International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 05 Issue: 10 | Oct - 2021 ISSN: 2582-3930

Numerical Performance Analysis of Centrifugal Blower Impeller with Airfoil Blades

Rahul Chavan[1], Sunil Patil[2]

[1]Student, Department of Mechanical Engineering, AISSMS COE, SPPU, Pune, India

[2]Professor, Department of Mechanical Engineering, AISSMS COE, SPPU, Pune, India

---

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. The relation among many of the parameters above is well explained in the literature but how the shape of impeller blades affects the centrifugal blower performance is not yet clear. When the shape of the blade is changed it results in the proper guidance of the air stream results in less resistance to airflow. Thus, it becomes important to find the proper blade shape to get better performance results. A numerical analysis is being carried out on the centrifugal blower to find the effect of the shape of the blade on its performance parameters. The numerical analysis is done by modifying the shape of the impeller blade. Initially the different solid models are prepared with help of modeling software CATIA V5 and then using the ANSYS Fluent software the numerical analysis is carried out.

Key Words: Centrifugal Blower, Airfoil Blade, Ansys, Numerical Analysis, CFD.

1.INTRODUCTION

A centrifugal blower with air bubbles is detected to capture gases with moderate pressure and flow rates. In a centrifugal blower, the air flow is inserted into the inlet axially and connected to the impeller. The rotation of each stream in a circular direction to the nearly circular side occurs where the planes that travel the collection of individual air currents occur in the chambers and eventually each air flow is converted into a single air stream leaving the blower radially. Here you find the rise and speed of gas.

Due to the widespread use of centrifugal strikers, the efficiency of these strikers can have a significant impact on energy efficiency. The kinetic conversion to compression force is carried out mainly on the sides of the impeller so the blower performance of the blower can be eliminated by altering the structure of the blade metal. Once test analysis is done, it is time-consuming and costly compared to quantitative simulation as prototypes are required based on trial and error. Therefore, for manufacturers, profit margins compared to statistical analysis are declining. Computational Fluid Dynamics (CFD) analysis is therefore used with the appropriate model of inexpensive damage and save time compared to experimental analysis. The behavior of the liquid inside the machine can be accurately predicted using numerical simulations. Therefore, an accurate performance analysis of a particular design can be performed correctly.

The design parameters of the centrifugal blower affect the performance of the outlet and the inlet width of the impeller, the number of impellers blows, the width of the impeller, the type of impeller blade, and the speed of the impeller. Design parameters such as impeller width, impeller outlet, and impeller width cannot be changed due to space constraints. But the shape of the blade can be changed, and the efficiency of the centrifugal blower can be studied without changing the size of the blower. Therefore, where the size of the blower cannot be changed due to space constraints then the shape of the blades can be adjusted to better adjust the performance of the blower. In this study, a collection of airfoil blade impeller blade is taken for research.

2. LITERATURE REVIEW

Jin-Hyuk Kim et al. [1] conducted research in Numerical Investigation on the Aerodynamic Performance of a Centrifugal Fan with Splitter Blades. This paper provides statistical analysis of aerodynamic performance based on the use of a separating device on the impeller of the centrifugal drill used in the waste collection system. The centrifugal fan price analysis was performed by adjusting the Reynolds scale - a standard Navier-Stokes scale for the shear transport trajectory model. Verification of numerical results is done by comparing the test data for pressure and efficiency. In all statistical analysis of a centrifugal fan with dividing particles, it has been found that circuits that lean on the metal side can be reduced by monitoring a large number of pages with a separating blade. The use of splitter blade for centrifugal fans leads to significant improvements in overall fan performance.

Vijay Rajpura, Vivek Brahmbhatt, and Gaurav Patel [2] learned about Performance Improvement of IND 25 Centrifugal Fan By Changing the Type of centrifugal Blade fan made using computational fluid dynamic method (CFD). The centrifugal flow impeller is made of Navier-stock shares. And curved functionality is available. As a first step, test setup was designed and modeled by fans to perform flow measurements, and after that, a computational fluid dynamics model was developed for the above set and the results were validated with a test scale.

Parth Shah, Ashwin Ganesh M and Thundil Kuruppa Ra [3] conducted research on the Design and Analysis of Airfoil-

Shaped Impeller Blades of Centrifugal Pump. This paper is related to a comparative study of the compressive force between conventional and conventional wind turbines and the centrifugal wind-driven imprint pump. While mounting is also important as an incense burner, current comparative analysis makes more money unnecessary in research, which is why it is ignored. All centrifugal pumps are usually constructed and constructed using reverse blades with equal cards on the upper and lower sides. The camber installed on the upper side is a good feature of the increased air flow.

S.R Patil et al.'s [4] studied the effect of volute tongue clearance variation on the performance of backward-curved blades centrifugal blower. In this paper, the volute tongues clearance decreases from 12.5% to 6% of impeller diameter, the total pressure and efficiency increases by 19.52 % and 21.90 % respectively at full discharge condition. The modified volute tongue clearance increases the total pressure and efficiency of the blower because the reverse airflow at the region near blower volute tongue is effectively reduced.

Rahul Chavan et. Al [5] studied the numerical analysis of a centrifugal blower with an airfoil blade. The results of the analysis were obtained from this study. The results of the 12 books are good. Production is reliable with these results.

Amjadimanesh et al [6] conducted a study in Numerical Study of the Blade Number Effect on the performance of 3D FC Centrifugal Fan. 3D forward curved fan (FC) centrifugal fan with airfoil blades is simulated with numbers to predict the flow of power and pressure of the fan. With this price simulation, the impact of the metal number on the performance of the fans is tested. In several analyzes, a limited volume (FVM) method is used to solve control statistics. Many results have been verified with test data. Obviously, the 20 blades works very well for this type of fans. And a small square method is used to enter the blade number. This adjustment predicts a total of 21 blades as the most efficient.

3. DETAILS OF CENTRIFUGAL BLOWER

The different parameters of the centrifugal blower are shown in Table 1. The centrifugal blower had twelve airfoil blades. The numerical analysis was performed on a centrifugal blower. The term modified blower centrifugal indicates a change in the shape of impeller blades. To keep all the limits of the centrifugal blower constant, the only change made in the shape of blades.

Table -1: 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

4. DESIGN OF CENTRIFUGAL BLOWER

For the design of the impeller, select the NACA (National Advisory Committee for Aeronautics) no.6512 airfoil blade. These blades are mostly used in blowers and pumps. These blades give required output, and they are efficient. Design the impeller using the CATIA V5 software. 2-D model of impeller is shown in Fig.1.

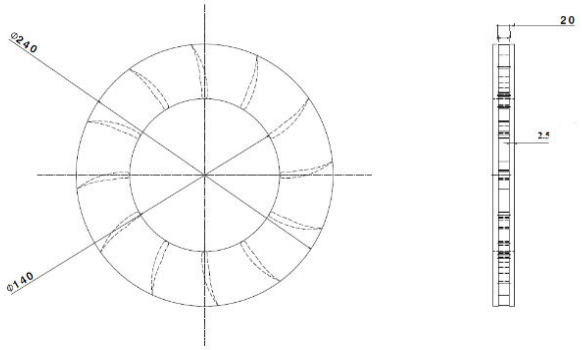

Fig.1. 2-D Model of Impeller

Design the centrifugal blower as per the specification. Using CATIA V5 software design the 3-D model of centrifugal blower. The model is shown in Fig.2.

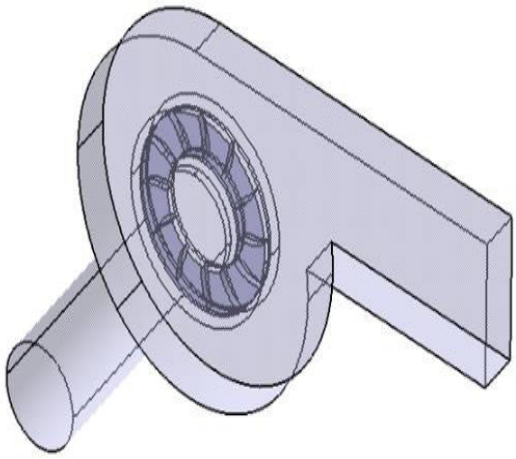

Fig.2. 3-D Model of Blower

5. NUMERICAL ANALYSIS

The commercial CFD Fluent package is used to mimic the different configuration of a centrifugal blower. Smooth solves Navier-Stokes' statistics using FVM, which is widely used in mechanical and engineering applications. It has been shown that the constant imitation of Fluent quasi-steady can be used to predict blower performance. Imitation results are compared to other number of bands. In ANSYS (Slide) there are three steps to problem solving.

a. Pre-processing

Geometric purification and Geometric performance

Initially, a solid blower model with 12 airfoil blade was introduced into the analysis tool which is currently ANSYS 16.0. Once the model has been imported into ANSYS, the first step is to clean up the geometry where the repair tool is used to remove the extra edges that are created when the model is imported from modeling software to software analysis. So, by using the tool, fixing the edges and solid edges additional lines are completed. So now the model has no extra lines and edges that can be a hindrance and distort the results.

Meshing

The next step in the previous operation involves combining the object. The center of importance should be well set and smooth should be placed at the top of the model rating settings. The next step is to determine the appropriate mesh size for our model. Now what happens when the size of the match has increased too much will result in negative border effects. There will be no suitable mixing, so we must move to a higher mesh size. If the meshing is more accurate the results will be more accurate, but it may result in higher calculation time. Take match size as 5 mm for analysis. With the right results.

b. Solution

The second step is the solver input solver is the mesh model of the centrifugal blower. In this resolution process first select the model and give the state of the parameters in this model and solve with sufficient repetition. First, the double specifying option is selected and then in the processing options, the same option is selected in 4 processes.

Multiple reference frame and setup

Centrifugal blower geometry has three distinct areas, incoming and outgoing areas, impeller, and volute. Here the impeller is a moving part, and the other parts are stationary.

c. Processing in the background

CFD comparisons are not limited to fluid flow calculations. From the use of the post, all predictions give users a complete understanding of their simulation results of fluid dynamics. Processing results after incorporation of velocity, general velocity structure, exit pressure structure, and total pressure. Sites offer the right values of total velocity, total pressure.

6. 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 above sections is used for the analysis of model.

Impeller speed= 2000 rpm

Numerical Analysis results gives 6.17 m/s outlet velocity. The overall pressure is 155.6357 Pa. The results obtained from the data is 268.23 m^3/hr flow rate.it gives 53.86.% efficiency. The Numerical analysis plots are shown in Fig. 3(a),3(b),3(c),3(d).

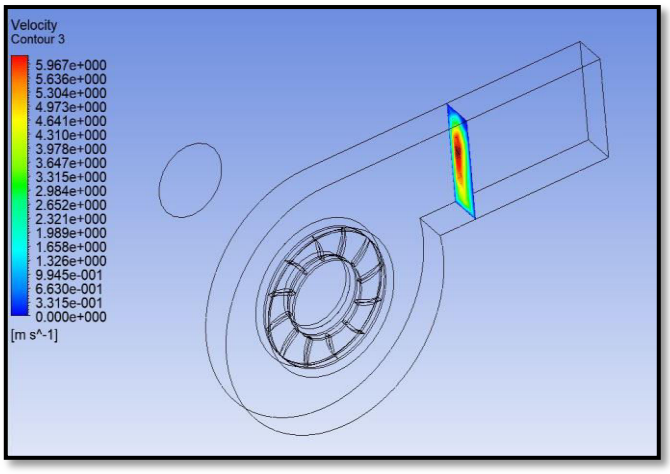

Fig.3(a). Outlet Velocity Plot

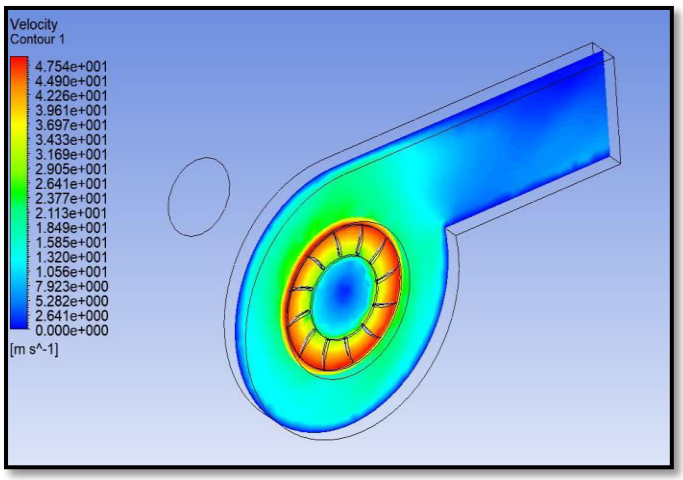

Fig.3(b). Overall Velocity Plot

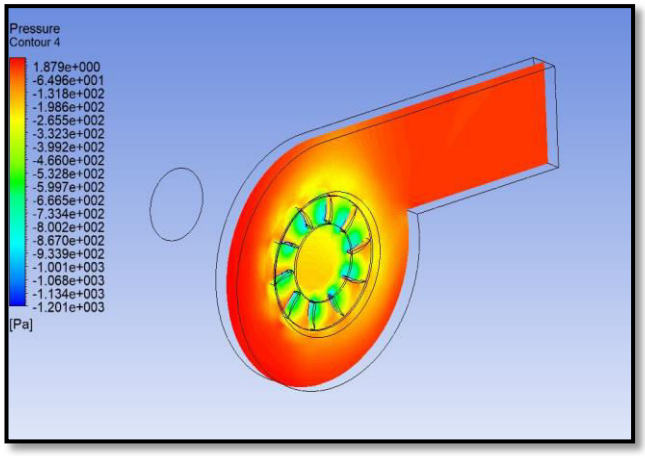

Fig.3(c). Overall Pressure Plot

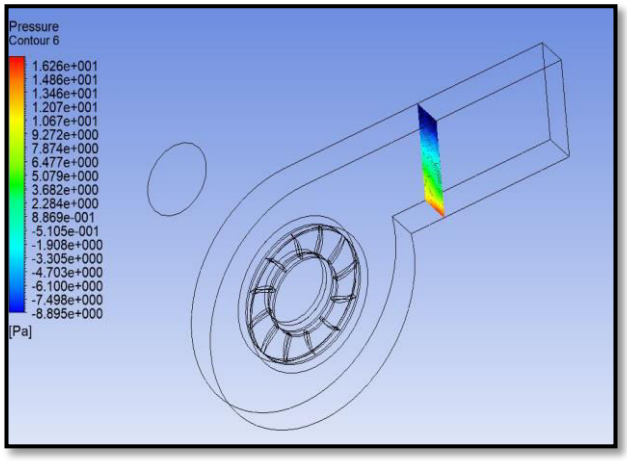

Fig.3(d). Outlet Pressure Plot

7. RESULTS AND DISCUSSION

The Numerical analysis results are shown in the Table 2. The blower with 12 airfoil blades gives the moderate pressure. The application of blower with airfoil blade commonly uses for moderate pressure and clean air. It gives the moderate velocity. The efficiency of blower is better for use.

Table -2: Numerical Analysis Result

Sr. No.	No. Of Blades	Flow Rate (m^3/hr)	Overall Pressure (Pa)	Outlet Velocity m/s	Efficiency %
1	12	268.23	155.63	6.17	53.86

8. CONCLUSIONS

From the numerical analysis 12 airfoil blade blower gives the following results.

The overall pressure of the blower is 155.63 Pa. The flow rate of blower is 268.23 m^3/hr. It gives the moderate pressure and velocity. The outlet velocity of blower is 6.17 m/s. the efficiency of blower is better. The overall efficiency of blower is 53.86 %.

REFERENCES

[1] Jin-Hyuk KIM, et. al, "Numerical Investigation on Aerodynamic Performance of a Centrifugal Fan with Splitter Blades",International Journal of Fluid Machinery and Systems International Journal of Fluid Machinery and Systems Vol. 5, No. 4, October-December 2012, pp-168-173.

[2] Vijay Rajpura, Vivek Brahmbhatt and Gaurav Patel, "Review Article On Performance Improvement Of IND 25 Centrifugal Fan By Changing Type Of Blade", International Journal of Engineering Research & Technology (IJERT) Vol. 2 Issue 7, July – 2013, pp-2385-2391.

[3] Parth Shah, Ashwin Ganesh M and Thundil Kuruppa Ra, "Design and Analysis of Airfoil-Shaped Impeller Blades of Centrifugal Pump" Applied Mechanics and Materials Submitted: 2016-03-16 ISSN: 1662-7482, Vol. 852, pp 539-544, 2016.

[4] 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.

[5] Rahul Chavan et. al 'Numerical Performance Analysis of Centrifugal Blower for Different Number of Blades', International Journal of Innovative Research in Science, Engineering and Technology (IJIRSET), Volume 10, Issue 8, August 2021, pp-11306-11313.

[6] 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.