the damping force. This way the breaking of water into fine droplets is treated as a mass-spring system. The CAB model also takes into consideration the differing sizes of the water droplets and their further breakup into daughter droplets. By using the CAB model, we can successfully simulate the breakup and hence the dust particle removal, as dust is captured in these droplets of water. Removal efficiency of dust is calculated using an inertial impaction mechanism, for which single particle collection efficiency is defined by Calvert [10]:

$$\eta = \left(\frac{\Psi}{\Psi+0.7}\right)^2 \tag{1}$$

where Ψ denotes Stoke's number, defined by Pak and Chang [6]:

$$\Psi = \frac{\rho_p d_p^2 (v_p - v_d)}{9\mu_g d_d} \qquad (2)$$

where

ρ_p = Density of particle
d_p = Diameter of particle
v_p= Velocity of particle
v_d= Velocity of droplet
μ_g= Viscosity of gas
d_d= Diameter of droplet

4. Simulation of force feed venturi scrubber

Geometry

The dimensions of the venturi scrubber are taken from Pak and Chang [6] and given in Table 1.

Table 1: Dimensions of venturi scrubber

Characteristics	Dimension
Pipe diameter (D) (Inlet & Outlet)	0.192 m
Inlet cylinder length (L_e)	0.1 m
Converging section length (L_c)	0.253 m
Throat length(L_t)	0.14 m
Throat diameter (d_t)	0.07 m
Diverging section length (L_d)	0.997 m
Exit cylinder length (L_o)	0.112m
Number of orifices	12
Orifice diameter	0.0025m

Twelve orifices are used in the throat section, from which water is introduced into the domain. Pak and Chang [6] made this geometry using the KIVA code; however, in the present case, the ANSYS 14.0 design modeler has been used. The geometry is made in a vertical direction, with the flow in the upward direction. The geometry of the venturi scrubber is shown in Figure 2 and Figure 3.

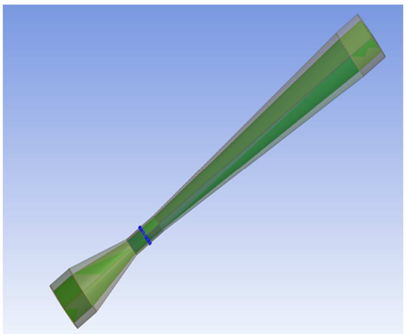

Figure 2: Overall geometry

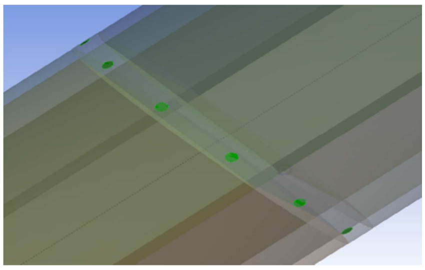

Figure 3: Orifices for water injection

Mesh

A combination of structured and unstructured mesh has been generated using ANSYS ICEM. Mesh around throat section is shown in Figure 4, while overall mesh of the domain is shown in Figure 5. Mesh independent study has also been performed and results got mesh independent at 95000 mesh elements.

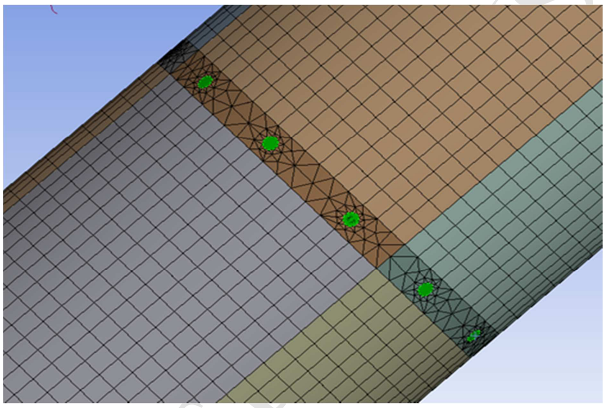

Figure 4: Regions of tetrahedral elements on throat

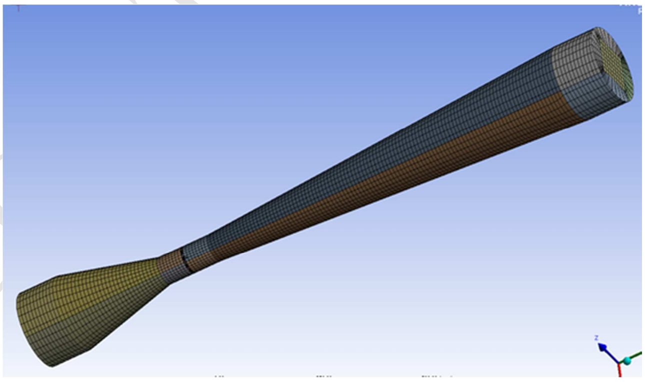

Figure 5: Overall mesh

Boundary conditions

At the inlet, gas mass flow is given. Dust particles having sizes of 1 micron are introduced as particle transport solid, while water is injected as particle transport fluid. At the outlet, constant pressure boundary condition is specified. Simulations are performed for two values of liquid to gas ratio i.e. 2 and 2.5 litre/m^3.

Solver

The high resolution advection scheme is selected due to its robustness [8].

Convergence criteria

During simulations the convergence criteria are set at 10^{-4} RMS for residuals.

Results

Removal efficiency is calculated using the Stokes number. First, the Stokes number is calculated from the post processing data; then, Equation 1 is used to calculate the removal efficiency. The comparison is shown in table 2. It can be observed that the results are almost the same as those predicted by Pak and Chang.

Table 2: Comparison of removal efficiency

Liquid to gas flow ratio (Litre/m^3)	**Efficiency (Present simulation) %**	**Efficiency (Simulation) given by Pak and Chang %**	**Efficiency (Experimental) given by Pak and Chang %**
2	97	97	99.2
2.5	98	97.5	99.3

Effectiveness of CAB model

The CAB model is a breakup model used to simulate the breakup of water into small droplets. The effectiveness of the CAB model was checked by running one simulation for the case of flow of 2 L/m^3 without using the CAB model. The comparison of the results is shown in Table 3. It can be seen that the CAB model improves the removal efficiency.

Table 3: CAB model effectiveness by efficiency comparison

Liquid to gas flow ratio (Litre/m^3)	**Efficiency with CAB model %**	**Efficiency without CAB model %**
2	97	91

5. Simulation of self-priming venturi scrubber

When switching from force feed to self-priming venturi, there arises a difference in the morphology of the phases. This is because, for the force feed case, the venturi is not submerged in water, i.e. initially there is no water present in the venturi; only when air flows through the venturi is water forced to flow through the orifices. Hence the use of the Eulerian approach for tracking air. In the self-priming case, water is present around the venturi even when no flow of air is provided at the inlet, making water the continuous phase and warranting the use of the Eulerian approach to track water.

In the case of the force feed venturi scrubber, the water flow rate was applied as a boundary condition at the water inlet. However, in the self-priming case, nothing is specified at the water inlet as a boundary condition. In fact, a tank is made around the venturi scrubber; tank is filled with liquid (water) domain. When the velocity of air at the throat reaches the maximum value, automatic suction of water takes place due to the pressure difference. Dimensions for the venturi are summarized in Table 3. The height of the tank has been set at 910 mm.

Table 3: Dimensions of self-priming venturi

Characteristics	Dimensions
Inlet cylinder length (L_e) = Pipe diameter (D)	30 mm
Converging section length (L_c)	30 mm
Throat length (L_t) = Throat diameter (d_t)	20 mm
Diverging section length (L_d)	80 mm
Exit cylinder length (L_o)	20 mm

Geometry

The geometry of the self-priming venturi scrubber has been created using an ANSYS design modeler. The tank has been made around it using revolve command. For this, a sketch has been made in such a way that there is a gap between the venturi wall and the tank; this gap serves as the wall thickness, as shown in Figure 6. The geometry of the venturi was changed from the forced case because the dimensions of the geometry used by Pak and Chang were quite large, i.e. in meters. This requires very fine mesh and more computational time. Such expensive calculations were made for the force feed case, in which experimental results were available for validation. However, for the self-priming case, there are no experimental results available as of now. Hence this research was used to establish the physics for the self-priming case by using a geometry that would help the solver to make the calculations. In conclusion, it was found that the modelling parameters worked well for the self-priming case, provoking no issues in terms of the physics of the problem.

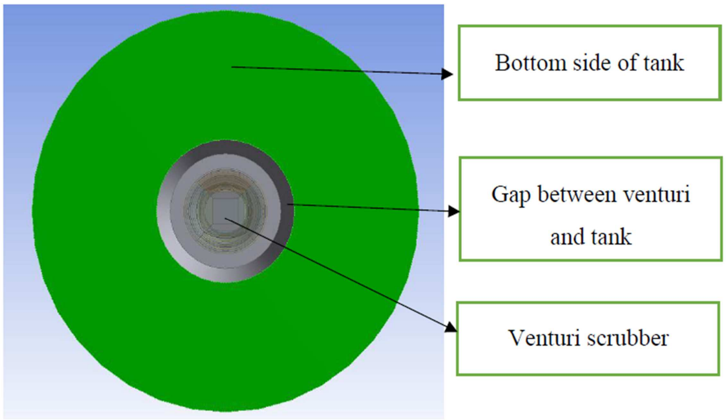

Figure 6: Bottom view of self-priming venturi scrubber

Figure 7 shows the inside boundary of the tank; the colored portion is the wireframe of the venturi scrubber. In the case of the self-priming venturi scrubber, a water inlet zone of 1 mm has been provided around the end of the throat section, as shown in Figure 8.

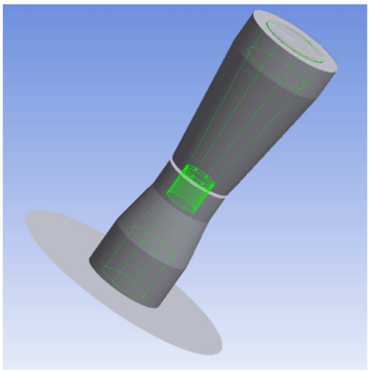

Figure 7: Inner boundary of liquid tank

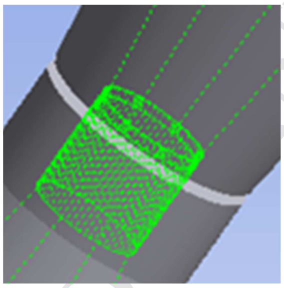

Figure 8: Passage for liquid injection

The final geometry of self-priming venturi scrubber is shown in Figure 9.

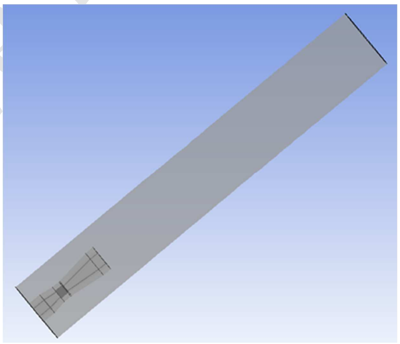

Figure 9: Final geometry of self-priming venturi scrubber

Mesh

The mesh of self-priming venturi scrubber with tank is shown in Figure 10. Mesh independent study was also performed and the results became independent of the mesh at around 2.3 million nodes, as shown in Figure 11.

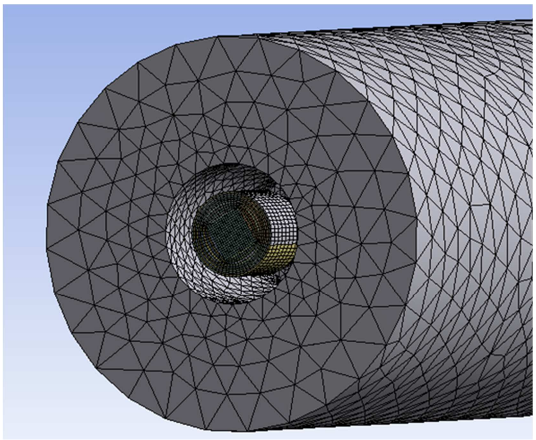

Figure 10: Mesh for complete geometry of self-priming venturi scrubber

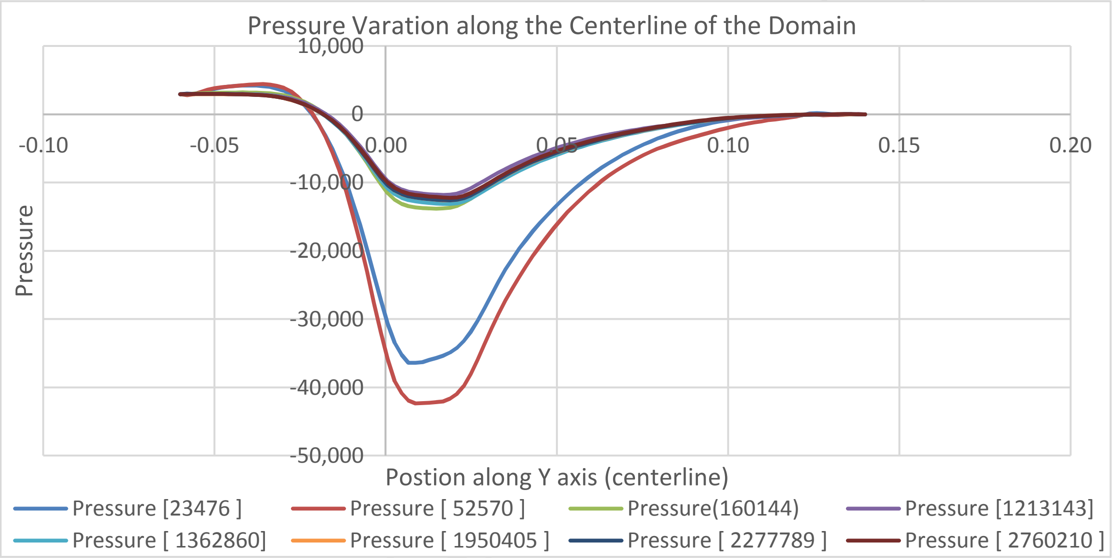

Figure 11: Mesh sensitivity study of self-priming venturi

Boundary conditions

Air and dust flow rates are specified at the inlet of the venturi, just as in the case of the force feed venturi. In this case, to simulate automatic suction of water from the tank into the venturi domain, no conditions are specified at orifices of the venturi. Atmospheric pressure condition is specified at the outlet.

Solver

High resolution advection scheme and steady state conditions are selected in the present case. (Transient simulations were to be performed after these steady state simulations)

Results

The morphology of phases was changed because of the submerged orientation of venture. Water acts as a continuous phase as it is present all around, even before the flow of air through inlet. Air is treated as a particle transport fluid. To see if the CAB model can still be used with the morphology of the phases changed, a comparison was made between the removal efficiency of the self-priming venturi with and without the CAB model. The results of this comparison are given in Table 4 below.

Table 4. Effectiveness of CAB model in self-priming case.

Removal efficiency with CAB model (%)	Removal efficiency without CAB model (%)
99.97	99.97

It is obvious that the CAB model has no effect on the removal efficiency in this case. This is because water is the continuous phase and is present everywhere in the domain. The CAB model in this case is being applied to air. This renders the breakup model ineffective in terms of dust removal. Another major finding from this comparison is the high values of removal efficiency even without using the CAB model. These high values of efficiency are due to the presence of water both inside and outside the venturi. Dust, upon encountering this presence of water, starts to slow down and remain suspended in water. This keeps most of the dust from escaping the scrubbing tank, which is filled with water. Ultimately this dust settles down inside the tank.

These findings motivated a change in morphology of air from particle transport fluid to dispersed fluid, to avoid the application of the CAB model to air. Keeping air in an Eulerian phase instead of a Lagrangian phase will help in water droplet formation. To do so, and at the same time by taking water as a continuous phase, the Eulerian-Eulerian model should be employed. This application along with its results is given in the sections below.

Boundary conditions (Eulerian-Eulerian approach)

Air velocity is specified at the inlet as a boundary condition. The volume fraction of air at the inlet is 0.95, while that of dust is 0.05. Constant pressure boundary condition is set at the outlet. Dust fraction given at the inlet is the fraction that is introduced into the venturi domain along with air to simulate contaminants. Initially dust comes along with air only, and no dust is mixed with water. Once water starts to slow down the dust particles, the dust will be mixed with water and can re-enter the venturi domain through the orifices. That is why a transient study has been performed. Since we are not applying any boundary conditions at the orifices and we are letting the physics take care of the water suction, dust captured in water is also simulated without any volume fraction specification at the orifices.

Solver

High resolution advection scheme and second order backward Euler transient scheme have been selected in the present case.

Time step

Time step taken for the simulation is 0.001 sec, while simulation was run for the total time of 3 seconds.

Results

In order to see whether the phenomenon of self-suction/priming occurs at the throat section or not, single phase simulation (water only) has been run by setting the velocity of water at the inlet of the venturi scrubber at 10 m/sec. Velocity vectors for this case are shown in Figure 12. It can be seen in the figure that the water velocity increases to its maximum value at the throat section and, as a result, negative pressure is generated at this location. Due to this, a pressure difference (i.e. pressure at the throat and hydrostatic pressure) occurs, which forces the water to enter the venturi scrubber.

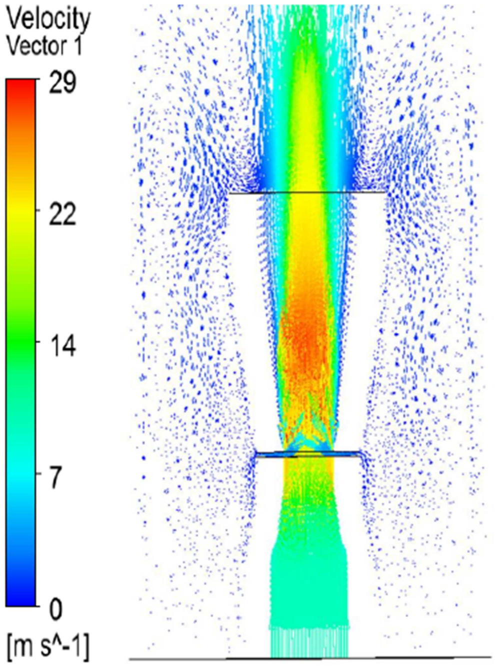

Figure 12: Velocity vector fields of water entering at 10 m/sec

The velocity vectors for the same conditions as shown in Figure 12 are shown for the orifices in Figure 13, while the pressure contours are shown in Figure 14.

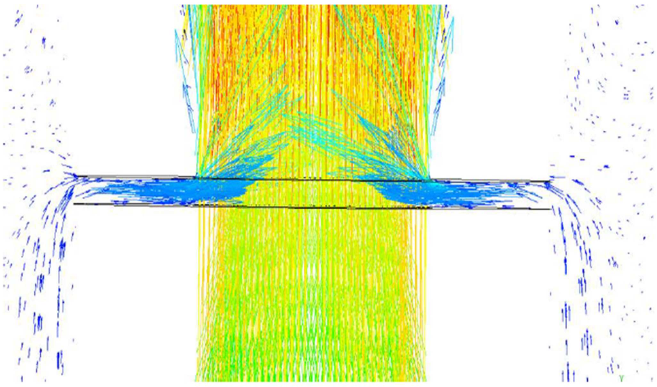

Figure 13: Velocity vectors starting from orifices

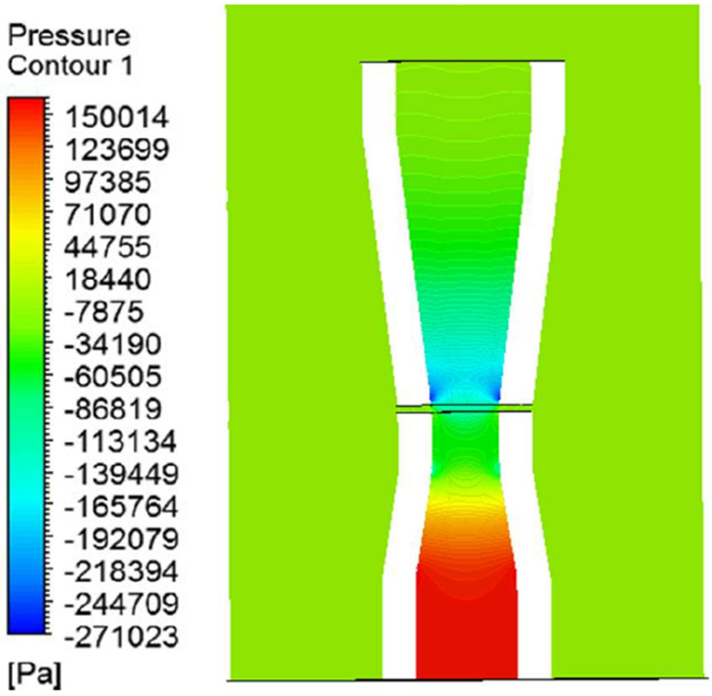

Figure 14: Pressure contours

After verifying the self-priming phenomenon, dust particle removal efficiency has been found using a multiphase approach. Simulations have been performed using two different values of inlet air velocity, i.e. 1 m/sec and 3 m/sec. The volume fraction of air after 2 seconds is shown in Figure 15.

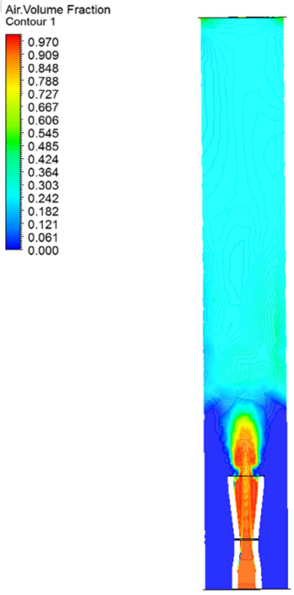

Figure 15: Volume fraction of air after 2 sec at 1 m/sec

Figure 16 provides an enlarged view at the throat section and shows the automatic suction of water at 1 m/s of air velocity.

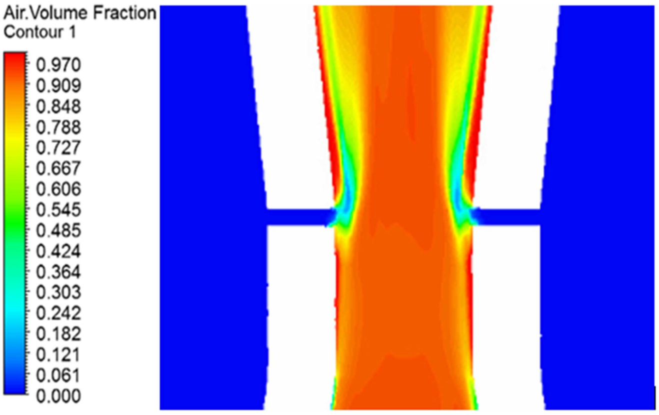

Figure16: Evidence of water suction in air volume fraction contours

Water is denser than air and air is forced from the inlet at a certain velocity, while water is automatically sucked in. The volume fraction of water carrying dust decreases as we move away from the venturi outlet because dust particles decelerate due to density difference and hence stay away from the outlet stream of air.

The way in which pressure differences force water injection at the throat can be seen in Figure 17, which clearly shows negative pressure being developed at the throat section, causing suction of water in the venturi scrubber.

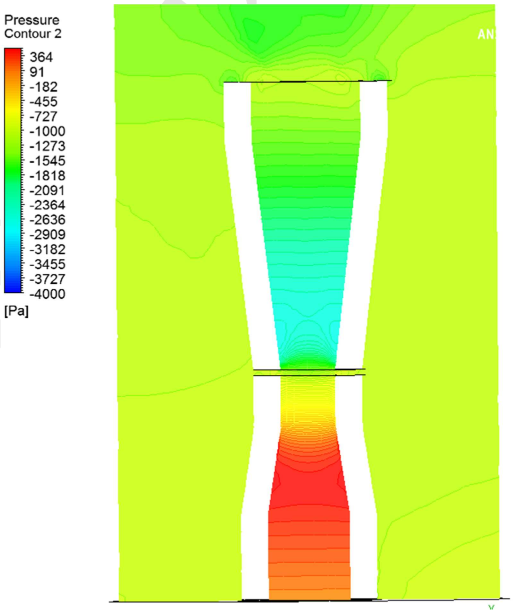

Figure 18: Pressure contours for three phase flow after 3 seconds

As mentioned earlier, simulations are performed for two values of air velocity at the inlet, i.e. 1 and 3 m/sec; the corresponding dust volume fraction for both cases along the centerline is compared in Figure 19. It can be seen that the dust volume fraction at the outlet for the case of 1 m/s is higher than it is in the case in which the inlet velocity is 3 m/s. In the case of higher inlet velocity, higher velocity is achieved at the throat, i.e. more negative pressure is developed as compared to the case for 1 m/s velocity. However, the hydrostatic head in both cases remains the same. Due to higher pressure difference in the case of velocity of 3 m/s, water suction is stronger, which results in an increase in the removal of dust from the air. As is obvious in Figure 19, dust fraction varies at the outlet section for the case of 1m/sec. The same is true for the case of 3 m/sec. Since at the 3 m/sec inlet velocity the dust fraction reaching the outlet is lower, we cannot easily see the variation in the dust fraction when comparing the two cases. Upon close observation, one can see a slight rise in the dust fraction towards the wall of the venturi, because maximum removal will occur towards the centre. This is because the velocity of the inlet stream is at its maximum in the centre of the venturi, leading to maximum removal of dust. To summarize, it is not true to say that dust fraction does not vary with position for any case; instead, dust fraction is different along the outlet section for all velocities of the inlet stream.

When air passes through the venturi domain, it creates suction for water and hence water flows in from the tank. Now, when we are not simulating the breakup of water, we cannot predict exactly whether the effect of air turbulence on water breakup is included in the simulation or not. Although we can come to conclusions identical to those of Lehner et al. [5] without simulating breakup of water into droplets, those conclusions would not be precise because the breakup of water into small droplets is not observed. This aspect has been added to our methodology by using the CAB model for the forced feed venturi and then representing air in Eulerian phase for breakup in the self-priming case. By simulating the breakup of water into parent droplets and then the breakup of those parent droplets into daughter droplets, the methodology for both cases takes care of all the different sizes of droplets formed and thus predicts the dust removal with added certainty. Velocity values at the throat also suggest a similar effect. For the case of the inlet stream entering at 1 m/sec velocity, water, air, and dust velocities have been probed on the line of the orifices. These values are given in Table 5. It can be seen that the dust moves with the velocity of water and not that of air. This means that dust particles are influenced by the presence of water and dragged down from the stream of air.

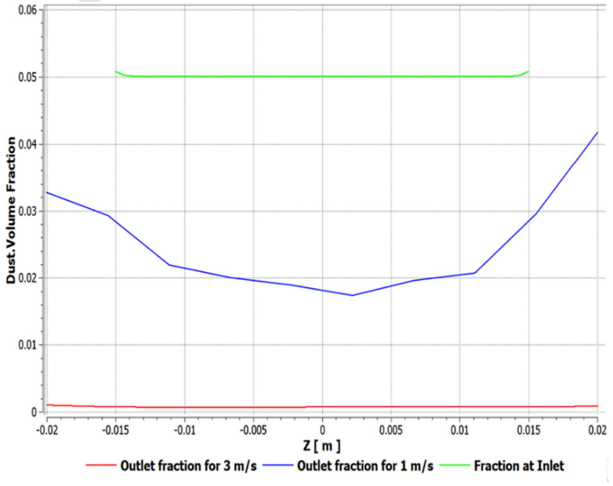

Figure 19: Dust volume fraction at inlet and outlet for inlet velocity of 1 m/s and 3 m/s

Table 5. Water, air and dust velocities at the throat.

Velocity of inlet stream (m/sec)	Velocity of water at throat (m/sec)	Velocity of air at throat (m/sec)	Velocity of dust at throat (m/sec)
1	0.225829	0.649428	0.225827

The dust particle removal efficiency has been calculated by taking the dust volume fraction (DVF) at both the inlet and outlet, as given below:

$$\eta = \frac{(DVF\ at\ inlet) - (DVF\ at\ outlet)}{(DVF\ at\ inlet)}$$

By using the above formula, removal efficiency for the case of 3 m/s inlet velocity comes out to be 96 %; removal efficiency ranges between 40-64 % for the case of 1 m/s after 2 seconds. The reason for the increased efficiency in the case of the higher inlet velocity is that this velocity generates more negative pressure at the throat and hence induces efficient suction.

6. Conclusions

In this work, CFD investigation of dust particle removal efficiency of self-priming venturi scrubber has been performed using ANSYS CFX 14.0. Current methodology was developed and validated by first employing it for the case of force-feed venturi. Experimental studies of force feed venturi are available in the literature and simulation results matched experimental results closely, validating the methodology used. Then, this methodology was applied to a self-priming venturi. The advantage of using this methodology and the reason behind the close resemblance of the force feed results and the experimental results is the usage of the CAB model to simulate water breakup.

For the self-priming venturi, because of the changes in morphology of phases, the CAB model could not simulate water breakup. This breakup was then achieved by treating air as an Eulerian phase. This gives the required effect of the CAB model. Dust removal efficiency has been calculated for two different values of air velocity at the inlet of the venturi scrubber, i.e. 1 and 3 m/s. It has been observed that the removal efficiency increases as the air inlet velocity is increased.

7. References

[1] B. Sehgal, “Accomplishments and challenges of the severe accident,” *Nuclear Engineering and Design,* vol. 210, no. 1-3, pp. 79-74, 2001.

[2] M. Ali, C. Yan, Z. Sun, H. Gu and K. Mehboob, “Dust particle removal efficiency of a venturi scrubber,” *Annals of Nuclear Energy,* vol. 54, p. 178–183, 2013.

[3] Nuclear Energy Agency and Committe on the Safety of Nuclear Installation, “Status Report on Filtered Containment Venting,” 2014.

[4] N. Gulhane, A. Landge, D. Shukla and S. Kale, “Experimental study of iodine removal efficiency in self-priming venturi,” *Annals of Nuclear Energy,* vol. 78, p. 152–159, 2015.

[5] 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.

[6] S. Pak and K. Chang, “Performance estimation of a Venturi scrubber using a computational model for capturing dust particles with liquid spray,” *Journal of Hazardous Materials,* vol. 138, no. 3, p. 560–573, 2006.

[7] Schmehl, R., Maier, G. and Wittig, S., “CFD Analysis of Fuel Atomization, Secondary Droplet Breakup and Spray Dispersion in the Premix Duct of a LPP Combustor,” in *Proceedings of the 8th International Conference on Liquid*, Pasadena, CA, USA, 2000.

[8] M. Ali, C. Yan, Z. Sun, J. Wang and H. Gu, “CFD simulation of dust particle removal efficiency of a venturi scrubber in CFX,” *Nuclear Engineering and Design,* vol. 256, p. 169–177, 2013.

[9] ANSYS, ANSYS CFX-Solver Theory Guide, Southpointe 275 Technology Drive Canonsburg, PA 15317 ANSYS,: ANSYS, Inc., 2013.

[10] S. Calvert, “Venturi and other atomizing scrubbers efficiency and pressure drop,” *AlChE Journal,* vol. 16, no. 3, p. 392–396, 1970.

Conflict of Interest Declaration:

“The author(s) declare(s) that there is no conflict of interest regarding the publication of this paper.”