

FLUID FLOW OPTIMIZATION OF SCRUBBER USING CFD

¹MAHADEVAPPA, ²Dr.V.M.KULKARNI,

¹M.Tech Student, ²Professor,

^{1&2}Department of Thermal Power Engineering,

^{1,2}Centre for PG studies, VTU, Kalaburagi, Karnataka, India.

Abstract: Scrubber systems are air pollution control devices that are used to remove coarse grained particulate matter and gases from industrial exhaust streams. The flow behavior of pollution causing particulate matter and gases mixture is essential for design of a scrubber. It also helps to know the velocity, pressure fields, velocity vectors at the inlet and outlet and thus essential for effective design of the scrubber and its system. For this purpose, computational fluid flow analysis for scrubber is carried out by using commercially available Ansys FLUENT® software. Two different models of scrubber geometry are considered in the analysis, where the location of the inlet pipe is perpendicularly located in model-1 and is tangentially located in model-2. The analysis is carried out for three different inlet velocities 0.32, 0.79 and 1.27 m/s and velocity and pressure fields, and turbulent kinetic energy are obtained accordingly. Maximum velocities of 0.41, 1.077 and 1.636 m/s were obtained from model-1 while model-2 gave maximum velocities as 0.488, 1.299 and 2.11 m/s. The corresponding maximum pressure values are 0.14, 1.050 and 2.1 N/m² for model-1 and 0.229, 1.636 4.327 N/m² for model -2 respectively. The corresponding maximum turbulence kinetic energy values of 0.011, 0.074 & 0.192 and 0.018, 0.123 and 0.324 were obtained from model-1 and model-2 respectively. From the analysis, it is observed that that model with tangential inlet (model - 2) is better in all the aspects for the proper fluid flow inside the scrubber as it results in better separation of the particles from the air.

Index Terms – Scrubber, Velocity, Pressure fields, Ansys FLUENT

I. INTRODUCTION

Scrubber arrangements are air pollution control equipment's used for removing undesirable particulates and gaseous mixtures from industrial exhaust flow line. Initially scrubbers were designed to remove of carbon dioxide from the air of submarine. Mainly scrubber has referred to pollution control equipment's that use liquid to ash undesirable pollutants from gas flow line. Nowadays scrubbers are also used for the systems that inject a dry reagent or slurry in to a polluted exhaust flow line to wash out acid gases. Scrubber can also be used for recovery of heat from hot gases by flue gas condensation. The main objective of design of scrubber is to decrease of ecological pollutants. Normally small sized plants do not have flue gas desulfurization units in future will be required to adapt pollution control devices. For SO₂ remove wet scrubber are mainly uses because of its low working expenses. However capital expenses for development are viewed as high. Hence forth a feasible improvement technique is needed to reduce the capital investment of optimization, and process changes and advancement is a continuous effort. Normally power plants uses lime slurry for splash response to understand of SO₂ discharge. With a specific end goal to enhance the scrubber, nozzle qualities and position must be upgraded to lessen the expenses of the frame work excursion and moderate dangers of lacking contamination decreases.

Researchers are making sincere efforts to find out suitable alternatives for scrubber with different geometry and varying parameters which effects on performance of scrubber. Some of the attempts made by various researchers to find out flow rate particle substance, liquid rheology, and sprinkles arrangement formation were investigated to fulfill SO₂ ejection over 90%.

K.J.Brown etc. all. (May 2013) [1] have concentrate on nozzle parameters and they were modeled to grant the grain capture over the system, by applying ANSYS FLUENT LANGRANGING grain capture form were used with warmth and mass transfer. A result obtained from this report is to determine the expected liquid – gas interacts relating to the system efficiency. For long term utilization of scrubber wall impact and flow pattern results were find out to decrease component plugging and corrosion due to their impact.

Bashir Ahmed Danzomo. (Dec 2012)[2]. Obtained from cement industry to design of a pilot scrubber system for pm10 (particulate matter) control by considering droplet sizes of 500µm, 1000µm, 1500µm and 2000µm. By increase of particle size 5µm and 10µm dust particles maximum accumulation efficiencies were found 99.985% and 100% again increasing size at 500µm droplet size and 2.71/m³ minimum efficiency were achieved 43.808% and again increasing droplet size 2000µm 0.7/m³ increased efficiency obtained 58.728%.

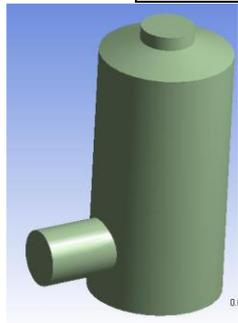
M. M. Toledo-Melchoretc.all.(2014)[4] By using three-dimensional mathematical simulation conducted of single-state and double state flow (air-water) in a venturi scrubber with an inlet and throat diameters of 250 and 122.5 mm. The mass discharge rate of liquid was 0.013 kg/s and 0.038 kg/s and the mass flow rate of gas was 0.483kg/s, 0.736kg/s, 0.861kg/s and 0.928kg/s gas discharge was checked in five geometries by changing angle of convergent and divergent and the two phases flow checked only one geometry. The obtained result shows that pressure trace build upon on gas discharge motion and the water discharge motion does not effect.

In present study computational fluid dynamics analysis is carried out for the scrubber for their different inlet velocity. Analysis is carried out for all three inlet velocity 0.32, 0.79 and 1.27m/s. Contours for velocity, pressure, turbulent kinetic energy, velocity vectors and stream lines colored by velocity are plotted to understand flow patterns inside the scrubber.

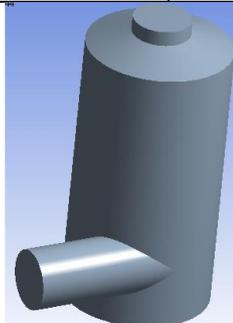
PROBLEM DESCRIPTION

Scrubber size	In ' mm'
Height of cylinder	1000
Diameter of the cylinder	500

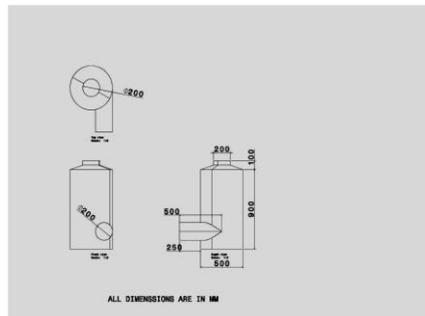
Diameter of nozzle	200
--------------------	-----



Model-1



Model-2



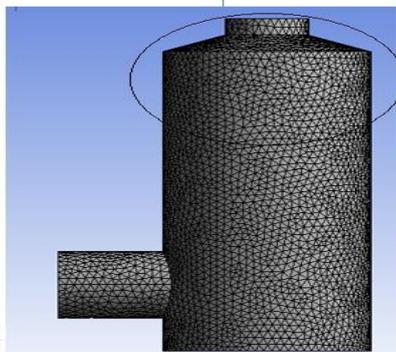
Geometry model-2

Meshing

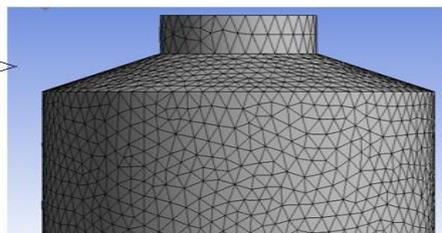
Mesh generation it is very necessary part of the work it should be started before any calculation of CFD. For this purpose grid generation has been done on workbench-mesh (CFD) automatic method is used for the creation of mesh. Model-1 mesh consists of 111325 elements and 21399 nodes and model-2 consist of 112941 elements and 21717 nodes.

Mashed model -1

Type	Tetrahedral mesh
Number of elements	111325
Number of nodes	21399



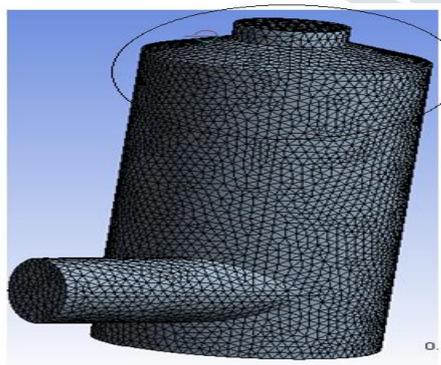
Mashed model -1



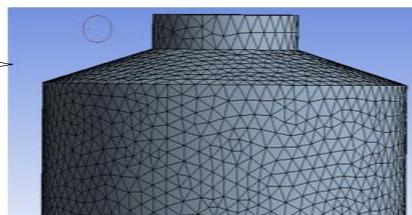
Mashed model zoom view

Mashed model -2

Type	Tetrahedral mesh
Number of elements	112941
Number of nodes	21717



Mashed model



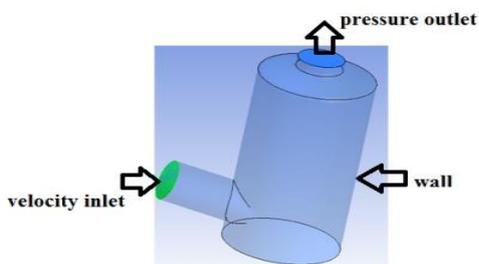
Mashed model- zoom view

Boundary conditions:

Analysis type	Steady state
Inlet	Velocities 0.32,0.79 and 1.27 m/s
Turbulence module	k-epsilon
Single phase	Air inlet
Out let	Pressure outlet

Fluid properties:

Working fluid	Air
Density	Compressible (ideal air)
Viscosity	1.78e-5 kg/ ms
Thermal conductivity	1006j/kg k

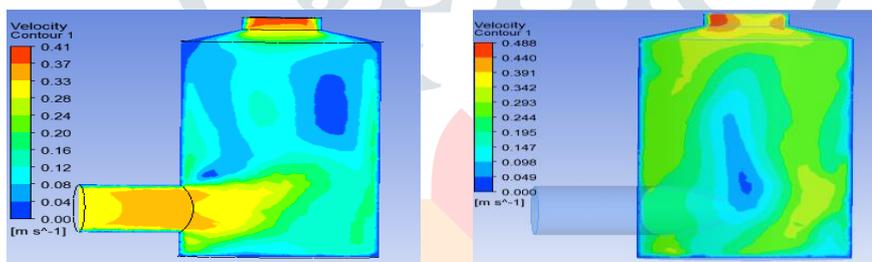


Boundary conditions

RESULTS AND DISCUSSION

CFD analysis is carried out for the scrubber for their different air inlet velocities. Analysis is carried for the inlet velocities of 0.32, 0.79 and 1.27 m/s. Contours for velocity, pressure, turbulent kinetic energy, velocity vectors and stream lines colored by velocity are plotted to understand flow patterns inside the scrubber. Objective is to understand the effect of inlet cross section location for different inlet velocities.

1. Velocity inlet at 0.32 m/s

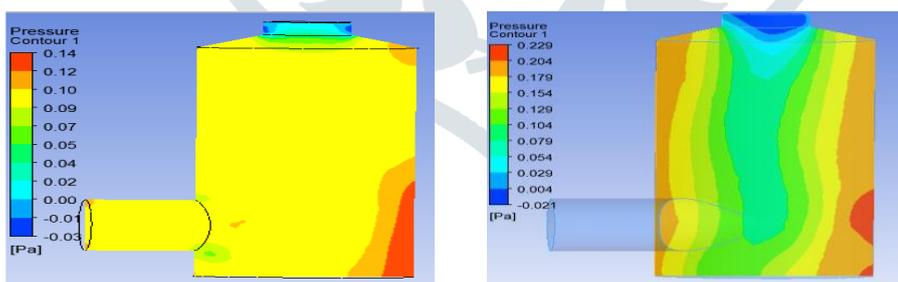


Model-1

Model-2

Contour of velocity for inlet flow 0.32 m/s

From the velocity contours it is observed that velocities are equally distributed near the wall for the model -2 since the inlets are tangential.

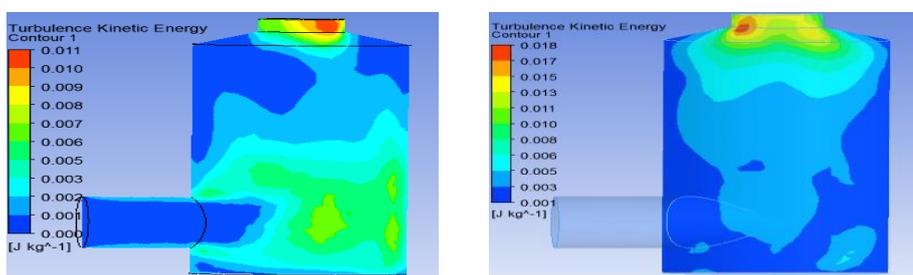


Model-1

Model-2

contours of pressure for inlet flow 0.32 m/s

From the pressure contours it is observed that pressure is higher near the wall for the model -2 due to tangential motion.

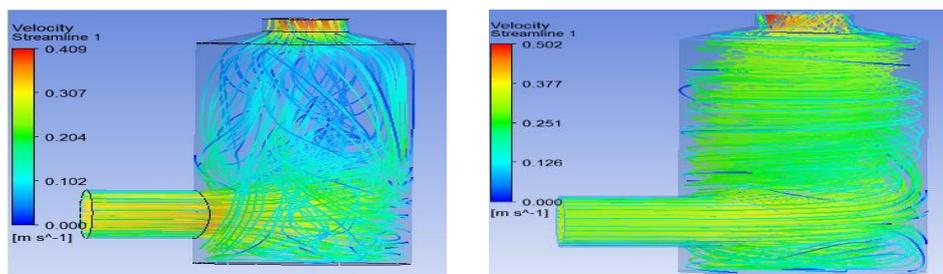


Model-1

Model-2

Contours of turbulent kinetic energy for inlet flow 0.32 m/s

From the turbulent kinetic energy contours it is observed that turbulence is higher for the model-2 in comparing with other model - 1.



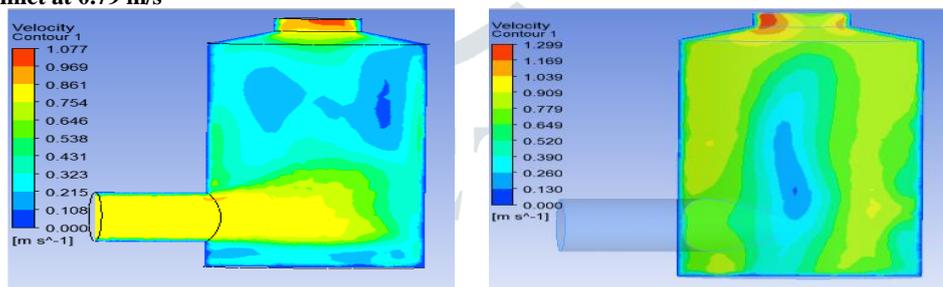
Model-1

Model-2

Streamlines colored by velocity for inlet flow 0.32 m/s

From the streamline plots it is observed that since the inlet for model – 2 is placed tangential to the domain flow of fluids touches the inner wall of the domain.

1. Velocity inlet at 0.79 m/s

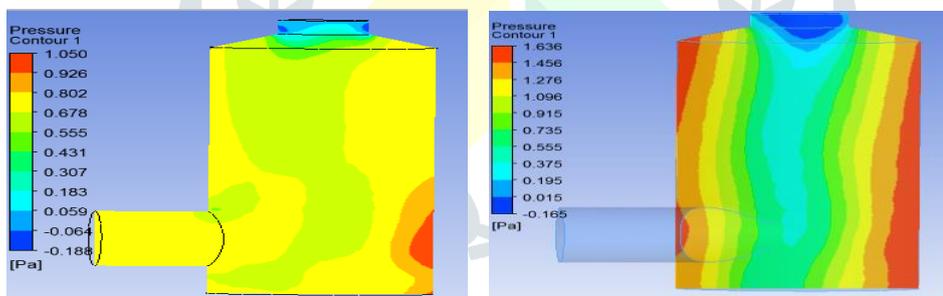


Model-1

Model-2

Contours of velocity for inlet flow 0.79 m/s

From the velocity contours it is observed that velocities are equally distributed near the wall for the model – 2 since the inlets are tangential. We can observe in model – 1 velocity at Centre of the model is more but in wall side it is less.

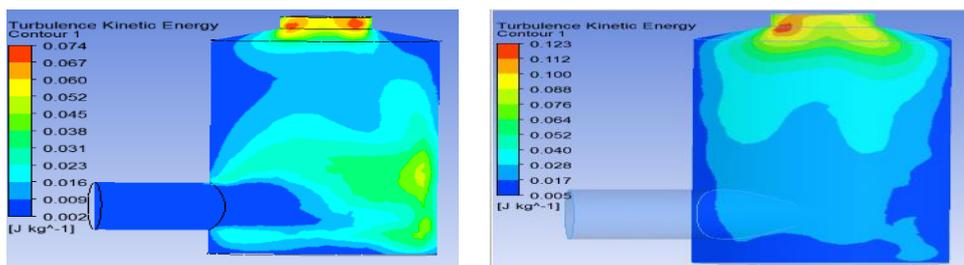


Model-1

Model-2

Contours for pressure for inlet flow 0.79 m/s

From the pressure contours it is observed that pressure is higher near the wall for the model – 2 due to tangential motion. Pressure in model – 1 is low as compared to model – 2 and we can also see pressure in model – 1 only at the striking of fluid to wall.

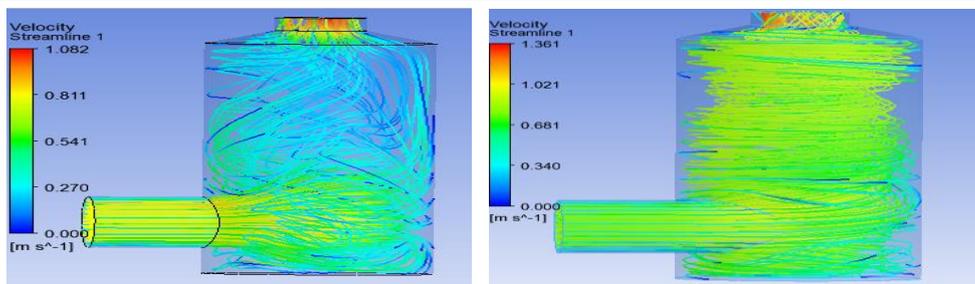


Model-1

Model-2

Contours for turbulent kinetic energy for inlet flow 0.79 m/s

From above fig the turbulent kinetic energy contours it is observed that turbulence is higher for the model 2 in comparing with other model.



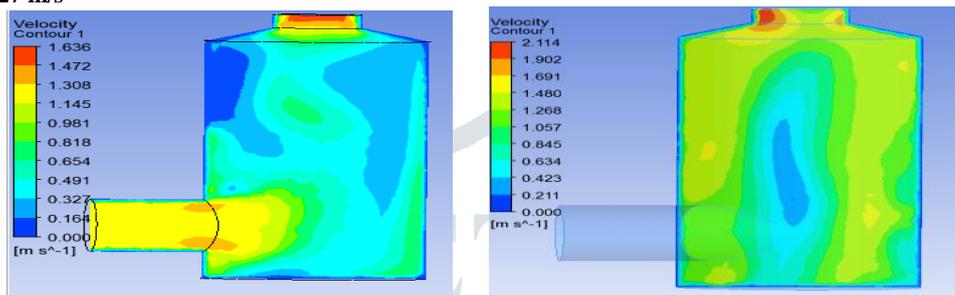
Model-1

Model-2

Streamlines colored by velocity for inlet flow 0.79 m/s

From the above fig the streamline plots it is observed that since the inlet for model – 2 is placed tangential to the domain flow of fluid of fluids touches the inner wall of the domain.

3. Velocity inlet 1.27 m/s

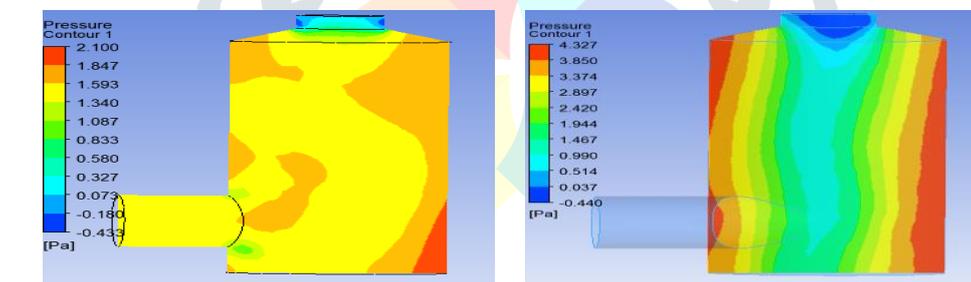


Model-1

Model-2

Contours for inlet velocity flow 1.27 m/s

From the above fig the velocity contours it is observed that velocities are equally distributed near the wall for the model – 2 since the inlet are tangential. We can observe in model – 1 velocity at the exit of the nozzle is more.

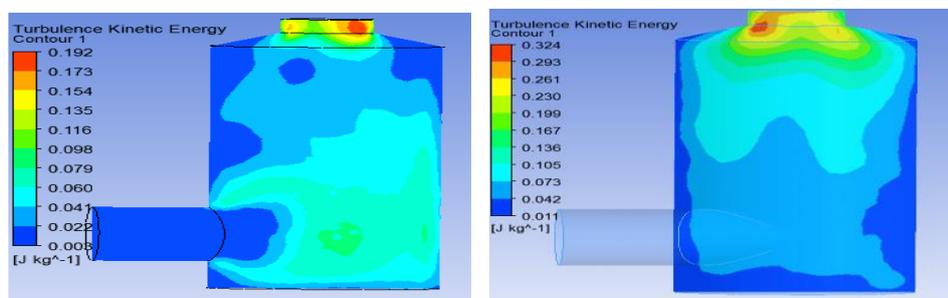


Model-1

Model-2

Contours of pressure for inlet flow 1.27 m/s

From the above fig the pressure contours it is observed that pressure is higher near the wall for the model – 2 due to tangential motion. We can observe model – 1 velocity near the wall is less as compared with model – 2.

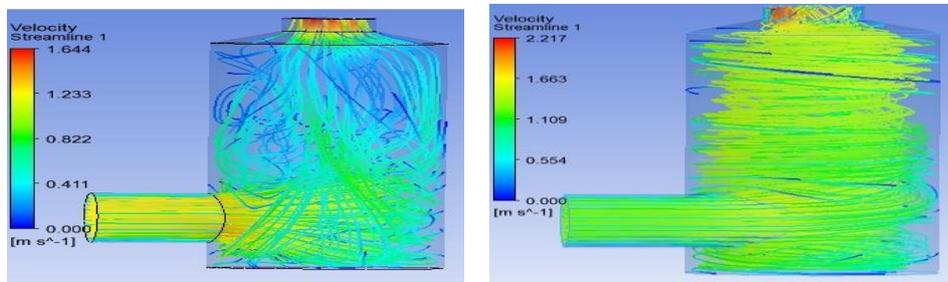


Model-1

Model-2

contours of turbulent kinetic energy for inlet flow 1.27 m/s

From the above fig the turbulent kinetic energy contours it is observed that turbulence is higher for the model – 2 in comparing with model – 1.



Model-1

Model-2

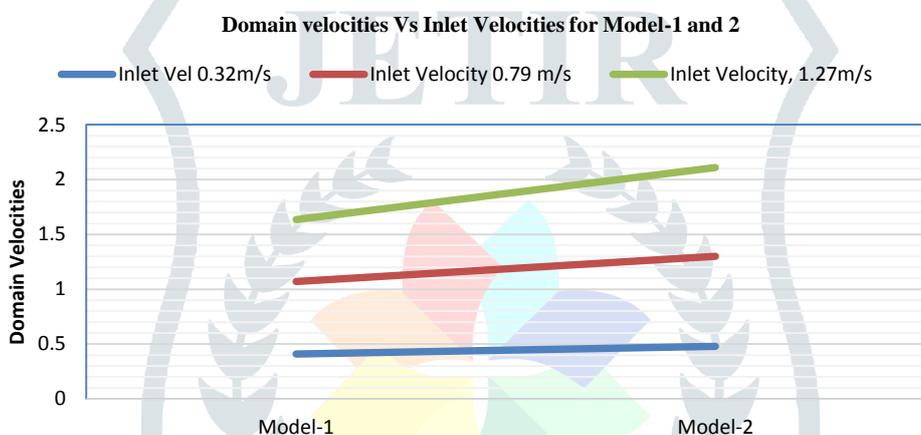
Streamlines colored by velocity for inlet flow 1.27 m/s

From the above fig streamline plots it is observed that since the inlet for model – 2 is placed tangential to the domain flow of fluid touches the inner wall of the domain.

GRAPHS:

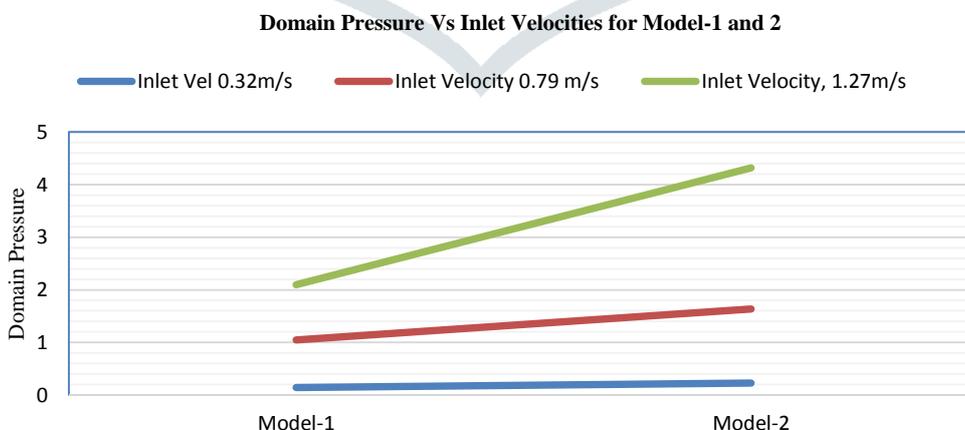
1. Domain velocities v/s inlet velocities for model – 1 and 2

Models	Domain velocity	Domain velocity	Domain velocity
Model -1	0.41	1.07	1.636
Model -2	0.48	1.299	2.11



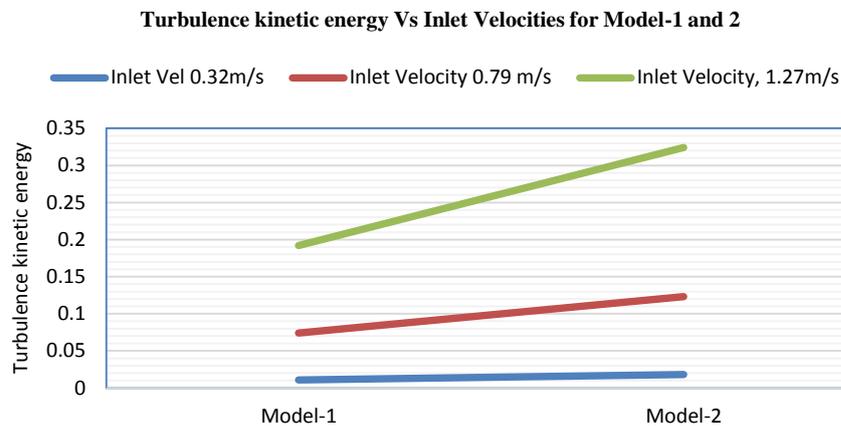
1. Domain pressure v/s inlet velocities for model – 1 and 2

Models	Domain pressure	Domain pressure	Domain pressure
Model -1	0.14	1.05	2.1
Model -2	0.229	1.636	4.32



2. Turbulence v/s inlet velocities for model – 1 and 2

Models	Turbulence	Turbulence	Turbulence
Model -1	0.011	0.074	0.192
8Model -2	0.018	0.123	0.324



CONCLUSION

CFD analysis is carried out for the fluid flow in scrubber with cylindrical geometry using commercially available Ansys FLUENT® software. Simulated results are validated against experimental values available in literature. The analysis is carried for single phase fluid flow in scrubber. Parametric numerical simulations studies were carried out by varying the inlet velocity as 0.32, 0.79 and 1.27 m/s. two different models were considered depending upon the inlet pipe location. In model-1 the inlet pipe is located perpendicular to the axis of the cylindrical scrubber. The inlet pipe is located at tangential direction to the cylindrical axis to the scrubber. Objective of the analysis is to understand the flow behaviour when the inlet location is tangential to the scrubber domain.

Contours of Velocity, Pressure, and Turbulent Kinetic Energy and stream lines are plotted for both models by considering the above mentioned three different velocities. From the CFD analysis the following main conclusion are drawn.

- It is observed that velocities are equally distributed near the wall, since the inlets are tangential.
- From the pressure contours, it is observed that pressure is higher near the wall due to tangential motion.
- Turbulent Kinetic Energy is observed to be higher due to tangential fluid motion.
- From the Streamline plots it is observed that since the inlet pipe is placed tangential to the domain, Flow of fluids touches the inner wall of the domain.
- It is observed from the streamline plots that the flow of fluids touches the inner wall of the domain since the inlet pipe is placed tangential to the domain.
- From the above observations, it is concluded that model with tangential inlet found to be better in all the aspects for the proper fluid flow inside the scrubber. It also results in better separation of the particles from the air.

SCOPE FOR FUTURE WORK

Further analysis can be carried out for multiphase, air and particle mixing to understand the multi-phase behaviour inside the scrubber.

REFERENCES

- [1] K.J. Brown, W. Kalata, and R. J. Schick (May 2013) "optimization of so2 scrubber using cfd modeling"
- [2] Bashir Ahmed Danzomo (DEC 2012) "performance evaluation of wet scrubber system for industrial air pollution control"
- [3] Bashir Ahmed Danzomo, and Momoh-Jimoh Enyiomika salami, RaisuddinMohdKha, neither MohdIskhandar Bin Mohd nor (July 2013) "CFD based parametric analysis of gas flow in a counter-flow wet scrubber system"
- [4] M. M. Toledo-Melchor, J. G. Barbosa-Saldana & Alonzo-garcial (2014) "Numerical Simulation of Flow Behavior within a Venturi Scrubber"
- [5] N. P. Gulhane, H. S. S. S. Kale & Kadam, (2015)"Analysis of Pressure and Velocity at the Throat of Self-Priming Venturi Scrubber"
- [6] A. M. Wala, S. Vytla, G. Huang & C. D. Taylor (NIOSH Pittsburgh Research Laboratory, Pittsburgh, PA, USA) "Study on the effects of scrubber operation on the face ventilation"
- [7] Mohammad Reza Ashouri, Majid Bayatian, and Rouhallah Mahmoudkhani (3-march-2016)
- [8] Majid Ali, Sun Zhongning, YanChangqi, Wang Jianjun, and Gu HaiFeng College of Nuclear Science and Technology, Harbin Engineering University, Harbin 150001, "Flow Behavior Simulation with Computational Fluid Dynamics in Spray Tower"

Scrubber” (3-march-2016) Research Journal of Applied Sciences, Engineering and Technology 4(19): 3830-3833, 2012 ISSN: 2040-7467

- [9] Kousalya Devi, Mr.Venkatesh, and Mr.Chandrasekaran, (APL 2015) (international journal of research and development organization)”performance improvement of venturi wet scrubber”
- [10] S.A. Dudek, J.A. Rogers and W.F. Gohara,(Aug 1999) “Computational Fluid Dynamics (CFD) Model for Predicting Two-Phase Flow in a Flue-Gas-Desulfurization Wet Scrubber”

