Numerical Investigation of the Flow Pattern with in the Impeller of a Centrifugal Blower

Beena D. Baloni¹, and Salim A. Channiwala²

Abstract—Centrifugal blower performance is strongly influenced by the design and geometry parameters of the volute casing. The effect of rotor-stator interaction affects the position and strength of the vortex that characterizes the operation of centrifugal blower. The paper presents a numerical investigation of flow field within the impeller at three different axial positions. Numerical investigations are done with the help of Gambit and FLUENT software of computational fluid dynamics. This study helps to understand the flow behaviour inside the impeller, and it is a part of an extensive work, by the same research group, aimed at establishing a general design of volute casing to improve blower performance by reducing the secondary flow losses.

Keywords—Backward blade impeller, Centrifugal blower, Computational fluid dynamics, Flow field

I. INTRODUCTION

CENTRIFUGAL blowers are widely used in industries for ventilation and process plants. In centrifugal blower, the volute has strong effect on the flow passing through the impeller and thus also on performance. The volute is a passage located circumferentially around the diffuser or impeller exit that collects the flow from the impeller and delivers it to the exit duct. It could be comprehended that, the work of the diffuser or even the impeller will be affected seriously due to the poor design of volute, as it leads to non-symmetrical pressure distribution [1]. This, stator-rotor interaction not only affects the flow at impeller outlet, but also leads to substantial changes to the secondary flow patterns inside the impeller passages.

Secondary flows are defined as the difference between the full three dimensional inviscid solution and the real viscous flow occurring in the impeller [2]. They redistribute the low energy fluid at impeller exit through the stream wise vorticity and influence the level of the inviscid core velocity and pressure by blockage. The different vortices inside the impeller passages are: passage vortices due to the flow turning in the vane to vane plane, blade surface vortices generated by the meridional curvature of impeller and vortices due to Coriolis forces. Hirsch et al. [3] had proposed an approximated integration of these three terms to evaluate the intensity of the generated vortices at impeller exit. These secondary flows were confirmed by the results of 3D Navier Stokes analysis of a radial impeller [2]. W. K. Ng et al. [4] had also reported secondary flow inside the impeller of industrial fan by the computational flow analysis.

Fatis et al. [5] investigated the response of the centrifugal impeller to downstream static pressure distortions imposed by volutes and analyzed the mechanisms that governed the unsteady flow field in the volute. According to Fatis et al. [5] the radial force of a centrifugal compressor can quite accurately be determined from the static pressure distribution at the impeller exit, if the impeller blades have high backsweep (40° in Fatis study). If the impeller has got radial blades, the pressure measurement alone is not adequate, since the radial momentum contributes to the radial force to a great extent. Wetche [6] and Iverson et al. [7] used the measured static pressure distribution to determine the impeller force of a centrifugal pump. Hillewaert and Braembussche [8] calculated the flow in a compressor with an external volute under off-design operating conditions to understand the strong interactions between radial impellers and volutes.

The aim of the present paper is to highlight the different features of flow field inside the impeller passage through the numerical simulation.

II. NUMERICAL SIMULATION

The Computational Fluid Dynamics programs are gaining more importance for the design and analysis of fluid turbomachinery. Numerical simulation offers flow visualization that allows the researchers to look inside the turbomachines during operation and provides invaluable insight, as to how the turbomachines operates and how it might be improved. The numerical analysis usually implies full 3D simulation to account for turbulent effects, secondary flows, and unsteady phenomena. The simulation work include geometry creation, mesh generation, zone assignment, development of fluent model, and convergence of present fluid domain.

A. Geometrical Model

Simulation work is based on the flow analysis inside the centrifugal blower. The domain consists of suction duct, impeller, vane less space and volute casing as shown in the figure 1. Air or gases enters the impeller axially through suction duct. Impeller of blower unit is backward curved. Inside and outside diameter of impeller are 30 cm and 42.5 cm respectively. Total 12 numbers of blades are there and rotating at angular velocity of 2900 rpm. Volute casing is based on

¹Beena Baloni is with Sardar Vallabhbhai National Institute of Technology-Surat, India. ( phone: +912612201994; e-mail: pbr@med.svnit.ac.in)
² Salim A. Channiwala is with Sardar Vallabhbhai National Institute of Technology-Surat, India. (e-mail: sac@med.svnit.ac.in)
constant velocity method design concept having base circle radius: 23.4 cm, and width of volute: 26 cm.

The mixing plane model is used in the present problem for stator rotor interactions [9]. Two mixing planes are defined, one between inlet duct outlet and impeller inlet and other between impeller outlet and vaneless inlet. At upstream zones the pressure outlet condition and at downstream zones pressure inlet condition are defined. The simulation is carried out at atmospheric pressure operating condition. At the inlet boundary, a uniform axial velocity (along the Z axis) is imposed based on the specified mass flow rate. For the turbulence model, inlet turbulent kinetic energy (k) is 5.61 m²/s² whereas turbulent dissipation rate (ε) is 103.97 m²/s³. The outlet boundary is specified as pressure outlet. All the stationary walls considered as moving wall having absolute motion and no slip shear condition. Wall roughness constant is 0.5. The walls of rotating volumes considered as moving wall with relative to adjacent cell zone motion and no slip shear condition. The air as fluid separated as air & moving air. Air having stationary motion type and moving air having moving reference frame motion type with rotational velocity equal to 2900 rpm.

The above fluid domain problem is solved for the flow, turbulence and energy equations by using SIMPLE pressure-velocity coupling. Standard discretisation method is used to solve pressure while others are solved by second order upwind discretisation method. Solution initialized from Inlet of the domain having absolute reference frame. Iteration carried out for converge down to maximum residual of less than e⁻⁰⁴. The present fluid domain numerical simulation is validated with the establish reference [10] and experimental work [11]. A good agreement is obtained between numerical and experimental results. The average value of pressure coefficient is 0.21 in case of experiment [10] and 0.22 in present CFD.

III. NUMERICAL ANALYSIS

The numerical simulation of a centrifugal blower stage was carried out with the help of FLUENT software. The results of converged simulation model are analyzed. The variations of flow parameters such as total pressure, static pressure and flow velocity are presented in form of contours inside the impeller. The velocity vector diagrams and pathlines were also plotted, to analyze the flow behavior inside the contours. The analysis is carried out at different axial locations. The axial distance is non-dimensionalised in terms of width of the impeller.
The motor side overhung portion as shown in Fig. 2 was considered as negative whereas, volute casing wall side considered as positive. The three axial locations were identified to observe the flow phenomenon inside the impeller. They are 15.4%, 54%, and 92.3% of the impeller width.

IV. RESULTS AND DISCUSSION

The variations in the total pressure inside the impeller are presented in Fig. 3. The figure reveals that high pressure contours are observed near the tip region, at pressure surface of impeller blade on motor overhung side. These pressure contour value decreases as we move away in tangential and axial direction from tip of blades. Near suction surface of impeller blade, low pressure contours observed at hub region. Except these two regions, variations in contours are less at all other locations of vane to vane passage.

Fig. 4 shows the variation in static pressure inside the impeller. High pressure region is observed at inlet and the pressure surface near the hub region of impeller. Whereas low pressure region, found at suction surface, near hub region of the impeller. This indicates non-uniform pressure distribution. Highest pressure contours observed near the middle passage of pressure side of blade. The variations in static pressure are less towards outflow direction near the tip region of impeller.

The variation in the velocity inside the impeller is presented in Fig. 5. The figure reveals that as we move from inlet to outlet of the impeller the velocity increases gradually. At the outlet surface three different regions of velocity contours are found. High velocity contours are observed at the tip region, near the pressure surface of blade, at the motor wall side. Volute wall side, comparatively less velocity contours are found. And, in rest of the outlet regions medium velocity contours are visualized. Pattern of stagnation pressure, static pressure, and velocity contours remain same at all circumferential locations of impeller outlet, which justify uniformity in flow at impeller outlet at designed flow rate of volute [11-19].

One way of looking to the secondary flow structure is by displaying the velocity vectors induced by the streamwise vorticity in impeller cross sectional planes [2]. Therefore, to visualized the real flow behavior inside the vane to vane passages with simulation results, the vector diagram at different axial positions inside the impeller are plotted as shown in Fig. 6. The vector length and colors indicates the magnitude of velocity at that particular location. Fig. 6 shows that as we move in axial direction from starting to end of impeller width the flow turning increases inside the vane to vane passage. Flow turning develops vorticity generation in the meridional plane and vane to vane plane [2]. The high velocity regions appeared (Sky blue colour) at pressure surface of the blade was disappeared slowly as we move from starting to end of the impeller. Very low velocity regions created near pressure side, at the impeller inlet regions are also decreased as we move from starting to end of the impeller. Thus, the Fig. 6 reveals that the flow behavior changes inside.
the vane to vane passages, as we move from starting to end of impeller. The behavior of flow is depended on the impeller dimensional configurations and speed.

Fig. 4.11 Variation of flow vectors inside the impeller vane to vane passage at different axial positions: a) 15.4% impeller width, b) 54% impeller width, c) 92.3% impeller width

Fig. 7 represents pathlines inside the impeller at three different axial positions from starting to end of the impeller width. Fig. 7 reveals that vortexes are formed inside the impeller blade passage at all the axial positions. The eddy formation observed near the suction surface at the hub region of impeller, covers almost 25-30% of passage area, in case of axial distance near the starting of Impeller width (refer: a). This justifies the statement [2] that, the vorticity transports low energy fluid from the pressure to the suction side. As we move towards axial direction of impeller i.e. end of the impeller width, the radius of eddies are increased. This produces very large eddy formation, which covers almost all the vane to vane passage area of impeller at the 54% of Impeller width (i.e. middle of impeller passage, refer: b). Whereas, near the end of the impeller, the large single eddy converted into two small eddies. These two eddies, one at hub region and other at tip region of the impeller, covers vane to vane passage area of impeller. However, these vortex flows are undesirable as they are responsible for head losses, flow non-uniformity and slip. These secondary vortices, leads to secondary flow losses and reduce the efficiency of the stage.

Fig. 4.12 Representation of pathlines inside the impeller vane to vane passage at different axial positions: a) 15.4% impeller width, b) 54% impeller width, c) 92.3% impeller width
V. CONCLUSIONS

The present results incorporate the flow phenomena inside the impeller. In the present paper, only numerical analysis has been conducted. However, they were sufficient to indicate the existence of the secondary flow which seriously affects the performance of turbomachine. The study reveals that the flow behavior changes inside the vane to vane passages, as we move from starting to end of impeller. The behavior of flow is depended on the impeller dimensional configurations and speed.

REFERENCES


