

Redesign and Validation of an Industrial Blower

C.N.Jayapragasan¹, M.Sai Phani Deep Kumar², Dr.K.Janardhan Reddy³

Research Scholar , VIT University, Chennai, India ¹

MTech Student, VIT University, Chennai, India ²

Associate Professor, VIT University, Chennai, India ³

ABSTRACT: Blowers play an important role in air-conditioning, furnaces, low pressure cooling and dust or fume extraction systems because they can achieve high pressures compared to fans. In this paper an industrial blower with three different volute geometries is investigated by using Computational Fluid Dynamics (CFD). And then two new volute geometries are modelled and analyzed based on the air flow through the blower. The numerical investigation revealed that blowers with modified volute geometries have better outlet velocities. For this work, impeller is modelled using Solid Works and meshed in ICEM CFD. The post processing is carried out using CFD POST. CFD simulations are performed using the commercial CFD software package FLUENT.

KEYWORDS: Computational fluid dynamics (CFD), Volute geometry, Industrial blower.

I. INTRODUCTION

Blower's main aim is to deliver the gas or air with an appreciable rise in pressure to overcome resistance in the flow and they can achieve high pressures than fans. Usually in many industries, blowers are used in dust extraction system. These blowers suck the air from the inlet and discharge them through the outlets in order to clean the machines. They also play an important role in air-conditioning, furnaces, low pressure cooling and dust or fume extraction systems. In industries, the centrifugal and positive displacement type blowers are mostly used. Parts like impeller, volute geometry, vane angle and vane type have an effect on performance of the blower

Volute is a passage located around the impeller which collects the flow from the impeller and delivers at the exit duct. Volute geometry varies based on the need of discharge. General centrifugal blowers have bell-mouth volute geometry. Volute geometry is one of the main performance parameter of a blower. In volute, tongue geometry and radial distance between impeller and volute are general performance parameters. In this paper, work is aimed at improvising the volute design of the blower to improvise the performance of an industrial blower.

II. LITERATURE REVIEW

Lot of research is done on areas like impeller design, volute geometry, vane angle and vane type and their effect on the performance experimental and theoretical approach. Volute geometry is one of the main parameter influence the efficiency of a blower. The volute geometry collects the flow from the impeller and delivers it to the exit duct. Poor volute design leads to the redistribution of low energy fluid at impeller exit which influence the velocity and pressure by blockage [1]. Impeller will be affected seriously due to the poor design of volute, as it leads to non-symmetrical pressure distribution. Volute design not only affects the flow at impeller outlet, but also leads to substantial changes in flow patterns inside the impeller passages [2]. Impeller and tongue interaction is also having an impact on centrifugal blower. Volute tongue creates unsteady forces on the impeller blades and also responsible for flow obstruction and noise generation. Centrifugal blower with different tongue geometries are analyzed numerically, and stated that increasing the distance between the impeller and the volute tongue causes a more favorable aerodynamic force distribution on the impeller which results less noise generation [4][13].

Centrifugal blowers generate noise and this noise can be divided in two ways as aerodynamically induced noise and vibration-induced noise. Several research works were done on different types of impeller vane types focusing on

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 6, June 2014

problems like noise, vibration and air flow. By numerical and experimental approach it was found that blower with backward impeller have more outlet velocities and flow rates compared with forward and radial impellers [7]. Lot of research was also done on radial gap between impeller and volute and found that reduction of radial gap will raise the pressure over the volute and causes more noise [8].

From the literature review, the importance of volute geometry in blower performance is studied. Hence in this study an industrial blower with three different volute geometries with backward vane impeller was investigated numerically by using the CFD method. The purpose of the present study is to investigate the effect of volute geometry on an industrial blower. Based on the simulation results, air flow through different volute geometries was investigated and analyzed.

III. MODELING AND MESHING DETAILS

Blower design consists of an impeller, suction cover, volute and its casing. These models were designed in solid works and ICEM. Impeller of Backward type and its vanes are designed in solid works and remaining parts are designed in ICEM geometry. Blowers with three different volutes are given below in Fig. 1.

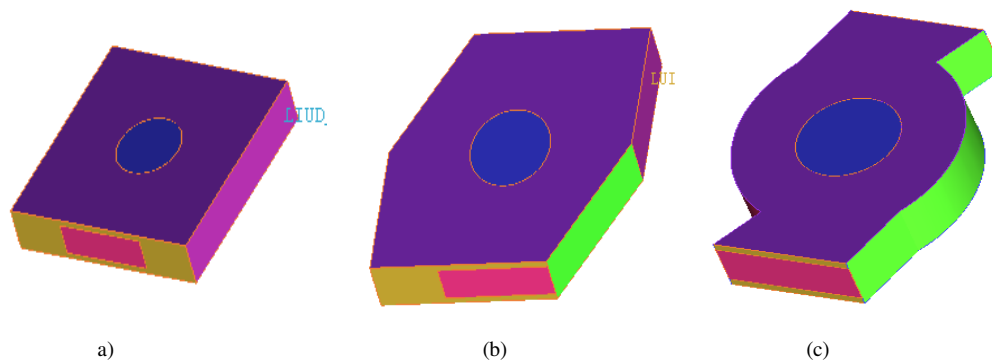


Figure 1: (a) Blower without volute, (b) Blower with Partition plate, (c) Blower with volute

After modeling blower with three different volutes, the parts are simplified using ICEM for sake of meshing. Two fluid domains are modeled one around the impeller and other one in the casing. Impeller is defined as a solid domain. The 3D CAD models are imported into ICEM CFD software for meshing.

The models are meshed using tetrahedral elements as they are easy to use for complex geometries as compared to other type of elements. Parts like impeller and volute geometries are of prime importance and hence they are fine meshed as compared to the other parts.

After successful grid generation, mesh independency test were carried out before moving onto the simulation and found out that mass flow rate is found be same for 2 million and 3.5 million mesh count. So to reduce the computational time mesh 2 million mesh count is selected.

Geometrical specifications for the blowers are tabulated below

Table: 1 Geometrical dimensions of the designed blowers		
S.No	Parameter	Dimension
1	Impeller diameter	380mm
2	Inlet diameter	180mm
3	Outlet dimensions	250X78mm
4	Impeller rotational speed	383.27 rad/sec
5	Blower casing	550X550X106mm
6	Partition plate dimensions	313X106mm
7	Volute diameter	440mm

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 6, June 2014

IV. COMPUTATIONAL APPROACH

Computational fluid dynamic (CFD) approach is an effective method to solve non-linear PDE equations in fluid flow and heat transfer problems etc. In the present problem impeller has a rotating motion. To solve such problems fluent incorporates moving reference frame.

The moving reference zone is a fluid domain created around the moving parts of a system. So impeller is assigned as a moving reference frame and remaining parts are assigned as stationary walls. For the proposed work assumptions and boundary conditions are tabulated below.

In present study for the turbulence model, SST k- ω turbulence model is selected to solve the transport equations for the kinematic turbulent viscosity used in the present calculations. It involves wall bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.

Table: 2 Assumptions and Boundary conditions

Materials	Air and aluminum
Turbulent model	k-omega Shear Stress Transport (SST) model
Navier strokes equation	Semi Implicit Method for Pressure Linked Equation (SIMPLE)
Moving reference frame	Impeller
Inlet conditions	Velocity inlet
Outlet condition	Pressure outlet
Flow	Steady state flow
Solver	Pressure based solver

V. NUMERICAL RESULTS AND DISCUSSIONS

Once the solution gets converged the results are post processed using CFD POST. Various values like total pressure, static pressure, velocity are obtained. Fig. 3-5 shows the velocity and pressure vectors obtained in different volute geometries

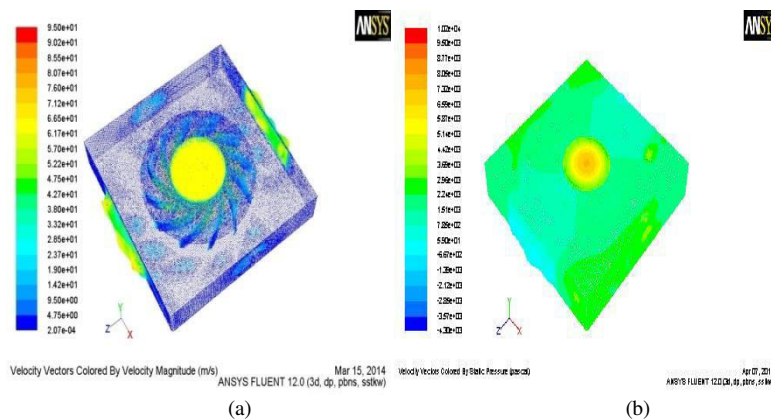


Figure: 3 (a) Velocity vectors and 3(b) pressure vectors in blower without volute geometries

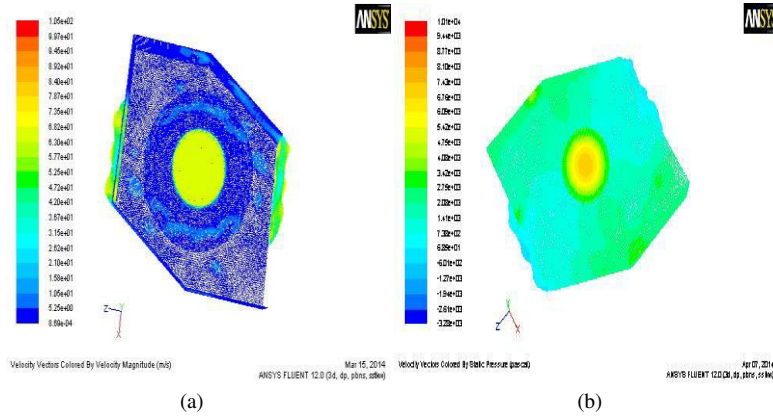


Figure: 4(a) Velocity vectors and 4(b) pressure vectors in blower with partition plate

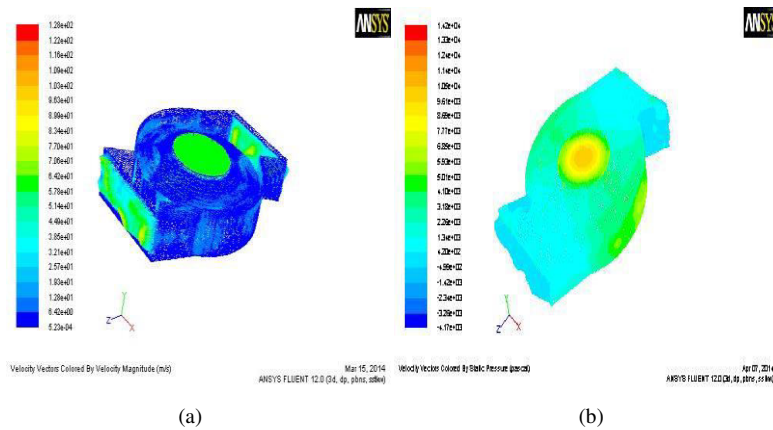


Figure: 5(a) Velocity vectors and 5(b) pressure vectors in blower with volute

From the above figures the air flow through different volute types is observed. And uneven pressure distribution is found in all the blower models. Velocities and mass flow rates in different volute geometries are tabulated below.

Table: 3 Inlet and outlet velocities and mass flow rates

Blower type	Inlet velocity (m/s)	Outlet1 (m/s)	Outlet2 (m/s)	Inlet mass flow rate(kg/s)	Outlet1 (kg/s)	Outlet2 (kg/s)
Blower without volute	77.94	65.8	64.05	2.42	1.19	1.22
	67	54.48	56.19	2.08	1.03	1.05
Blower with partition plate	77.94	60.89	63.67	2.42	1.19	1.23
	67	52.5	54.7	2.08	1.03	1.05
Blower with volute	77.94	63.8	63.7	2.42	1.20	1.22
	67	58.27	55.51	2.08	1.02	1.05

By observing the flow in blower with volute geometry it is found that air is getting obstructed between impeller and volute tongue geometry. Further new volute geometries are designed to avoid the air obstruction in the blower by modifying volute geometry parameters like radial distance between impeller and volute and tongue geometry.

By considering parameter like tongue geometry, different tongue geometries connecting outlet and volute is designed. From the numerical results, the tongue geometry 90° with both outlet and volute gave better results.

**International Journal of Innovative Research in Science,
Engineering and Technology**

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 6, June 2014

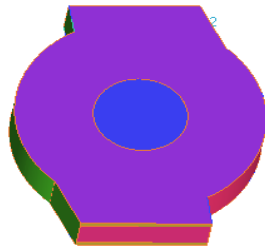
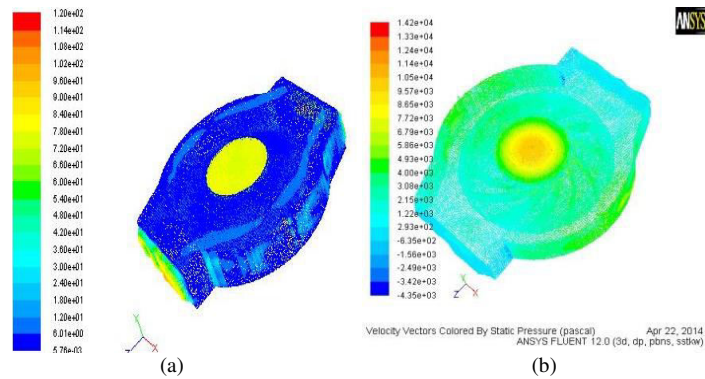


Figure:6 Volute geometry with changed tongue geometry



Considering the other parameters, the volute geometries with different radial distance is designed and is numerically found that volute geometry with 10mm increased radial distance is having better outlet velocities.

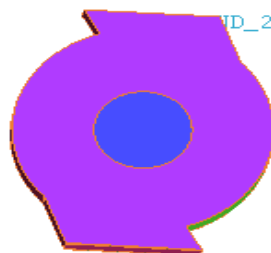


Figure: 8 Volute geometry with increased radial gap

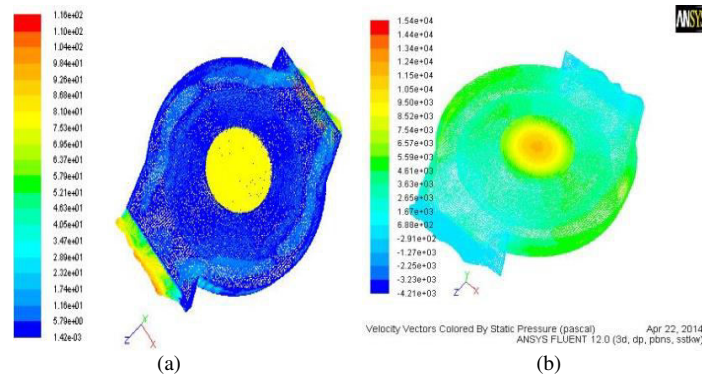


Figure: 9(a) Velocity vectors and 9(b) pressure contours in volute geometry with increased radial gap

Table: 4 Velocities and mass flow rates in new volute designs

Blower type	Inlet velocity (m/s)	Outlet1	Outlet 2	Inlet mass flow rate (kg/s)	Outlet1	Outlet2
Volute with change in tongue geometry	77.94	65.5	67.8	2.42	1.19	1.23
Volute with increase in radial distance	77.94	73.5	76.7	2.42	1.20	1.21

VI. CONCLUSION

In this study a steady state CFD simulation conducted on an industrial blower with three different volute geometries (without volute, with partition plate, with volute) having same impeller type and vane angle was investigated using computational fluid dynamics. From the numerical simulation air flow through different volute geometries are studied. It was found that air gets obstructed between the tongue geometry and the impeller.

To overcome these problems volute parameters like radial distance between impeller and volute and tongue geometry are modified. Different types of tongue geometries were designed and found that tongue geometry making 90° to the outlet is giving better result compared to others. And other parameter like radial distance between volute and impeller was also investigated and found that radial gap of 10mm between impeller and volute giving better result.

REFERENCES

- [1] Beena D. Baloni and Salim A. Channiwala(2013). Numerical investigation of the flow pattern within the impeller of a centrifugal blower. 3rd International Conference on Mechanical, Automotive and Materials Engineering, pp. 92-96, Singapor.
- [2] Chehhat Abdelmajid, Si-Ameur Mohamed and Boumeddane Boussad(2013). CFD analysis of the volute geometry effect on the turbulent air flow through the turbocharger compressor. Terra green conference, pp. 746-755.
- [3] Fluent 14.0. User's guide. Fluent Inc; 2013.
- [4] Fernández Oro.J.M, Pereiras García.B, González.J, Argüelles Díaz.K.M and Velarde-Suárez.S(2013). Numerical methodology for the assessment of relative and absolute deterministic flow structures in the analysis of impeller–tongue interactions for centrifugal fans. Computers and Fluids, Vol. 86, pp. 310–325.
- [5] Honggeng Zhu, Rentian Zhang, Guoqiang Luo and Bin Zhang(2012). Investigation of Hydraulic Characteristics of a Volute-type Discharge Passage based on CFD. International Conference on Modern Hydraulic Engineering, Vol. 28, pp. 27-32.
- [6] Jayapragasan C.N., Sumedh J. Suryawanshi, Janardhan Reddy K., (2014). Design Optimization of Centrifugal Fan of Travelling Cleaner, International Conference on trends in Product Lifecycle, Modeling, Simulation and Synthesis, PLMSS-2014, Jan6-8, VIT, Vellore.
- [7] Krishna.V, Naresh kumar.K and Prasanth kumar.M(2013). Numerical analysis of centrifugal blower using CFD. International journal of engineering and technology, Vol. 2, pp. 1461-1463.
- [8] Kui Jiao, Harold Sun, Xianguo Li, Hao Wu, Eric Krivitzky, Tim Schram, and Louis M.Larosiliere(2009). Numerical simulation of air flow through turbocharger compressors with dual volute design. Applied Energy, Vol. 86, pp. 2494-2506.

International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 3, Issue 6, June 2014

- [9] Li Chunxi, Wang Song Ling and Jia Yakui(2011). The performance of a centrifugal fan with enlarged impeller. Energy Conversion and Management, Vol. 52, pp. 2902-2910.
- [10] Mona Golbabaei Asl, Rouhollah Torabi , Ahmad Nourbakhsh.S (2009). Experimental and FEM failure analysis and optimization of a centrifugal-pump volute casing, Engineering Failure Analysis, Vol. 16, pp.1996-2003.
- [11] Qi Datong, Mao Yijun, Liu Xiaoliang and Yuan Minjian (2009). Experimental study on the noise reduction of an industrial forward-curved blades centrifugal fan. Applied Acoustics, Vol.70, pp. 1041–1050.
- [12] Oyelami.A.T, Adejuyigbe.S.B, Waheed.M.A, Ogunkoya.A.K and Iliya.D(2012). Analysis of radial-flow impellers of different configurations. The pacific journal of science and technology, Vol. 13, No. 1, pp. 24-33.
- [13] Raúl Barrio, Jorge Parrondo and Eduardo Blanco(2010). Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. Computers and Fluids, Vol. 39, pp. 859-870.
- [14] Sandra Velarde-Sua´rez, Rafael Ballesteros-Tajadura, Carlos Santolaria-Morros and Bruno Pereiras-García (2008). Reduction of the aerodynamic tonal noise of a forward-curved centrifugal fan by modification of the volute tongue geometry. Applied Acoustics, Vol. 69, pp. 225-232.
- [15] Sunsheng Yang, Fanyu Kong and Bin Chen (2011). Research on Pump Volute Design Method Using CFD. International Journal of Rotating Machinery. Vol.2011, pp. 1155-1162.
- [16] Shenbo Yu and Jun Li (2011). Design of Low-Noise Centrifugal Fan for Motorized Spindle Cooling System. International Conference on Power Electronics and Engineering Applications, Vol.23, pp. 380-386.
- [17] Srivastava.S.L and Mehra.S.M (1973). Tribological Design Considerations in Fans and Blowers. First world Conference on Industrial Tribology, pp. 245-253, New Delhi.
- [18] Thomas.R.N, Kostrzewsky.G.J, and Flack.R.D (1986). Velocity measurements in a pump volute with a non-rotating impeller. International Journal of Heat and Flow, Vol. 7, No.1, pp. 11-19.
- [19] Thundil Karuppa Raj.R and Ganesan.V (2008). Study on the effect of various parameters on flow development behind vane swirlers. International Journal of Thermal Sciences, Vol. 47, pp. 1204-1225.
- [20] Vibhakar.N and Channiware.S.A (2012). Three dimensional CFD analysis of backward curved radial tipped blade centrifugal fan designed as per unified methodology with varying number of blades. International journal of engineering trends in engineering and development, Vol. 1, No.2, pp. 12-20.
- [21] Yadav.R and Yahya.S.M (1980). Flow Visualisation Studies and the Effect of Tongue Area on the Performance of Volute casings of Centrifugal Machines. International Journal of Mechanical Science, Vol. 22, pp. 651-660.