Performance Evaluation of a Centrifugal Blower for Different Rotational Speeds of an Impeller by Numerical Analysis

[1]Sumit M. Naik, [2]S.R.Patil [3]Kalpesh W. Shirsath
[1]PG Student, [2]Assistant Professor, [3]PG Student
[1]Department of Mechanical Engineering,
[1]AISSMS COE, SPPU, Pune, India

Abstract : A centrifugal blower, in case of space constraints the rotational speed of an impeller is a design parameter to fine tune the performance of it. This study has been undertaken to investigate how the rotational speed of an impeller affects the performance of centrifugal blower by numerical analysis. Solid modelling software SOLIDWORKS 2016 is used to create the solid model and numerical analysis is carried by using analysis software ANSYS 16.0. The performance is evaluated for speeds of 1500 rpm, 2000 rpm, 2500 rpm and 2800 rpm. From numerical analysis the different performance parameters flow rate, total pressure, shaft power and efficiency are obtained. The experimental analysis was performed by the previous researcher on the same configuration setup as considered for the numerical analysis in the present work. The numerical analysis results are validated with the experimental analysis results. It is observed that there is an increase in the value of the performance parameters as the speed is increased from 1500 rpm to 2800 rpm.

***IndexTerms* - Centrifugal blower, Rotational speed, Numerical analysis, Performance parameters.**

1. INTRODUCTION

The blowers are used to achieve higher pressures as compared to the fans. In the industrial vacuum systems their use includes producing the negative pressures. Depending upon how the air is flowing through the blower, the two primary types include axial and centrifugal type. In the axial type of blowers, the flow is parallel to axis of the blower. They are used for applications which have relatively lower pressures and higher flow rates. Sickle, airfoil, variable pitch and paddle are the blade shapes of these blowers. Centrifugal blowers produce high pressure with high efficiency and can also be used where the operating conditions are harsh. Pressure is created to move air against a blocking caused by dampers, ducts and other components. The pressure of the supplied air is increased owing to kinetic energy of the impeller blades. They are quiet, reliable, sturdy and able to operate in different range of conditions. They are used in various industries to carry the materials or gases and in ventilation systems of the buildings.

The major parts of the centrifugal blower include the housing or volute casing, drive shaft, outlet ducts, impeller blades, inlet ducts and a drive mechanism. The inlet and outlet ducts are attached to the casing of the centrifugal blower with the help of nut and bolt arrangement. The energy is transferred owing to the rotary motion of impeller. The impeller is followed by stationary casing in which energy transformation takes place. The casing decides the pressure rise and size in the system. In a centrifugal blower, the air enters in radial direction and goes out in tangential direction thus providing high discharge and static pressures. Owing to the rotation of the impeller inside the casing, low pressures are created at the inlet thus allowing the air to flow inside. Then the air travels through the blade passages and moves out. The air moves out due to the deflection of blades and centrifugal force and thus proceeds to the volute casing. The volute casing maintains the discharge of the flow and guides out the air through the outlet duct. Depending upon the application, the impeller blade can be radial, forward or backward type. In this study a backward type impeller is used.

Experimental analysis is time consuming and expensive due to constructing and testing physical prototypes in a hit or miss process thus proving to be non-profitable to the manufacturers. Thus without the need to manufacture the prototypes, the performance can be predicted by various computational methods. These methods not only save time and expenditure but also offer reliable solutions. For this reason Computational Fluid Dynamics (CFD) analysis with suitable turbulence modelling currently has more benefits than the experimental works. A complete performance evaluation of a specific design can be obtained by an engineer by the numerical simulation. The fluid behaviour in the machine can also be predicted correctly with help of numerical simulation.

When the speed is varied, the performance of the blower can be approximately predicted by the fan laws. It is due to the fact that in fan laws the speed of the impeller is in some relation with the performance parameters flow rate, total pressure and shaft power. The fan laws equations are as follows

Fan law 1 states that the flow rate is directly proportional to the rotational speed of the impeller as shown in Equation 1,

$$\frac{Q_2}{Q_1} = (\frac{N_2}{N_1})^1 \tag{1}$$

where Q represents the flow rate while N represents the speed of blower.
Fan law 2 states that the pressure is directly proportional to square of the rotational speed of the impeller as shown in Equation 2,

$$\frac{P_2}{P_1} = (\frac{N_2}{N_1})^2 \tag{2}$$

where P represents the total pressure of the blower.
Fan law 3 states that the power is directly proportional to cube of rotational speed of the impeller as shown in Equation 3,

$$\frac{kW_2}{kW_1} = (\frac{N_2}{N_1})^3 \tag{3}$$

where kW represents the shaft power of the blower.
Thus when the above three equations are fulfilled it is said that centrifugal blower is obeying the fan laws [1].

2. LITERATURE REVIEW

Tito Mwinuka [2] studied the effect of number of impeller blades and impeller speed on the power consumption and air flow rate of centrifugal blower. The description of how the blade number and impeller speed affect the power consumption and air flow rate was described with the help of experimental analysis. When impeller blades were increased from 4 to 8, it was observed that there was rise in the flow rate while the power consumption also increased slightly. However there was a notable increase in the flow rate observed when the speed was increased from 580 rpm to 2990 rpm.

Jadhav et al. [3] studied the effect of rotational speed and volute tongue clearance on the centrifugal blower performance. An experimental analysis was carried out by varying the rotational speeds from 1500 rpm to 2800 rpm while the volute tongue clearances were also varied. The results showed that when the rotational speed of impeller is increased there is rise in the centrifugal blower performance. Also the modified tongue clearances increase the efficiency and total pressure of the centrifugal blower.

Dhande et al. [4] evaluated the centrifugal blower performance of air assisted sprayer which was used for application of orchard pesticides. The forward type of impeller blade was used and the variation was made in the blade number and blower speed. Blower B with 40 blade number was observed to be giving the best performance. It was also observed that when the blower speed was raised from 2050 rpm to 2450 rpm there was increase in the air discharge, total pressure and input power of the centrifugal blower.

Jayapragasan et al. [5] made the study and optimization of radial type blower for travelling cleaner application by using CFD. The 3D model of the blower was created using SOLIDWORKS while CFD Fluent was used to solve the flow inside the centrifugal blower. Taguchi orthogonal array based design of experiments technique was used to determine the required experimental trials. The optimum combination of the parameters were observed to be blade number, fan outlet diameters and fan blade angle as 8, 190 mm and 60° respectively, which would help to improve the fan performance.

Chaudhari et al. [6] made the geometric optimization and parametric study of the centrifugal blower. For this purpose a centrifugal blower with backward type blade was taken into consideration for the study and analysis. The CFD analysis validation was done with the help of the experimental results. Of the different combinations the one with blade number 16, outlet angle 76º and radial gap 85 mm was found to be the one with best results. As compared to the existing blower the optimal model had 23.07 % higher efficiency.

Chunxi et al. [7] studied the centrifugal fan performance by enlarging the impeller. The increment in outlet diameters with 5% and 10% was done and comparisons were made with the original impeller. The total pressure was observed to be increased with the impeller enlargement and also the flow distributed less uniformly in the volute thus resulting into lower fan efficiency.

Krishna et al. [8] made the use of numerical analysis software CFD Fluent for the study of the centrifugal blower. The objective was to find the aeroaccoustic and aeroaccoustic parameters. The parameters included velocity, noise and pressure of the centrifugal blower. The three different types were forward, backward and radial type of impellers. It was noticed that the backward impeller blower had more pressure and velocity as compared to the radial and forward types of impellers. The results also revealed that backward impeller blower was producing less noise compared with the radial and forward types of impellers.

S. Wagh and D. Panchgade [9] made the use of CFD to study impeller width effect on mass flow rate. The type of impeller was forward type with 10 number of impeller blades. It was observed that with the increase in impeller width of the centrifugal blower, there was an rise in mass flow rate upto 20 mm impeller width but when the width is further increased to 21.5 mm the mass flow rate starts to decrease.

Karthik V. and Rajeshkannah T. [10] made the experimental investigations of a centrifugal blower with the help of CFD. The objective was to maximize the efficiency, pressure and discharge with the consideration of power consumption. In the study CFD was used to observe the optimum combination operating variables of the blower. The comparison of the results from CFD analysis was made with the experimental analysis results in order to conform the validity. Thus finally the optimum parameters were finalized and it was found that there was increase in discharge by 39.7% and increase in the pressure by 16% compared to the existing model.

Kusekar S. K. and Lavnis A. K. [11] made the modal and CFD analysis for the critical parts optimization of the centrifugal blower. The earlier centrifugal blower was manufactured by using MS material which had the problem of corrosion. The above problem was solved using finite element analysis. The different materials were used in order to compare with the original material. It was found due to the high stiffness and layup sequence in the blower, the vibration problem of centrifugal blower was reduced. It was also found that the blower discharge increased from the present value of 10.84 m^3/s to 13.89 m^3/s.

3. DETAILS OF ORIGINAL BLOWER

The centrifugal blower is manufactured according to IS: 4894-1987. The different parameters of the original centrifugal blower are as shown in Table 1. The centrifugal blower has 12 number of backward type blades. The numerical analysis is carried out on the blower by varying the speed of the blower. Keeping all the parameters of the original centrifugal blower constant, only change done is in the rotational speed of the impeller.

Table 1. Parameters of original centrifugal blower

Sr. no.	Parameters	
1	Impeller material	IS 2062
2	Impeller Outlet Diameter (mm)	280
3	Impeller Inlet Diameter (mm)	140
4	Number of Blades	12
5	Impeller Blade Type	Backward type
6	Impeller Width (mm)	20
7	Casing Width (mm)	65
8	Casing Inlet Diameter (mm)	130
9	Casing Outlet B×L (mm)	65 × 186

The different levels at which the numerical analysis is to be carried is given notation as shown in Table 2. Thus the rotational speed is varied and performance for each level is observed with the help of numerical analysis.

Table 2. Different levels for numerical analysis

Sr. no.	Level	Impeller Speed (rpm)
1	N_0	1500
2	N_1	2000
3	N_2	2500
4	N_3	2800

Numerical simulation is carried out for the centrifugal blower once the different levels are identified.

4. NUMERICAL ANALYSIS

The numerical analysis of the centrifugal blower is done with the help of the commercial available CFD package ANSYS Fluent. To solve the partial differential equations, Finite Volume Method (FVM) is widely used. The ANSYS Fluent solves the Navier-Stokes equation with help of the FVM. In order to predict the performance of the blower, the qausi-steady simulation is used. The three-dimensional centrifugal blower solid model is created with the help of modelling software SOLIDWORKS 2016. Fig. 1 shows the centrifugal blower solid model in SOLIDWORKS 2016.

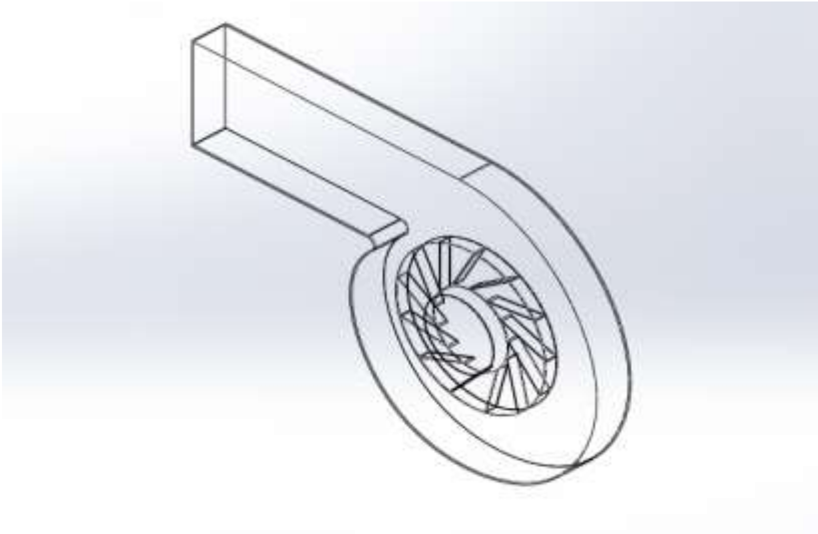

Fig. 1. Solid model of centrifugal blower

Once the solid modelling is done, the solid model is imported from SOLIDWORKS 2016 to ANSYS 16.0 software. Geometry clean-up is done which is followed by the mesh generation. Tetrahedral elements are used for the meshing of the rotating impeller while the hexahedral elements are selected for the meshing of inlet duct, casing and outlet duct. Meshing size of element is taken as 5 mm. Fig. 2 shows the meshed model for the centrifugal blower.

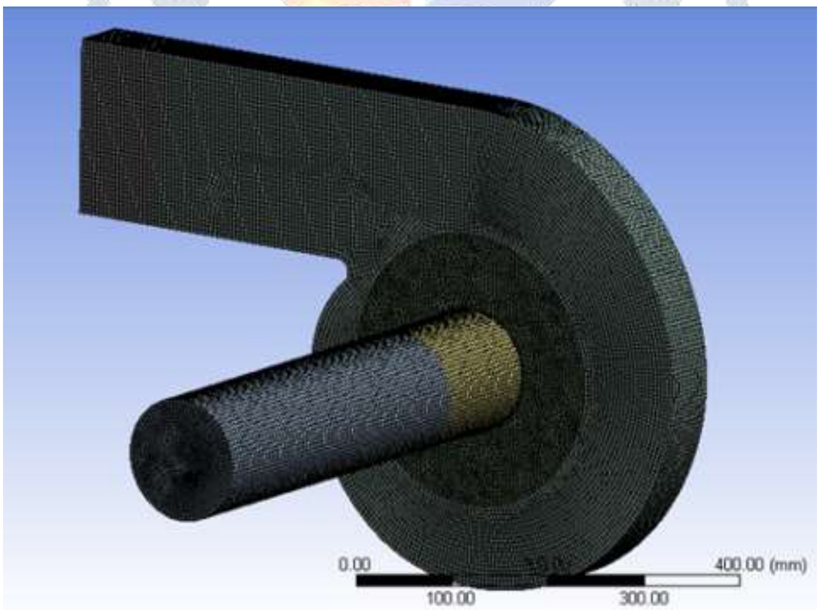

Fig. 2. Mesh model centrifugal blower

Boundary conditions are given to the centrifugal blower model in which inlet, outlet and impeller of the blower are given the boundary conditions. The outlet and inlet boundary conditions are specified at atmospheric pressure. Rotational motion is given to impeller and considered as moving wall. The problem set-up is given wherein the k-Ɛ turbulence model is selected to solve the given problem. Hybrid initialization is done and the convergence of the solution is observed by giving the required number of iterations to solve the given problem. Thus after several iterations there will be negligible difference in the values of the parameters, so it can be said that the solution is now converged.

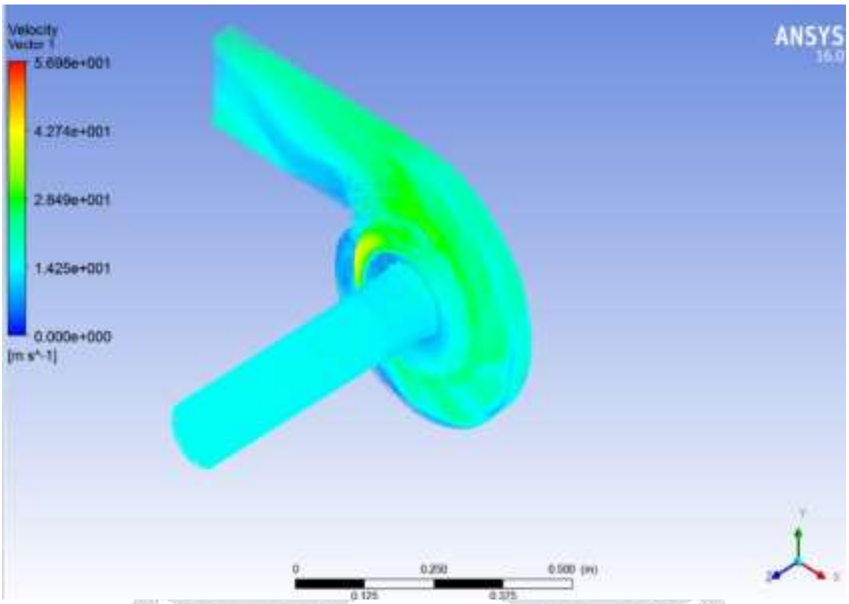

Fig. 3. Velocity vector plot for centrifugal blower

After the converging of the solution, post-processing of the centrifugal blower model is done in which different results are obtained. The numerical analysis results show the velocity vector plot, pressure counter plot, graphs, streamlines etc. With the help of this plot the flow pattern around rotor, separation phenomenon, pressure, velocity and streamlines can be easily recognized. Fig. 3 shows the velocity vector plot for the centrifugal blower wherein the velocity of particles at different locations in the centrifugal blower is observed.

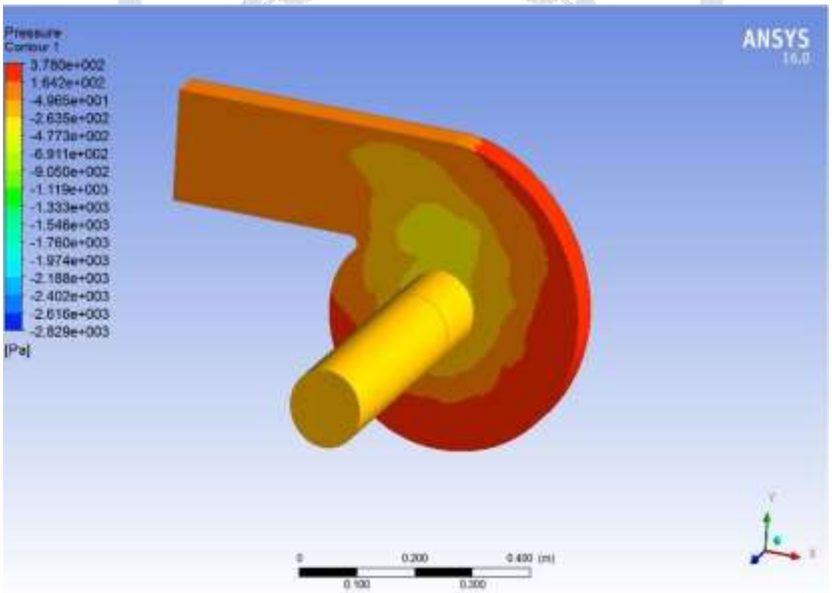

Fig. 4. Pressure counter plot for modified blower

Fig. 4 shows the pressure counter for the centrifugal blower wherein the pressure at different locations of the centrifugal blower is observed.

5. RESULT AND DISCUSSIONS

The results obtained after the numerical analysis of the centrifugal blower for different levels are as shown in Table 3. Flow rate obtained from numerical analysis for the blower with level N_1 is 349.49 m^3/hr. It is observed that with the increase in the speed to level N_2 the flow rate increases upto 463.97 m^3/hr. While at the rated speed of the blower which is level N_4, the flow rate obtained is 652.34 m^3/hr. Thus at the level N_4, the maximum flow rate is observed. Similarly with respect to the total pressure, the value tends to increase from level N_1 to N_4. Initially the value observed is 130.34 Pa while at the rated speed which is level N_4 the value obtained is maximum which is 452.07 Pa. Finally with respect to the shaft power it is observed that at level N_1 the shaft power obtained is 0.0248 kW. With the increase in the level to N_4 the value is 0.1591 kW. Thus with the increase in levels there is increase in the shaft power of the blower.

Table 3. Numerical analysis results of different speed levels of the centrifugal blower for performance parameters

Sr. no.	Speed levels	Flow rate (m^3/hr)	Total pressure (Pa)	Power input (kW)	Blower efficiency (%)
1	N_1	349.493	130.34	0.0248	50.99
2	N_2	463.977	228.16	0.0568	51.78
3	N_3	580.1912	356.641	0.1124	51.15
4	N_4	652.34	452.07	0.1591	51.48

The validation of the numerical analysis is done by comparison of the numerical results with the experimental results obtained from the experiemntal analysis reported in reference [3].

Table 4. Validation of the numerical results with experimental results

Sr. no.	Performance parameters	Experimental results	Numerical Results	% Deviation
1	Flow Rate (m^3/hr)	630.44	652.34	3.47
2	Total Pressure (Pa)	472.74	452.07	4.57
3	Shaft power (kW)	0.15367	0.1592	3.59

As seen from the Table 4, the numerical values obtained from numerical simulation are close to the values obtained from experimental data. Fig. 5 to Fig. 7 shows the variation of experimental and numerical analysis results for the speed levels from N_1 to N_4. The graphs are for performance parameters total pressure, flow rate and shaft power. According to Fig. 5 to Fig. 7, the numerical and experimental results are in good agreement and the maximum deviation observed is 4.57 % which is within the permissible limits of 10%.

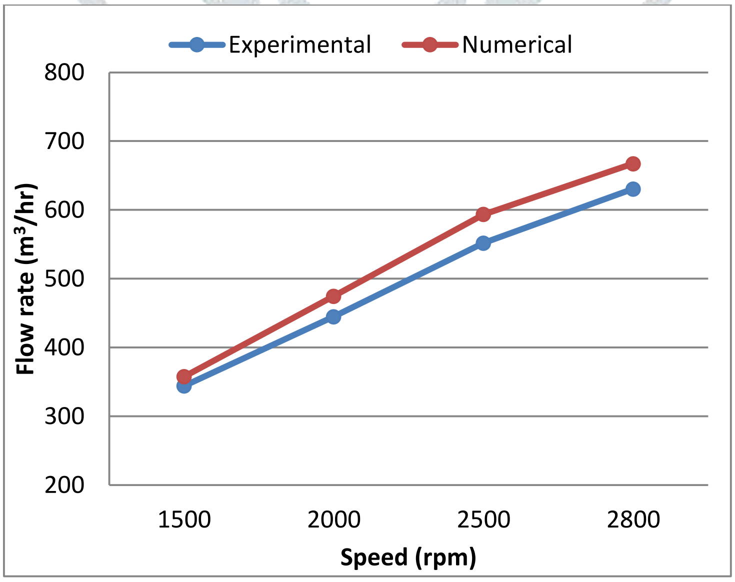

Fig. 5. Experimental validation for flow rate

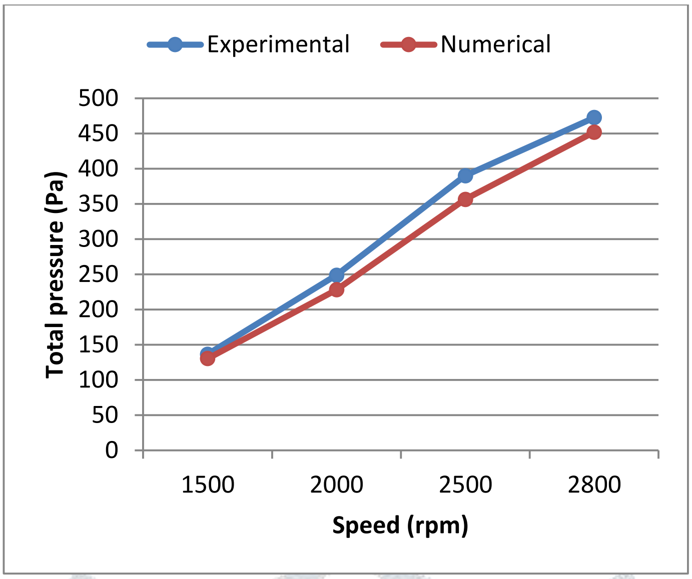

Fig. 6. Experimental validation for total pressure

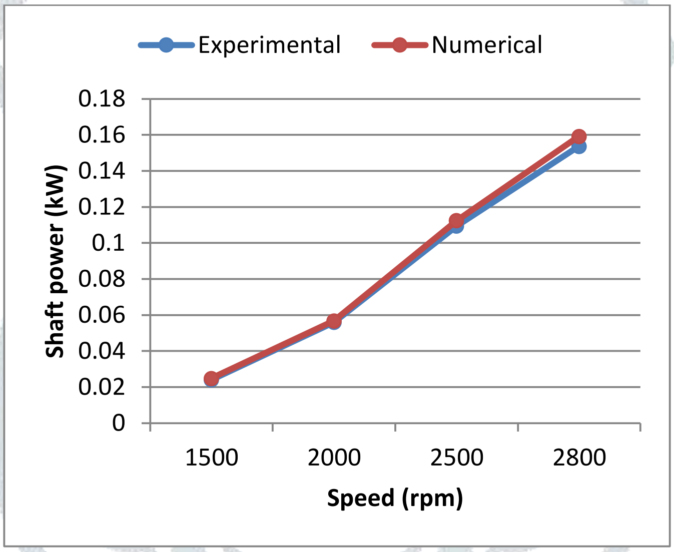

Fig. 7. Experimental and numerical validation for shaft power

Thus it can be observed that in both the experiemental and numerical analysis of the centrifugal, the performance parameters are noticed to be increasing with increase in the speed of the impeller.

6. CONCLUSION

From the numerical analysis of centrifugal blower it is observed that the performance parameters of the centrifugal blower increase with the increase in the rotational speed. There is rise in the flow rate from 357.54 m^3/hr to 652.34 m^3/hr as the speed is increased from 1500 rpm to its rated speed 2800 rpm. There is rise in the total pressure from 142.84 Pa to 452.07 Pa as the speed is increased from 1500 rpm to its rated speed 2800 rpm. There is rise in the shaft power from 0.0248 kW to 0.1592 kW as the speed is increased from 1500 rpm to its rated speed 2800 rpm. Thus without compromise in the efficiency of the centrifugal blower which is around 51%, increase in the performance parameters is obtained by increasing the speed from 1500 rpm to 2800 rpm. It is also observed that the numerical results obtained are in good agreement with the experimental results.

REFERENCES

[1] Bureau of Energy Efficiency, “Fans and Blowers”, Page no. 99.
[2] Tito Mwinuka, Apr. 2016, Experimental Determination of the Effect of Number of Impeller Blades on the Air Flow Rate and Power Consumption of Centrifugal Blowers, American Scientific Research Journal for Engineering, Technology and Sciences, Vol. 17, No. 1, Page no. 81-88.
[3] Nitin Jadhav and S.R. Patil, Jul. 2016, Effect of volute tongue clearance and rotational speed on performance of centrifugal blower, International Journal of Scientific research in Science, Engineering and Technology, Vol. 2, Issue 4, Page no. 261-266.

[4] K. G. Dhande, Ravi Mathur and Ajay Sharma, Aug. 2016, Performance Evaluation of a Centrifugal Blower of Air Assisted Sprayer for Orchard Pesticide Application, International Journal of Agricultural Science and Research, Vol. 6, Issue 4, Page no. 119-130.

[5] Jayapragasan.C.N, Manojkumar A. and Dr Janardhan Reddy, Apr. 2015, Design Optimization and Parametric Study on the Alternative Blower of Travelling Cleaner, International Journal of Innovative Science, Engineering and Technology, Vol. 2 Issue 4, Page no. 76-84.

[6] V. Chaudhari, Varun Parmar and M.J.Zinzuvadia, 2017, Parametric Study and Geometrical Optimization of Centrifugal Blower, International Conference on Research and Innovations in Science, Engineering and Technology, Vol. 1, Page no. 96-102.

[7] Li Chunxi, Wang Song Ling and Jia Yakui, 2011, The performance of a centrifugal fan with enlarged impeller, Energy Conversion and Management 2011, Page no. 2902-2910

[8] V. Krishna, K. Naresh Kumar and M. Prasanth Kumar, Aug. 2013, Numerical Analysis of Centrifugal Blower Using CFD, International Journal Of Engineering Research and Technology, Vol. 2 Issue 8, Page no. 1461-1463.

[9] Sushant Wagh and Dhananjay Panchagade, Jun. 2014, Analyzing effect of impeller width on mass flow rate of centrifugal blower using CFD, International Journal of Engineering and Advanced Technology, Vol. 3, Issue 5, Page no. 236-238.

[10] Karthik V. and Rajeshkannah T., Jul. 2014, Experimental Investigation of a centrifugal blower by using CFD, International Journal of Mechanical Engineering and Robotics Research, Vol. 3, No. 3, Page no. 39-44.

[11] Kusekar S.K. and Lavnis A.K., Dec. 2014, Optimization of critical parts of centrifugal blower by Modal & CFD Analysis, International Journal of Innovative Research in Advanced Engineering, Vol. 1, Issue 12, Page no. 69-82.