Numerical Analysis of Backward Inclined Impeller of Centrifugal Blower

Dr. Sunil R. Patil

Department of Mechanical Engineering, AISSMS College of Engineering, Pune-411001, Maharashtra, India

Abstract
A centrifugal blower is designed to handle compressible fluids like air and gases and is useful in ventilation, power stations, workshops etc. This paper focuses on numerical analysis of a backward centrifugal blower at different speeds of the impeller. A computational fluid dynamics model was developed in Ansys Fluent. The CFD simulation approach makes it possible to visualize the flow condition inside a centrifugal blower and solve for the required performance parameters. The k-ε turbulence method was modeled and solved using Ansys Fluent. Parameters like blower total pressure, flow rate, efficiency, and power were observed.

Keywords: Centrifugal blower ;CFD; Blower performance ;Impeller

1. Introduction

Blowers are one of the types of turbo machinery that are used to move air continuously with a slight increase in static pressure. A blower, which is also sometimes called a fan, develops an appreciable rise in pressure of the gas flowing through it. In order to overcome pressure losses of the gas during its flow through various passages, a pressure rise is required. The pressure developed by blowers can be more than 1600 mm H_2O in certain applications like power plant boilers and mine ventilation systems. [1]
A centrifugal blower includes an impeller that has fixed blades between the inner and outer diameters. The action of the impeller imparts energy to the gas inside it and delivers it at a high pressure and velocity to the casing. This causes a reduction in pressure at the inlet and sucks the gas inside axially. Thus, unlike the axial blower type, the centrifugal energy also contributes to the pressure rise. The spirally shaped casing that is also known as a scroll or volute collects the gas from the impeller blades. The static pressure can further be increased by the scroll. The outlet passage after the scroll can also take the form of a conical diffuser. [1]
The peripheral speed of the impeller and blade angles determine the pressure rise and flow rate in centrifugal blowers [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 inclined/curved, forward inclined/curved, or radial, depending on the type of blades [1].
Performance of a centrifugal blower is affected by various factors like fan type & size, airflow rate, static pressure, density of the air to be moved, system resistance, temperature range, space constraints, material to be handled, variations in operating conditions, etc. [2]. Centrifugal fans have varied designs, such as backward inclined, radial blade, forward curved, airfoil design, etc., depending on fan blade configuration. These fans deliver different airflow rates at different pressures. [4]. Choosing the right

size of fan in system design is important to have good energy efficiency and system performance. Not using a proper fan for an application increases the cost of the system while decreasing fan reliability [5]. A CFD-based computational tool, Ansys Fluent, was used to analyse fluid flow in the centrifugal blower. The results obtained by the CFD analysis came out to be satisfactory [6-7]. The following paper compares the performance of a centrifugal blower when using a backward-inclined impeller at different speeds.

2. CFD Process

The process is done in three parts i.e., pre-processing, numerical solution and post- processing.

Pre-processing

Setting up the domain: This step involved the creation of a 3D model of the blower and geometry suitable for the numerical analysis. The geometry consists of a casing with inlet and outlet ducts as one part and an impeller with a flow region as the second part.

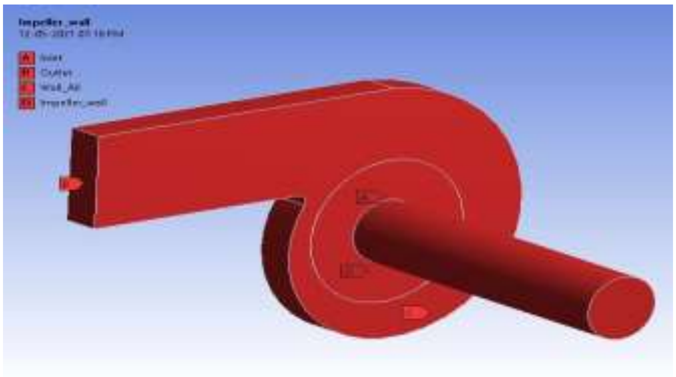

Fig. 1: Geometry

Creation of Mesh: Due to the presence of curved geometry and a complex impeller region, tetrahedral meshing was selected. Finer mesh was given in the impeller zone. Maximum element size in the impeller zone was 5 mm and 10 mm for the rest of the geometry.

.

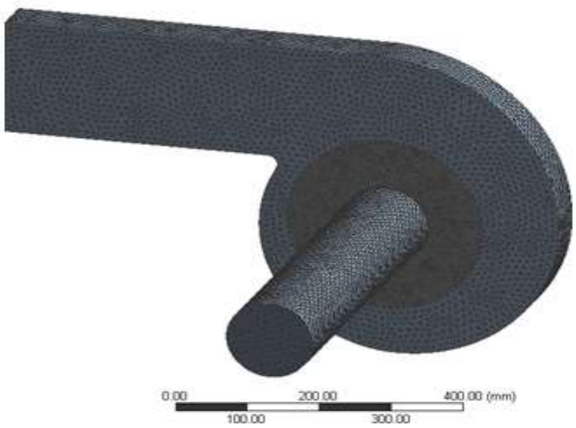

Fig. 2: Meshing

Set-Up: The moving reference frame method was chosen to solve this problem. Correct material properties of air and operating conditions were fed to the solver. 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 CFD problem was solved in Ansys fluent solver using the k-ε turbulence model. It was solved at different impeller speeds of 2800, 2500, 2100 and 1400 rpm. 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 proved to be very useful tool for visualization of the results. Pressure contours, velocity vectors and streamlines were plotted. These plots were checked for expected trends to be observed in the centrifugal blower. Torque of the impeller was obtained using the in-built calculator. The solver gave values of pressure and velocity over the surfaces of the inlet and outlet.

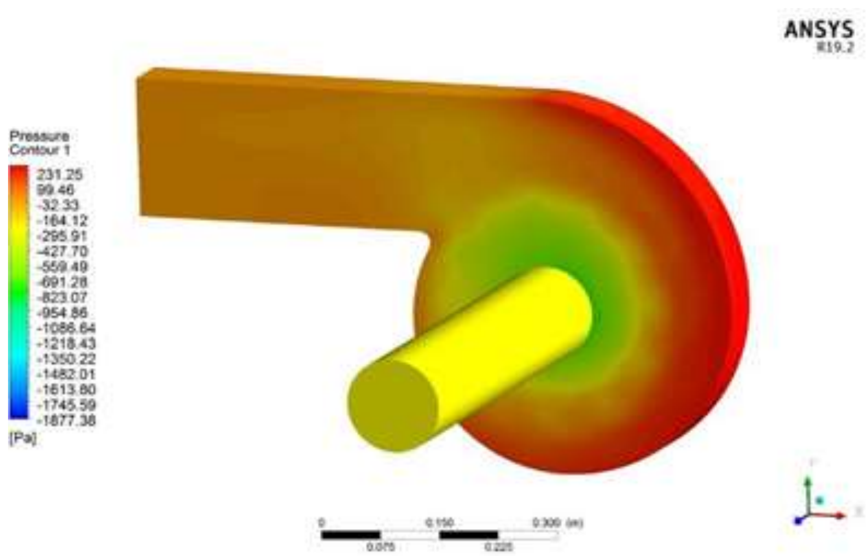

Fig.3. Pressure Contours for Backward Inclined Impeller at 2800 rpm

Pressure contours are shown in fig. 3, which shows that the centrifugal blower with a backward-inclined impeller created a higher pressure. The bright red zone to the right shows a higher suction pressure being created inside the blower, which matches with the expected trends.

.

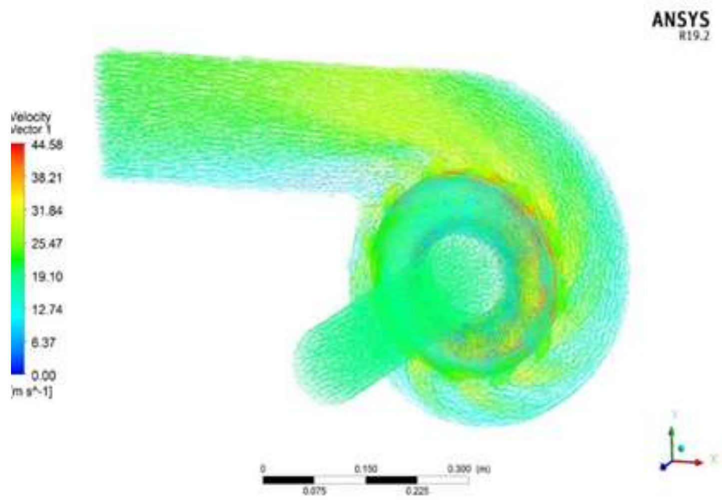

Fig.4 Velocity Vectors for Backward Inclined Impeller at 2800 rpm

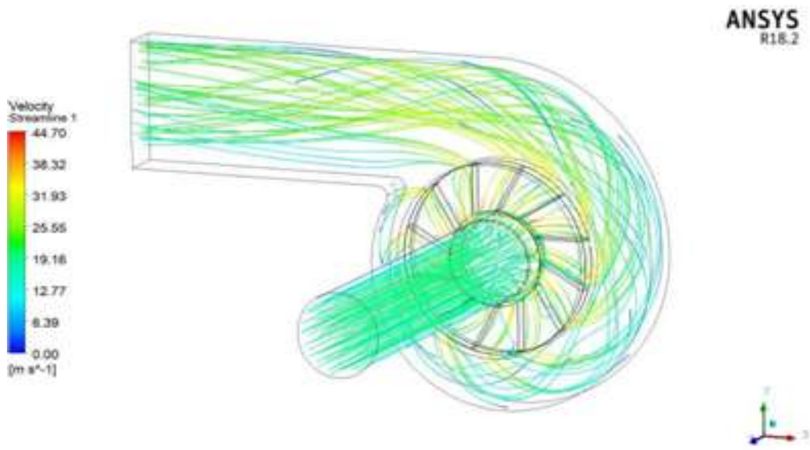

Fig.5 Streamlines for Backward Inclined Impeller at 2800 rpm

The concentration of higher velocity vectors of the backward-inclined impeller is giving much better air delivery, which is observed in fig. 4.
Streamlines show that the flow of air from the impeller zone to the outlet duct of the backward-inclined impeller in fig 5.
The pressure contours and velocity vectors for speeds 2500, 2100, and 1400 rpm clearly show that the total pressure, outlet velocity, and thus power required and efficiency decrease with decreasing rpm.

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	480.85	19.90	0.012025	0.24	1.19	189.23	115.07	60.81
2	2500	374.33	17.64	0.012025	0.21	0.95	134.81	79.40	58.90
3	2100	260.71	14.69	0.012025	0.18	0.67	80.44	46.06	57.26
4	1400	116.16	9.76	0.012025	0.12	0.31	24.83	13.63	54.89

4. Conclusion

The paper numerically analyse the effect of changing the speed of the impeller on the performance of a centrifugal blower. The results from CFD show that the blower's total pressure and efficiency are higher for a backward-inclined impeller at higher speeds. Also, shaft power and air power required for a backward-inclined-blade impeller centrifugal blower are more at higher speeds.

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. V.S.Thangarasu, G.Sureshkannan,N.V.Dhandapani, Design and Experimental Investigation of Forward Curved, Backward Curved and Radial Blade Impellers of Centrifugal Blower. *Australian Journal of Basic and Applied Sciences*, 9(1) January 2015, Pages: 71-75.
5. Bureau of Energy Efficiency (BEE), Government of India ,*Energy Efficiency Guide Book*, chapter 5. 2004, pp. 93-112.
6. Beena D. Baloni, Yogesh Pathak, S.A. Channiwala, Centrifugal blower volute optimization based on Taguchi method, *Computers & Fluids*,Volume 112, 2 May 2015, Pages 72-78.
7. 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.