International Journal of Research in Advent Technology, Vol.6, No.9, September 2018
E-ISSN: 2321-9637
Available online at www.ijrat.org

Effect of Flow Distortion in Inlet Duct on the Performance of Centrifugal Blower

Kalpesh W. Shirsath[1], S.R. Patil[2], Sumit M. Naik[3]
PG Student [1,3], Assistant Professor[2]
Department of Mechanical Engineering [1,2,3], AISSMS COE, SPPU, Pune, India[1,2,3].
Email: kalpeshshirsath17@gmail.com[1], srpatil@aissmscoe.com[2], smtnk72@gmail.com[3]

Abstract- In this paper, the effect of flow distortion in the inlet duct on the performance of centrifugal blower has been numerically and experimentally investigated. Numerical analysis was carried out on centrifugal blower with hub and tip distorter with 25 % blocked area in inlet duct. ANSYS Fluent 16.0 software was used for numerical analysis. Flow rate, total pressure, blower efficiency and discharge head were taken as performance parameters. Numerical analysis was also done for original centrifugal blower having straight circular inlet duct. According to IS: 4894-1987 experimentation was carried out on original centrifugal blower and modified centrifugal blower with tip and hub distorter. Performance parameters were evaluated from numerical and experimental results. Validation of numerical results with experimental results was done. The results indicated that flow distortion degrades the performance of centrifugal blower in terms of flow rate, blower efficiency and discharge head.

Keywords- Centrifugal Blower; Hub distorter; Numerical analysis; Performance parameters; Tip distorter.

1. INTRODUCTION

The centrifugal blowers are nothing but one of the types of turbo machines which are primarily used for accelerating air continuously to accommodate large volume of air with slight increase in static pressure. Blowers are extensively used in industrial and commercial applications such as shop ventilation, material handling, boiler applications and vehicle cooling systems.

Distortion in inlet duct refers to the non-uniform flow upstream of turbo-machines. Inlet flow distortion can be detrimentally affecting the performance of the fans, blowers and compressors in terms of static efficiency and pressure rise. Inlet flow distortions can also affect the stability limits of these turbo-machines in terms of aerodynamic and aero-elastic properties. [1] The problem of determining how much a turbo-machine is affected by poor inlet flow conditions is an old one. It arises in HVAC design and installation, layout of hydraulic pumping systems and in the design of inlet configurations for both ground-based and airborne gas turbine compressors. [2]

The flow non-uniformity is frequently generated at the entrance of impeller called as inlet distortion, which is divided mainly into two types of distortion. One is radial distortion and other is circumferential distortion. The radial distortion is further subdivided into tip distortion and hub distortion corresponding to the regions where obstacle is placed. Hub distortion is observed when axisymmetric obstacles are placed at a center portion of an inlet duct, such as a tachometer pick up and a hub cover. Axisymmetric boundary layers of an inlet duct or a return channel or an orifice plate are responsible for tip distortion. Non-axisymmetric obstacles are mainly responsible for circumferential distortion. [3-4] In turbo-machineries such as compressors, due to limited space curved inlet duct are generally used to provide and guide air to casing. Such a curved inlet duct can generate flow distortion in compressor therefore deteriorate compressor stage performance. [5]

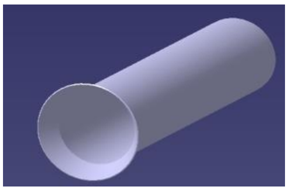
Inlet flow distortions occurred due to various inlet duct configurations including obstacles, struts and bending duct can be seriously affected performance of centrifugal compressors. Kim *et al.* have conducted experimentation on centrifugal compressor with straight circular duct and a 90° curved duct with nozzle shape to compare stage efficiency. [6] Patil *et al.* have done experimentation and numerical simulation to investigate the effect different volute tongue clearances such as 6 %, 8 %, 10 % and 12.5 % of impeller diameter on the performance of centrifugal blower. Numerical simulation was done using ANSYS Fluent software. They used mathematical model as standard k-ε turbulence model for solving 3D Reynolds Averaged Navier Stokes equations. [7] Girish *et al.* have done CFD simulation of centrifugal blower using ANSYS CFX. They have obtained velocity vector and pressure counter plot for different blower speeds to evaluate performance parameters. [8] Naseri *et al.* have obtained experimental results which determine adverse impact of inlet total pressure distortion on the performance of a micro gas turbine. Experimentation was carried out on the gas turbine which was exposed to inlet gas flow with 60^0, 120^0 and 180^0 circumferential distortion patterns. The performance of the gas turbine has been evaluated and compared with straight inlet gas flow case. [9]

2. DETAILS OF CENTRIFUGAL BLOWER

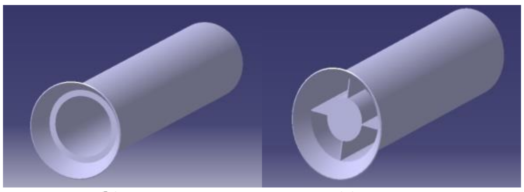
The centrifugal blower setup is specified according to IS: 4894-1987. Main components of original centrifugal blower contain casing, straight circular inlet duct, outlet duct and impeller. In this paper original centrifugal blower is indicated by ‘M_0’. The various parameters of original centrifugal blower are given in Table 1.

Table 1. Parameters of original centrifugal blower

Sr. No.	Parameters	
1	Impeller outlet diameter (mm)	280
2	Impeller inlet diameter (mm)	140
3	No. of blades	12
4	Impeller blade type	Backward inclined
5	Inlet duct diameter (mm)	130
6	Inlet duct length (mm)	550
7	Motor speed (RPM)	2800



(a)



(b) (c)

Fig. 1. Inlet duct of (a) centrifugal blower M_0, (b) centrifugal blower M_1, (c) centrifugal blower M_2

To know the effect of distorted inflows on performance of centrifugal blower, obstacles such as tip and hub distorter were placed inside inlet duct without changing other parameters. The tip and hub distorter with 25 % blocked area in inlet duct of centrifugal blowers are defined as modified centrifugal blowers ‘M_1’ and ‘M_2’ respectively. Fig. 1 differentiates the inlet duct of centrifugal blowers. The changes made in inlet duct of centrifugal blower are shown in Fig. 1 (b) and (c).

3. NUMERICAL ANALYSIS

ANSYS Fluent 16.0 software was used for numerical analysis of centrifugal blowers. The three dimensional model of centrifugal blower having only casing and impeller were first created in Solid Works 2015 software. Further modification such as modeling of inlet duct, outlet duct and passages were done in Design Modular of ANSYS Fluent. Rotating zone and stationary zone were solved by Multiple Frames of Reference in rotating reference frame and stationary reference frame respectively. A rotating reference frame was set to impeller wheel with rotational speed and other parts were referred as stationary frame.

In meshing process, tetrahedral elements were selected for the elements of rotating impeller and hex elements were defined for the inlet duct, outlet duct and casing of centrifugal blower. Meshing size of element was taken as 5 mm.

The flow of air inside centrifugal blower is generally turbulent flow. Hence to solve 3D Navier-Stokes equations standard k-epsilon model was selected. Here k represents turbulent kinetic energy and epsilon denotes turbulent dissipation. Inlets, outlets and impeller of original and modified centrifugal blowers were taken as boundary conditions. The atmospheric pressure was defined as the inlet boundary condition at the entrance of the inlet duct. Also outlet boundary condition was set as atmospheric pressure at the exit of outlet duct. Rotational motion was given to impeller and considered as moving wall. Sufficient iterations were given until solution was converged.

After convergence of solution numerical results were obtained in vector plots, counter plots and streamlines. Velocity vector and pressure counter plots for centrifugal blowers M_0, M_1 and M_2 are shown in Fig. 2 to Fig. 7 respectively.

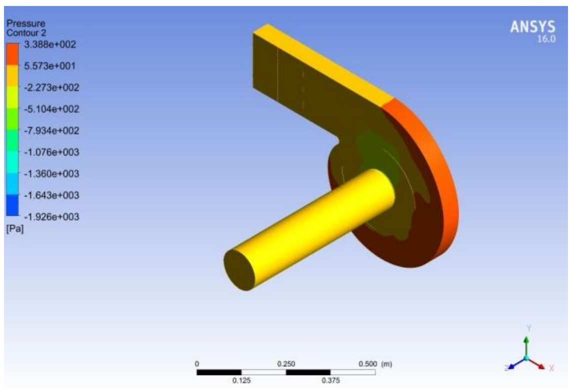


Fig. 2. Pressure counter plot of centrifugal blower, M_0

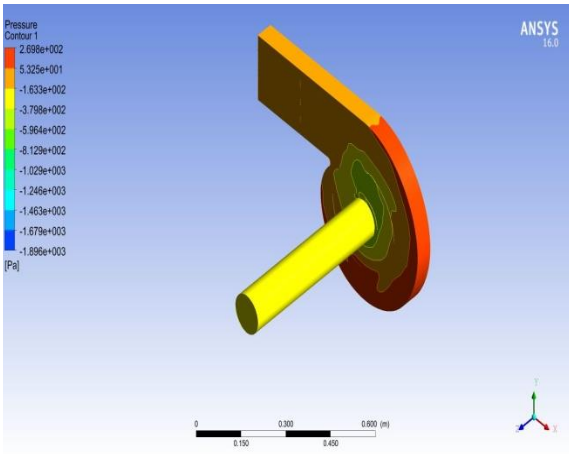


Fig. 3. Pressure counter plot of centrifugal blower, M_1

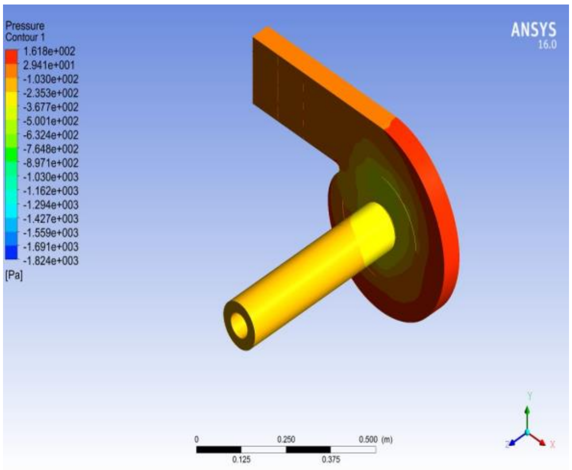


Fig. 4. Pressure counter plot of centrifugal blower, M_2

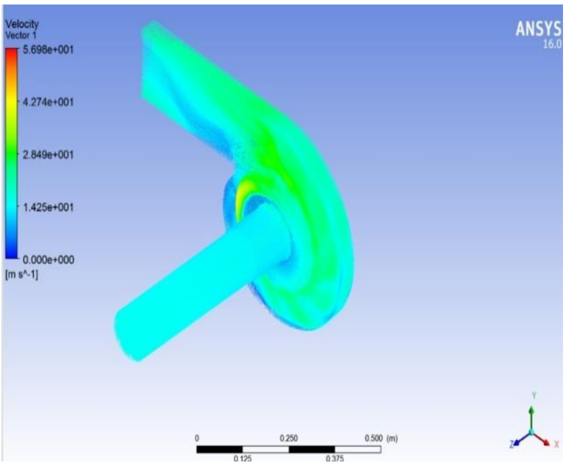


Fig. 5. Velocity vector plot of centrifugal blower, M_0

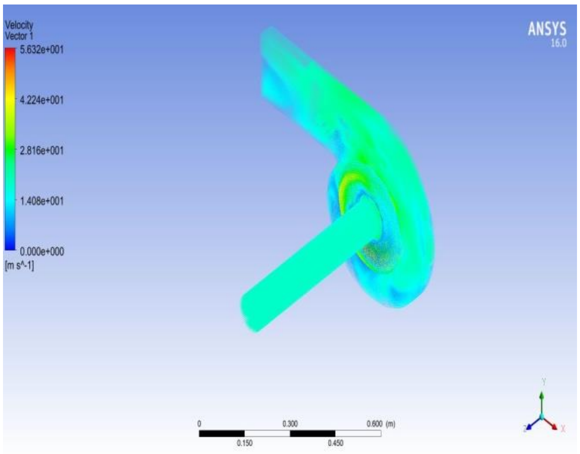


Fig. 6. Velocity vector plot of centrifugal blower, M_1

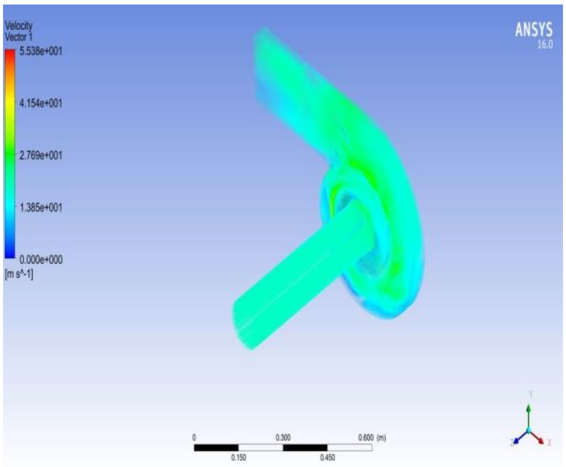


Fig. 7. Velocity vector plot of centrifugal blower, M_2

4. EXPERIMENTATION

Experimentation was carried out on original and modified centrifugal blowers. An experimental setup of centrifugal blower was according to IS: 4894-1987. Photograph of experimental setup of blower is shown in Fig. 8. An electric motor of 0.5 HP with three-phase was used to drive the centrifugal blower. The input of four way inlet and one way outlet valve was connected to inlet duct and also output of this valve was connected to one limb of the manometer. U-tube manometer was connected through four side tappings at the entrance side of the inlet duct to measure static pressure which later used to calculate flow rate as shown in Fig. 8.

At exit side of inlet duct, measurement of average static pressures was done by using U-tube manometer. It was also connected at the outlet of the blower to calculate outlet total pressure. Dimmer-stat was used to vary voltage from zero to rated value of 415 V; hence rated rpm of motor is achieved. Current and voltage measurements were done by analogue ammeters and voltmeter respectively.

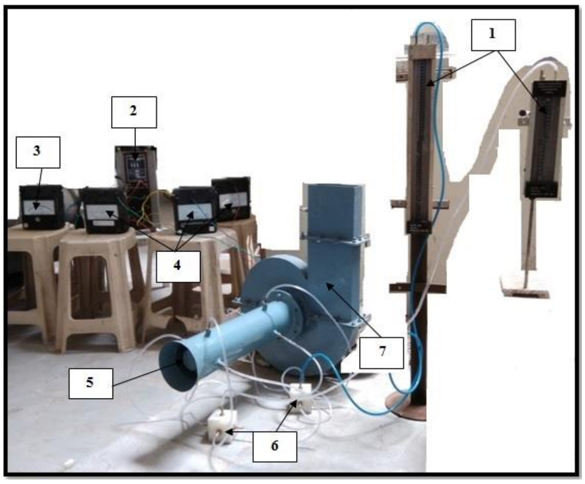


Fig. 8. Photograph of the experimental setup
1: U-tube manometers, 2: Dimmer-stat,
3: Voltmeter, 4: Ammeters, 5: Inlet duct hub distorter,
6: four way inlet one way outlet valves,
7: Modified centrifugal blower, M_2

5. RESULT AND DISCUSSION

Numerical and experimental results of original and modified centrifugal blowers for flow rate, total pressure, blower efficiency and discharge head are shown in Fig. 9 to Fig. 12 respectively. .

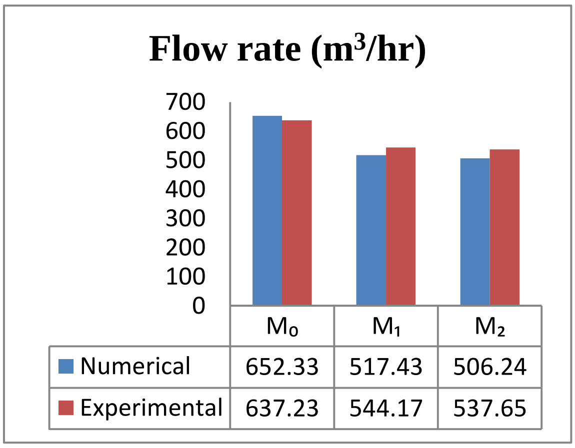


Fig. 9. Numerical and experimental results for flow rate

Both numerical and experimental results show that flow rate is maximum in original centrifugal blower, M_0 as effective flow area is more in case of original centrifugal blower. Modified centrifugal blower M_2 shows least flow rate as compared to others. From Fig. 10 it can be seen that total pressure is occurred to be more in case of modified centrifugal blower, M_1. Increment in total pressure is seen in modified centrifugal blowers because of inlet static pressure along with velocity pressure is increased.

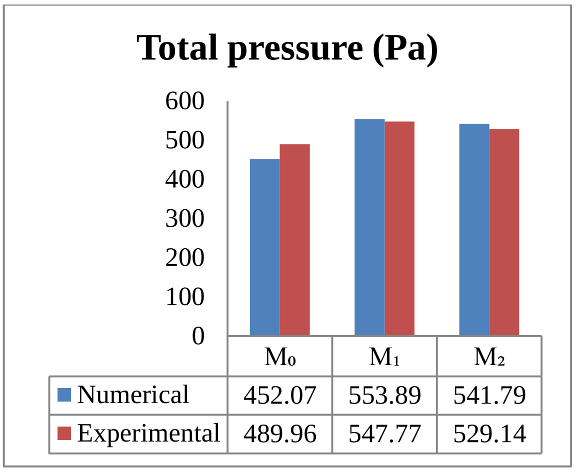


Fig. 10. Numerical and experimental results for total pressure

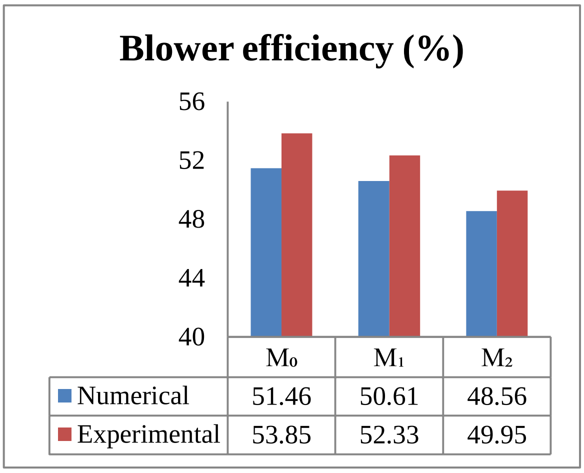


Fig. 11. Numerical and experimental results for blower efficiency

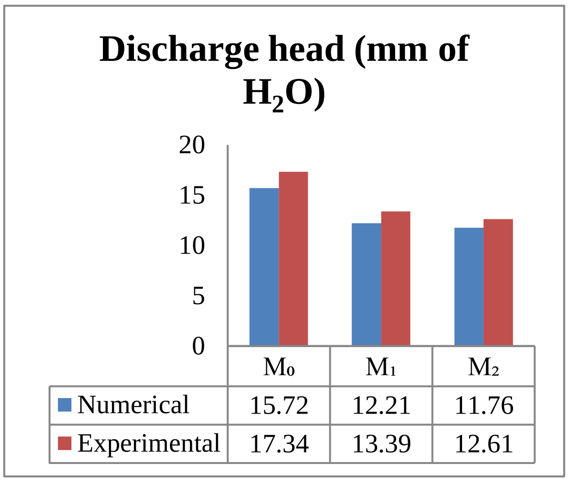


Fig. 12. Numerical and experimental results for discharge head.

Use of hub and tip distorter degrades performance of centrifugal blower in terms of blower efficiency. Modified centrifugal blowers M_1 and M_2 shows less efficiency as compared to original centrifugal blower M_0. Efficiency is occurred to be minimum in modified centrifugal blower M_2 as shown in Fig. 11 Reduction in discharge head is occurred in both modified centrifugal blowers M_1 and M_2 which can be seen in Fig. 12. Hence modified centrifugal blowers with tip and hub distorter are unable to build up static pressure rise.

Comparison of numerical results with experimental results is done for validation purpose. From Table 2 and Table 3, it can be seen that numerical results has well matched with experimental results of original and modified centrifugal blowers. As deviations occurred between numerical results and experimental results are within 6 %, hence it ensures validity.

Table 2. Comparisons of numerical results with experimental results of modified centrifugal blower, M_1 for performance parameters

Performance parameters	Numerical results	Experimental results	Deviation
Flow rate (m^3/h)	517.43	544.17	4.91 %
Total pressure (Pa)	553.89	547.77	1.12 %
Blower efficiency (%)	50.61	52.33	3.28 %

Table 3. Comparisons of numerical results with experimental results of modified centrifugal blower, M_2 for performance parameters

Performance parameters	Numerical results	Experimental results	Deviation
Flow rate (m^3/h)	506.24	537.65	5.85 %
Total pressure (Pa)	541.79	529.14	2.39 %
Blower efficiency (%)	48.56	49.95	2.78 %

6. CONCLUSION

The study was conducted on how inlet flow distortion affects the performance of centrifugal blower. In modified centrifugal blower with hub distorter minimum flow rate is occurred and it is decreased by 22.39 % than original centrifugal blower at rated rpm. Modified centrifugal blower with tip and hub distorter shows increment in total pressure than original centrifugal blower by 22.52 % and 19.84 % respectively. Modified centrifugal blower with hub distorter shows least efficiency as compared to others. It is decreased by 5.63 % with respect to original centrifugal blower. Modified centrifugal blower with hub distorter shows larger drop in discharge head and which is 25.19 % with respect to original centrifugal blower. Compared with undistorted case performance of centrifugal blower falls in order of tip and hub distortions.

REFERENCES

[1] Wenqiang Zhang, Mehdi Vahdati, “Influence of the Inlet Distortion on Fan Stall Margin at Different Rotational Speed”, Proceedings of Shanghai, Global Power and Propulsion, 2017.

[2] T. Wright, S. Madhavan, J. Dir, “Centrifugal fan performance with distorted inflows”, Journal of Engineering for Gas Turbines and Power, volume 106, pages 895-900, 1984.

[3] Ariga I., Kasai N., Masuda S., Watanbe Y., Watanbe I., “The effect of inlet distortion on performance characteristics of a centrifugal compressor”, Journal of Engineering for Gas Turbines and Power, pages 223-230, 1983.

[4] Dr.Basharat Salim, “Performance of a radial blower with curved inlet duct”, International Journal of Emerging Technology and Research, volume 2, Issue 4, pages 99-118, 2015.

[5] Ben Zhao, Harold Sun, Leilei Wangb, Manxiang Song, “Impact of inlet distortion on turbocharger compressor stage performance”, Applied Thermal Engineering, volume 124, pages 393-402, 2017.

[6] Y Kim, A Engeda, R Aungier and G Direnzi, “The influence of inlet flow distortion on the performance of a centrifugal compressor and the development of an improved inlet using numerical simulations”, Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, pages 323-338, 2001.

[7] Sunil R. Patil, Sandip T. Chavan, Nitin S. Jadhav, Shivaraj S. Vadgeri, “Effect of volute tongue clearance variation on performance of centrifugal blower by numerical and experimental analysis”, Materials Today: Proceedings, volume 5, pages 3883-3894, 2018.

[8] Girish A R, Prajwal Sandyal, Lokesh. K.S, Varun N, "CFD simulation of centrifugal blower using ANSYS CFX", National Conference on Advances in Mechanical Engineering Science, pages 34-38, 2016.
[9] Alireza Naseri, Shervin Sammak, Masoud Boroomand, Alireza Alihosseini, Abolghasem M. Tousi, "Experimental Investigation Of Inlet Distortion Effect On Performance Of A Micro Gas Turbine", Journal of Engineering for Gas Turbines and Power, 2017.

(A.1)