IJNRD.ORG ISSN : 2456-4184
INTERNATIONAL JOURNAL OF NOVEL RESEARCH AND DEVELOPMENT (IJNRD) | IJNRD.ORG
An International Open Access, Peer-reviewed, Refereed Journal

DESIGN EVALUATION OF CENTRIFUGAL FANPERFORMANCE AND THE EFFECT OF INLET DIFFUSER IN FAN EFFICIENCY AND POWER

V. Rajangam*[1], Dr. A. Premkumar[2], N. Chandran[3], D. Rajesh[4]

[*1,2,3,4]Assistant Professor, Department of Mechanical Engineering, Panimalar Engineering College, Chennai,Tamilnadu, India
rajangam.accet@gmail.com[*1],
prem.mitan@gmail.com[2],
chandrannagendran008@gmail.com[3],
rajesh2energy@gmail.com[4]

Abstract

This project outlines the areas that need to be considered during the design and development of centrifugal fans, and in particular the effect of changes made in inlet diffuser. During this project a new design of diffuser has been developed for the existing fans which offerthe following improvements over the existing: Increased efficiency and associated power, savings, Stress analysis and bending moment. This Project describes various fan types available and their uses, Fan design concepts and their effect of diffuser on performance, Evaluation of specific speed and performance coefficients: Flow Coefficient (Kq), Pressure Coefficient (Kp), Power Coefficient (Kpw)

Keywords: Centrifugal fan, Diffuser, Fan efficiency

1. Introduction

Fans are one of the types of turbo machinery which are used to move air continuously with slight increase in static pressure. Fans are widely used in industrial and commercial applications from shop ventilation to material handling, boiler applications, transporting gas or materials and most use in the HVAC industry today. The performance of the centrifugal fan is analyzed by its performance curves. The flow between the blades is always complicated for understanding. The losses created like entry losses at impeller, impeller losses, leakage loss and volute losses alwaysoccur in the centrifugal fan. Hence, by reducing the losses of centrifugal fan, performance of centrifugal fan can be improved.The impeller is the main moving part of a centrifugal fan, and the structural parameters of the impeller include the blade shape, blade profile, outlet width, number of blades, inlet and outlet diameter, etc. An excellent impeller design is helpful to improve the aerodynamic performance of the fan.

Different field industries use centrifugal fans for various reasons as mentioned above. The centrifugal fan that is to be analyzed for power efficiency is from the application of boiler and it is of bagasse fired boiler type. The fan utilized in this is to be made a prototype and the performance incurred with the power is to be noted and a suitable suggestion of power efficiency adjustments is to be provided on the basis of the data. One main modification done to the centrifugal fan is the change of its inlet duct. Inlet duct plays some vital role in power efficiency. It manages how the air/gas flow enters inside the centrifugal fan. Centrifugal fan in the factory site consisted of straight profiled which produced a certain efficiency relating to power output. This project focuses on the change of the inlet damper to a smooth profiled cone and comparing the power efficiency with the latter straight profiled inlet damper on the impeller blades.

Stefano Castegnaro [8] on the paper titled Aerodynamic Design of Low-Speed Axial- Flow Fans: A Historical Overview presents a historical overview of the developments of aerodynamic design methods for low-speed axial-flow fans. This historical overview starts from the first fan applications, dating back to the 16th century, and arrives to the modern times of computer-based design techniques, passing through the pioneering times of aerodynamic theories and the times of designing before computers. The overview shows that the major achievements in the axial fan design discipline have actually been related to other technological fields, such as marine and aeronautical propulsion, as well as to the development of wind tunnels. Yu- Taj and et.al. [10] have presented a method for redesigning a centrifugal impeller and its inlet duct. The double-discharge volute casing is a structural constraint and is maintained for its shape. The redesign effort was geared towards meeting the design volute exit pressure while reducing the power required to operate the fan. Given the high performance of the baseline impeller, the redesign adopted a high-fidelity CFD-based computational approach capable of accounting for all aerodynamic losses. The present effort utilized a numerical optimization with experiential steering techniques to redesign the fan blades, inlet duct, and shroud of the impeller. The resulting flow path modifications not only met the pressure requirement, but also reduced the fanpower by 8.8% over the baseline. A refined CFD assessment of the impeller/volute coupling and the gap between the stationary duct and the rotating shroud revealed a reduction in efficiency due to the volute and the gap. The calculations verified that the new impeller matches better with theoriginal volute. Model-fan measured data was used to validate CFD predictions and impeller design goals. The CFD results further demonstrate a Reynolds-number effect between the model- and full-scale fans.

Hongchang Ding, Tao Chang, Fanyun Lin [2] takes centrifugal fan as the research objectand establishes five impeller models with different blade outlet angles. By means of computational fluid dynamics (CFD), the external characteristics of the centrifugal fan and the internal characteristics, including the velocity, pressure, and turbulent energy distribution, at the middle span plane of the impeller or fan were obtained and compared. In addition, the pressure fluctuations surrounding the impeller outlet were also analyzed. The results showed that the change of the blade outlet angle of the centrifugal fan had a great influence on the performance; the total pressure and efficiency of the fan were the highest when the outlet angle of the blade was increased to 29.5 under the design flow rate; and the influence of the outlet angle on the fan performance was different in design conditions. On the other hand, at different flow rates, the change of the internal flow field with the increase of the outlet angle was different For the pressure fluctuation of the fan, by increasing the blade outlet angle properly under high flow conditions, the fluctuation amplitude of the fan at the blade frequency and its frequency multiplication could be reduced, which is conducive to decreasing the impeller noise.Keyur K Patel [4] studied the performance of the centrifugal fan by analyzing its performance curves. The flow between the blades is always complicated for understanding. The losses created like entry losses at impeller, impeller losses, leakage loss and volute losses always occur in the centrifugal fan. Hence, by reducing the losses of centrifugal fan, performance of centrifugal fan has been improved.

Li et al. [5] studied the influence of blade shapes on the performance of high specific speed centrifugal fans, and found that the blockage phenomenon at the blade outlet of impeller with plate blade and the turbulent kinetic energy inside the volute were weakened under the condition of a large flow rate, so that the performance of the fan with plate blade was better than that of the airfoil blade. Wu et al. [9] compared the performance of a centrifugal fan with different blade profiles, and found that the centrifugal fan with a double-arc blade was higher in the efficiency and total pressure under the design condition, but the axial power consumed by equal deceleration blade was smaller. However, under the condition that other parameters of the fan remain unchanged, the internal flow of the fan with an equal deceleration blade was more uniform under the condition of low flow.Jian et al. [3] found that when the blade outlet width changed, various losses of the fan increased, and the efficiency decreased. With the decrease of the blade outlet width, the flow-pressure curve of the fan shifted to the lower left and the pressure decreased with the increase of the flow rate. This provides a reference for the design of the outlet width of the impeller and the reconstruction of the impeller.Liu et al. [6] found that the aerodynamic performance of the fan can be improved by increasing the number of blades and thediameter of the blade outlet.

The optimized fan with a 12-blade number and increased blade outlet diameter was better than the prototype fan in terms of the total pressure and efficiency. HEO et al. [14] analyzed the aerodynamic characteristics of a centrifugal fan with additional splitter blades in the impeller by using three-dimensional Reynolds-averaged Navier–Stokes (RANS). The global Pareto optimal frontier for centrifugal fan design was obtained by using a hybrid multi-objective evolutionary algorithm and response surface approximation model.

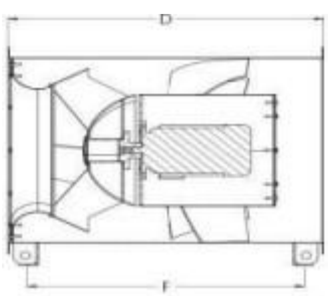


Fig. 1 Mixed Flow Fan

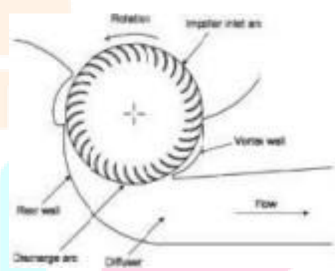


Fig.2 Cross flow fan

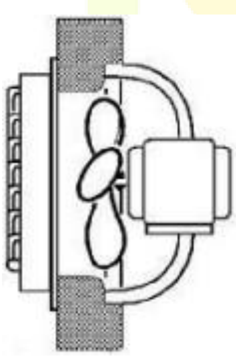


Fig.3 Propeller Fan

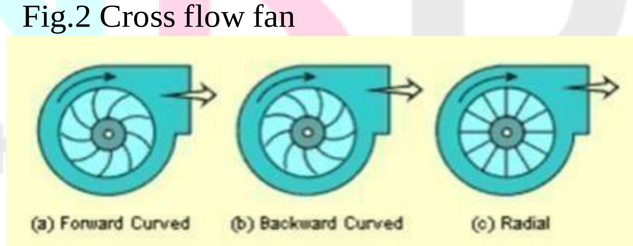


Fig.4 Types of Implellers

2. Materials and Methods

The centrifugal fan uses the centrifugal power supplied from the rotation of impellers to increase the kinetic energy of air/gases. When the impellers rotate, the particles near the impellers are thrown off from the impellers, then move into the fan casing. As a result, the kinetic energy of air/gas is measured as pressure because of the system resistance offered by the casing and duct. The air/gas is then guided to the exit via outlet ducts. After the air/gas is thrown-off, the gas pressure in the middle region of the impellers decreases. The gas from the impeller eye rushes in to normalize this. This cycle repeats and therefore the air/gas can be continuously transferred.

A diagram called a velocity triangle helps us in determining the flow geometry at the entry and exit of a blade.

A minimum number of data are required to draw a velocity triangle at a point on blade. Some component of velocity varies at different point on the blade due to changes in the direction of flow.

Hence an infinite number of velocity triangles are possible for a given blade.To describe the flow using only two velocity triangles, we define mean values of velocity and their direction.

Velocity triangle of any turbo machine has three components

U Blade velocity

Vr Relative Velocity

V Absolute velocity

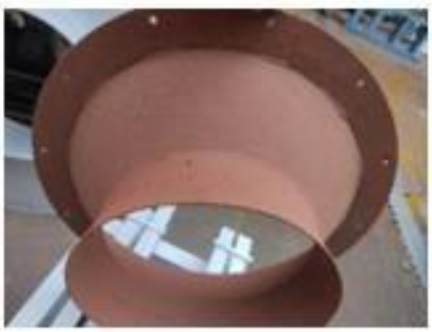
These velocities are related by the triangle law of vector addition

$V = U + Vr$

This relatively simple equation is used frequently while drawing the velocity diagram. The velocity diagram for the forward, backward face blades shown are drawn using this law. The angle α is the angle made by the absolute velocity with the axial direction and angle β is the angle made by blade with respect to axial direction.

Prototype of Wind Tunnel

Prototype of Impeller Casing



Prototype of Smooth Profiled Inlet Cone

Prototype of Smooth Profiled Inlet Cone

3. Results and discussion

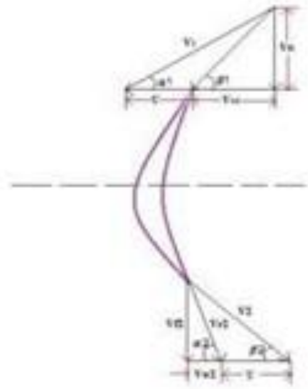
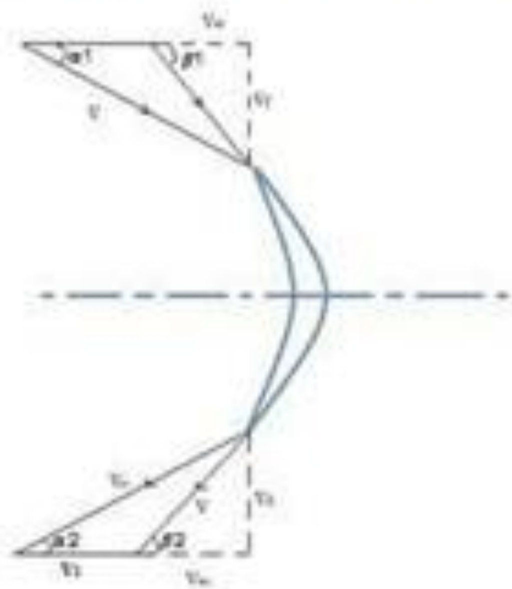
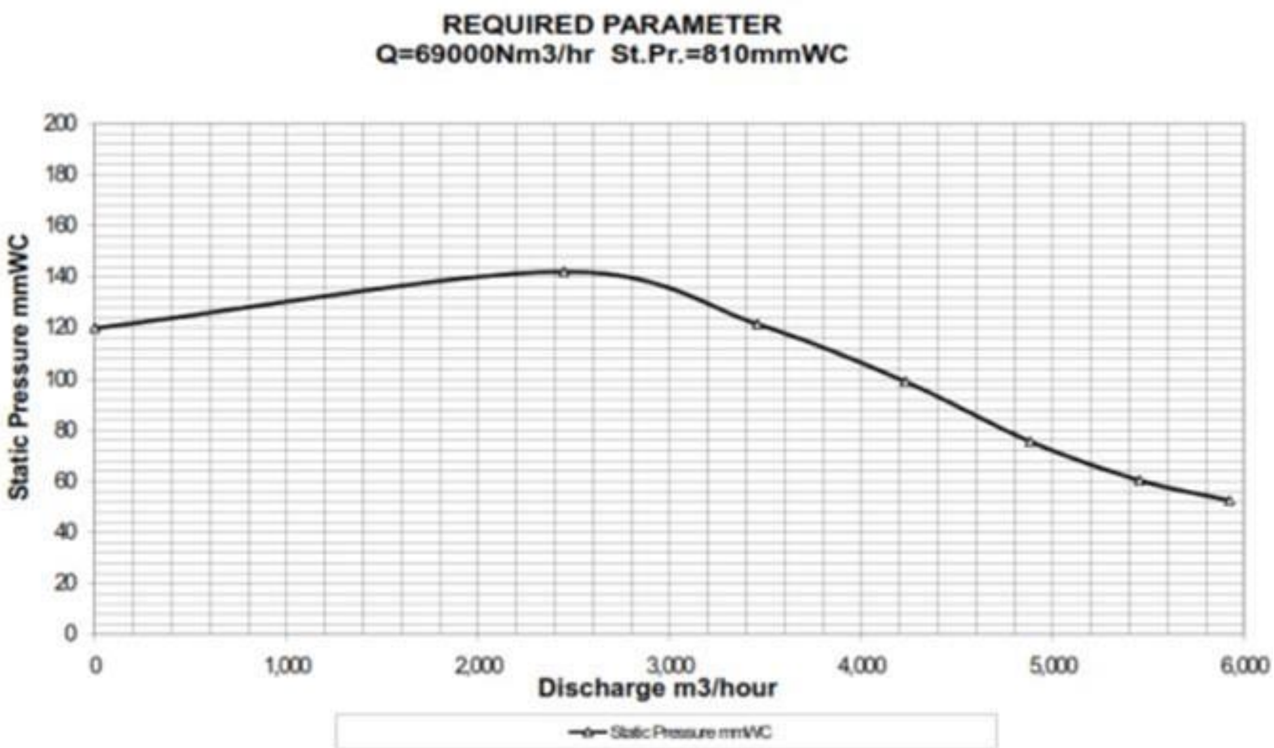


Figure 3.9 Velocity Triangle For Forward Curved Impeller

Figure 3.10 Velocity Triangle For Backward Curved Impeller





Performance Curve

The performance of a fan in terms of pressure, volume flow and power absorbed depends on a number of factors, the most important being;

- The design and type of fan
- The point of operation on the volume flow / pressure characteristic.
- The size of fan
- The speed of rotation of the impeller
- The condition of the air or gas passing through the fan

It is customary for a manufacturer to make a range of fans of varying sizes to a single design, thus producing a series of geometrically similar fans. It is convenient to be able to compute the performance of each fan from the minimum test data. The pressure/volume flow relationship is not generally capable of being expressed as a simple mathematical function. However, by considering any single point of operation on the characteristic curve, it is possible to derive somesimple relationships known as fan laws.

Q = kq.da.ub.δc .µd (1)

P = kp.deuf.δg.µh (2)

Where ;

d = Impeller diameter

u = Impeller peripheral

speedδ = Air density

μ = Coefficient of viscosity kq= Flow coefficient

kp= Pressure coefficient Equating the dimensions of equation (1) ;

$I{-}1t{-}2 = (l)(lt{-}1)b(ml{-}3)c(ml{-}1t{-}1)d$

$=la{+}b{+}3c{-}dtb{-}dmc{+}d$

Equating indices;

Therefore;

m : 0 = c + d

t : -1 = - b - d

l : 3 = a + b - 3c – d

a = 2 - d b = 1 - d c = - d Equation (1) becomes ;

$Q = kq \times d2{-}du1{-}d \times \delta{-}d \times \mu d$

Q = kq.d2.u.$(du/\delta\mu){-}d$ (3)

Equating the dimensions of equation (2)

m$I3t{-}1 = (l)(lt{-}1)f(ml{-}3)g(ml{-}1t{-}1)h$

$=le{+}f{-}3g{-}ht{-}f{-}hmg{+}h$

Equating indices ;

Therefore ;

m : 1 = g + h t : -2 = - f - hl : 3 = e + f - 3g – h

e = - h f = 2 - h l = 1 - h

Equation (2) becomes ;

$P = kp \times d-h \times u2-h \times \delta 1-h\mu h$

$P = kp \times u2 \times \delta \times (du/\delta\mu)-h\ldots\ldots$ (4)

In equations (3) and (4), the term (duδ/μ) is seen to be of the same form as Reynolds Number. It may be taken as the Reynolds Number of the fan (Re) based on the impeller diameter and peripheral velocity and equations (3) and (4) becomes

; $= kq \times d2 \times u \times f1(Re)\ldots\ldots$(5)

$P = kp \times u2 \times \delta \times f2(Re) \ldots ..$ (6)

Where $f1$(Re) and $f2$(Re) are variable factors based on Reynolds Number. If the speed of rotation of the impeller is n, then $u = \pi \times d \times n$ and equation (5) and (6) can be written as ;

$Q = kq \times d3 \times n \times f1(Re) \ldots \ldots \ldots \ldots .$ (7)

$P = kp \times d2 \times n2 \times \delta \times f2(Re) \ldots \ldots$ (8)

In practice it is found that Reynolds number has little effect over quite wide ranges of impeller diameters, speed and air density. The above equations are therefore normally simplified as follows

$Q = kq \times d3 \times n$ ….. (9)

$P = kp \times d2 \times n2 \times \delta\ldots\ldots$ (10)

$Pw = P \times Q$

$Pw = kpw \times d5 \times n3 \times \delta$

Where kpw is the Power coefficient. The coefficients kq, kp, kpw will be constant for a range of geometrically similar fans, and for a particular point of operation on the pressure / volume characteristic.

4. Conclusion

The main aim of this project was to select a fully functioning centrifugal fan at site and to analyze the characteristics and to arrive with its power and its associated efficiency by using means of prototype and hence suggesting suitable alterations to the design and developing prototype for it also and likewise analyzing it. The result that was obtained from the suggested design change was compared with the original product. The obtained result was satisfactory and within the safe limits. For the same profile of impeller, casing and other components of centrifugal fan and by only changing the inlet cone to a smooth profile, the efficiency is improved from 72.3% to 74.3% i.e., approximately 2% efficiency improvement incentrifugal fan characteristics.

References

[1] Heo, M.W.; Kim, J.H.; Kim, K.Y. " Design optimization of a centrifugal fan with splitter blades", *International Journal of Turbo Jet Engines* 2015, 32, 143–154

[2] Hongchang Ding , Tao Chang and Fanyun Lin, " The Influence of the Blade Outlet Angle on the Flow Field and Pressure Pulsation in a Centrifugal Fan" *Processes*, Nov0220.

[3] Jian, X.S.; Chen., Q.; Sai, Q.Y.; Wu, H.Y.; Zhu, Z.N, " Effect of the blade outlet width on the performance of small high speed backward curved centrifugal fans", *Compress. Blower Fan Technol*. 2017, 59, 71–76.

[4] Keyur K Patel, "Performance Analysis And Optimization Of Centrifugal Fan" *International Journal of Emerging Trends in Engineering and Development* Issue 3, Vol.2 (May 2013) .

[5] Li, X.; Wang, W.; Li, J.L.; Luo, X.Q.; Huang, Y.J. "Analysis of the influence of different blade types on the performance of high specific speed centrifugal fan" *In Proceedings of the Symposium on the development strategy of hydraulic machinery discipline and the* 11*th National Annual Conference on hydraulic machinery and its systems*, Beijing, China, 19–21 October 2018.

[6] Liu, D.; Li, J.; Ling, X. "Performance improvement of MVR high pressure centrifugal fan with designed number and outlet diameter of blades" *Fluid machinery*. 2018, 046, 44–48.

[7] Shi, Y.J.; Ge, A.X.; Fu, Y.X.; Wang, X. "Retrofit analysis of the impeller and guide vanes of a centrifugal fan based on CFD", *Compress. Blower Fan Technol*. 2017, 59, 27–31.

[8] Stefano Castegnaro , "Aerodynamic Design of Low-Speed Axial-Flow Fans: A Historical Overview" *Designs*, June 2018

[9] Wu, R.L.; Wu, P.J.; Qin, G.L. "Numerical research on the effect of blade profile on performance of centrifugal fan", *Compress. Blower Fan Technol*. 2014, 0, 31–36.

[10] Yu-Tai Lee, Vineet Ahuja, Ashvin Hosangadi, Michael E. Slipper, Lawrence P. Mulvihill, Roger Birkbeck, and Roderick M. Coleman, *International Journal of Rotating Machinery*, Vol 2011, Article ID 537824, doi:10.1155/2011/537824

International Research Journal
IJNRD
Research Through Innovation