International Journal of Engineering Research
Volume No.4, Issue No.4, pp : 216-221

ISSN:2319-6890)(online),2347-5013(print)
01 April 2015

Numerical Analysis and Experimental Verification of an Industrial Cleaner

Jayapragasan C N , Dr. K Janardhan Reddy
VIT University, Chennai, India.
jayapragasan.cn2013@vit.ac.in, kjanardhanreddy@vit.ac.in

Abstract: Cleaner blowers play an important role in the proper functioning of fluffs collection system for industries. 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 modeled and analyzed based on the air flow through the blower. The numerical investigation revealed that the blowers with modified volute geometries have better outlet velocities. For this work, volute is modeled using Solid Works and meshed in ICEM CFD. The post processing is carried out using CFD POST. The results obtained were then validated and compared the experimental values of the different models of the blowers.

Keywords- Centrifugal fan, Volute geometries, Computational Fluid Dynamics (CFD), Experimental methods

I. Introduction

Cleaner main aim is to deliver the air with an appreciable rise in pressure to overcome resistance in the flow and they can achieve high pressures. The industrial cleaners are used to keep the inside the plant and overall machines clean by travelling over the machinery. These Cleaner blowers suck the air from the inlet and discharge them through the outlets in order to clean the machines and maintain the quality of the product. Parts like fan, volute geometry, vane angle and vane type have an effect on performance of the cleaner. Volute is a passage located around the fan which collects the flow from the fan and delivers at the exit attachment. Volute geometry is one of the main performance parameter of a cleaner and it is varies based on the need of discharge. In volute, tongue geometry and radial distance between impeller and volute are general performance parameters. In this paper, the cleaner construction was analysed to by modifying the volute design of the cleaner blower and aimed to compare the analytical and Experimental results to conclude the performance of an industrial cleaner.

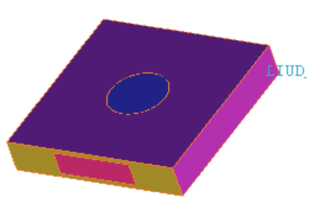
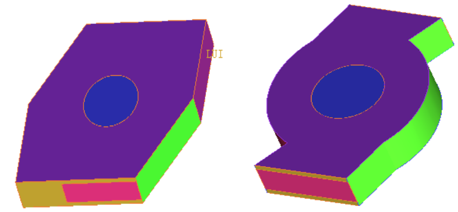


Fig. 1 Cleaner without volute



(a) (b)

Fig. 2 (a) Cleaner with Partition plate (b) Cleaner 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.

Notation	Nomenclature used
CFD	Computational Fluid Dynamics
rpm	Rotations per minute
mm	millimeters
m	meters
MRF	Moving Reference Frame
mmwc	Millimeters of water column
Q	Discharge
D	Outer diameter of impeller
N	Fan speed
PTF	Total pressure developed by fan
m/s	Meter per second
cfm	Cubic feet per meter
P_s	Suction Pressure

Table. 1 Geometrical dimensions of the designed blowers

S.no	Parameter	Dimension
1	Impeller diameter	380mm
2	Inlet diameter	180mm
3	Outlet dimensions	250X78mm (Dia 122 mm Outlet)
4	Impeller rotational speed	383.27 rad/sec
5	Blower casing	550X550X106mm
6	Partition plate dimension	313X106mm
7	Volute diameter	440mm

II. Literature Survey

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 [I]. 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 [II]. 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 [IV].

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 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 [VII]. 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 [VIII].

Prior to mesh the fan is modeled in SLIDOWOKS. The fan needs to be modeled correctly and geometry cleanup required to be done in ICEM CFD for deleting unwanted surfaces. The selection of meshing technique plays a vital role in the analysis. The different mesh parameters are varied to get the best meshing possible [V]. The flow is then simulated by using the popular RANS (Reynolds Averaged Navier Stokes) equation with a proper turbulence model [IV].

Table. 2 Boundary Condition

Materials	Air and aluminum
Turbulent model	k-omega Shear Stress Transport
Navier strokes equation	SIMPLE algorithm
Moving reference frame	Impeller
Inlet conditions	Velocity inlet
Outlet condition	Pressure outlet
Flow	Steady state flow
Solver	Pressure based solver

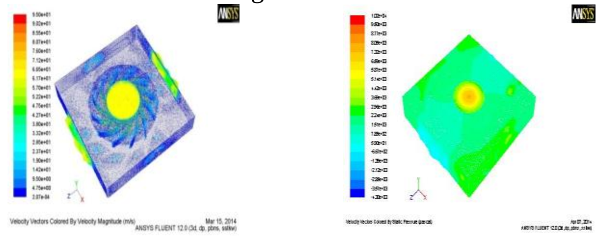
III. METHODOLOGY

In this study following methodology was tagged with:

- Literature survey
- Analytical results and discussion
- Experimental Study of the volute, partition plate and modified volute

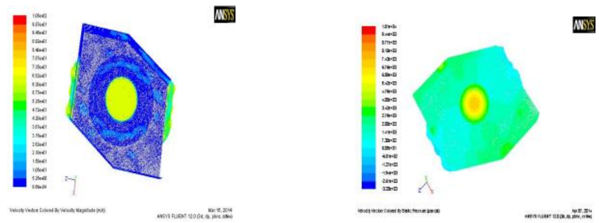
A. Analytical Results and discussion

Various values like total pressure, Mass flow rate, velocity are obtained. Fig. 3-5 shows the velocity and pressure vectors obtained in different volute geometries



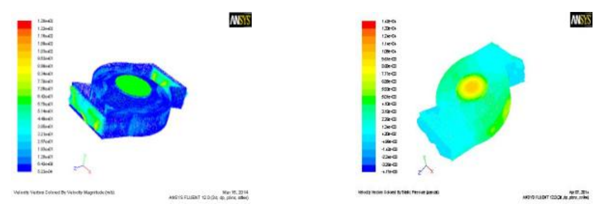
(a) (b)

Fig. 3 (a) Velocity and 3(b) pressure vectors in blower without volute



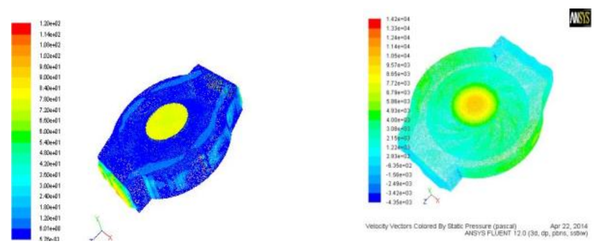
(a) (b)

Fig. 4(a) Velocity and 4(b) pressure vectors in blower with partition plate



(a) (b)

Fig. 5(a) Velocity and 5(b) pressure vectors in blower with volute



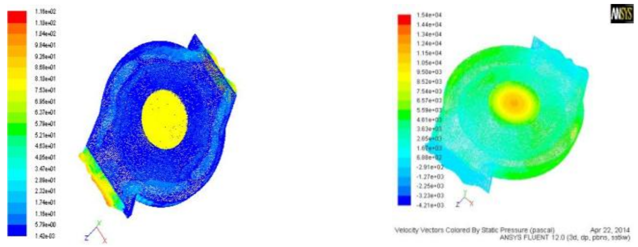
(a) (b)

Fig. 6(a) Velocity and 6(b) pressure vectors in in changed tongue geometry

Table. 3 Analytical results of Inlet and outlet velocities of 2.2 and 1.5 kw

Blower type	Inlet Power (kw)	Inlet velocity (m/s)	Outlet1 (m/s)	Outlet2 (m/s)
Blower without volute	2.2	77.94	65.8	64.05
	1.5	67	54.48	56.19
Blower with partition plate	2.2	77.94	60.89	63.67
	1.5	67	52.5	54.7
Blower with volute	2.2	77.94	63.8	63.7
	1.5	67	58.27	55.51

 ISSN:2319-6890)(online),2347-5013(print)
01 April 2015



(a) (b)

Fig. 7(a) Velocity and 7(b) pressure contours in volute geometry with increased radial gap

Table. 4 Analytical results of Velocities in new volute designs for 2.2 kw

Blower type	Inlet Power (kw)	Inlet velocity (m/s)	Outlet1	Outlet 2
Volute with change in tongue geometry	2.2	77.94	65.5	67.8
Volute with increase in radial distance		77.94	73.5	76.7

B. Post Processing

Once the convergence was reached in FLUENT, the post processing is done in CFD-POST to obtain the values like static pressure, various velocity contours and total mass flow rate.

C. Experimental Setup

As per the Standard Procedure the Experimental set up was developed and determining the air delivery and pressure of the blower. The fan inlets shall be attached with a parallel duct having the same cross section as the fan outlet and length equal to twice its diameter.

The four side tappings at plane shall be equally spaced at 90 deg. on the cylindrical duct. The four side tappings shall be connected to the manometer, each connection being of the same length, bore and arrangement of tubing to minimize the effect of flow due to difference of pressure at the tappings. The other limb of the manometer shall be opened to the ambient atmosphere and the manometer reading shall be taken as equal to the average static pressure in the airway. A resistance comprising a screen having evenly spaced aperture of uniform size, not exceeding D/20 should be filled at a distance D from the commencement of the cylindrical portion of the inlet. The screen may be composed of one or more layers of even wire or fabric supported by a wire guard.

Fig. 8 Experimental Setup

Fig. 9(a) with volute 9(b) without volute

D. Mathematical Expression

Density of air at conical inlet is taken as a density of air at ambient condition.

$$\rho c = \frac{P_b \times m_e}{R \times T_a} = kg / m^3$$

R = universal gas constant

The flow rate through the blower was obtained from the discharge head characteristics of the orifice,

$$V_a = \frac{\sqrt{2(P_1 - P_2)}}{\sqrt{\rho_{air}}\,(1-(d/D)^4)}$$

Average velocity of air through the duct

$$U_t = \frac{q_a}{A}$$

Velocity Pressure at the test section

$$q = C_d \times (\pi/4)\, d^2 \times V_a$$

Static Pressure readings at the beginning of the two discharge pipes are just outside the chamber.

$$P_{air} = p/RT$$

Static pressure of the blower Exit = Mean of the static pressure at the two delivery.

$$\text{velocity head} = v^2/2g$$

Blower output power = 9.81 x q x Δh

With the help of the above mathematical formulae the flow rate at test conditions, motor input power, flow velocity in the blower outlet duct at test conditions, average velocity of air through the duct are calculated and were used for further calculations. Later on the analytical and experimental values were compared to find out the optimum result giving blower volute parameters.

E. Experimental Data

Table. 5 Experimental Data of Blower without volute 2.2kw

Sl.No	Test speed (rpm)	Blower Head hs (static)ΔP (mm of wc)		Flow rate (m3/hr)	ht (mmwc)	Motor Output power (w)	Blower efficiency (%)
		Duct1	Duct 2				

1	3695	224	217.8	486	224	1042	33.5
2	3672	192.4	188.9	799	200	1170	43.7
3	3660	176.4	171.3	1118	188	1255	53.7
4	3674	153.4	149.2	1305	170	1330	52.2
5	3661	132.8	128.4	1521	160	1375	56.7
6	3648	104	98.7	1657	143	1421	53.5
7	3679	76.8	72.1	1780	128	1449	50.3
8	3678	44.5	39.4	2032	103	1550	43.3
9	3682	38.4	33.5	2201	83	1589	36.7
10	3699	21.5	17.4	2361	72	1704	31.9

Graph. 1 Performance of without volute 2.2kw

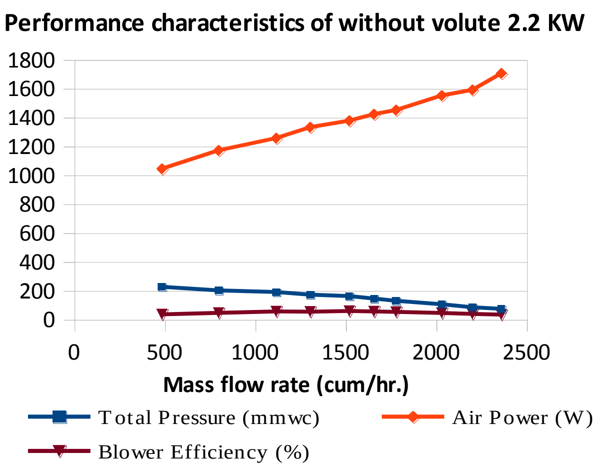


Table. 6 Experimental Data of Blower with partition plate 2.2kw

Sl.No	Test speed (rpm)	Blower Head hs (static)ΔP (mm of wc)		Flow rate (m3/hr)	ht (mmwc)	Motor Output power (w)	Blower efficiency (%)
		Duct1	Duct 2				
1	3687	208	201	424	208	1065	26.5
2	3669	196	191	621	201	1151	34.7
3	3667	184	178	823	192	1212	41.7
4	3672	164.5	157.8	1269	173	1291	54.6
5	3664	142.7	134.6	1424	163	1334	55.7
6	3641	132	128	1543	166	1403	58.5
7	3687	98.7	91.4	1742	137	1446	52.8
8	3682	65.5	59.4	2030	117	1508	50.6
9	3689	51	44.8	2197	93	1565	41.8
10	3699	34.7	29.5	2397	84	1681	38.3

Graph. 2 Performance of with partition plate 2.2kw

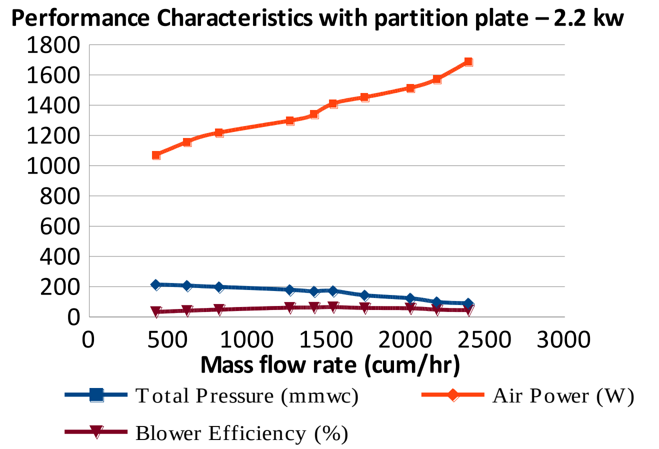
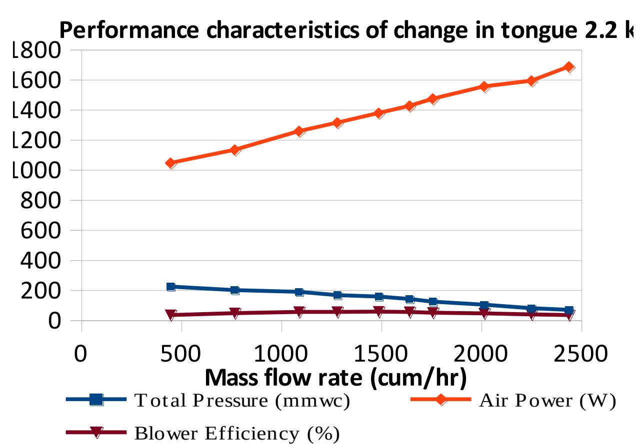


Table. 7 Experimental Data of Blower with change in tongue 2.2kw

Sl.No	Test speed (rpm)	Blower Head hs (static)ΔP (mm of wc)		Flow rate (m3/hr)	ht (mmwc)	Motor Output power (w)	Blower efficiency (%)
		Duct1	Duct 2				
1	3695	220	215	448	220	1042	30.3
2	3672	190.1	187	769	196	1130	42.8
3	3662	175.6	169	1091	185	1254	51.5
4	3674	150.5	148.5	1281	163	1310	51.1
5	3664	130.5	127.6	1489	154	1374	53.3
6	3645	101	97.5	1643	137	1422	50.8
7	3679	75.5	72	1759	120	1469	45.9
8	3675	42.5	38.5	2016	100	1551	41.5
9	3680	36	32	2252	76	1589	34.5
10	3695	19.5	18	2440	65	1683	30.1

Graph. 3 Performance of change in Tongue 2.2kw



ISSN:2319-6890)(online),2347-5013(print)
01 April 2015

Graph. 4 Performance of increase in radial clearance 2.2kw

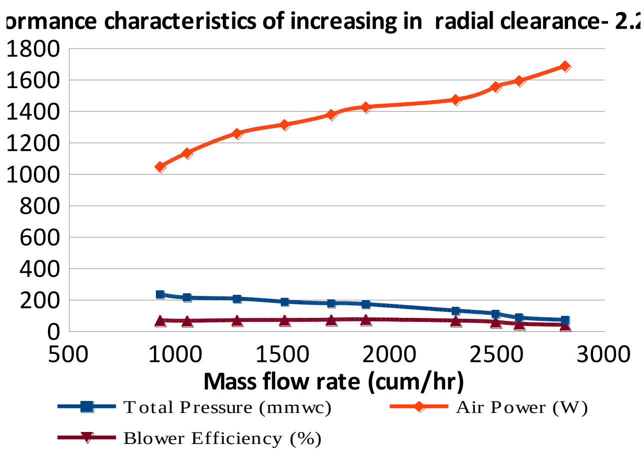


Table. 8 Experimental Data of Blower with increase in radial clearance 2.2kw

Sl.No	Test speed (rpm)	Blower Head hs (static)ΔP (mm of wc)		Flow rate (m3/hr)	ht (mmwc)	Motor Output power (w)	Blower efficienc y (%)
		Duct1	Duct 2				
1	3695	231	228	927	232	1039	66.1
2	3672	207.5	198	1054	210	1130	62.9
3	3662	197.6	184.5	1287	203	1256	66.9
4	3674	174.5	167.5	1509	184	1314	68
5	3664	156.7	142	1727	174	1379	70
6	3645	132.1	128.6	1888	169	1422	71.8
7	3679	84.3	79	2308	127	1467	64.1
8	3675	51.2	46.8	2495	108	1549	55.7
9	3680	43.5	39.1	2605	83	1586	43.6
10	3695	24.2	21.2	2819	69	1680	36.9

Table. 9 Comparison of Analytical and Experimental results

Blower type	Analytical		Experimental	
	Mass flow rate cum/hr	Total Pressure in mmwc	Mass flow rate cum/hr	Total Pressure in mmwc
Without volute	2780	240	2361	224
With partition plate	2600	240	2394	208
Change in tongue	2783	240	2440	220
Increase in radial clearance	3136	240	2819	232

Table. 10 Comparison of Experimental results

Blower type	Input power in kw	Motor output in watts (Max)	Efficiency % (Max)
Without volute	2.2	1704	56.4
	1.5	1462	45.5
With partition plate	2.2	1681	58.5
	1.5	1444	37.7
Change in tongue	2.2	1683	53.3
Increase in radial clearance	2.2	1680	71.8

IV. RESULTS AND DISCUSSION

Various parameters like static pressure, total pressure, air power, velocity were calculated with available experimental data and the analytical data available with us. Table 5 to Table 10 gives us the details of those calculations.

Following graphs show the air delivery characteristics of the air of various models based on the experimental data. The graphs are plotted with total pressure, power and efficiency verses mass flow rate. It has been observed from the graphs that as the mass flow rate at the outlet increases the total pressure and efficiency decreases accordingly. It has been also observed that there is a change in velocity.

V. CONCLUSION

This paper deals with the approach of comparing the analytical and experimental results for various models of blowers developed by us. The CFD approach helps to improve the results in the present system. Result validation helps us to substantiate our physical model by exploiting the analytical software ICEM-CFD, CFD-FLUENT and CFD-POST. The obtained results are satisfactory in the increase in radial clearance model of the blower as the range of increased mass flow rate is 19.3% and efficiency is 5.1% without increasing the power. Also from the available results, both analytical and experimental, it can be concluded that the increased in radial distance model gives the optimum results and can be used in the industry for better functionality in real working conditions due to high pressure and mass flow rate requirement.

VI. FUTURE SCOPE

This area of research has a vast scope for further studies and implementation. The design optimization of the model can be done by conducting many trials with available models, better results can be obtained and validation for the same can be done. For different parameters and levels of the models available with us good results can be obtained. The blower performance can be checked continuously after varying the parameters and experimental values can be compared with the obtained analytical values. With the help of satisfactory results the design variation can be implemented in the current model.

REFERENCES

i. *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.*

ii. *Chehhat Abdelmadijid, 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.*

iii. *Fluent 14.0. User's guide. Fluent Inc; 2013.*

iv. *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.*

v. *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.*

vi. *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.*

vii. *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.*

viii. *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.*

ix. *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.*

x. *X. 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.*