International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 8 Aug 2021, pp: 952-959www.ijaem.net ISSN: 2395-5252

NumericalAnalysisofCentrifugal Blower by Changing Speed and Angle ofAttackto Study the Performance Parameters

Abhilasha V. Mane[1], Sunil R. Patil[2]
[1]P. G. Scholar,Department of Mechanical Engineering, AISSMSCOE, Pune, Maharashtra, India,
[2]Assistant Professor, Department of Mechanical Engineering, AISSMSCOE, Pune, Maharashtra, India
Corresponding author: Abhilasha V. Mane

Submitted: 01-08-2021 Revised: 14-08-2021 Accepted: 17-08-2021

ABSTRACT: The performance ofcentrifugal blowerdepends onvarious design parameters likemotor power,rotational speed, various angles,number ofblades, etc. Different types of blades and the different angles of blades affect the performance parameters of the centrifugal blower. Performance parameters include flow rate, total pressure, blower efficiency. In this work, a numerical analysis is done to determine the effect of different angles of attack at 0^0, 8^0, 14^0 on theperformance ofcentrifugal blower. Angle of attack is angle between camber line and fluid direction. Initially, a centrifugal blower solid model is designed in Catia V5, and analysis is done by Computational Fluid Dynamics (CFD) Fluent analysis in ANSYS 20.0. The effect of speed on performance parameters is analyzed and the speed at which the maximum performance is obtained is determined and considered for further analysis.

Keywords-The centrifugal blower, Airfoil curved blades, Angle of Attack, Numerical analysis, Performance parameters.

I. INTRODUCTION

The centrifugal blower is amechanical device which uses to increase pressure of air. These require large volume air at low pressure for operation. It consists of blade, impeller, casing, driveshaft, inlet ducts, outlet ducts, etc. The air enters through the inlet duct axially and strikes on the impeller. Then air circulates from axially to circular with the rotating action of the impeller and kinetic energy converts into pressure energy. The collection of air then circulates in the housing and at the end, the individual air stream is converted to a single-stream air stream which leaves the blower radially. Thus, the pressure and velocity of air increase. By increasing the efficiency, the performance of the blower can be increased. Hence, this can be achieved by modifying the angle of attack of the blade. For the experimental analysis, thistakes more time and is expensive as compared to numerical analysis. Hence, the Computational Fluid Dynamics analysis with a suitable turbulence model is utilized which is inexpensive and time-saving as compared to experimental analysis. It predicts the behavior of thefluid inside the machine correctly with the use of numerical simulations. Thus, the most accurate performance analysis is carried out.

Some design parameters like inlet and outlet diameters of impeller and impeller width cannot be changed because of space constraints.But other parameters like thenumberof blades, type of blade, angles of attack of the blade impeller can be changed. Thus, to study the performance of centrifugal blower airfoil curved blades are used. Performance of the airfoil curved centrifugalblower is studied by changing speeds and angles of attack.

[1]. Numerical simulation and analysis of backwardcurved airfoil centrifugal blowers are numerically analyzed. By varying,a static pressure performance is studied.Flowrate and the variation of the efficiency are studied. Here,a 7.9% improvement in static pressure and a 1.5% improvement in efficiency is observed. This simulation is done for the development and improvement of the blower.Theobtained results are compared with measured.Also,the effects of blade angle, blade number, tongue length, and scroll contour are numerically studied.

[2]. Study of performance of centrifugal blower is done by varying volute tongue clearance. Four types of casings are taken with volute clearances 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-e turbulence model. The parameters total pressure, flow rate and efficiency are calculated.

[3]. Studied design of the blade for regions of low wind power density for selecting suitable airfoil. Here, NACA 4412 airfoil profile is taken for analysis. The design of the airfoil is created using GAMBIT 2.4.6. Numerical analysis is done using CFD FLUENT 6.3.26 at different angles of attack. For NACA 4412 the coefficient of lift and drag is calculated.

[4].Comparison is done between theconventional and normal blade impeller and airfoil curved blade impeller. The outlet pressureenergy is compared. An increased camber on the top side is an ideal trait forlift generation. 3D analysis of the centrifugal pump impeller is designed in SOLIDWORKS® and analyzed using ANSYS® CFX. Values are plotted on a graph where the difference in slope of the two graph points is evident. Comparative analysis is showed that the airfoil design provides subtly more hydrodynamic energy compared to the conventional design. The conclusion and inference hold high importance in industries and other sectors to reduce power consumption for the pumping process.

[5]. Stresses and deflections are analyzed for the modified and pre-modified model andoptimization. Optimization is done to increase the fan efficiency by optimizing the thickness of the various components in the fan impeller. The impeller of default thickness resulted in maximumweight which increases vibrations and failure. So, the analysis is done by comparingvarious thicknesses. From this analysis, the most efficient thickness of the impeller parts is found for safe stress and strain limits. With the help of this analysis weight of the impeller is reduced and minimum vibrations are observed.

II. METHODOLOGY

The methodology used in this paper is to create amodel of the centrifugal blower in CATIA V5 and analyzing in Ansys 20.0.

a) **Design model of the centrifugal blower in CATIA V5:**The design of a model is done by using CATIA V5 and this is used here to determine theperformance parameters of the centrifugal blower.
b) **Analysis in Ansys 20.0:**Performance of different parameters of the centrifugal blower aredetermined using Ansys 20.0. Analysis of the centrifugal blower is the fluid analysis. Hence, for this analysis, CFD is used.

III. DETAILS OF CENTRIFUGAL BLOWER

The centrifugal blower design parametersare as shown in Table I. Numerical analysis is carried out of centrifugal blower with different speeds and angles of attack.

TABLE I
PARAMETERS OFCENTRIFUGAL BLOWER

Sr. no.	Parameters	
1	Impeller outlet diameter (mm)	280
2	Impeller inlet diameter (mm)	140
3	Number of blades	12
4	Impeller blade type	Airfoil curved
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)	2800

Different blowers with different angles of attack are named as shown in Table II. Here,angles of attackare increased and the performance of eachblower is observed with the help ofnumerical analysis.

TABLE II
CONFIGURATIONS OF THE CENTRIFUGAL BLOWER

Sr. no.	**Blower name**	**Angleof attack (degree)**
1	M_0	0^0
2	M_1	8^0
3	M_2	14^0

The centrifugalblower with different speeds named areas shown inTable III. Thus,speeds of the impeller are increased and the performance of eachblower is observed with the help of numerical analysis.

TABLE III
CONFIGURATIONS OF THE CENTRIFUGAL BLOWER

Sr. no.	**Level**	**Speed (rpm)**
1	N_0	1500
2	N_1	2000
3	N_2	2800

IV. DESIGN MODEL OFTHE CENTRIFUGAL BLOWER

The centrifugal blower and impeller geometry arecreated in the modeling software CATIA V5. It is shown in Figure1.

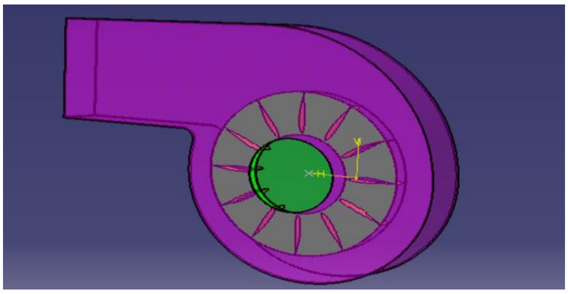

Figure 1: CATIA model of centrifugal blower

V. NUMERICAL ANALYSIS

The commercialCFD is used to simulate different configurations of the centrifugal blower. CFDsolves the Navier-Stokes equation using the finite volume method, which has been applied widely in fluid mechanics and engineering applications. Also, aFluent quasi-steady simulation can be used to study the performance of the blower. In CFD there are three steps to solve the problem,

i. Pre-processing
ii. Solver
iii. Post-processing

Numerical analysis of centrifugal blower is done with varying parameters i.e. speed and angle of attack.

a) Pre-processing:
The solid model created in themodeling software wherein the casing, outlet duct, impelleraredrawn as shown in Figure 2.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 8 Aug 2021, pp: 908-914www.ijaem.net ISSN: 2395-5252

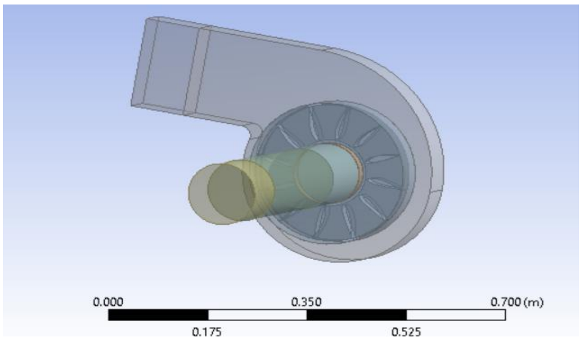

Figure 2: Centrifugal blower solid model withairfoil-curved impeller blades

After the modeling, to study a model meshing important. This meshing process helps to study the model in detail by the discretization method. For the meshing process,meshing size is taken 2 mm for passage and 5mm for all other parts. The meshing model is shown in Figure 3.

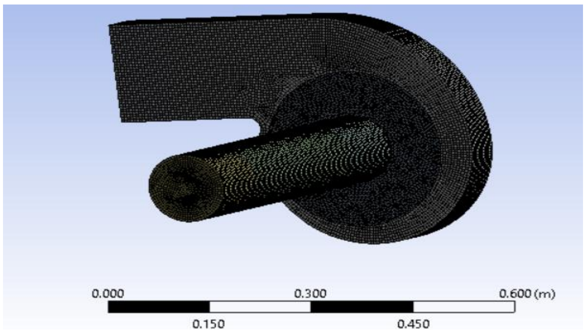

Figure 3: Meshing modelof centrifugal blower

b) Solver:

The second step is a solver.The input of the solver is a mesh model ofthe centrifugal blower. In this solving process, first of all, select the model and give the boundary conditions to this model and solve with sufficient iterations. In this work,atmospheric pressure is set as the inlet boundary condition and static pressure equal to atmospheric pressure as outlet boundary condition.Multiplereference frame solvesthe rotating zone and stationary zone in rotating reference frame and stationary reference frame respectively.Rotational motion is given to impeller and other parts are considered as stationary. Tosolve the continuityequation and Reynolds-Averaged Navier-Stokes equation standard k - ε model is used.

c) Post-processing:

Post-processing includesresults and reports. Results are in the form ofpressure contour, velocity vectors, plots, streamlines, etc.

The numerical analysis is done for the centrifugal blower for different speeds and different angles of attack of the blade.

RESULTS FOR DIFFERENT ANGLES OF ATTACKOFCENTRIFUGAL BLOWER:

Numerical analysis results for various angles of attack are as shown in Figure 4 to Figure 9. All blowers are analysed at same speed.

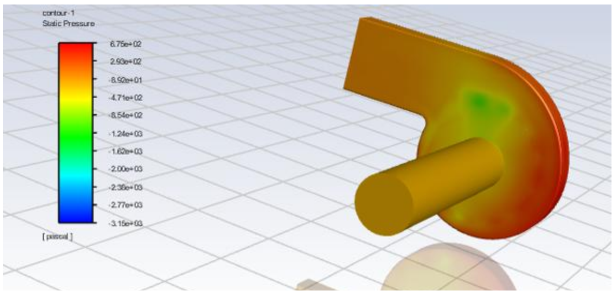

Figure 4. Pressure contour for blower M_0

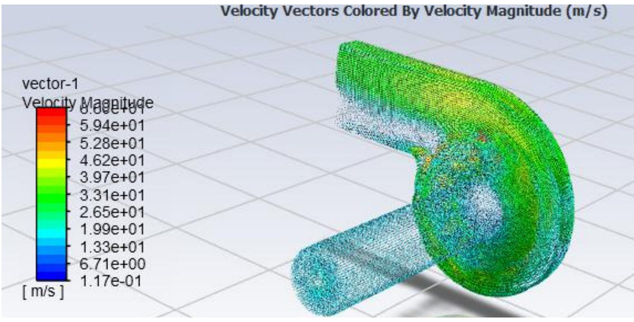

Figure 5. Velocity vector for blower M_0

The pressure contour and velocity vector for blower M_0 with speed 2800 rpm are as shown in Figure 4 and Figure 5.Here, centrifugal blower is analyzed at 2800 rpm and results are obtained. The velocity of the fluid and total pressure of the blower are 14.18 m/s and 568.59 Pa respectively.

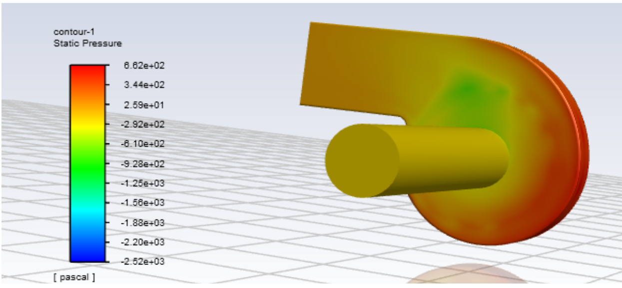

Figure 6. Pressure contour for speedM_1

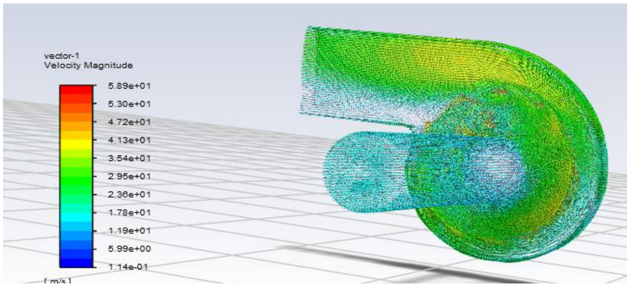

Figure 7. Velocity vector for speedM_1

The pressure contour and velocity vector for speedN_2 are as shown in Figure 6 and Figure 7. Here centrifugal blower is analyzed at 2800 rpm and results are obtained. The velocity of the fluid and total pressure of the blower are 17.1 m/s and 598.78 Pa respectively.

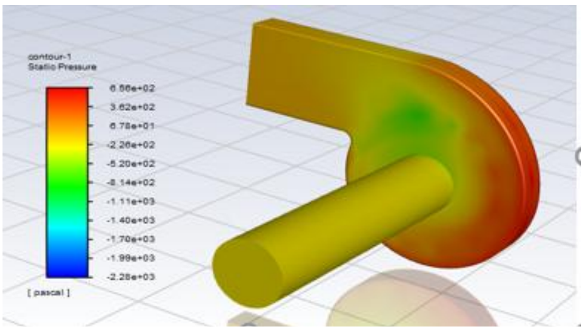

Figure 8. Pressure contour plot for blower M_2

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 8 Aug 2021, pp: 908-914www.ijaem.net ISSN: 2395-5252

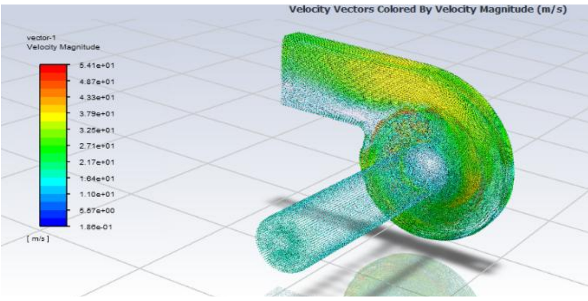

Figure 9. Velocity vector for modified blower M_2

The pressure contour and velocity vector for the blower M_2 are as shown in Figure8 and Figure9.Here, centrifugal blower is analyzed at 2800 rpm and results are obtained. The velocity of the fluid and total pressure of the blower are 15.09 m/s and 564.05 Pa respectively.

From the analysis it is observed that blower $M_1$is more efficient. Hence, for the further analysis this blower M_1 is taken.

RESULTS FOR DIFFERENT SPEEDS OF CENTRIFUGAL BLOWER:

Numerical analysis results for various speeds are shown in Figure 10. to Figure 11.For this analysis, blower M_1 is taken which has angle of attack of 8^0.

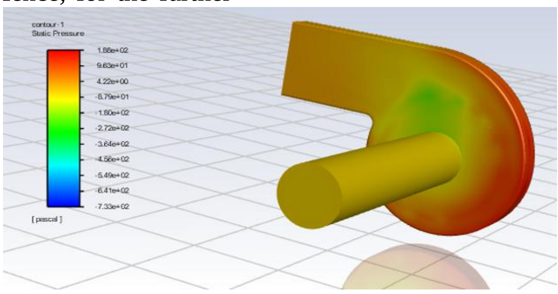

Figure 10: Pressure contour for speed N_0

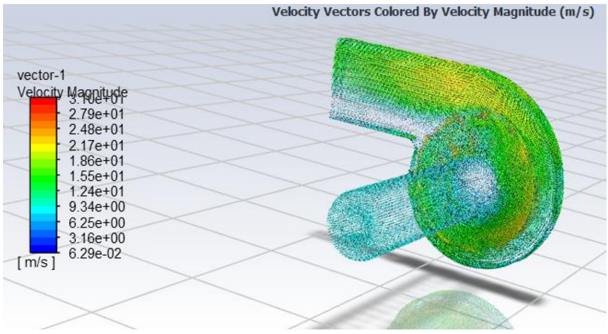

Figure 11:Velocityvector for speed N_0

The pressure contour and velocity vector for speed$N_0$are as shown in Figure 10 and Figure11. Here centrifugal blower is analyzed at 1500 rpm and results are obtained. The velocity of the fluid and total pressure of the blower are 10.1m/s and 184.99Pa respectively.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 8 Aug 2021, pp: 908-914www.ijaem.net ISSN: 2395-5252

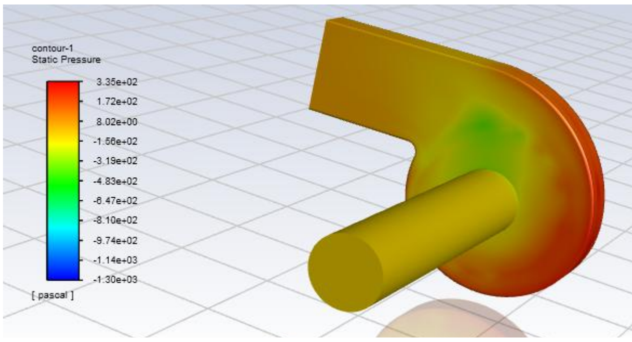

Figure 12: Pressure contour for speed N_1

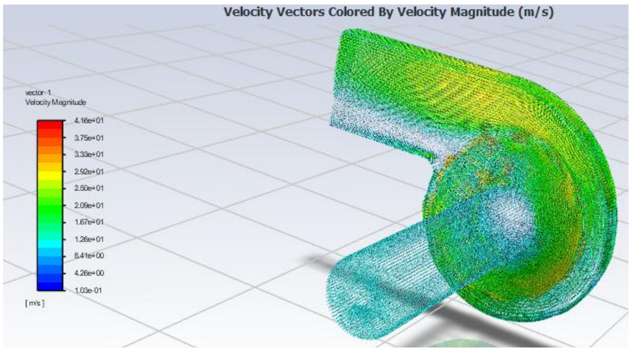

Figure 13. Velocity vector for speed N_1

The pressure contour and velocity vector for the blower speed $N_1$is as shown in Figure 12 and Figure13.Here, centrifugal blower is analyzed at 2000 rpm and results are obtained.The velocity of the fluid and total pressure of the blower are 13.32m/s and 324.59Pa respectively.

Results for speed N_2 are already shown in Figure 6 and Figure 7.Fromthe above-obtained results, it is observed that performance parameters are highest at speed 2800 rpm.

VI. RESULTS AND DISCUSSION

Numerically analyzed results are shown in Table IV and V. From the velocity, flow rates are calculated. The results are obtained from numerical analysis for blowers $M_{0,}$ $M_1$to M_2. The flow rate of the blower M_0 is 677.70m^3/h, for M_1 is 755.45m^3/h and for $M_2$is 720.99 m^3/h. From obtained results, it is observed that the flow rate increases at 8^0 angle of attack and decreases at 14^0. Here, the flow rate of blower M_1 is maximum. Now, the value of total pressure for blower M_1 is maximum and forM_0, M_2isminimum with values 598.78Pa and 564.05 Pa respectively. According to the efficiency, blower M_1 is good with a value of 65.14% while the efficiency of blower M_2 decreases again with a value of60.54%. Thus, from the analysis,it is observed that the blower $M_1$is more efficient blower.

TABLE IV
NUMERICAL ANALYSIS RESULTS OF BLOWERS FOR DIFFERENT ANGLE OF ATTACK

Sr. no.	**Different blowers**	**Flow rate (m^3/h)**	**Total pressure (Pa)**	**Blower efficiency (%)**
1	M_0	677.70	568.59	58.05
2	M_1	755.45	598.78	65.14
3	M_2	720.99	564.05	60.54

TABLE V
NUMERICAL ANALYSIS RESULTS OF THE BLOWER FOR DIFFERENT SPEEDS FOR 8^0ANGLE OF ATTACK

Sr. no.	Speed levels	Flow rate (m^3/h)	Total pressure (Pa)	Blower efficiency (%)
1	N_0	446.20	184.99	55.21
2	N_1	588.45	324.59	58.15
3	N_2	755.45	598.78	65.14

VII. CONCLUSION

- The total pressure for 8^0angle of attack is 598.78 Pa where total pressures for 0^0 and 14^0angle of attack are 568.59 Pa and 564.05 Pa which shows 8^0 angle of attack has greatest total pressure.
- The centrifugal blower with angle of attack 8^0gives high flow rate value i.e. 755.45 m^3/h which is greater than 0^0 and 14^0angle of attack.
- The centrifugal blower having angle of attack 8^0 is the most efficient blower.
- As speed of impeller increases the parameters total pressure, flow rate and efficiency increase.

REFERENCES

[1]. Chen-Kang Huang, Mu-En Hsieh, The Performance Analysis and Optimized Design of Backward curved Airfoil Centrifugal Blowers,HVAC&R RESEARCH, Vol. 15, Number 3, MAY (2009)461-482.
[2]. Sunil R. Patilet.al Effect of Volute Clearance Variation on Performance of Centrifugal Blower by Numerical and experimental Analysis, Vol. 5, Issue2,(2018)3883-3894.
[3]. Mayurkumarkevadiyaet.al., 2-D Analysis of NACA 4412 Airfoil, International Journal. of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 5,May (2013) 1686-1691.
[4]. Parth Shah et.al., Design and Analysis of Airfoil-Curved Impeller Blades of Centrifugal Pump, Trans Tech Publications, Switzerland,Vol.852, July (2016) 539-544.
[5]. G.Rathinasabapathiet.al., Design Optimization of Airfoil Radial Fan Impeller, International Journal of Engineering Research & Technology (IJERT), Volume 6, Issue 02,(2018).
[6]. S.Dajani, M.Sehadehet.al., Numerical Study of a Marine Current Blade Performance under varying Angle of Attack, Energy Procedia 119 (2017) 898–909.
[7]. R.K. Rathore et.al., Study of Functional and Aerodynamic Design with Blade Parameter of NACA 4412, International Journal of Engineering Research & Technology (IJERT), ISNCESR-Volume 3, Issue 20,(2015) 1-4.
[8]. Y Wu et al., Effect of attack angle on the flow characteristic of centrifugal fan, IOP-Conference Series: Materials Science and Engineering, (2016) 81-86.
[9]. Lorenzo Battisti et.al., A generalized method to extend airfoil polars over the full range of angles of attack, Renewable Energy 155 (2020) 862-875.
[10]. AtrePranav C. and Thundil Raj R., Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Curved Impellers, Research Journal of Recent Sciences, Vol. 1(10), October (2012)07-11.
[11]. Chhagan Lal Kharol, Performance Analysis of Centrifugal Fan with Airfoil Curved BladeusingCFD,INTERNATIONALJOURNALRESEARCH CULTURE SOCIETY, Volume - 3, Issue - 5, May (2019)82-89.