Performance Analysis of Backward Curved Centrifugal Fan in Heating Ventilation and Air Conditioning

Sharad Chaudhary[1], Santosh Kansal[2]

[1] Department of Mechanical Engineering, Institute of Engineering & Technology, Devi Ahilya Vishwavidyalaya, Indore (M.P), India

[2]Department of Mechanical Engineering, Institute of Engineering & Technology, Devi Ahilya Vishwavidyalaya, Indore (M.P), India

Abstract: ***In this paper, a performance analysis of the backward curved centrifugal fan has been carried out. Centrifugal fan is mostly used in HVAC, and in this a theoretical analysis is done using CFD. As a first step, modeling based on specifications is done. Then, CFD TOOL Autodesk CFD 2013 is used for performance analysis of the fan. During analysis blade angles is varied, and its effect on static pressure and total pressure of the fan is investigated. Also, theoretical analysis on the same lines has been developed. In this study, theoretical results obtained are compared with that obtained using CFD tool by drawing a performance curve for different blade angles. This is helpful is selecting the appropriate fan. The purpose of this paper is to give a better understanding of the performance behavior under different aspects to improve the design of the fan.***

Keywords: Centrifugal fan; Simulation Model; CFD; Performance curve; Blade Angle

1. Introduction

A fan is used for pumping or circulating the air through the duct system and the cooled space. The centrifugal fan is the fluid machinery used in duct air conditioning system. Centrifugal fans are utilized in all kind of engineering application such as air supply in process, and consumer electronics. The centrifugal fans have simple impeller construction with backward curved or forward blades.

The backward curved blades must be operated at a much higher speed of rotation than the forward curved blades. The centrifugal force created by the rotating impeller moves the air outward along the blade channels. The outward moving air streams are combined by the scroll into a single large air stream. This air stream leaves the fan through the discharge outlet. In order to increase the flow rate and static pressure of the centrifugal impeller it is necessary to change the parameters like shape of a fan, by changing a shape of blade, pitching angle, tip clearance, blade chord angle, number of blades. In this study a fluid dynamic tool CFD is used and mathematical calculation to satisfy the design parameters such as flow rate, static pressure and efficiency.

Fan capacity or the fan volume flow rate, m^3/s, is defined as the rate of volume flow measured at the inlet of the fan, corresponding to a specific fan total pressure. It is usually determined by the product of the duct velocity and the area of the duct connected to the fan inlet, according to the test standard Air Movement and Control Association (AMCA) Standard 210-85 and ASHRAE Standard 51-1985

Fan total pressure, expressed in mm of height of water column (or Pa), is the total pressure rise of a fan, i.e., the pressure difference between the total pressure at the fan outlet and the total pressure at the fan inlet

Fan characteristics can be described by certain interrelated parameters such as volume flow rate, pressure, power, and efficiency.

Nomenclature

bhp = brake horsepower, W
g = acceleration of gravity, m/s^2
H = net head, m
m= mass flow rate, kg/s
Q = volume flow rate, m^3/minute
r = radius, m
T = torque, N·m
Vb = tangential speed, m/s
V = velocity, m/s
W = work, J
Greek Symbols
β = blade angle, °
η = efficiency, %
b= impeller wheel width, m
ρ = density, kg/m^3
ω = rotational speed, rad/s

2. Theoretical Model

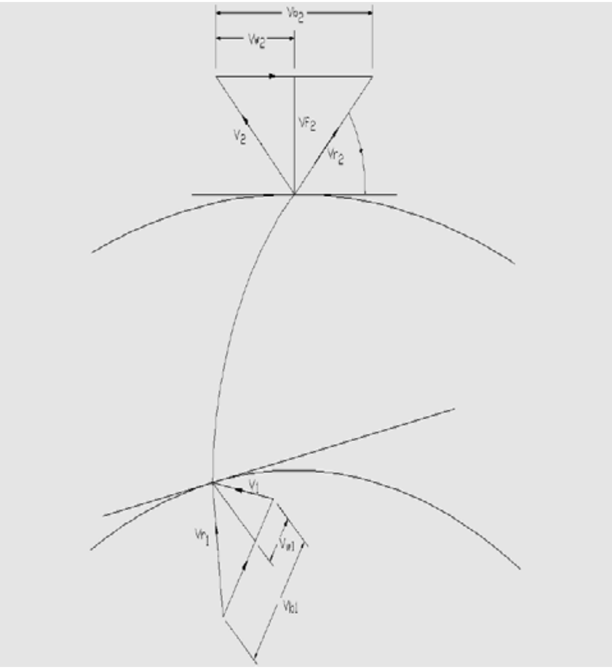


Figure 1: Velocity Triangles for Moving Blades of a Centrifugal Fan

Velocity Triangles for Moving Blades of a Centrifugal Fan: The velocity triangles at the inlet and outlet tips of the blade

V_{b1}	Linear or tangential velocity of the moving at inlet
V_1	Absolute velocity of air entering the blade
V_{f1}	Velocity of flow at inlet
V_{r1}	Relative velocity of air to the moving blade at inlet
V_{w1}	Velocity of whirl at inlet
β_1	Blade angle at inlet
V_{b2}, V_2, V_{f2}, V_{r2}, V_{w2}, β_2	Corresponding values at outlet of the blade tip

Angular momentum entering the impeller per second,
$= m\ V_{w1}\ R_1$
Angular momentum entering the impeller per second,
$= m\ V_{w2}\ R_2$
Torque, $T = m\ (V_{w2}\ R_2 - V_{w1}\ R_1)$
Work done per second = Torque (T)x Angular velocity
$= m\ (V_{w2}\ R_2 - V_{w1}\ R_1)\ w$
Since $\omega = V_{b1} / R_1$
$\omega = V_{b2} / R_2$
Work done per second, $= m\ (V_{w2}\ V_{b2} - V_{w1}\ V_{b1})$…..(*i*)
Total head developed by fan,
If the fluid enters the impeller radially then $V_{w1} = 0$
Total head developed by fan
$H = V_{w2}\ V_{b2} / g$ ….. (*ii*)
From outlet velocity triangle
$V_{b2} - V_{w2} = V_{f2} \cot \beta_2$ …..(*iii*)

3. Model construction

A model of fan was created in pro-e- software for this study as shown in Fig 2. A number of such models were created by changing the blade angle and keeping the rest of the specifications as same and exported the model in Autodesk CFD 2013 software. Autodesk CFD 2013 software is a CFD tool

Specifications of the fan-

Type – Backward curved centrifugal fan
Outlet diameter of fan – 560 mm
Inlet diameter of fan - 380 mm
Width of the blade – 180 mm
No. of blade – 14
Volume flow rate - 350 m^3/ minute
Blade angle at the outlet - 40° to
Blade angle at the outlet - 60

Procedure -: In this study CFD Autodesk CFD 2013 is used to simulate backward curved centrifugal fans. Different types of equations like Conservation of Mass and Naviour stoke equation related to fluid mechanics involved have been solved with appropriate boundary conditions. The finite volume method and grid generation in the CFD environment provided by the software was used. Following is the brief outline of the method adopted

- Create working fluid of air and select material of the impeller
- Assign a rotating region material to the volume surrounding the fan. The main construction of model is rotating region. Rotating region are envelope the impeller of fan. This is not torching the surface of fan.
- Now create the Boundary Conditions at the inlet and outlet. At the inlet pressure – 0 Pascal gauge pressure and volume flow rate 350 m^3/ minute.
- Set the Known Rotational Speed.
- Define the mesh distribution using Automatic-Sizing was gives in this study.
- Solving the application to obtain the result.

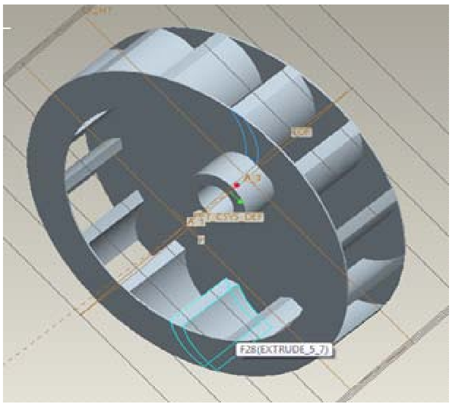


Figure 2: A CFD Model of the Centrifugal fan

4. Result and Discussion

The results of the analysis are shown in Fig. 3, Table 1 and Fig 4 the results obtained are also compared with the ideal performance curve using the theory of turbo machines.

4.1 Pressure Variations across the Centrifugal Fan

In, Fig 3.the results obtained by the CFD software after giving the boundary conditions as mentioned earlier in the procedure adopted. The values of the maximum static pressure is about 100 mm of water and minimum value is about 90mm of water

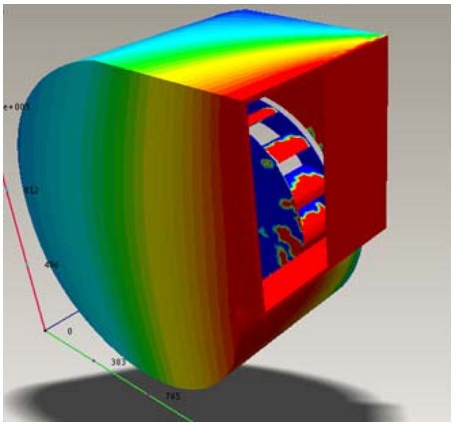


Figure 2: The pressure variations across the centrifugal fan

4.2 Effect of the Blade Angle on the Static Pressure

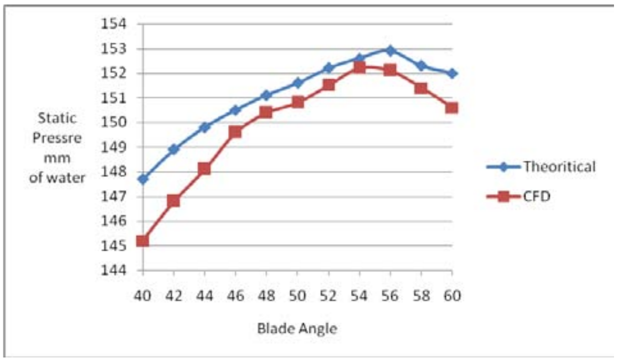
The effect of changing the blade angle on the static pressure obtained is tabulated in Table 1. It can be concluded that the initially increases and then it decreases. So for the given geometry of the fan the optimum angle comes out to be 54^o-56^o which is consistent with the theoretical results obtained

Blade angle	Static pressure mm of water (Theoretical)	Static pressure mm of water (CFD)
40°	95.7	93.2
42°	96.9	94.8
44°	97.8	96.1
46°	98.5	97.6
48°	99.1	98.4
50°	99.6	98.8
52°	100.2	99.5
54°	100.6	100.2
56°	100.9	100.9
58°	101.3	100.3
60°	101.0	99.7

Table 1 Effect of the Blade Angle on the Static Pressure

4.3 Comparison of the theoretical and CFD based performance curves

In figure 4 a comparison between the two approaches is shown. It is found that static pressure increases as outlet angle increases. When outlet blade angle is 40° then value of static pressure is obtained minimum and if the angle is varied till a 54° maximum value of static pressure is obtained after which any further increases the blade angles results in decreases static pressure.



.

5. Conclusions

In this paper, investigations on the effect of blade angle of centrifugal fan on performance have been presented through theoretical and CFD simulation. It is shown that for backward curved fan an increase blade angle results increase in static pressure and accompanied by increase in total head of fan. Further blade angle is increases than static pressure decreases. The maximum static pressure of the fan was observed to be at blade angle of 54°. The results obtained are in line with the ones predicted theoretically. This analysis helps in proper selection of fan in HVAC application.

References

[1] Singh O.P. 2011, “Parametric study of centrifugal fan performance”. International Journal of Advance Engineering & Technology 1,(2),pp 33-50.

[2] Bulaga Robert 2002 , “ Performance study of a Ducted fan system

[3] Stein Jeff 2003, “Development and Testing of the Characteristic Curve Fan Model. ASHRAE. 1996. 1996 ASHRAE Handbook—Air-Conditioning Systems and Equipment. Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

[4] AMCA/ASHRAE. 2007. ANSI/AMCA Standard 210-07/ANSI/ASHRAE Standard 51-07, Laboratory Methods of Testing Fans for Certified Aerodynamic Performance Rating. Arlington Heights, IL: Air Movement and Control Association and Atlanta: American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.

[5] Dilin, P., T. Sakai, M. Wilson, and A. Whitfield. 1998. A Computational and Experimental Evaluation ofthe Performance of a Centrifugal Fan Volute. Proceedings of the Institution of Mechanical Engineers, Part A —Power & Energy 212(4):235–46.