Plennum Chamber Optimization of Turbo ventilator Test Rig

[1]Mr.Ganesh K. Jadhav, [2]Dr. P. M. Ghanegaonkar, [3]Mr. Sharad Garg, [4]Aakash Khatale, [4]Manoj Gosavi, [4]Nikhil Sali, [4]Jayendra Ingale

[1] Research Scholar, Sinhgad College of Engineering, Pune
[2] Professor, Mechanical Engg. Dept. Dr. D.Y. Patil Inst.of Engg. & Tech. Pimpri, Pune
[3] Design Engineer, Continuum Technologies LLP
[4] BE Students

Abstract- Turbo ventilator is wind driven conventionally used in industries for proper ventilation. Rotating turbo-ventilators are cost effective environmental friendly natural ventilation devices, which are used to extract polluted air from a building. For its performance testing, test rig is developed in which plenum chamber is act as the room from which stale air is to be thrown outside. Mass flow rate of the turbo ventilator is measured at the suction pipe of the plenum chamber. This paper aims at optimizing the shape and size of plenum chamber to be used during performance testing. From CFD analysis, deviation in differential pressure at different mass flow rate for different volumes is studied in this paper.

Index Terms- CFD, Ventilation, Turbo ventilator, Plenum chamber

I. Introduction

Turbo ventilator is wind driven effective method for natural ventilation in industries. This type of device is usually installed on the rooftop to extract air extract air flow from a building to improve air quality and comfort. It is cost effective and maintenance free solution for industrial ventilation. Project aims to design and development the experimental setup for testing the performance of turbo ventilator. Setup is used to measure the mass flow rate of commercial available as well as modified turbo ventilator. Turbo ventilator is mounted on top of the plenum chamber though the throat. Wind velocity is simulated by the variable speed fan mounted near ventilator. Metrological data shows range of wind velocity min 2 Km/hr (approximately) in major part of India. When we simulate wind velocity interestingly at what minimum wind velocity turbo ventilator starts functioning will recorded.

II. Plennum Chamber

Plenum is a big room from which stale air is to be thrown outside by turbo ventilator. Turbo Ventilator is placed on horizontal roof of plenum. It should be volume of 5 to 8 times of flow rate of handled by a commercial Turbo ventilator in m^3/h. Plenum is a big room from which stale air is to be thrown outside by turbo ventilator. It should be air tight as air should only come into the plenum chamber from the suction pipe. From literature review we come to know that no any researcher have focused on the design and optimization of plenum chamber.

This paper focused on the size optimization of plenum chamber. The Computational Fluid Dynamics (CFD) is the tool used to optimize the size of plenum chamber.

CFD analysis was carried out for different volumes of plenum chamber. According to that it is performing that optimization of plenum chamber for different volumes. While doing analysis, we assume that turbo ventilator rotates with uniform speed and considering uniform flow of air over turbo ventilator.

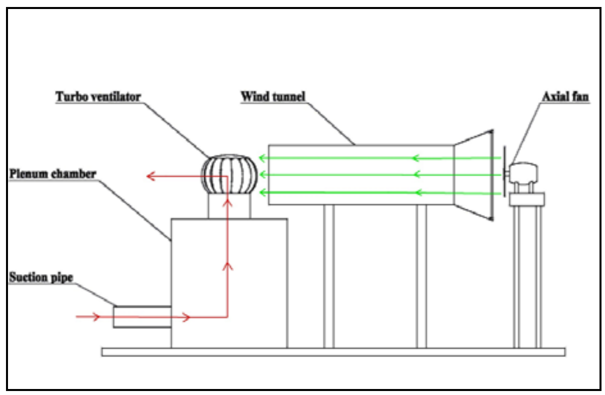

Fig.1 Schematic Diagram of Experimental Test Ring

III. Methodology

CAD geometry of plenum chamber was made in Uni-graphic as shown in Fig 2. The different models were made by adding or subtracting equal amount of volume from base setup. Also we consider a pipe with negligible friction losses connecting inlet and outlet area. The parameters that were kept constant are

- Inlet and outlet area
- Inlet and outlet datum

Three dimensional models are imported in to the STAR CCM + which is the CFD tool. STAR CCM+ is used for meshing as well as simulation. For modeling of the rotating motion of the turbo ventilator, the multiple reference frames (MRF) meshing in STAR CCM+ is used. For proper meshing and good resolution the realizable k- epsilon model is used. K-ε model is very popular for industrial applications due to its good convergence rate and relatively low memory requirements. A fine mesh of polyhedral scheme is used as this generates a fine mesh of at the wall boundaries particularly at the surfaces of plenum chamber as shown in Fig. 3.

Boundary condition

At the inlet i.e. suction pipe of the plenum chamber giving the different mass flow rate and atmospheric temperature. Differential pressure is measured at the outlet of the plenum chamber. Different volume models are simulated by the CFD software. For each model performed the several iterations on CFD code to get differential pressure readings for different mass flow rates at outlet. In CFD analysis several iterations for the different volumes of plenum chamber has been carried out. Plenum chamber is increased or decreased from the base size.

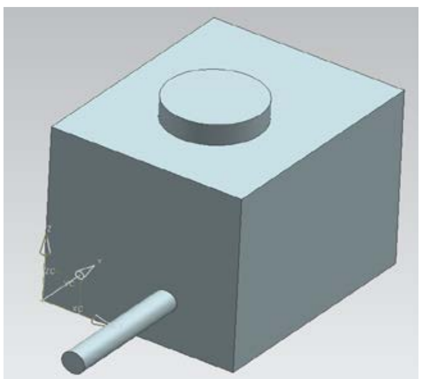

Fig. 2 CAD Model of Plenuum Chamber

IV. Results

Simulation time using 32 GB ram Dual core processor was approximately 2 Hrs.

Figure 4 and 5 shows the velocity contour and pressure contour respectively. It is clear from the velocity diagram; the negative pressure is created at the out let of the plenum chamber which helps in extracting the stale air inside the plenum chamber. Value of the suction pressure is depends on the wind velocity impingement on the turbo ventilator.

Fig. 3 Mesh Model of Plenuum Chamber

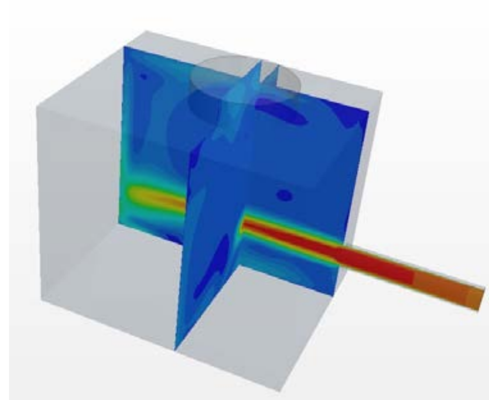

Fig. 4 Velocity Contour

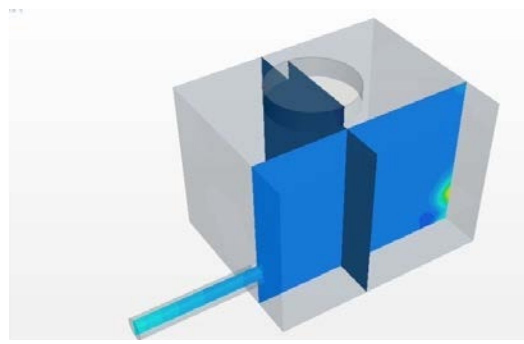

Fig. 5 Pressure Contour

The results obtain from the CFD simulation at different mass flow rate like 20, 40, 60, 80 lit/sec. are shown in the table 1 and Fig. 6 shows the between the mass flow rate and differential pressure of different volumes.

Volume (m^3) ⇨	**0.44 (Pipe)**	**1.48**	**1.72**	**1.95 (Base)**	**2.19**	**2.34**
Mass flow rate (lps) ⇩	Differential pressure (Pa)					
20	0.98	1.10	1.09	1.08	1.08	1.10
40	3.81	4.21	4.17	4.11	4.20	4.31
60	8.35	9.28	9.25	9.44	9.33	9.50
80	14.98	16.27	16.2	16.14	16.39	16.42

Table No. 1

Comment are drown about the optimization of turbo ventilator from the obtained results are as follows

1. For approximately 10% and 20% increase or decrease in base volume for different mass flow rate there is no significant change in differential pressure.

2. By 75% reduction in volume (considering pipe) only 10% decrease in differential pressure.

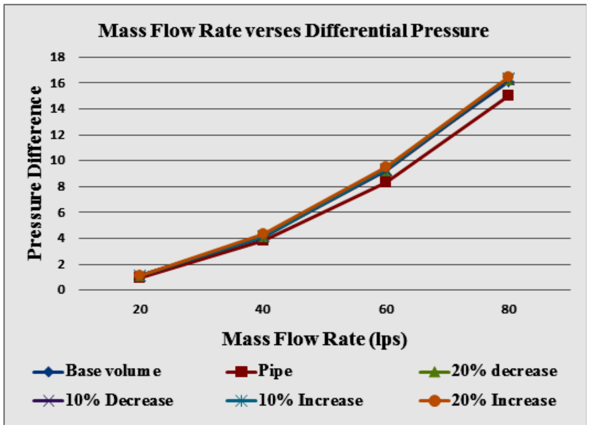

Fig. 6 Mass Flow Rate verse Differential Pressure

V. Conclusion

Different parameter affects the performance of turbo ventilator. Plenum chamber being part of experimental setup does not have a standard model or parametric study available in any of research work done till date. Based on result obtained and plot it is concluded that size of plenum chamber does not play a pivotal role in performance testing of turbo ventilator and Base size of the plenum chamber gives the correct mass flow rate of the turbo ventilator. Selection of shape of the plenum chamber is cubical because it is easy to manufacture as well as shape of all the industry or factories are cubical.

References

[1] Meadows, V. H. Rotary ventilator. US Patent 1, 857, and 762. (1929).

[2] Naghman Khan et al., A review on wind driven ventilation techniques Energy and buildings 2008; 40:1586-60

[3] Shao-Ting J. Lien "Numerical simulation of rooftop ventilator flow" Building and Environment 45 (2010) 1808-1815

[4] Farahani et al. (2010). Simulation of airflow and aerodynamic Forces Acting on a Rotating Turbine Ventilator. American Journal of Engineering and Applied Sciences, 3(1), 159-170.

[5] Jason Lien and Noor Ahmed "Numerical evaluation of wind driven ventilator for enhanced indoor air quality" Evolving Energy-IEF International Energy Congress (IEF-IEC2012) Procedia Engineering 49 (2012) 124 – 134

[6] Computational Fluid Dynamics by John D. Anderson, McGraw-Hill International editions.

[7] Computational Fluid Dynamics by JiyuanTu (RMIT University, Australia), Guan HengYeoh (Australian Nuclear Science and Technology Organization), Chaoqun Liu (University of Texas, Arlington), ELSEVIER

Authors

First Author – Mr.Ganesh K. Jadhav, Research Scholar, Sinhgad College of Engineering, Pune, ganeshjadhavcoep@gmail.com
Second Author – Dr. P. M. Ghanegaonkar, Professor, Mechanical Engg. Dept. Dr. D.Y. Patil Inst.of Engg. & Tech. Pimpri, Pune, praving21863@gmail.com
Third Author – Mr. Sharad Garg, Design Engineer, Continuum Technologies LLP, Pune sharadnhn@gmail.com
Other Authors – Aakash Khatale, Manoj Gosavi, Nikhil Sali, Jayendra Ingale, BE Students