NUMERICAL ANALYSIS TO STUDY PERFORMANCE PARAMETERS OF CENTRIFUGAL BLOWER BY USING CFD FLUENT

Abhilasha V. Mane[1], Sunil R. Patil[2]

[1]P. G. Scholar, Department of Mechanical Engineering, AISSMS COE, Pune, Maharashtra, India,
[2]Assistant Professor, Department of Mechanical Engineering, AISSMS COE, Pune, Maharashtra, India

---

Abstract - *The important parameters for design of centrifugal blower include impeller outlet and inlet diameter, rotational speed of impeller, various angles of blade, etc. The relation among many of the parameters is well explained in the literature but how angle of attack of impeller blades affects the centrifugal blower performance is not yet clear. Angle of attack is the angle between chord line and uniform velocity. Due to curves of airfoil blade, there is pressure difference between both upper and lower surfaces. Hence, this creates lift and carries blade easily. As angle of attack increases, the lift also increases upto some angle of attack. After that, it decreases. Thus, it becomes important to find the proper angle of attack in order to get better performance results. A numerical analysis is being carried out on the centrifugal blower to find the effect of angle of attack of blades on its performance parameters with values 0^0, 8^0 and 14^0. The numerical analysis is done by modifying the angle of attack of impeller blades. 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. The performance parameters flow rate, total pressure and efficiency of the centrifugal blower are calculated.*

Key Words: The centrifugal blower, Airfoil curved blades, Angle of Attack, Numerical analysis, Performance parameters.

1. INTRODUCTION

The centrifugal blowers are generally used to carry the fluid at higher pressures and flow rates. In a centrifugal blower, the airflow enters in the inlet duct axially and strikes on the impeller. The air deflects in a spiral pattern to an almost circumferential direction in the impeller blades. The collection of all air streams circulates in the housing and at the end leaves the blower radially through outlet duct. Thus, the velocity and pressure of fluid increases. With the rotating movement of the impeller, kinetic energy converts into pressure energy. The experimental analysis is more expensive and takes more time compared to numerical simulation. Hence the Computational Fluid Dynamics (CFD) analysis with suitable turbulence model is utilized. The behavior of the fluid inside the machine can be correctly predicted with use of numerical simulations. Thus, accurate performance analysis of a particular design can be efficiently carried out. The centrifugal blower requires large volume air at low pressure for operation. It consists of blade, impeller, casing, driveshaft, inlet ducts, outlet ducts, etc. The design parameters like the inlet diameter of impeller, outlet diameter of impeller and impeller width cannot be changed because of the space constraints. But the angle of attack, number of blades, etc. of the impeller can be changed. The performance of the centrifugal blower is studied by changing the angle of attack. In this paper the selection of airfoil curved impeller blade is done for the study.

[1]. Numerical simulation and analysis of backward curved airfoil centrifugal blowers are numerically analyzed. By varying, a static pressure performance is studied. Flow rate 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. The obtained 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 the conventional and normal blade impeller and airfoil curved blade impeller. The

outlet pressure energy is compared. An increased camber on the top side is an ideal trait for lift 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 and optimization. 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 maximum weight which increases vibrations and failure. So, the analysis is done by comparing various 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

2. METHODOLOGY

The methodology used in this paper is as given below:

a) **Design model of the centrifugal blower in CATIA V5:** A model is created by using CATIA V5 and used here to study the performance parameters of the centrifugal blower.

b) **Analysis in Ansys 20.0:** Analysis of the centrifugal blower is the fluid analysis. Hence, for this analysis, CFD is used. The performance of centrifugal blower with different parameters are determined using Ansys 20.0.

3. DETAILS OF CENTRIFUGAL BLOWER

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

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	12
4	Type of blade impeller	Aifoil 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

The blowers with different angles of attack are named as shown in Table II. Here, the performance of each blower is observed with increased angles of attack with the help of numerical analysis.

Table -2: Configurations of The Centrifugal Blower

Sr.no.	Blower	Angle of attack
1	M_1	0^0
2	M_2	8^0
3	M_3	14^0

4. DESIGN MODEL OF THE CENTRIFUGAL BLOWER

The centrifugal blower geometry is created in the modeling software CATIA V5, which is assembly of casing and impeller. It is shown in Fig -1.

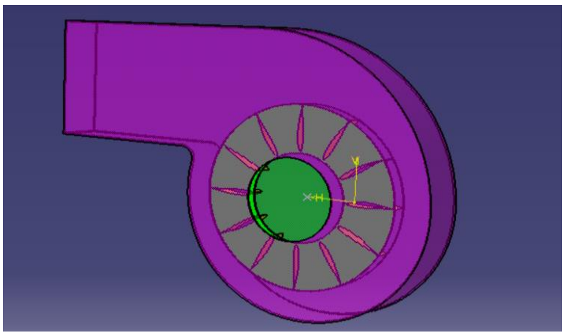

Fig -1: CATIA model of centrifugal blower

5. NUMERICAL ANALYSIS

The numerical analysis of the centrifugal blower is done with the help of the commercial CFD package Fluent. In the engineering and fluid mechanics applications, the Finite Volume method (FVM) is widely used. Thus, using this FVM the Fluent solves the Navier-Stokes equation. In order to predict the performance of the blower, the Fluent qausi-steady simulation is used. The three-dimensional centrifugal blower model is first modelled in CATIA V5 software. In CFD, there are three steps to clear up the problem,

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

a) Pre-processing:

The solid model created in the modeling software wherein the casing, outlet duct, impeller are drawn as shown in Fig -2.

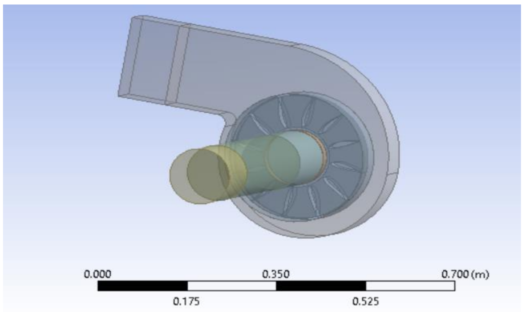

Fig -2: Centrifugal blower solid model with airfoil-curved impeller blades

After the modeling, meshing of model is very important. It is a method used to study the model in detail. For this study, discretization is important, which splits the model in infinite number of parts. Here meshing size is taken 2 mm for passage and 5 mm for all other components. The meshing model is as shown in Fig - 3.

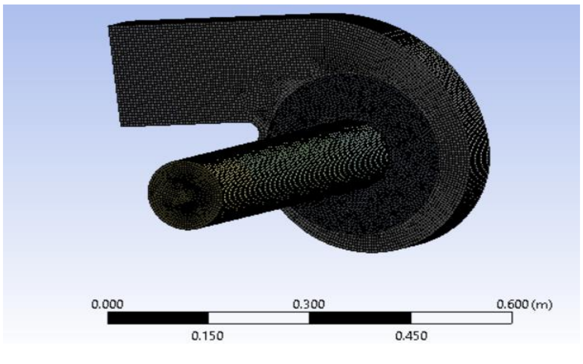

Fig -3: Meshing model of centrifugal blower

b) Solver:

Solver is the second step in processing. This step involves mainly the boundary conditions for model. Boundary conditions for this model are atmospheric pressure is set as the inlet boundary condition and static stress equal to atmospheric pressure as outlet boundary condition. After the selection of boundary conditions, it solves the solution with enough iterations. Here, multiple reference frame is used to solve the rotating frame zone and stationary zone where impeller is considered as rotating zone and other than impeller is considered as stationary. For solving the continuity equation and Reynolds-averaged Navier-stokes equation k - ε model is used.

c) Post-processing:

Post-processing gives results and reports. Results are in the form of pressure 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 ATTACK OF CENTRIFUGAL BLOWER:

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

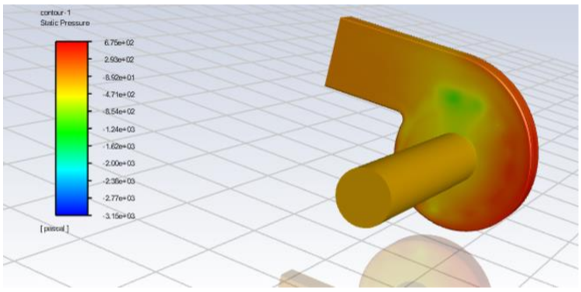

Fig -4. Pressure contour for blower M_0

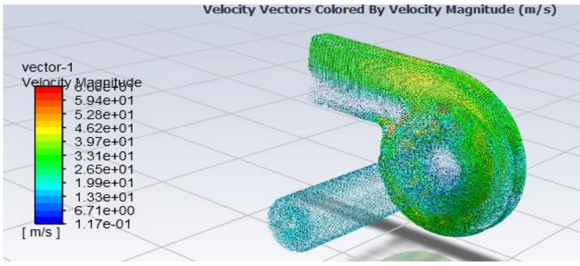

Fig -5. Velocity vector for blower M_0

For blower M_0, the pressure contour and velocity vector are as shown in Fig -4 and Fig -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.

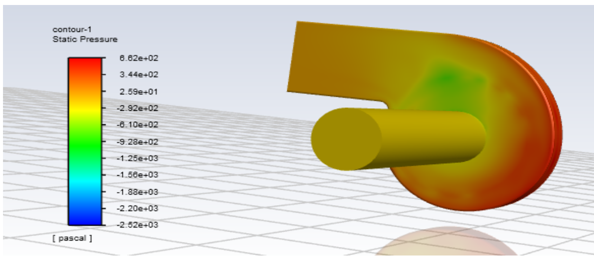

Fig -6. Pressure contour for speed M_1

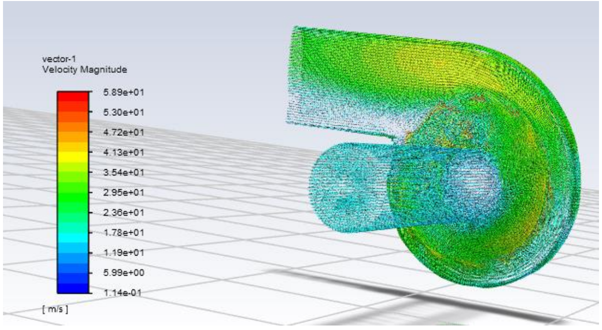

Fig -7. Velocity vector for speed M_1

The pressure contour and velocity vector for speed N_2 are as shown in Fig -6 and Fig -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.

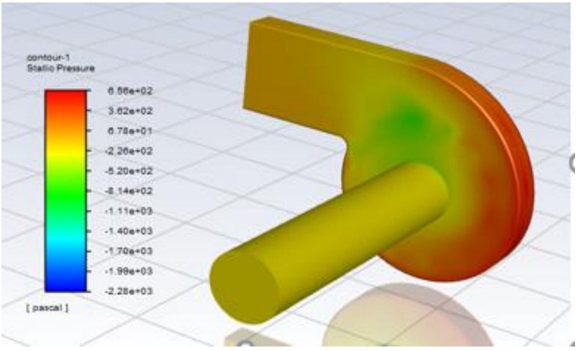

Fig -8. Pressure contour plot for blower M_2

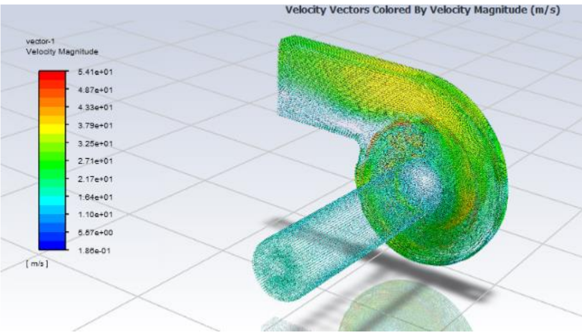

Fig -9. Velocity vector for modified blower M_2

The pressure contour and velocity vector for the blower M_2 are as shown in Fig -8 and Fig -9. 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.

6. RESULTS AND DISCUSSION

Numerically analyzed results are shown in Table III. Velocity and total pressure are obtained in form of readings. Flow rate and efficiency are calculated from obtained readings. The flow rates of the blower M_0, M_1 and M_2 are 677.70 m^3/h, 755.45 m^3/h and 720.99 m^3/h respectively. By comparing results of all blowers, it is observed that the flow rate increases at 8^0 angle of attack and decreases at 14^0. Now, the value of total pressure for blower M_1 is maximum and for M_0, M_2 is minimum with values 598.78 Pa and 564.05 Pa respectively. Blower M_1 is more efficient with a value of 65.14 % than blower M_0 and M_2 having the values 58.05 % and 60.54 % respectively.

Table -3: Numerical Analysis Results of Blowers for Different Angle of Attack

Sr.no.	Different Blowers	Flow Rate (m^3/h)	Total Pressure (Pa)	Efficiency (%)
1	M_1	677.70	568.59	58.05
2	M_2	755.45	598.78	65.14
3	M_3	720.99	564.05	60.54

7. CONCLUSIONS

- The centrifugal blower M_1 having angle of attack 8^0 is the most efficient blower compared to 0^0 and 14^0 angles of attack.
- The total pressure of blower of 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.

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. Patil et. al Effect of Volute Clearance Variation on Performance of Centrifugal Blower by Numerical and experimental Analysis, Vol. 5, Issue 2, (2018)3883-3894.

[3] Mayurkumar kevadiya et.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.Rathinasabapathi et.al., Design Optimization of Airfoil Radial Fan Impeller, International Journal of Engineering Research & Technology (IJERT), Volume 6, Issue 02, (2018).

[6] S.Dajani, M.Sehadeh et.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] Atre Pranav 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.