International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8 Issue-6, March 2020

Effect of Impeller Parameters on the Flow inside the Centrifugal Blower using CFD

Yogesh R Pathak, Kailas D Deore, Rohan R Ozarkar

Abstract: ***A numerical analysis is carried out to understand the flow characteristics for different impeller configurations of a single stage centrifugal blower. The volute design is based on constant velocity method. Four different impeller configurations are selected for the analysis. Impeller blade geometry is created with point by point method. Numerical simulation is carried out by CFD software GAMBIT 2.4.6 and FLUENT 6.3.26. GAMBIT work includes geometry definition and grid generation of computational domain. This process includes selection of grid types, grid refinements and defining correct boundary conditions. Processing work is carried out in FLUENT. The viscous Navier-Stokes equations are solved with control volume approach and the k-ε turbulence model. In this three dimensional numerical analysis is carried out with steady flow approach. The rotor and stator interaction is solved by mixing plane approach. Results of simulation are presented in terms of flow parameters, at impeller outlet and various angular positions inside the volute. Also, the contours of flow properties are presented at the outlet plane of fluid domain. Results suggest that for the same configurations of centrifugal blower, as we change geometrical parameter of impeller the flow inside the blower get affected.***

Keywords: ***Backward swept shrouded Impeller, CFD, centrifugal blower, k-ε turbulence model, mixing plane approach, Navier-Stokes equations, Numerical simulation***

I. INTRODUCTION

Centrifugal blowers are widely used in industrial applications and consume a large amount of power. It is therefore important to minimize the energy consumption and widen the operation range. For these reasons, the flows inside the impellers should be studied in detail in order to optimize the performance of the blower. The flow inside centrifugal impeller is very complex for computation and measurement because of the rotation, viscous effects and complex geometry. Some special phenomena, such as jet-wake structure and separation, have great effect on the losses inside the impeller which greatly affects the range of operation. Therefore, numerical simulation is a good tool as it offers flow visualization that allows the researchers to look inside the turbomachines during operation and provides invaluable insight as to how the turbomachines operate and how it might be improved. This paper deals with a numerical study of the flow through the centrifugal blower for different impeller configurations. The volute design is based on constant velocity method and volute width is twice of the impeller width.

In recent years, a number of researchers have done considerable work on the computation and optimization centrifugal impellers including Xu-wen Qiu [3], Tahsin Engin [4], Tushar Goel [5] and Sheam-Chyun Lin [6].Their achievements contribute significantly in revealing the flow mechanisms inside centrifugal impellers and are most valuable for optimizing the centrifugal impellers with good performance. Zang bin et.al. [1] had conducted the blade optimization design and performance investigations of an ultra-low specific speed centrifugal blower.

The results revealed that the total pressure rises of three blades (backswept blade, straight blade, optimized blade) increased orderly in the same operating range. Choon-Man Jang et. al. [2] did the the shape optimization of an impeller used for two-stage high pressure ring blower. The results showed that hub height is effective to increase pressure in the ring blower. The three dimensional, compressible, steady flow computations were carried out for alternative volute designs by D. Pan et. al. [7].

The objective of the present work was to carry out the numerical simulation to analyze the effect of various geometrical parameter of rotor part. The centrifugal blower stage for different impeller shape i.e. taper and straight with different number of blades is simulated and comparative analysis is done with the simulation results.

II. NUMERICAL MODEL AND METHODOLOGY

The present investigation of flow analysis was carried out for single stage centrifugal blower. The domain consists of suction duct, impeller, vaneless space and volute casing as shown in figure 1. Blade profile of impeller is generated with point by point method [8]. GAMBIT and FLUENT are two CFD programmes used for meshing and simulation. The co-ordinates of each node are spatially created and then two dimensional lines and curves are generated. From lines, faces are created and then from faces volumes are generated.

Air or gas enters the impeller axially through the suction duct. Backward curved impeller with designed flow rate Q = 2.733 m^3/s and head H = 272.12 m of air is used.

Manuscript received on February 10, 2020.
Revised Manuscript received on February 20, 2020.
Manuscript published on March 30, 2020.
* Correspondence Author
Yogesh R Pathak*, is currently working as Assitant Professor in department of Mechanical Engineering at R C Patel Institute of Technology, Shirpur India Email-yogeshpathak123@gmail.com
Kailas D Deore, is currently working as Assitant Professor in department of Mechanical Engineering at R C Patel Institute of Technology, Shirpur India Email-kailasdeore1@gmail.com
Rohan R Ozarkar, is currently working as Assitant Professor in department of Mechanical Engineering at R C Patel Institute of Technology, Shirpur India Email-rohanozarkar@gmail.com

Volute casing was based on constant mean velocity method design concept. Detailed specification of an impeller and volute casing are as shown in table 1.

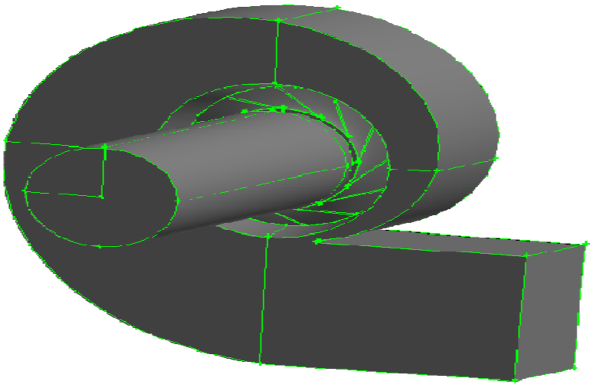

Fig. 1. Modeling of blower in GAMBIT

Table 1 Specification of impeller and volute casing

Specification of impeller		Specification of volute casing	
Parameter	size (cm)	Parameter	size (cm)
Inlet Diameter	30	Volute base circle diameter	46.75
Outlet Diameter	42.5	Volute exit diameter	125.26
Inlet blade angle	20^0	Volute width	26
Outlet blade angle	48^0	Throat length	39.26
Thickness of blade	0.5	Tongue angle	21.84^0
Number of blades	12	Tongue radius	25.51

III. GRID GENERATION

The meshing of the complete domain is done using structure grid except impeller. Meshing of the impeller is done using unstructured type of mesh. Two different types of meshing schemes are applied as mixing plane model being used for interaction between rotor and stator. In mixing plane model, stator and rotor meshes do not have to be conformal. Impeller meshing is done by unstructured grid.

The inlet and outlet faces of the suction duct are meshed by using Quad/Tri wedge primitive meshing scheme and other faces are meshed by map quadratic scheme. Vaneless space is meshed with map quadratic scheme while volute is meshed with combination of quadratic map and sub map scheme. Impeller is meshed by using TGrid scheme with interval size of 0.4. In the domain total 7, 09,757 nodes, 4, 90,520 mixed cells, 9, 27,528 hexahedral cells are present. The meshing of the complete domain is as shown in figure 2.

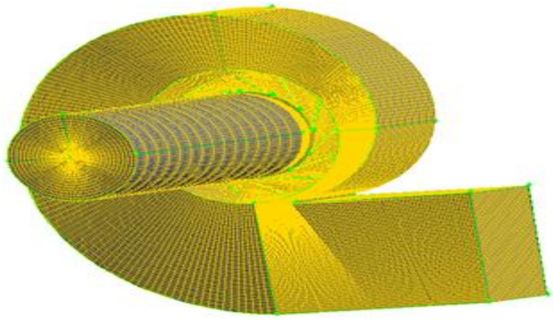

Fig. 2. Meshing of blower

IV. SIMULATION DETAILS

To simulate the present fluid domain, pressure based implicit 3-D steady absolute formulation solver is selected. Initially 100 iterations are done by standard *k- ε* model and then realizable *k-ε* model is selected for rest of the iterations. The near wall modelling is done with the help of standard wall functions. The mixing plane model is used in present problem for rotor stator interactions. Two mixing planes are defined, one between inlet duct outlet and impeller inlet and other between impeller outlet and vaneless inlet. At upstream zone the pressure outlet condition and at downstream zones pressure inlet conditions are imposed.

The simulation is carried out at operating condition of atmospheric pressure. At inlet boundary, uniform axial velocity is imposed based on the specified mass flow rate and that value is 38.68 m/s. For turbulence model, a turbulence intensity of 5% and a reference length equal to 7 % of the inlet diameter. Inlet turbulent kinetic energy is $5.61 m^2/s^2$ whereas turbulent dissipation rate is $103.97 m^2/s^3$. The outlet boundary is specified as pressure outlet. The outlet gauge pressure calculated from designed pressure ratio of fluid domain and that value is 2837 pascal. All the stationary walls were considered as moving walls having absolute motion and no slip shear condition. Wall roughness constant is 0.5. The walls of rotating volumes are considered as moving wall with relative to adjacent cell zone motion and no slip shear condition. Air is in stationary motion type and moving air is in moving reference frame motion type with rotational velocity 2900 rpm. The above fluid domain problem is solved for the flow, turbulence and energy equations by using SIMPLE pressure velocity coupling. Standard discretization method is used to solve pressure while others were solved by second order upwind discretization method. Solution initialized from inlet of the domain with absolute reference frame. Iterations carried out for convergence down to maximum residual of less than 10^{-4}. The selected configuration of fluid domain is already validated in earlier research work.

The numerical simulation is carried out for the different impeller configurations as shown in table 2.

Table 2 Different configurations of impeller

ase No	Width at outlet(cm)	Width at inlet(cm)	Number of blades
1	17	13	12

2	13	13	12
3	17	13	10
4	13	13	10
5	17	13	8
6	13	13	8

V. RESULTS AND DISCUSSION

The results are presented at angular periphery of impeller outlet i.e. 0 to 360^0 and also at various angular positions inside the volute i.e. 90^0, 180^0, 270^0 and 360^0. Also the contours of stagnation pressure and velocity are plotted at the outlet of fluid domain.

A. Flow at impeller outlet

Fig. 3 to 5 represents the flow property variations near impeller exit at different angular positions. Fig 3 depicted the variation of average stagnation pressure at exit of impeller at various angular positions. The average values of the stagnation pressure, static pressure and velocity were determined based on area average. It is clear from figure that there is slight increase in the value in the stagnation pressure, as we move from 0 to 360^0. However, the variation in the stagnation pressure is non uniform in case of tapered impeller with 12 blades compared to other 5 cases. Also in case of straight impeller i.e. case 2, 4 and 6 more uniform nature is observed compare to tapered impeller.

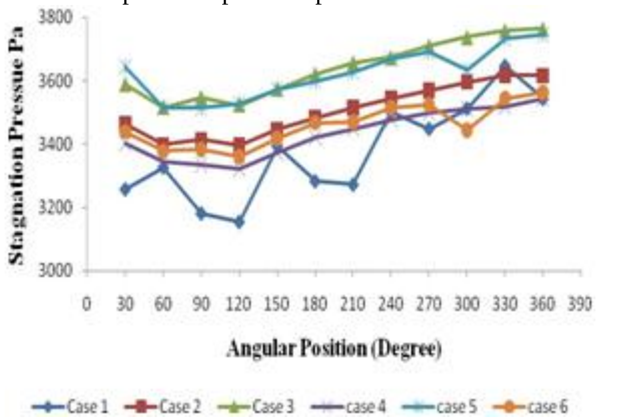

Fig 3 Variation of average stagnation pressure at exit of impeller at various angular positions

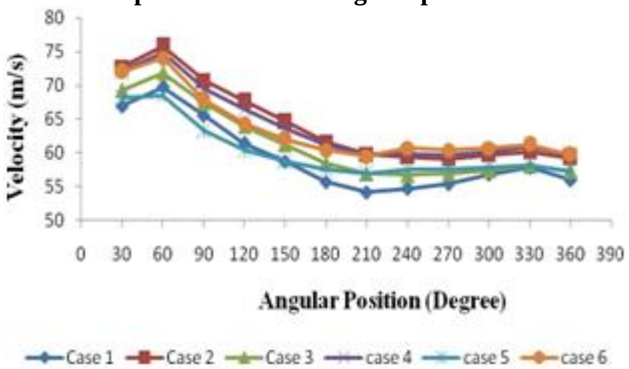

Fig.4 Variation of average velocity at exit of impeller at various angular positions

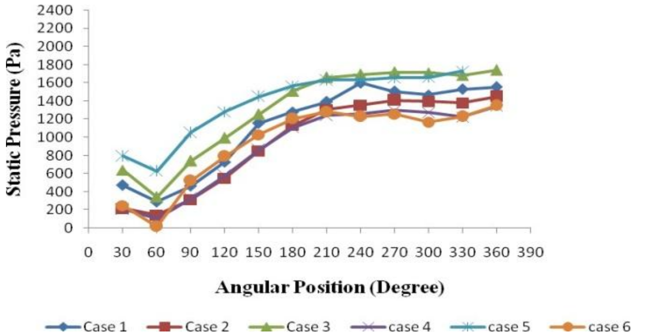

Fig. 5 Variation of average static pressure at exit of impeller at various angular positions

B. Flow inside the volute

As stated earlier, flow properties are plotted at four angular positions of volute i.e. 90^0, 180^0, 270^0 and 360^0 which are shown in figure 6 and 7. As we move from suction to exit of the volute i.e. 90^0 to 360^0 angular position, it is observed from figure 6 that velocity of flow is decrease. It is observed that there is more conversion of kinetic energy take place in case of straight impellers. The maximum percentage of conversion of kinetic energy from 90^0 to 360^0 is 37.5% in case of straight impeller while 31.03% in case of tapered. Figure 7 shows the variation of average static pressure at various angular positions inside volute. It reveals from figure that as we move from suction to exit of volute static pressure increases in all cases. This suggests conversion of kinetic energy into pressure energy inside the volute. Taper impeller with 8 blades shows higher static pressure followed by 10 blades.

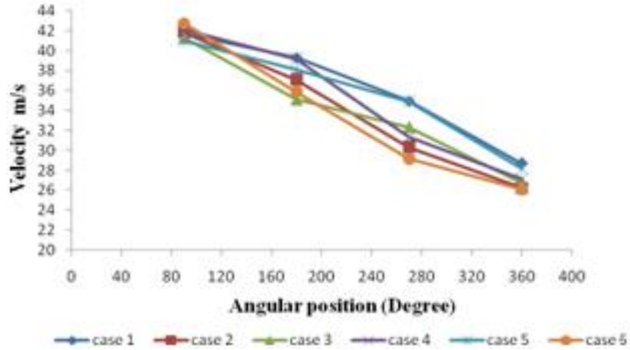

Fig. 6 Variation of average velocity at various angular positions inside volute

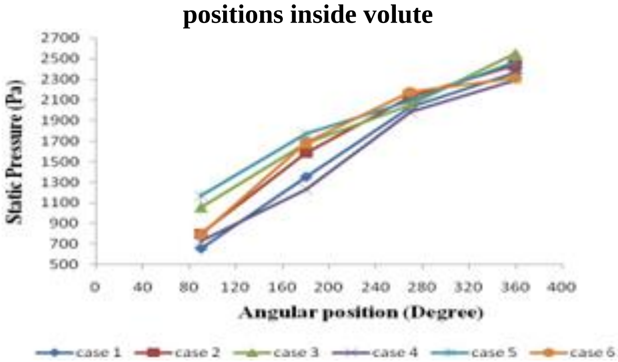

Fig. 7 Variation of average static pressure at various angular positions inside volute

C. Flow at the outlet of fluid domain

To get more clarification about the effect of impeller configuration on the flow properties, the average value of total pressure and velocity are calculated at outlet of fluid domain.

The same are represented in table 3.It is observed from the table that though the boundary condition at inlet and outlet are same, if we change the configuration of impeller, other flow properties are changed. At outlet, comparatively less value of velocity and total pressure is observed in case of straight impeller configuration. Maximum value of total pressure is observed with tapered impeller having 8 blades followed by 10 blades. Maximum velocity is observed in case of tapered impeller with 12 blades which is not desirable.
The average values of total pressure and velocity at the outlet plane are given in the table 3. Static pressure at outlet is same as it is applied as outlet boundary condition.
To get more idea of flow properties at outlet plane the contours of flow properties are plotted in figure 8 and 9.

Case	Total Pressure (Pa)	Velocity(m/s)
1	2917.3	28.69
2	2920.25	26.23
3	2871.94	26.08
4	3247.44	28.19
5	3207.24	23.06
6	3228.94	23.74

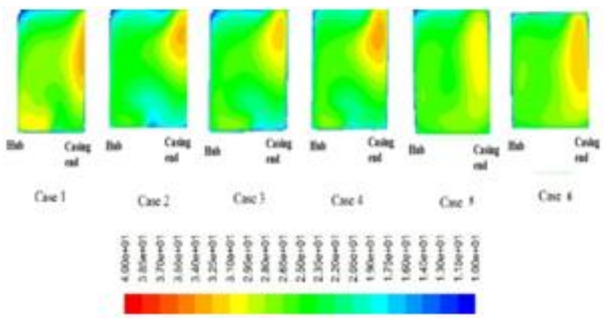

Fig. 8 Contours of velocity at outlet

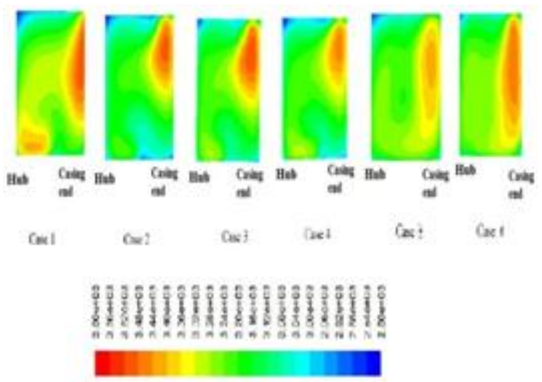

Fig. 9 Contours of stagnation pressure

VI. CONCLUSION

CFD analysis has been done for six cases of different blade configurations to observe the effect of blade geometry on the flow of the centrifugal blower. Following conclusions are deduced from the numerical simulation:

1. At impeller outlet, high stagnation pressure achieved by tapered configuration compared to straight configuration for all blades. Also, the variations are less in case of tapered impeller compared to straight impeller. This suggests less stagnation pressure loss with tapered impeller configurations.
2. The tapered impeller with 10 blades is best configuration for present fluid domain amongst all.
3. As we move from suction end (i.e. tongue) to exit of volute, the static pressure continuously goes on increasing at the expense of the velocity which suggests conversion of kinetic energy in pressure energy. This suggests that volute satisfies its function well.

REFERENCES

1. Zhang Bin, Wang Tong, GU Chuan Gang, Shu Xin Wei, Blade optimization design and performance investigations of an ultra-low specific speed centrifugal blower, Science China, January 2011 Vol.54 No.1: 203–210
2. Choon-Man Jang, Gi-Young Han, Enhancement of Performance by Blade Optimization in Two-Stage Ring blower, Journal of thermal science Vol.19, No.5 (2010) 383−389.
3. Xu-wen Qiu, Mark Anderson, David Japikse, An integrated design system for turbomachinery, J. Hydrodynamics, 347-354.
4. Tahsin Engin, Mesut Gur, Reinhard Scholz, Effects of tip clearance and impeller geometry on the performance of semi-open ceramic centrifugal fan impellers at elevated temperatures, Experimental Thermal and Fluid Science 30 (2006) 565–577.
5. Tushar Goel, Daniel J. Dorney , Raphael T. Haftka, Wei Shyy, Improving the hydrodynamic performance of diffuser vanes via shape optimization, Computers & Fluids 37 (2008) 705– 723.
6. Sheam-Chyun Lin, Chia-Lieh Huang, An integrated experimental and numerical study of forward– curved centrifugal fan, Experimental Thermal and Fluid Science 26 (2002) 421–434.
7. D. Pan, A Whitefield and M Wilson, “Design considerations for the volutes of centrifugal fans and Compressors”, Instn Mech Engrs Vol 213 Part C, 1996.
8. Church AN. Centrifugal pumps and blowers. Joftr witey and Sons, Hemisphere Publishing Corporation; 1989.
9. S.M.Yayha “Turbines compressors and fans” third edition 2005 by Tata McGrawHill publishing company limited Delhi.

Retrieval Number: F8973038620/2020©BEIESP
DOI:10.35940/ijrte.F8973.038620
Journal Website: www.ijrte.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication