INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume3, Issue2)
Available online at: www.ijariit.com

Numerical Analysis of Aerofoil Section of Blade of Axial Flow Fan at Different Angle of Attack

Anil Yadav
Mechanical Engineering Department, NRIIST, Bhopal M.P. (India)

Prof Meghna Pathak
Mechanical Engineering Department, NRIIST, Bhopal M.P. (India)

Dr. P. K. Sharma
Mechanical Engineering Department, NRIIST, Bhopal M.P. (India)

Abstract: Fans are used all over the world in a wide verity of industries and other purposes. Some of the important applications are in steam power station, ventilation system, cooling of electric motor and generator, and many industrial processes. Many researchers and engineers are making their efforts to design fans to fulfill the particular requirement of application in the most efficient way. The criterion of cost of fan, ease in manufacture and conservation of energy are other also to be considered in design. Several studies are available of various researchers in analysis and simulation of axial and centrifugal fans. Axial flow fans have also been designed and simulated by the researchers. Simulation of performance of axial flow fans and design of various blade sections of the axial flow fans have been studied experimentally or numerically. The present work comprises the numerical study of the axial flow fan section aerofoil. The objective of the study is to simulate the flow features around the aerofoil of particular design for three different values of the striking angle. The results are obtained using FLUENT in the form of velocity vectors at the leading edge, across the aerofoil and at the trailing edge. The contours of pressure and turbulence are also shown for the three cases.

Keyword: Axial Flow Fan, Aerofoil, Angle of attack, Lift and Drag, CFD.

1. INTRODUCTION

The basic purpose of a “fan” is to move a mass of gas or vapor at the desired velocity. For achieving this objective there is a slight increase air the gas pressure across the fan rotor or impeller. However, the main aim remains to move air or gas without any appreciable increase in its pressure. The total pressure developed by fan is of order of a few millimeter of water gauge.

Fans are used all over the world in a wide verity of industries. Some of the important applications are in steam power station, ventilation system, cooling of electric motor and generator, and many industrial processes.

The fan play vital role in creating cooling effect through heat exchanger. The efficiency of the machine depends mostly on cooling effect. More is cooling, more efficient machine may be, and therefore proper design selection of the fan in heat exchanger is very important. The heat exchangers consist of condenser and evaporator which are mostly used in air conditioning units, refrigerator, Boiler and condenser in thermal power plants. The heat exchanger used in automobile is radiator and oil cooler. The uses of heat exchanger are mostly in chemical and other industries.

In its simplest form, axial flow fan stages consist of rotor made of a number of blade fitted to the hub. When it is rotated by an electric motor or any other drive, a flow is established through the rotor causes an increase in stagnation pressure of air or gas across it.

The design procedure for an axial flow fan applicable in heat exchangers and other engineering systems has been presented. The design calculations are performed with presumption of flow through cascade of blade and it is the main governing factor of the design too. The profiled blade theory is used for designing. Calculations are tabulated for different parameter of radius and angles. On profiling, the increase of outlet angle has been shown which gives ultimate angle opening. The small corrections are therefore obtained. The important design parameters with particular values as obtained through calculation are listed. The data of important design parameters can be used for design optimization and energy conservation in application of axial flow fans.

1.1 FANS

Fans provide air for ventilation and industrial process requirements. Fans generate a pressure to move air (or gases) against a resistance caused by ducts, dampers, or other components in a fan system. The fan rotor receives energy from a rotating shaft and transmits it to the air.

There are two general classifications of fans: the centrifugal or radial flow fan (see ED-2400) and the propeller or axial flow fan. In the broader sense, the air passes through the impeller. The propeller or axial flow fan propels the air in an axial direction (Figure 3.1) with a swirling tangential motion created by the rotating impeller blades. In a centrifugal fan the air enters the impeller axially and is accelerated by the blades and discharged radially (Figure 3.2).

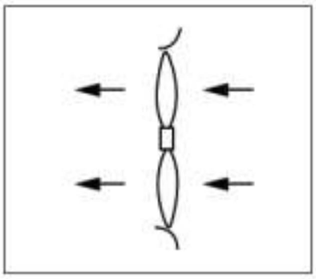


Fig. 1.1 Axial Flow

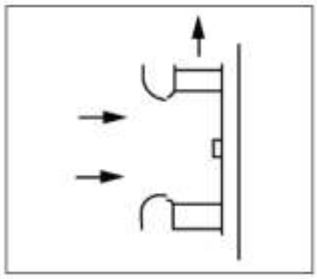


Fig. 1.2 Centrifugal Flow

The axial flow fan increases the air velocity through rotational or tangential force which produces velocity pressure (VP), kinetic energy, with a very small increase in static pressure (SP), potential energy. The centrifugal fan induces airflow by the centrifugal force generated in a rotating column of air producing potential energy (IP) and also by the rotational (tangential) velocity imparted to the air as it leaves the tip of the blades producing kinetic energy (VP).

1.2 Axial Flow Fan

The term “axial flow fan” like the “radial flow fan” originates from the main flow path through the rotor. The rotor is in the path of the axis of the rotation. Accordingly, the rotor of a hub, which is fitted with aero foil in such a way that all particle are given the increase in energy and the unavoidable losses are kept as low as possible.

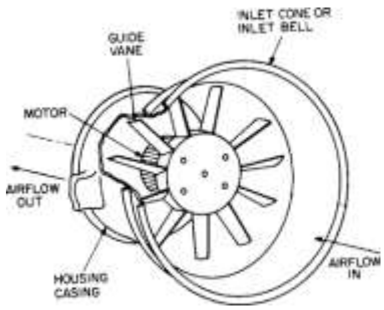


Fig. 1.3 Axial Flow Fan

In general application, the fan, according to fig. becomes the “armature of a duct”. By its introduction into a duct the axial flow an simplifies the design. This is because owing to the basically axial flow path, the part of the duct externally.

The following components are mainly present axial flow fans:

1) A piece of duct constricted into a nozzle and a duct expanded into a diffuser. In many cases, in the interest of efficiency and convenience, it is necessary for the diameter of the rotor to be less than that of the duct.
2) Rotor consists of a hub and aerofoil blade, the number of which generally varies from 4 to 8, the limits lie between 2 to 50 blades.
3) Upstream & downstream guide vanes.

As the flow through the fan is symmetrical to the axis, uniform flow condition will be on any random section of the cylinder. Therefore it is advisable to develop this cylinder on a plane. This is shown in fig guide vanes and rotor appear here as a cascade of blade of infinite length. Each section of the cylinder therefore will have a different appearance. If we look at a section AB close to the hub, cascade of blade are seen, the pitch of which is less than at the periphery, and their blade cross section according to length form and angle must look different from there since, of courses, the peripheral speed varies from radius to radius. It will presume that the flow through the cascade of blade will be the governing factor for the designing of fan this kind. In actual fact the knowledge of the so cascade flow is the basis for the whole circulation.

1.2.1 Type of Axial Flow Fan

Propeller Fans: Sometimes called as the panel fans, propeller fans are the lightest, least expensive and most commonly used fans. These fans normally consist of a flat frame or housing to be mounted in a wall or in partition to exhaust air from a building. This exhausted air has to be replaced by fresh air, coming in through other openings. If these openings are large enough, the suction pressure needed is small. The propeller fans, therefore, are designed to operate in the range near free delivery, to move large air volumes against low static pressures. These fans can be built both direct drive and belt drive (Figure 1.4 and Figure1.5). In direct drive arrangement, an electric motor is directly mounted to fan wheel, while a belt and pulley configuration is used to transfer the rotation from motor shaft to fan wheel in belt drive arrangement. Belt drive results inflexibility in performance, since any rotational speeds can be obtained for the fan wheel by selection of proper pulley ratio. In large sizes, belt drive is preferable since it will keep the speed of the fan wheel low or moderate while keeping the motor speed high, for lower cost because high-speed motors are less expensive than the low-speed motors of the same horsepower. The direct drive arrangements have lower number of components resulting in lower cost and require no maintenance and regular checkups for adjustment of the belt. Direct drives are more efficient than the belt drives since some of the power is consumed in the belt pulley arrangement.

Figure 1.4: Propeller fan with direct drive

Figure 1.5: Propeller fan with belt drive

Tube-axial Fans: A tube-axial fan is a glorified type of propeller fan with a cylindrical housing about one diameter long, containing a motor support, a motor and a fan wheel. The motor can be located either on upstream or downstream of the fan wheel. The fan wheel of a tube-axial fan can be similar to that of a propeller fan. It often has a medium sized hub diameter, about 30 to 50% of the blade outside diameter. The units are designed to operate in the ranges of moderate static pressures, higher than for a propeller fan. A tube-axial fan can be connected to an inlet duct or an outlet duct or both but the best application is exhausting from an inlet duct because any length of outlet duct results in larger pressure losses after the fan wheel due to presence of air spin. Figure 1.6 is a typical tube-axial fan.

Figure 1.6: Tube-axial Fans

Vaneaxial Fans: A vaneaxial fan is a more elaborate unit than the previous ones. It has the outside appearance of a cylindrical housing at least one diameter long. As in a tube-axial fan, this housing contains the motor support, the motor, and the fan wheel but the vaneaxial fan housing includes a set of guide vanes and sometimes an inner ring, a converging tailpiece, and an expanding diffuser for static regain. The guide vanes at the downstream of the fan wheel removes the rotational component of the air, slowing it down, and converting some of the excess velocity pressure into more useful static pressure . The hub diameter of a vaneaxial fan is larger than that of a tube-axial fan, usually between 50 to 80 % of the blade outside diameter. The vaneaxial fans are designed to operate in the range of fairly high static pressures. Figure 1.7 shows an example of vaneaxial fan.

Figure 1.7: Vaneaxial fan

Single -Stage Axial Fans: Single-stage (one rotor and one set of blades) axial fans are typically used in a forced draft situation on a balanced draft steam generator. When axial fans are designed for induced draft service, the higher pressure requirements normally.

Figure 1.8: Single-stage axial flow fan

Two-Stage Axial Fans: Two-stage axial fans are sometimes a good solution for applications where higher static pressure is required. There are two ways to design a two-stage axial flow fan; two fan wheels rotating in the same direction with guide vanes placed between the two stages and two counter-rotating fan wheels with no guide vanes at all. By either method, the static pressure is doubled. In the first method, the guide vanes pick up the helical flow produced by the rotating blades of the first stage and reverse the rotational component to the opposite direction, and prepare the flow to second stage. In other words, they behave as the outlet vanes for the first stage and inlet vanes for the second stage. This configuration has the advantage that the same fan wheel shaft can be used for both stages with adjustments on blade angles for the second stage. In the second method, two fan wheels run in opposite directions and are driven by two separate motors. The air spin produced by the first stage is more or less neutralized by the deflection produced by the second stage. As a result, no guide vanes are needed, which reduces the manufacturing cost and compensates for the possible extra expense of two motors instead of one. Another advantage of this configuration is that in case that one motor should fail, the unit can still deliver some air with only one stage running.

Figure 1.9: Two-stage axial flow fan

1.3 Performance of Axial Flow Fans

Figure 1.10 shows the shape of a typical pressure versus flow rate curve. Starting from the free delivery, the pressure value rises to a peak value. This is the good operating range for an axial flow fan. As the air volume decreases due to increasing restrictions, the axial air velocity decreases as well, resulting in an increased angle of attack and increased lift coefficients. The increase in the lift coefficient is responsible for the increase in the pressure. After the maximum lift angle is reached, the flow can no longer follow the upper contour of the blades, thus separate from the surface of the blade. Separated flow results in a decrease in lift coefficient, thus a decrease in pressure occurs. This phenomenon is called stall. After the stalling, the axial flow fan starts acting like an inefficient and noisy mixed flow fan. As the airflow approaches the fan inlet, the blades throw the air outward by centrifugal force and in this way produce the static pressures of the stalling range, which keeps until the point of no delivery is reached At this point, the blades act as a paddle wheel creating radial flow only. Figure 1.11 is a sketch of the flow behavior at different points on the pressure curve.

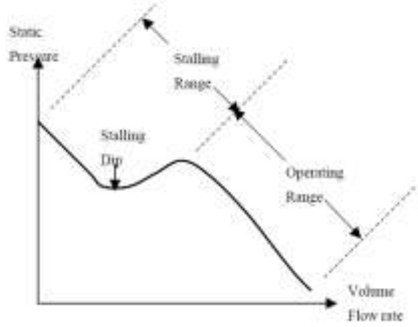


Figure 1.10: Static Pressure vs. Volume flow rate of an axial fan

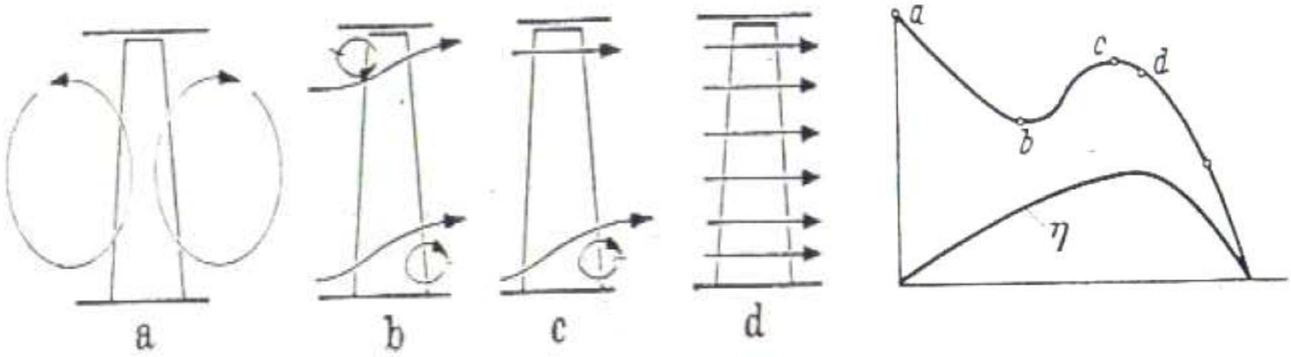


Fig. 1.11: Effect of different flow conditions on axial fan performance

In order to have a good operation with axial flow fans, one should provide a safety margin in order not to cause the fan to work in the inefficient and noisy stalling range. A good practice is that the peak pressure of the operating range should be 30to 50 % higher than the pressure required for the application. This pressure safety margin will allow for possible errors that may have been made in the determination of system resistance and to allow for possible fluctuations of the system.

2. METHODOLOGY

2.1 Method of analysis

In this work, the Computational Fluid Dynamics (CFD) analysis of the airfoil section of the blade of an axial flow fan has been done. The effect of angle of attack on the velocity distribution, pressure distribution and turbulence is numerically simulated. The NACA 747A415aerofoil is used in the study. CFD software ANSYS Fluent 6.3.26 is used to perform the numerical simulation of airflow around the selected aerofoil sections. This study is based on the finite volume method (FVM) in which the domain is discretised into a finite set of control volumes (or cells). Conservation equations for mass and momentum are solved in this study to determine the pressure distribution and therefore fluid dynamic forces acting on the wing as a function of time. Three values of angle of attack as 0°, 5° and 10° have been used for the simulation of an aerofoil section. The suitable airfoil section and the angle of attack have been chosen based on an extensive literature review for 2-D simulation for the blades of an axial flow fan. This aerofoil is used in aeroplanes, wind turbines, high velocity rotors, sail-planes and rotorcrafts etc. Figure 2.1 shows the section of the aerofoil

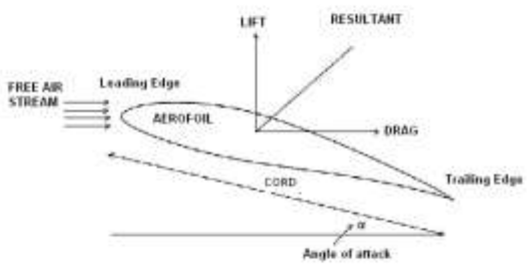


Figure 2.1: Section of the aerofoil

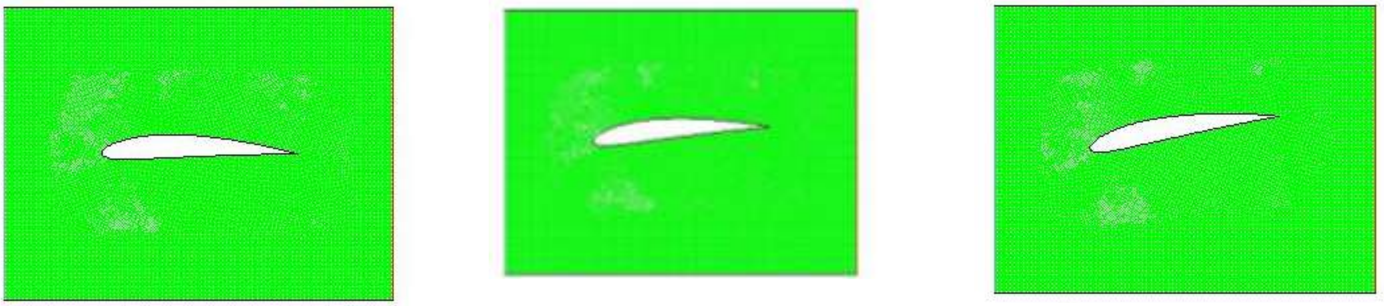
Boundary Conditions: The boundary conditions for the numerical analysis are given in table (2.1). The flow velocity of the air is chosen from the literature survey and taken as 15 m/s.

Table (2.1)

S No.	Requried	Quantity
1	Velocity of Air Flow(V)	15 m/s
2	Density of air(ρ)	1.225 kg/m3
3	Temperature	300 k
4	Viscosity of the air(μ)	1.7894 × 10–5 kg/ms
5	Reynolds no (Re)	3×10^6

3. RESULTS AND DISSCUSSIONS

In this dissertation work, numerical study on aerofoil section of the blade of axial flow fan has been done for three values of angle of attack. The NACA 747A415aerofoil has been used and computational domain is shown in figure 3.1. The results of numerical simulation are obtained in the form of velocity vectors, pressure distribution and turbulence around the aerofoil section. The velocity vectors at the leading edge, across aerofoil and trailing edge are shown to represent the velocity distribution around the section of the aerofoil.



(a) **(b)** **(c)**

Figure 3.1: Aerofoil Sections for angle of attack (a) 0°, (b) 5° and (c) 10°

3.1Angle of attack of 0°

The results of numerical simulation are obtained in the form of velocity vectors, pressure distribution and turbulence around the aerofoil section for the angle of attack of 0°.

3.1.1 Convergence. The figure (4.2) shows the residuals of the solution for this problem.

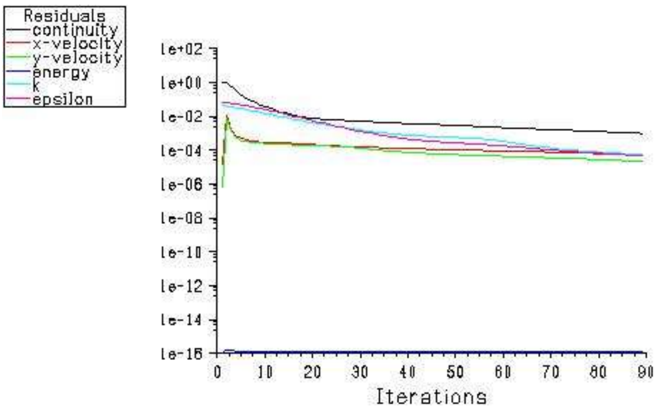
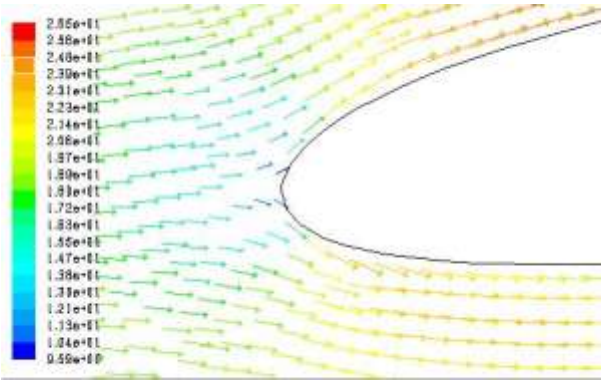
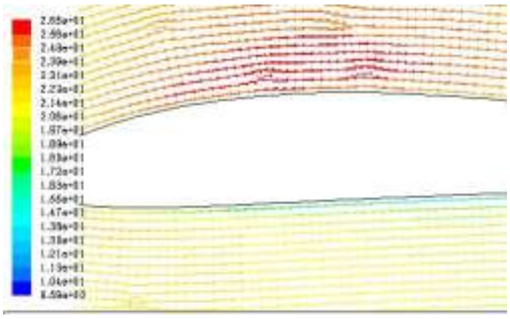


Figure 3.2

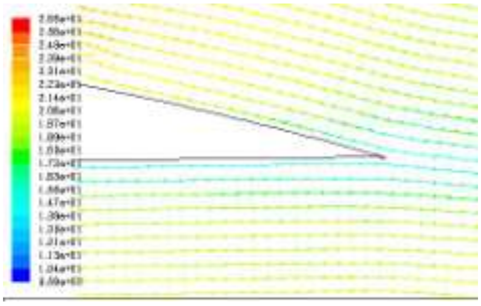
3.1.2 Velocity Vectors: Velocity vectors around the section of the aerofoil are obtained from CFD analysis and are shown in figure (3.3)



(a)



(b)



(c)

Figure (3.3): Velocity vectors (a) At leading edge, (b) across aerofoil and (c) trailing edge

3.1.2 Pressure Distribution: Pressure distribution on the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.4)

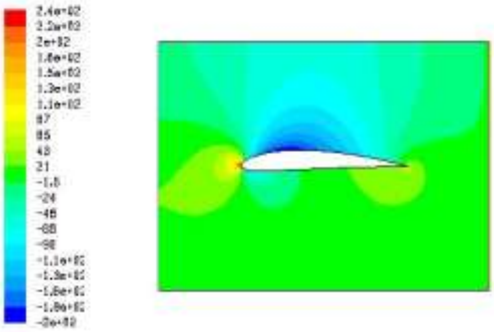


Figure (3.4): Pressure Distribution at 0^0

To show the variation in pressure around the aerofoil with the angle f attack of 0^0, pressure contour is obtained and is shown in figure 4.4. The contour shows that the pressure distribution and at the leading sharp pressure is visible due to striking of air at the leading edge. it is than distributing over and above the aerofoil surface. Pressure values are much higher at the middle surface of the aerofoil and it is depleting near the trailing edge.

3.1.2 Turbulence: Turbulence for the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.5)

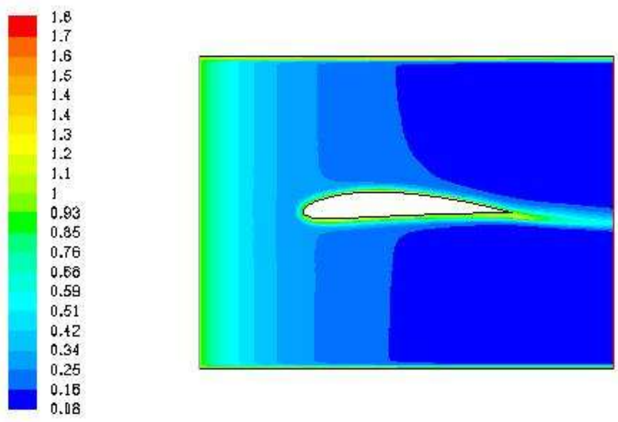


Figure (3.5): Turbulence at 0^0

Figure 3.5 shows the turbulence at the angle of attack of 0. Sharp turbulence is visible at the trailing edge.

3.2 Angle of attack of 5°

The results of numerical simulation are obtained in the form of velocity vectors, pressure distribution and turbulence around the aerofoil section for the angle of attack of 5°.

3.2.1 Convergence. The figure (3.6) shows the residuals of the solution for this problem.

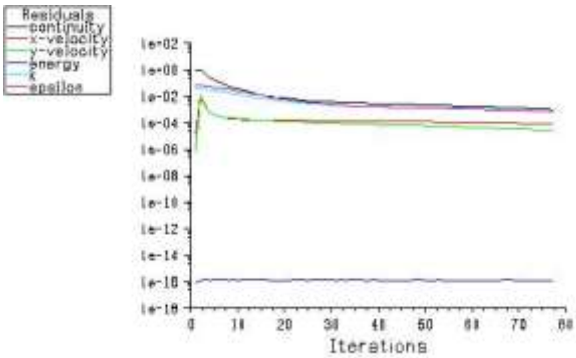
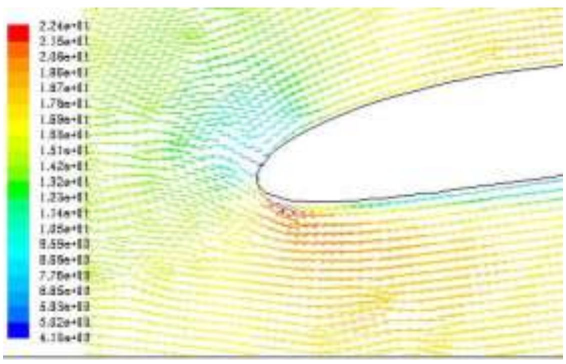
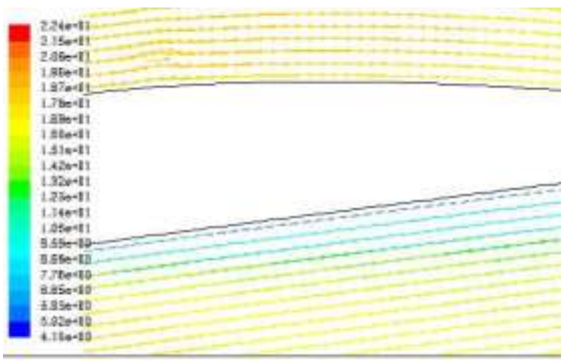


Figure 3.7

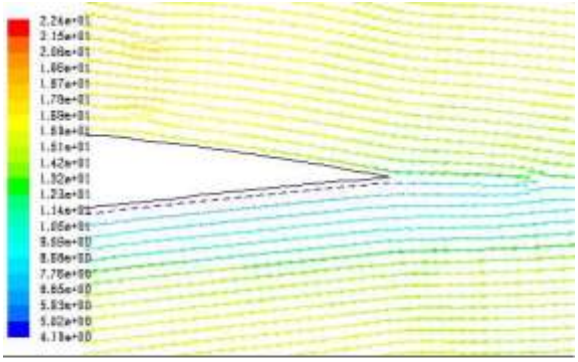
3.1.2 Velocity Vectors: Velocity vectors around the section of the aerofoil are obtained from CFD analysis and are shown in figure (3.8)



(a)



(b)



(c)

Figure: 3.8 Velocity vectors at 5^0 (a) At leading edge, (b) across aerofoil and (c) trailing edge

3.1.2 Pressure Distribution: Pressure distribution on the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.9)

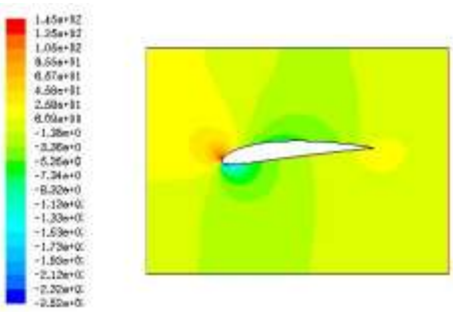


Figure: 3.9 Pressure Distributions at 5^0

3.1.2 Turbulence:Turbulence for the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.10)

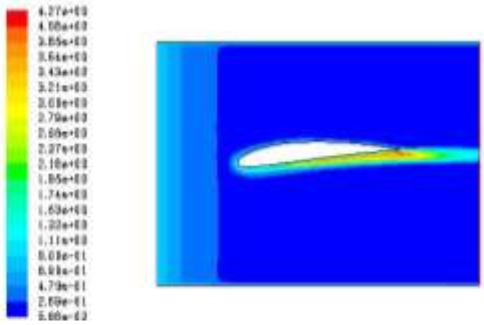


Figure: 3.10 Turbulence at 5^0

3.3 Angle of attack of 10°

The results of numerical simulation are obtained in the form of velocity vectors, pressure distribution and turbulence around the aerofoil section for the angle of attack of 10°.

3.3.1 Convergence. The figure (3.11) shows the residuals of the solution for this problem.

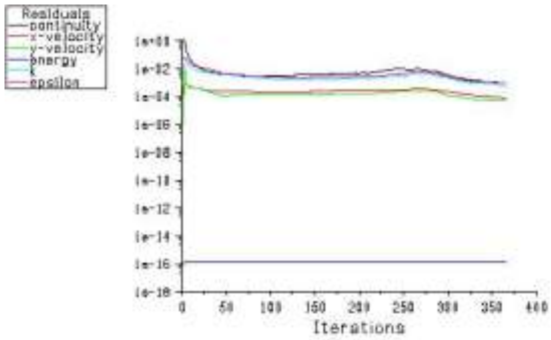
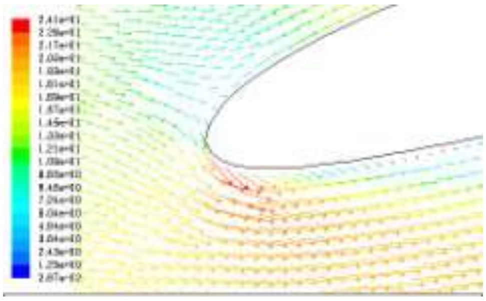
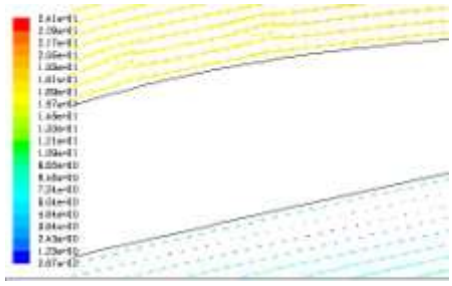


Figure: 3.11

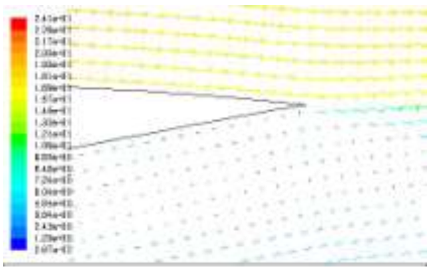
3.1.2 Velocity Vectors: Velocity vectors around the section of the aerofoil are obtained from CFD analysis and are shown in figure (3.12)



(a)



(b)



(c)

Figure: 3.12 Velocity vectors at 10^0(a) at leading edge, (b) across aerofoil and (c) trailing edge

3.1.2 Pressure Distribution: Pressure distribution on the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.13)

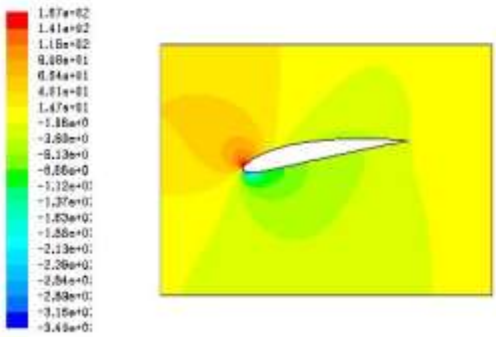


Figure: 3.13 Pressure Distributions at 10^0

3.1.2 Turbulence: Turbulence for the section of the aerofoil is obtained from CFD analysis and is shown in figure (3.14)

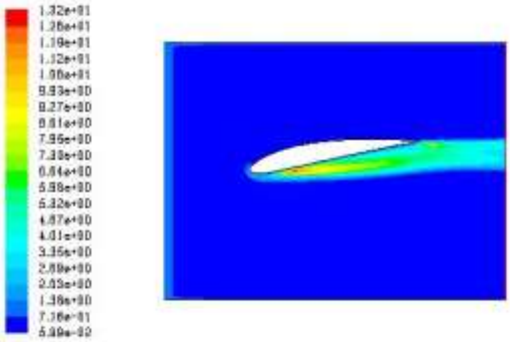


Figure: 3.14 turbulence at 10^0

DISCUSSION

The results of numerical simulation of flow features around the aerofoil section of an axial flow fan have been compared for the three different values of the angle of attack as 0°, 5° and 10°. The different contours are compared for velocity variation, pressure distribution and turbulence to get the complete picture of flow around the blade section and these may help in the better design of the blade in particular and fan in total.

The velocity vectors at three location of the aerofoil i.e. at the leading edge, across the aerofoil and the trailing edge are obtained to simulate the magnitude and direction of the flowing air around the aerofoil. The vectors show that the velocity magnitude is changing over and around the aerofoil. Boundary separation is not seen in the simulation and velocity variations are higher across the aerofoil as compared to the leading and trailing edge for the low or zero value of angle of attack. As the angle of attack is increased for 50 and 100, the velocity vectors show increasing value of velocity at the upper side of the aerofoil and decreasing values of velocity at the lower side. The velocity vectors at the trailing edge also show the same pattern.

To show the variation in pressure around the aerofoil with the angle f attack of 0°, 5° and 10° pressure contours are obtained. The contours of pressure show that the pressure distribution at the leading edge is sharply varying at the leading edge as the angle of attack is increasing. The values of pressure are more at the upper side of the aerofoil section than the lower side of the aerofoil section. With the increase of the angle of attack, the pressure distribution around the aerofoil is becoming uniform at trailing edge. The optimum value of the angle of attack seems to be greater than 5° and lower than the 10° for the greater values of lift and drag ratio.

The contours as obtained from the numerical simulation have also been compared and give the important information of turbulence. The figures show that the flow is very smooth for the low and zero value of angle of attack. As the angle of attack is increasing, the turbulence is increasing at the lower side of the blade section. The flow at the upper side is very smooth. The maximum turbulence is seen a the trailing edge for the angle of attack of 5° and it is maximum near the leading edge for the angle of attack of 10°. The optimum values of the turbulence may also fall in between the angle of attack of 5°-10° for the smooth flow of the air around the aerofoil.

REFERENCE

1. David lávi_ka, Richard matas, “Computational of darg and lift co efficients for simple two-dimensional objects with reynolds number Re=420000”,Owned by the authors, published by EDP Sciences, 2012 .
2. Durga Charan Panigrahi and Devi Prasad Mishra (2014). CFD simulations for the selection of an appropriate blade profile for improving energy efficiency in axial flow mine ventilation fans. Journal of Sustainable Mining, 13(1), 15–21. doi: 10.7424/jsm140104
3. Ahmed F. Abdel Azim El–Sayed & Mostafa Mohamed M. Ibrahim, “Numerical Investigation of Different Tip Clearance Shape Effects on Performance of an Axial Flow Compressor Stage”, The Online Journal on Power and Energy Engineering (OJPEE).
4. LI Yang & LIU Jie, “Internal flow mechanism and experiments research of low pressure axial fan with forward skewed blades”, Journal of hydrodynamics science directs.
5. Hussain NOURI, Florent RAVELET, Farid BAKIR, and Christophe SARRAF, “Experimental investigation on ducted counter-rotating axial flow fans”, Author manuscript, published in "Proceedings of ASME-JSME-KSME Joint Fluids Engineering Conference 2011, Hamamatsu Shizuoka : Japan (2011)."
6. C. Sarraf, H. Nouri, F. Ravelet,, F. Bakir, “Experimental study of blade thickness effects on the overall and local performances of a Controlled Vortex Designed axial-flow fan”,Author manuscript, published in "Proceedings of ASME-JSME-KSME Joint Fluids Engineering Conference hal-00507786, version 2 - 25 Nov 2010.
7. Eck, B., “Fans”, Pergamon Press, 1973.
8. Ali Aktürk & CengizCamci, “Axial Flow Fan Tip Leakage Flow Control Using Tip Platform Extensions”, Journal of Fluids Engineering Copyright © 2010 by ASME May 2010, Vol. 132 / 051109-1.8

9. Laszlo MOLNÁR1, Dr. JánosVad, "Design of a small –scale axial flow fan rotor of high specific performance",
10. D. N. So. rensen, M. C. Thompson & J. N. So.rensen, "Toward Improved Rotor-Only Axial Fans—Part II: Design Optimization for Maximum Efficiency", Journal of Fluids Engineering JUNE 2000, Vol. 122 Õ 325.
11. Eimad E. Elhadi & Wu Keqi, "Numerical simulation and modification 3d flow phenomena in an axial flow fan", Task quarterly 7 No 2 (2003), 199–213.
12. Young-Kyun Kim, Tae-Gu Lee, Jin-HuekHur, Sung-Jae Moon, and Jae-Heon Lee, "Flow Characteristics Impeller Change of an Axial Turbo Fan", International Journal of Aerospace and Mechanical Engineering 4:1 2010.
13. S.J. van der Spryt T.W. von Backstrdml, "Performance of rotor-only axial fans designed for minimum exit kinetic energy", R & D Journal, Vol. 18, No. 3, Nov. 2002.
14. AlirezaFalahat[ISSN: 2045-7057], "Numerical and Experimental Optimization of Flow Coefficient in Tubeaxial Fan", International Journal of Multidisciplinary science & Engineering , Vol. 2, No. 5, Aug. 2011.
15. Subrata Roy, Phillip Cho[ISSN 0148-7191I], "Designing Axial Flow Fan for Flow and Noise", International Off-Highway &Powerplant Congress & Exposition Indianapolis, Indiana September 13-15, 1999.
16. Oday I. Abdullah, Josef Schlattmann, "Stress Analysis of Axial Flow Fan", Adv. Theor. Appl. Mech., Vol. 5, 2012, no. 6, 263 – 275.
17. MahajanVandana N., Shekhawat Sanjay P. "Analysis of Blade of axial flow fan using Ansys", International Journal of Advanced Engineering Technology E-ISSN 0976-3945.
18. Gifford N. L., Hunt A. G., "Experimental Study of Low-Pressure Automotive Cooling Fan Aerodynamics under Blocked Conditions", CSME 2006 Forum 8.
19. TolgaKokturk, A Thesis of "Design and performance analysis of a reversible axial flow fan", Middle East Technical University in June 2005.

