

# Analyzing Effect of Impeller Width on Mass Flow Rate of Centrifugal Blower using CFD

Sushant Wagh, Dhananjay Panchagade

**Abstract**— to predict the mass flow rate from the centrifugal blower, Computational Fluid Dynamics (CFD) analysis is one of the advanced tools used in industry. A detailed CFD analysis was done to predict the mass flow rate at outlet of blower for different widths of impeller. The CFD analysis was done using Fluent software. The CAD models of the blower were modeled using CAD modeling software ProE WF2. Three different CAD models were made with impeller width of 17, 20 and 21.5 mm at the tip. The CFD results were validated experimentally for 17 mm width impeller by measuring the outlet velocity with the help of an anemometer and then calculating mass flow rate. It was observed that CFD results were having very good correlation with measured values.

**Keywords**—CFD, Blower, Mass flow Rate, Impeller Width

## I. INTRODUCTION

Fans and blowers are turbo machines which deliver air at a desired high velocity (and accordingly at a high mass flow rate) but at a relatively low static pressure. The pressure rise across a fan is extremely low and is of the order of a few millimeters of water gauge. The upper limit of pressure rise is of the order of 250mm of water gauge. The rise in static pressure across a blower is relatively higher and is more than 1000 mm of water gauge that is required to overcome the pressure losses of the gas during its flow through various passages. A blower may be constructed in multi stages for still higher discharge pressure. A large number of fans and blowers for relatively high pressure applications are of centrifugal type. A blower consists of an impeller which has blades fixed between the inner and outer diameters. The impeller can be mounted either directly on the shaft extension of the prime mover or separately on a shaft supported between two additional bearings. Air or gas enters the impeller axially through the inlet nozzle which provides slight acceleration to the air before its entry to the impeller. The action of the impeller swings the gas from a smaller to a larger radius and delivers the gas at a high pressure and velocity to the casing. The flow from the impeller blades is collected by a spiral-shaped casing known as volute casing or spiral casing. The casing can further increase the static pressure of the air and it finally delivers the air to the exit of the blower. A centrifugal fan impeller may have backward swept blades, radial tipped blades or forward swept blades. The backward swept blades are employed for lower pressure and lower flow rates. The radial tipped blades are employed for handling dust-laden air or gas because they are less prone to blockage, Dust erosion and failure.

**Manuscript Received on June 2014.**

**Sushant Wagh**, Mechanical Department, D Y Patil College of Engineering, Pune, India.

**Prof. Dhananjay Panchagade**, Mechanical Department, D Y Patil College of Engineering, Pune, India.

## II. CENTRIFUGAL BLOWER DETAILS

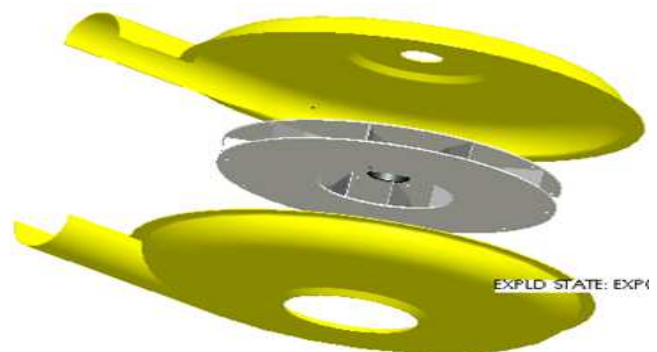
- Blower/ Pump: Centrifugal blower
- Impeller : Forward curved
- Blower inlet diameter: 95 mm
- Blower outlet diameter: 60 mm
- Blade thickness: 1 mm
- Impeller diameter (D1): 95 mm
- Impeller diameter (D2): 240 mm
- Number of blades: 10
- Casing diameter: 308 mm

## III. METHODOLOGY

Computational fluid dynamics is part of fluid mechanics that uses numerical method and algorithm to solve and analyze problem that related to fluid flow.

**A-Geometry Creation using CAD software (Pro-E WF 2.0)**

**Pro/ENGINEER** is a parametric, integrated 3D CAD/CAM/CAE solution created by Parametric Technology Corporation (PTC). The application runs on Microsoft Windows platform, and provides solid modeling, assembly modeling and drafting, finite element analysis, direct and parametric modeling, sub-divisional and nurbs surfacing, and NC and tooling functionality for mechanical engineers. CAD models were made with three different configurations of impeller viz, 17 mm, 20 and 21.5 mm width.



**Fig. 1: Exploded View Blower in Pro-E**

## B- Meshing

Meshing is done in Ansys workbench itself. For volume meshing, a tetrahedral mesh generally provides a more automatic solution with the ability to add mesh controls to improve the accuracy in critical regions. Conversely, a hexahedral mesh generally provides a more accurate solution but is more difficult to generate. Path independent method was used for meshing which uses top down approach (creates volume mesh and extracts surface mesh from boundaries). The patch-independent method uses the geometry only to

associate the boundary faces of the mesh to the regions of interest thereby ignoring gaps, overlaps and other issues that present other meshing tools with countless problems. The advanced size function is the default for fluids applications and is designed to accurately capture the geometry while maintaining a smooth growth rate between the regions of curvature and/or proximity. Maximum skewness observed was 0.6.

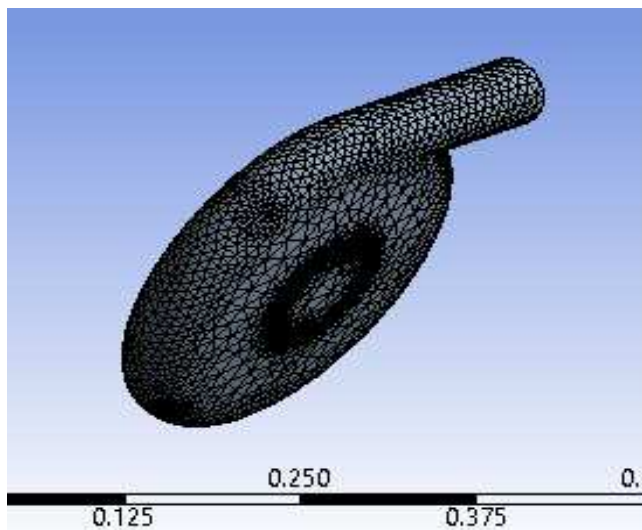


Fig. 2: Meshed Blower Geometry

**C- CFD Model**

In the CFD model, the Standard k- $\epsilon$  turbulent model is selected which is one of the most common turbulence models. It is a two equation model that means it includes two extra transport equations to represent the turbulent properties of the flow. This two equation model accounts for history effects like convection and diffusion of turbulent energy.

**D- CFD Result**

Mass flow rate at the outlet of blower was 0.118 kg/sec. Below fig. shows the air velocity plot

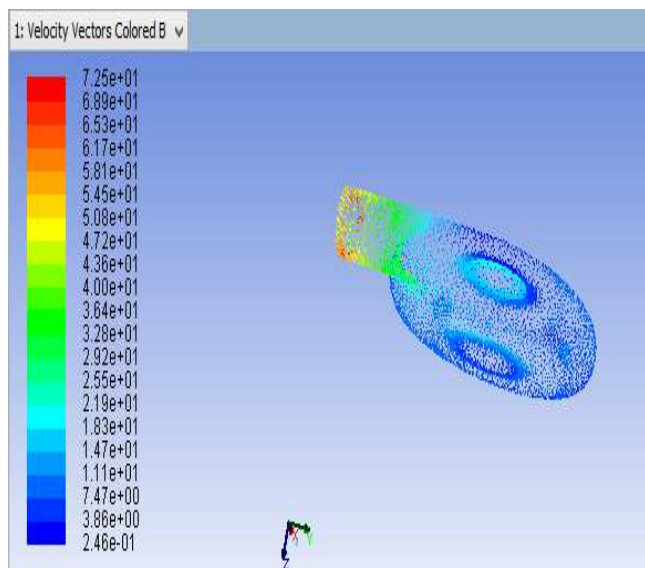


Fig. 3: Velocity Plot at 2500 RPM

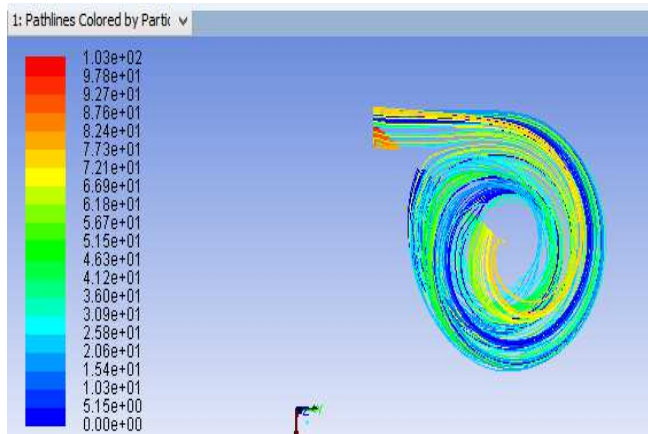


Fig. 4: Path Lines of Fan

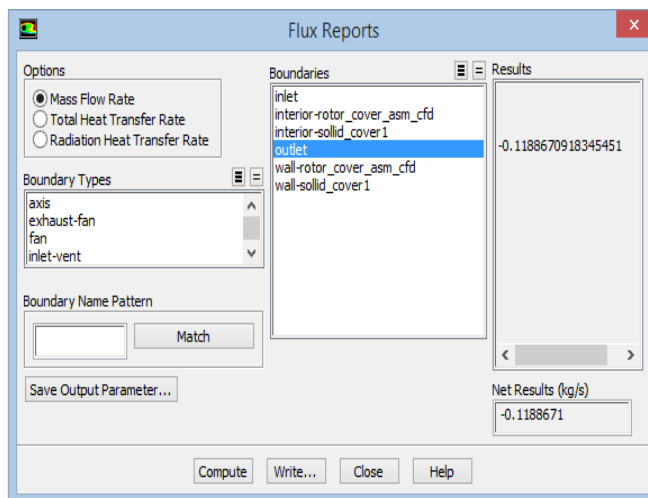


Fig. 5: Mass Flow Rate at Outlet at 2500 RPM

Following mass flow rates were observed with different impeller configurations in CFD.

Impeller Width, mm	17	20	21.5
Mass flow rate, kg/sec	0.118	0.134	0.126

Table1: Effect of Impeller Width on Mass Flow Rate

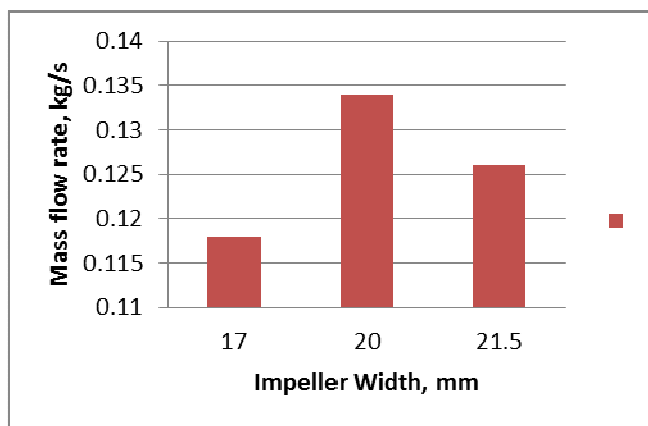


Fig. 6: Effect of impeller width on mass flow rate

**IV. EXPERIMENTAL METHODOLOGY**

An anemometer was used to record the outlet velocity from the blower whereas pulse engine tachometer was used to set the engine speed at 2500 RPM. The velocity observed at outlet of blower was 28.54 m/sec.



**Fig. 7: Digital Anemometer**

Mass flow rate = Outlet area \* Outlet velocity \* Air density  
Experimental mass flow rate = 0.0992 kg/sec

### V. CONCLUSION

Based on the CFD results we can conclude that width of the impeller plays important role in air discharge of centrifugal blower. It is observed that when impeller width is increased from 17 mm to 20 mm mass flow rate has increased from 0.118 to 0.134 kg/sec. Also when impeller width is further increased by 1.5 mm mass flow rate decreases to 0.126 kg/sec. There is good correlation between CFD derived mass flow rate and measured mass flow rate of blower. Thus CFD analysis is an effective tool to calculate quickly and inexpensively the effect of design and operating parameter of blower. By properly designing blower impeller the outlet mass flow rate can be improvised.

### REFERENCES

- [1] O. P. Singh & R. Khilwani, "Parametric Study of Centrifugal Fan Performance: Experiments and Numerical Simulation", International Journal of Advances in Engineering & Technology, Vol.1, Issue # 2, pp. 33-50, May 2011
- [2] V. Ahuja, A. Hosangadi & M. Slipper "Impeller Design of a Centrifugal Fan with Blade Optimization", International Journal of Rotating Machinery, Vol. 2011, Article ID 537826, May 2011
- [3] H.Dumitrescu & V. Cardos, "Flow Prediction in a Blower Casing,"University of Targoviste, 2007
- [4] C.Jang & K. Choi, "Optimal Design of Splitters Attached to Turbo Blower Impeller by RSM", Journal of Thermal Science, Vol.21, pp 215-22, 2012
- [5] Jin-Hyuk Kim1, Kyung-Hun Cha, Kwang-Yong Kim & Choon-Man Jang, "Numerical Investigation on Aerodynamic Performance of a Centrifugal Fan with Splitter Blades", International Journal of Fluid Machinery and Systems, Vol. 5, Issue #4, Dec 2012
- [6] A. Manivannan, "Computational fluid dynamics analysis of a mixed flow pump impeller", International Journal of Engineering, Science and Technology, Vol.2, Issue# 6, pp. 200-206, 2010
- [7] K. Vasudeva Karanth & N. Yagnesh Sharma, "CFD Analysis on the Effect of Radial Gap on Impeller-Diffuser Flow Interaction as well as on the Flow Characteristics of a Centrifugal Fan", international Journal of Rotating Machinery, Vol. 2009, 2009