



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 10, Issue 8, August 2021

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 7.569

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Numerical Performance Analysis of Centrifugal Blower for Different Number of Blades

Rahul Chavan¹, Prof. Sunil Patil²

P.G. Student, Department of Mechanical Engineering, AISSMS COE, SPPU, Pune, India¹

Professor, Department of Mechanical Engineering, AISSMS COE, SPPU, Pune, India²

ABSTRACT: The important parameters for the design of a centrifugal blower include impeller outlet and inlet diameter, the rotational speed of the impeller, impeller width, impeller blade type, and the number of impeller blades. The relation among many of the parameters above is well explained in the literature but how several impeller blades affect the centrifugal blower performance is not yet clear. When the number of blades is less it results in improper guidance of air stream while more number results in resistance to airflow. Thus, it becomes important to find the proper blade number to get better performance results. A numerical analysis is being carried out on the centrifugal blower to find the effect of number of blades on its performance parameters. The numerical analysis is done by modifying the number of impeller blades from 10,12,14. Initially the different solid models are prepared with help of modelling software CATIA V5 and then using the ANSYS Fluent software the numerical analysis is carried out.

KEYWORDS: Centrifugal Blower, Airfoil Blade, Impeller, Numerical Analysis, Number of Blades.

I. INTRODUCTION

The centrifugal blower with airfoil blades is used to carry the gases at moderate pressure and flow rates. In a centrifugal blower, the airflow is entered into the inlet duct axially and meets the impeller. The deflection of each stream in a spiral pattern to an almost circumferential direction occurs in the impeller blades the collection of each of the air streams then happens in the housing and at the end the individual air stream is converted to a single air stream which leaves the blower radially and increases the velocity and pressure of gases.

Because of the wide industrial application of centrifugal blowers, the efficiency of these blowers may have a great influence on energy consumption. The transformation of kinetic to pressure energy is mainly done in the impeller blades thus an improvement in the blower performance can be achieved by modifying the impeller blades number. When the experimental analysis is done, it consumes much time and is expensive as compared to numerical simulation as prototypes are required to be prepared on a trial-and-error basis. Thus, for the manufacturers, the profit makings as compared to numerical analysis get reduced. Hence the Computational Fluid Dynamics (CFD) analysis with a suitable turbulence model is utilized which is inexpensive and timesaving as compared with experimental analysis. The behaviour of the fluid inside the machine can be correctly predicted with the use of numerical simulations. Thus, accurate performance analysis of a particular design can be efficiently carried out.

The design parameters which influence the performance of the centrifugal blower are outlet and inlet diameter of impeller, impeller blades number, impeller width, impeller blade type, and impeller speed. The design parameters like the inlet diameter of the impeller, outlet diameter of the impeller, and impeller width cannot be changed because of space constraints. But the number of blades of the impeller can be changed and the performance of the centrifugal blower can be studied without changing the overall dimensions of the blower. Thus, when the size of the blower cannot be changed due to space constraints then the number of blades can be modified to fine-tune the performance of the blower. In this paper, the selection of airfoil blade impeller blade is done for the study.

II. LITERATURE REVIEW

Chen-Kang Huang and Mu-En Hsieh [1] made an experimental study on the airfoil blade type centrifugal blower. Studied the effects of blade angle, blade number, tongue length, scroll contour. Compared analytical data with measured data it was found that the abnormality of the static pressure curve at each flow rate was within 4.8% and the abnormality of the efficiency curve was within 15.1%. An optimized design is performed with a 7.9% development in static pressure and a 1.5% improvement in incompetence. The full process reproduces backward curved airfoil centrifugal blowers successfully and is a powerful design tool for blower development and improvement.



Kyung Jung Lee et. al. [2] study on the 3D blades of centrifugal fan. The optimized model discovered that the static pressure in the outlet had been increased equally and steadily, and the flow split-up at the leading edge was significantly reduced. According to replications, the three-dimensional blade calms down the fluid flow and the flow friction was reduced by controlling the flow separation as much as likely so that both the static pressure and the static efficiency were improved in comparison with those of the innovative model. As a result, the constant efficiency was increased by 6.3% compared with that of the original model.

S.R.Patil et. al. [3] Study of performance of centrifugal blower is done by varying volute tongue clear Four types of casings are taken with volute clearance of 6%, 8%, 10% and 12.5% of impeller diameter. Numerical analysis is done by using computational fluid dynamics. For solving Reynolds-averaged Navier-Stokes equations are used with k- ϵ turbulence model. The parameters total pressure, flow rate and efficiency are calculated.

S. Chakraborty and Krishna Murari Pandey [4] studied the performance testing of Numerical Studies on Effects of Blade Number Differences on Performance of Centrifugal Pumps at 4000 RPM. In this study, the performance of impellers with the same outlet width with different blades of centrifugal pump blades was thoroughly tested. The width of the impeller out Ellor, angle blade, and metal numbers are the most critical parameters affecting the performance of centrifugal pumps. The pump has a design rotational speed of 4000 rpm and an impeller with numbers of 4,5,6,7,8,9,10,12 leaves is counted. Internal flow fields and centrifugal pump signals with various metal numbers are simulated and predicted using Ansys Fluent software. With the increase in the value of the blade, the head and stationary pressure of the model increases, but the flexible control of the efficiency is complex, but there are large numbers of numbers for each instrument. The area of the low-pressure circuit on the blade gets pull steadily increases and the vertical pressure increases gradually with the increase in page numbers.

Amjadimanesh et. al. [5] conducted a study in the Numerical Study of the Blade Number Effect on the performance of 3D FC Centrifugal Fan. A 3D forward curved centrifugal fan with airfoil blades is created numerically to calculate the flow of the force and pressure of the fan. With this price simulation, the impact of the metal number on the performance of the fans is tested. In numerical analysis, a finite volume method (FVM) is used to solve the control figures.

B. Jafarzadeh et. al. [6] studied the simulation of centrifugal pump flow rate. In this paper, we present a standard three-dimensional analogy of turbulent weed flow to predict the field strength and pressure of a centrifugal pump. Commercial CFD code was used to resolve the flow of field flow. In addition, the effect of several skills on pump performance was studied. Finally, it was also observed that the position of the letters about the volute language had a significant effect on the beginning of the division. Therefore, to analyse the effect of blade number on pump signals, blade position and language should be the same.

Tito Mwinuka [7] is studied the Effect of Impeller Blades on the Air Flow Rate and the Power Consumption of Centrifugal Blowers. In this practice, a test method to define the effect of the mushroom number on the impeller is explained. The blower was tested at three different speeds using five mechanical sprays with 4, 5, 6, 7, and 8 respectively. It was found that, in all 3 selected blower blockages, the airflow rate increased by about 19% by increasing the number of impeller pipes from 4 to 8. It was also found that the blower speed increased with a higher rate flow; that is, moreover that 80% increases the number of impeller blades. It is also found that there is a slight change in power use due to changes in the number of belts. It is therefore recommended that converting the number of flight tubes can only be used to fine-tune the blower formation within a range of 15-20% flow rates. However, the automatic flow rate should be established using blower speed and size. Power consumption when found to increase slightly with a maximum variation of 3% in increasing the blade number from 4 to 8.

III. DETAILS OF CENTRIFUGAL BLOWER

The different parameters of the centrifugal blower are as shown in Table I. The centrifugal blower has 12 airfoil blades. The numerical analysis is carried out on the modified centrifugal blower. The term modified centrifugal blower indicates the changes made in the number of impeller blades. Keeping all the parameters of the centrifugal blower constant, the only change done is in the number of blades.



TABLE I
PARAMETERS OF CENTRIFUGAL BLOWER

Sr. No.	Parameters	
1	Impeller outlet diameter (mm)	280
2	Impeller inlet diameter (mm)	140
3	Number of Blades	10, 12, 14
4	Impeller Blade Type	Airfoil blade
5	Impeller width (mm)	20
6	Casing width (mm)	65
7	Casing inlet diameter (mm)	130
8	Casing outlet B*L (mm)	65*186
9	Motor Speed (rpm)	2000

The centrifugal blower is changed with an airfoil blade impeller. Thus, the number of blades is increased, and the performance of each modified blower is observed with the help of numerical analysis. In a modification, the number of blades of the impeller are changed. For analysis taking 10,12 and 14 blades.

IV. DESIGN OF CENTRIFUGAL BLOWER

a. Design of Blade

For the design of the impeller, NACA (National Advisory Committee for Aeronautics) no.6512 airfoil blade is selected. The standard specifications of the blade taken from "Fan Handbook".

b. Design of Impeller

From the study, the parameters and geometry, use CATIA V5 software for design the impeller. 2-D model is as shown in Fig1.

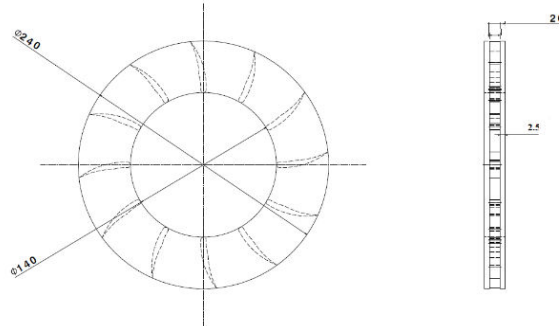


Fig.1. 2-D Model

c. Design of Blower

Design of centrifugal blower is on CATIA V5. The Blower Shown in Fig.2 As shown in Fig.2 design the blower with 10 and 14 number of blades.

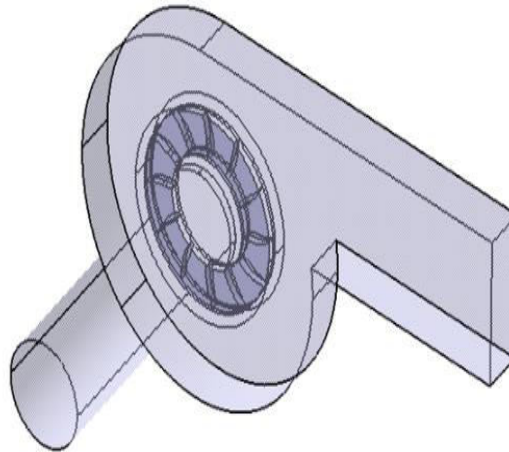


Fig.2. Blower with 12 number of blades

V. NUMERICAL ANALYSIS

The commercial CFD package Fluent is used to simulate the different configurations of the centrifugal blower. Fluent solves Navier-Stokes's equation using the FVM, which has been applied widely in fluid mechanics and engineering applications. It has been indicated that the Fluent quasi-steady simulation can be used to predict blower performance. The simulation effects are compared with the other number of blades. In ANSYS (Fluent) there are three steps to solve the problem.

a. Pre-processing

I. Geometry clean-up and geometry operations:

Initially, the solid model of blower with 10,12 and 14 number of blades is imported to the analysis tool which in this case is ANSYS 16.0. Once the model is imported to the ANSYS, the first step is the geometry clean-up wherein a repair tool is used to remove the extra edges which are generated when the model is imported from the modelling software to analysis software. Hence by using the tool, repair edges and hard edges the extra lines are eliminated. Hence now the model is without the extra lines and edges which might prove to be a hindrance and deviate the results.



II. Meshing:

Pre-processing includes the meshing of the component. The relevance center is to be set to fine and smoothing is to be set to high in the sizing settings for the model. The next step is to determine the appropriate mesh size for our model. Now what happens is when the mesh size is too coarse then it will result in faulty results of the parameters. There would not be proper convergence and thus we must move for higher mesh size. In case when the meshing is too fine then the results will be accurate, but it might result in higher computational time. Take the mesh size as 5 mm for the analysis. For the appropriate results.

b. Solver

The second step is the solver the input of the solver is the mesh model of the centrifugal blower. In this solving process first select the model and give the boundary condition to this model and solve with sufficient iterations. First, the double-precision option is selected and then in the processing options, the parallel option is selected with 4 processes.

I. Multiple reference frame and setup

Centrifugal blower geometry consists of three different zones, the inlet and outlet regions, the impeller, and the volute. Here impeller is the moving part and other parts are stationary.

c. post-processing

CFD simulations do not end with the fluid flow calculation. From the post-processing, all the predictions give to user complete insight into their fluid dynamics simulation results. Post-processing results include the View of the geometry and influence volumes. Vector plots showing the direction and magnitude of the flow at various planes in the volute casing. Pressure plots shows the overall pressure in the blower. Velocity plot shows the velocity in the centrifugal blower.

VI. NUMERICAL ANALYSIS RESULTS

The numerical analysis is performed for the different cases and the results will be discussed in the following sections. The numerical analysis process as discussed in the previous section is used for the analysis of each model.

Results for different types of centrifugal blowers:

Impeller speed=2000rpm

1) For 10 Blades:

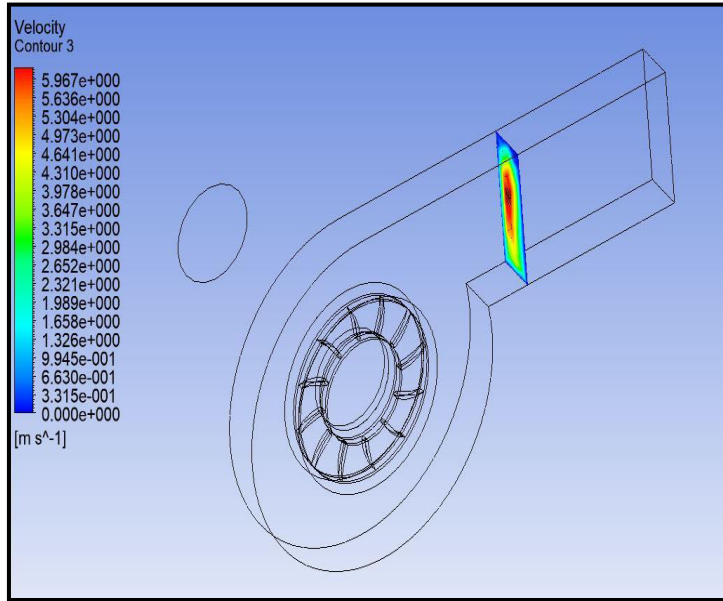
From the analysis, the result of outlet velocity is 6.47 m/s. The overall pressure is 43.65 Pa. From the calculation the flow rate is 281.99 m³/h. The overall efficiency of centrifugal blower is 51.26 %.

2) For 12 Blades:

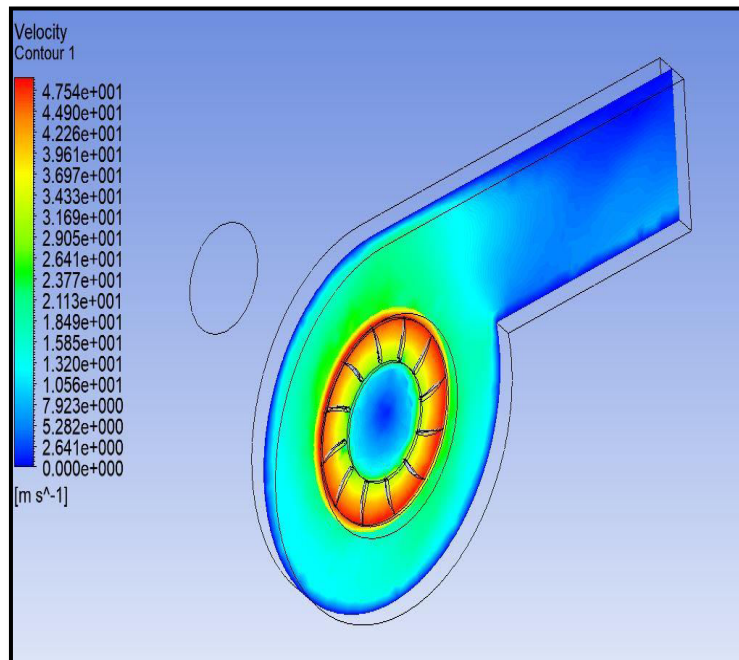
From the analysis the result of outlet velocity is 6.17 m/s. The overall pressure is 155.63 Pa. From the calculation the flow rate is 268.23 m³/h. The overall efficiency of centrifugal blower is 53.86 %. Sample analysis results shown in Fig.3.

3) For 14 Blades:

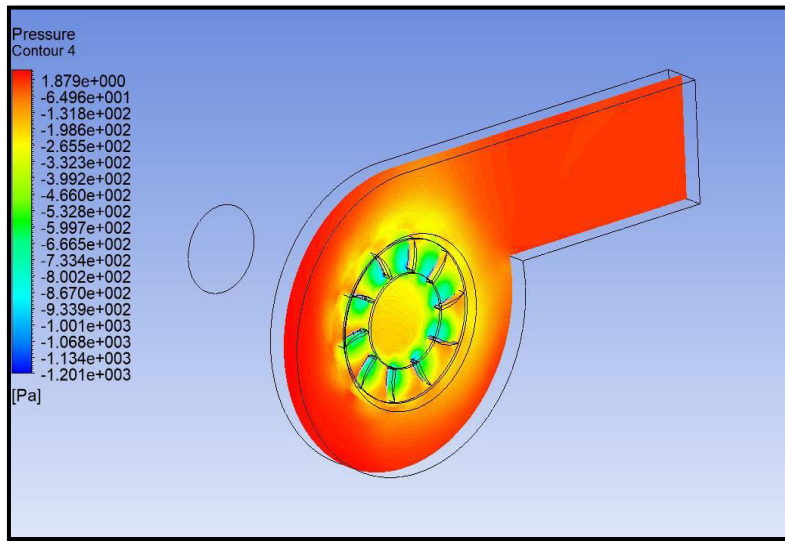
From the analysis the results obtained are outlet velocity is 91.00 m/s. The overall pressure is 301469 Pa. From the calculation the flow rate is 3961.01 m³/h. The overall efficiency of centrifugal blower is 69.53 %.



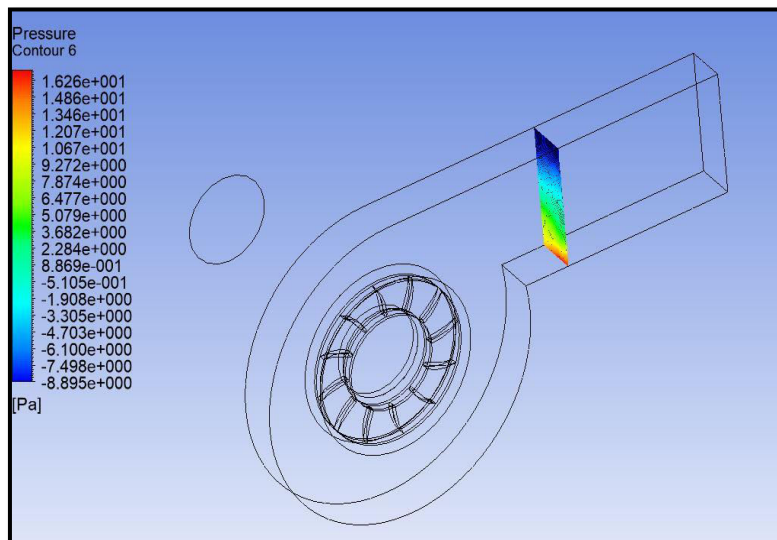
(a). Outlet Velocity Plot



(b). Overall Velocity Plot



(c). Overall Pressure Plot



(d). Outlet Pressure Plot

Fig.3. Results for different number of blades

VII.RESULTS AND DISCUSSION

The numerical analysis results are shown in Table II. The blower of 10 blades gives less pressure but its discharge is more. With an increasing number of blades, the pressure will increase, and discharge decrease but in small amounts. When increasing the number of blades pressure will increase but decrease the flow rate as pressure is inversely proportional to the flow rate. That's for the 12 number of blades it gives moderate pressure and moderate velocity. Again, if the number of blade increase, the area between blades reduces and it gives more turbulence and not feasible pressure and velocity.



TABLE II
NUMERICAL ANALYSIS RESULTS

Sr. No.	No. of Blades	Flow Rate (m ³ /h)	Overall Pressure (Pa)	Efficiency (%)
1	10	281.99	43.65	51.26
2	12	268.23	155.63	53.86
3	14	3961.01	301469.00	69.53

VIII. CONCLUSION

The centrifugal blower with 10 blades gives a 281.99 m³/h flow rate, 43.65 Pa overall pressure, and 53.86 % efficiency. The pressure of the blower is less as compared with other results. The centrifugal blower with 12 blades gives a 268.23 m³/h flow rate, 155.63 Pa overall pressure, and 53.86 % efficiency. The centrifugal blower with 12 blades gives good efficiency and pressure. The blower with 14 blades does not give feasible results. The 12 blades give good results.

REFERENCES

- [1] Chen-Kang Huang, Mu-En Hsieh, "Performance Analysis and Optimized Design of Backward-Curved Airfoil Centrifugal Blowers", HVAC&R RESEARCH, VOLUME 15, NUMBER 3, MAY 2009. pp- 461-488
- [2] Kyung Jung Lee et. al, "Optimal Design of a Plenum Fan with Three-Dimensional Blades", Applied science, May 2020. pp-1-14
- [3] Sunil R. Patil et. al Effect of Volute Clearance Variation on Performance of Centrifugal Blower by Numerical and experimental Analysis, Vol. 5, Issue 2, (2018) pp-3883-3894.
- [4] S. Chakraborty and Krishna Murari Pandey, "Numerical Studies on Effects of Blade Number Variations on Performance of Centrifugal Pumps at 4000 RPM", International Journal of Engineering and Technology · January 2011 pp-72-78.
- [5] A. Amjadimanesh, H. Ajam and A. Hossein Nezhad "Numerical Study of Blade Number Effect on the Performance of a 3D FC Centrifugal Fan", International Journal of Mechatronics, Electrical and Computer Science, Vol. 5(15) Apr. 2015, pp- 2109-2119.
- [6] B. Jafarzadeh a, A. Hajari b, M.M. Alishahi a, M.H. Akbari, "The flow simulation of a low-specific-speed high-speed centrifugal pump", Applied Mathematical Modelling 35 (2011) pp-242-249.
- [7] Tito Mwinuka, "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 (ASRJETS) (2016) Volume 17, No 1, pp 81-88.



**Impact Factor:
7.569**



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  ijirset@gmail.com



www.ijirset.com

Scan to save the contact details