

Numerical Analysis of Forward Curved Impeller of Centrifugal Blower

Dr. Sunil R. Patil Department of Mechanical Engineering,
AISSMS College of Engineering, Pune- 411001, Maharashtra, India

Abstract

Centrifugal blowers are extensively employed for conveying compressible fluids, such as air and gases, in ventilation systems, power plants, and various industrial applications. This study focuses on the numerical analysis of a forward-curved centrifugal blower operating at different impeller speeds. A Computational Fluid Dynamics (CFD) model was developed using Ansys Fluent to investigate the flow characteristics and performance of the blower. The CFD simulation enables detailed visualization of the internal flow behaviour and facilitates the prediction of key performance parameters. The standard $k-\epsilon$ turbulence model was employed to simulate the turbulent flow within the blower. Performance characteristics such as total pressure, flow rate, efficiency, and power consumption were evaluated and compared at various impeller speeds. The results provide valuable insights into the influence of rotational speed on blower performance and can assist in optimizing centrifugal blower design and operation.

Keywords: Centrifugal blower; CFD; Blower performance; Impeller

1. Introduction

A blower is a type of turbomachine used to move air or gas continuously while producing a moderate increase in static pressure. Blowers are commonly used to overcome pressure losses in systems such as power plant boilers and mine ventilation networks, where pressures can exceed 1600 mm H₂O. [1] A centrifugal fan's performance is influenced by a number of variables, including fan size and type, airflow rate, static pressure, density of the air to be moved, system resistance, temperature range, space limitations, material to be handled, changes in operating circumstances, etc. [2] The losses and performance also vary with the blade geometry. [3] The blade can be either of uniform sheet metal of uniform thickness or of aerofoil section. The impeller can be either backward curved, forward curved, or radial, depending on the type of blades. [1]. Selecting the appropriate fan size during system design is crucial for optimal system performance and energy efficiency. When a fan is not used appropriately for a given application, the system's cost rises and fan reliability declines. [4]. Fluid flow in the centrifugal blower was analysed using Ansys Fluent, a computational tool based on CFD. The CFD analysis produced good results. [5]. The following paper compares the performance of a centrifugal blower when using a forward curved impeller at different speeds.

2. CFD Process

Pre-processing, numerical solution, and post-processing are the three stages of the procedure. Prior to processing Creating a 3D model of the blower and geometry, appropriate for the numerical analysis, was the first step in setting up the domain. The two components of the geometry are an impeller with a flow zone and a casing with inlet and exit ducts.



Fig. 1: Geometry

Creation of Mesh: Tetrahedral meshing was used because of the complex impeller region and curved geometry. In the impeller zone, finer mesh was provided. The impeller zone's maximum element size was 5 mm, while the remaining geometry's maximum element size was 10 mm.

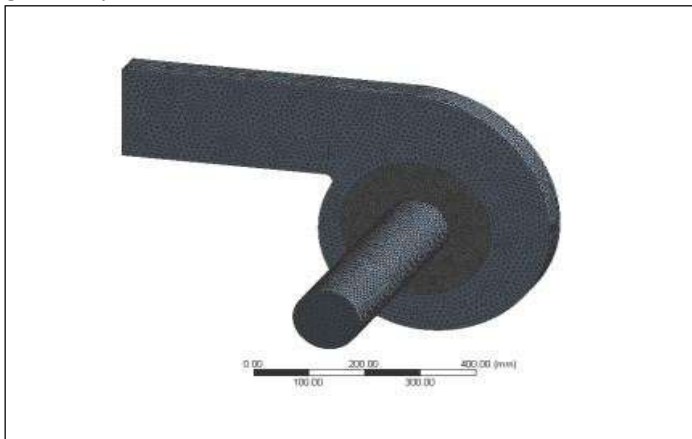


Fig. 2: Meshing

Set-Up: To address this issue, the moving reference frame approach was selected. The solver was provided accurate air material properties and operational conditions. Following assumptions were made for this simulation:

1. Steady-State flow.
2. Flow is incompressible.
3. Negligible effects of gravity.
4. Fluid properties are not a function of temperature.
5. No slip boundary condition.

Numerical Solution: The k- ϵ turbulence model was used to solve the CFD problem in Ansys Fluent Solver. Different impeller speeds of 2800, 2500, 2100, and 1400 rpm were used to solve it. The k- ϵ turbulence model consists of following governing equations:

1. Reynold's Continuity Equations.
2. Reynold's Averaged Navier Stokes (RANS) equations.
3. Conservation equation for k (turbulent kinetic energy).
4. Conservation equation for ϵ (turbulent dissipation).

After running the solver for sufficient number of iterations, convergence was obtained. A check for discretization error was performed using mesh refinement studies.

Post-Processing :Ansys CFD-Post turned out to be a really helpful tool for results visualisation. Plotting was done for

streamlines, velocity vectors, and pressure contours. The centrifugal blower's anticipated trends were examined in these figures. The built-in calculator was used to determine the impeller's torque. The pressure and velocity values across the inlet and outflow surfaces were provided by the solver.

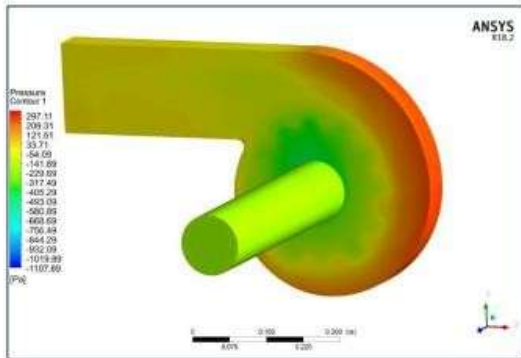


Fig.3. Pressure Contours for forward curved Impeller at 2800 rpm

Pressure contours are shown in fig. 3, it demonstrates that the centrifugal blower with a forward curved impeller produced an adequate pressure. Higher suction pressure is being produced inside the blower, as indicated by the bright red zone to the right, which is consistent with the predicted trends.

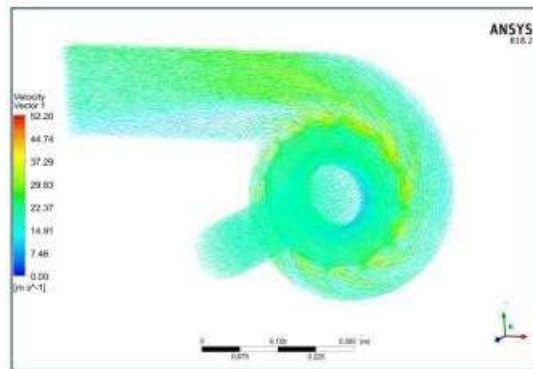


Fig.4. Velocity Vectors for forward curved Impeller at 2800 RPM

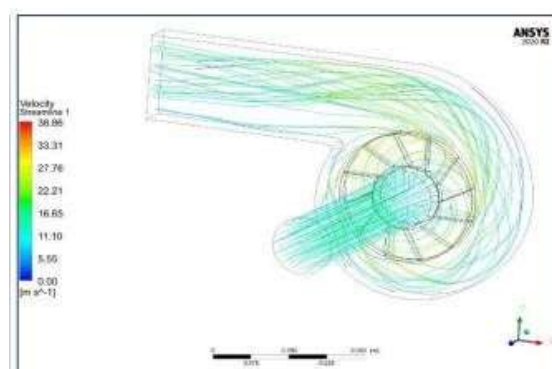


Fig.5. Streamlines for forward curved Impeller at 2800 RPM

The concentration of higher velocity vectors in the forward curved impeller enhances air delivery, as seen in Fig.4. The air flow from the impeller zone to the forward curved impeller's exit duct is shown by streamlines in Fig.5. The pressure contours and velocity vectors at 2500, 2100, and 1400 rpm make it clear that as rpm decreases, the total pressure, outlet velocity also decreases and consequently the power required and efficiency.

3. Results

Sr. No.	Speed (rpm)	Total Pressure (Pa)	Outlet Velocity (m/s)	Area (m ²)	Flow Rate (m ³ /s)	Torque (N/m)	Shaft Power (W)	Air Power (W)	Blower Efficiency %
1	2800	446.37	19.09	0.012	0.23	1.17	187.05	102.44	54.77
2	2500	357.07	16.99	0.012	0.20	0.94	133.50	72.94	54.64
3	2000	248.38	14.17	0.012	0.17	0.66	78.58	42.32	53.86
4	1500	104.50	9.30	0.012	0.11	0.30	23.81	11.69	49.07

4. Conclusion

This paper presents a numerical analysis of the effect of varying the rotational speed of a forward curved impeller on the performance of a centrifugal blower. The CFD results reveal that the total pressure and efficiency of the blower, lower as the impeller speed decreases. However, at lower working speeds, the forward curved impeller centrifugal blower's shaft power and air power requirements also drop.

5. References

1. Turbines, Compressors and Fans, S MYahya, Tata McGraw Hill Publications,1983.
2. Jasmina Jovanovic , Bozidar Bogdanovic,Zivan Spasic,Centrifugal Fans with Backward-curved and Forward-curved Blades,17th Symposium on Thermal Science and Engineering of Serbia, Sokobanja,Serbia,October 20-23,2015.pp. 290-297
3. Aung Ko Latt, Design Comparison of Forward and Backward Curved Radial Tip Blades for Centrifugal Blower, IRE Journals, Volume 2, Issue 5,2018, pp.138-145.
4. Bureau of Energy Efficiency (BEE), Government of India, Energy Efficiency Guide Book, chapter 5. 2004, pp. 93-112.
5. Yogesh R Pathak, Kailas D Deore, Rohan R Ozarkar, Effect of Impeller Parameters on the Flow inside the Centrifugal Blower using CFD, International Journal of Recent Technology and Engineering ,Volume-8 Issue-6, March 2020, pp 3977-3780.