Computational Review of Aerofoil Sections in Axial Flow Fan Blade Design

Prof. Emily S. Thompson, Dr. Benjamin A. Harris, William J. O'Connor*

Guide, Mechanical Engineering Department, University of Cambridge, Cambridge, UK
Co-guide & Head, Mechanical Engineering Department, University of Cambridge, Cambridge, UK
M.Tech Research Scholar, Mechanical Engineering Department, University of Cambridge, Cambridge, UK

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.

KEYWORDS: Axial Flow Fan, Aerofoil, Angle of attack, Lift And Drag, CFD.

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.

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).

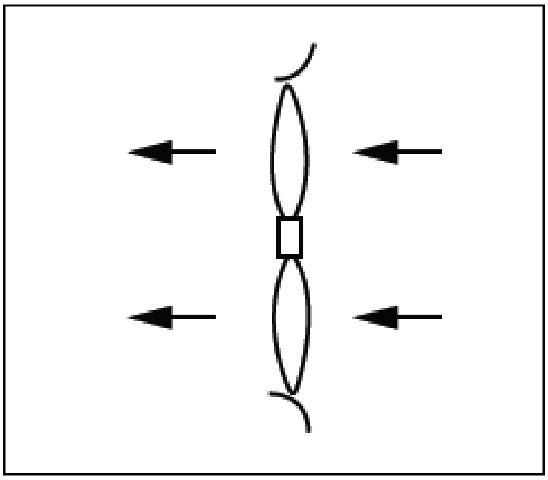

Fig. 1.1 Axial Flow

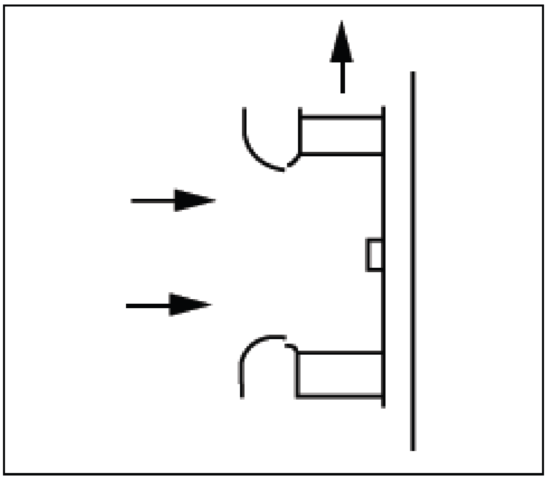

Fig. 1.2 Centrifugal Flow

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).

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.

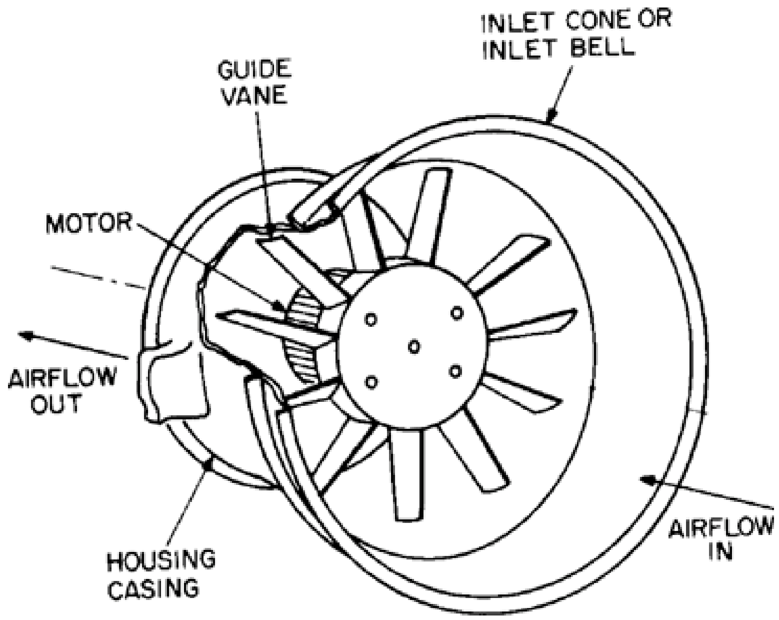

Fig. 1.3 Axial Flow Fan

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 hole circulation.

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 a 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.

F*igure 1.4: Propeller fan with direct drive*

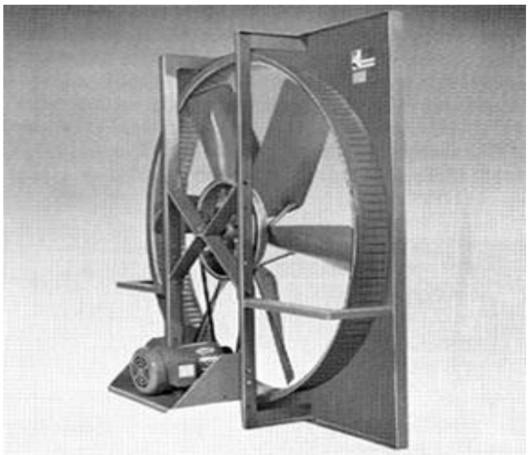

Figure 1.5: Propeller fan with belt drive

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

CONCLUSION

In this dissertation, numerical study on the aerofoil section The NACA 747A415 aerofoil 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). The results obtained in the form of velocity vectors at the leading edge, across the section of the aerofoil and the trailing edge of the aerofoil section are presented.

SCOPE FOR FUTURE WORK

In the present distribution work, the numerical study on the aerofoil section of the blade of an axial flow fan has been done. Simulation results in the form of velocity vectors, pressure distribution and turbulence are also presented for three values of the angle of attack as 0°, 5° and 10°. Numerical turbulence model K-ϵ in Standard form is used for the analysis.

In future the following work may also be done

1. The numerical analysis may be performed on other section of the aerofoils.
2. Other turbulence models may also be used for the other sections of aerofoil.
3. Experimental analysis may also be done for the numerical study performed in this dissertation work.

LITERATURE REVIEW

David lávi_ka, Richard matas [1] The article deals with comparison of drag and lift coefficients for simple two-dimensional objects, which are often discussed in fluid mechanics
fundamentals books. The commercial CFD software ansys/fluent 13 was used

for computation of flow fields around the objects and determination of the drag and lift coefficients. The flow fields of the two-dimensional objects were computed for velocity up to 160 km per hour and Reynolds number Re = 420 000.
Main purpose was to verify the suggested computational domain and model settings for further more complex objects geometries. The more complex profiles are used to stabilize asymmetrical ('z'-shaped) pantographs of high-speed trains.
The trains are used in two-way traffic where the pantographs have to operate with the same characteristics in both directions.
Results of the CFD computations show oscillation of the drag and lift coefficients over time. The results are compared with theoretical and experimental data and discussed. Some examples are presented in the paper

Durga Charan Panigrahi1 and Devi Prasad Mishra2 [2] This study focuses on one of the key design aspects of mine ventilation fans, i.e. the selection of an appropriate aerofoil blade profile for the fan blades in order to enhance the energy efficiency of axial flow mine ventilation fans, using CFD simulations. Computational simulations were performed on six selected typical aerofoil sections using CFD code ANSYS Fluent 6.3.26 at angles of attack varying from 0^0 to 21^0 at an interval of 3^0 and at Reynolds number $Re = 3 \times 10^6$, and various aerody-namic parameters, viz. coefficients of lift (*Cl*) and drag (*Cd*) as a function of angle of attack (α) were determined to assess the efficiency of the aerofoils The study revealed that the angle of attack has a significant effect on the lift and drag coefficients and stall condition oc-curred at α values of 12^0 and 15^0 in most of the aerofoils. Based on the criterion of higher lift to drag ratio (*Cl*/*Cd*), a blade profile was chosen as the most efficient one for mine ventilation fans. This study forms a basis for selecting appropriate blade profiles for the axial flow fans used for ventilation in mining industry. The application of an appropriate aerofoil blade profile will impart energy efficiency to the mine ventilation fans and thereby result in energy saving in mine ventilation.

Ahmed F. Abdel Azim El–Sayed at el [3]. An efficient numerical method is applied to investigate the influence of different tip clearance of compressor rotor on the performance of an axial flow compressor stage. The steady, viscous, compressible flow three dimensional governing equations representing the flow field with standard *K* turbulence model are solved using commercial code Fluent 6.3.26. Tip clearance shapes of compressor rotor in this paper include zero gap, different uniform gap and linearly varying gap. The analysis of the results shows, that the rotor blade with zero gap has higher efficiency and higher pressure ratio. Increasing tip clearance, leads to decrease of both efficiency and pressure ratio. Surge margin decreases also with increasing the tip clearance. Linearly expanding gap leads to increase both compressor efficiency and pressure ratio compared with linearly shrinking gap.

LI Yang, LIU Jie et al. [4] This article presents the flow mechanism analysis and experimental study of a forward-skewed impeller and a radial impeller in low pressure axial fan. The forward-skewed blade was obtained by the optimization design of the radial blade and CFD technique. Measurement of the two blades was carried out in aerodynamic and aeroacoustic performance. Compared to the radial blade, the forward-skewed blade has demonstrated the improvements in efficiency, total pressure ratio, Stable Operating Range (SOR) and less aerodynamic noise. Detailed flow measurement and computation were performed for outlet flow field for investigating the responsible flow mechanisms. The results show the forward-skewed blade can cause a span wise redistribution of flow toward the blade mid-span and reduce tip loading. This results in reduced significantly total pressure loss near hub and shroud end wall region, despite the slight increase of total pressure loss at mid-span.

HussainNouri, FlorentRavelet, FaridBakir et al. [5] an experimental study on counter-rotating axial-flow fans was carried out. The fans of diameter D = 375 mm were designed using an inverse method. The counter-rotating fans operate in a ducted-flow configuration and the overall performances are measured in a normalized test bench. The rotation rate of each fan is independently controlled. The distance between the fans can vary from 10 to 50 mm by steps of 10 mm. The results show that the efficiency is strongly increased compared to a conventional rotor or to a rotor-stator stage. The effects of varying the rotation rates ratio on the overall performances are studied and show that the system is highly efficient on a wide range of flow-rates and pressure rises. However, the change of the axial distance between rotors from 10 to 50 mm does not seem to change the overall performances. This system has thus a very flexible use, with a large patch of high efficient operating points in the parameter space. Further local studies including velocity measurements and wall-pressure fluctuations in the space between the rotors are needed to better understand the interactions between the rotors and to optimize the system.

C. Sarrafa, H. Nouri et al [6] the purpose of this work is to study the effects of blade thickness on the performances of an axial-flow fan. Two fans that differ only in the thickness of their blades were studied. The first fan was designed to be part of the cooling system of an automotive vehicle power unit and has very thin blades. The second fan has much thicker blades compatible with the rotor moulding conception process. The overall performances of the fans were measured in a test bench designed according to the ISO-5801 standard. The curve of aerodynamics characteristics (pressure head versus flow-rate) is slightly steeper for the fan with thick blades, and the nominal point is shifted towards lower flow-rates. The efficiency of the thick blades fan is lower than the efficiency of the fan with thin blades but remains high on a wider flow-rate range. The mean velocity fields downstream of the rotors are very similar at nominal points with less centrifugation for the thick blades fan. Moreover, the thick blades fan maintains an axial exit-flow on a wider range of flow-rates. The main differences concern local properties of the flow: Phase-averaged velocities and wall pressure fluctuations strongly differ at the nominal flow-rates. The total level of fluctuations is lower for the thick blades fan that for the thin blades fan and the spectral decomposition of the wall fluctuations and velocity signals reveal more harmonics for the thick blades fan, with less correlation between the different signals. For this kind of turbo machinery, the use of thick blades could lead to a good compromise between aerodynamic and acoustic performances, on a wider operating range.

Eck, B [7]This book related to the design of Axial flow fans and centrifugal fan.
Ali Aktürk & Cengiz Camci et al [8] Performance of an axial flow fan unit is closely related to its tip leakage mass flow rate and level of tip/casing interactions. The present experimental study uses a stereoscopic particle image velocity meter to quantify the three dimensional mean flow observed near the blade tip, just downstream of a ducted fan unit. After a comprehensive description of the exit flow from the baseline fan, a number of novel tip treatments based on custom designed pressure side extensions are introduced. Various tip leakage mitigation schemes are introduced by varying the chord wise location and the width of the extension in the circumferential direction. The current study shows a proper selection of the pressure side bump location and width are the two critical parameters influencing the success of each tip leakage mitigation approach. Significant gains in the axial mean velocity component are observed when a proper pressure side tip extension is used. It is also observed that a proper tip leakage mitigation scheme significantly reduces the tangential velocity component near the tip of the axial fan blade. Reduced tip clearance related flow interactions are essential in improving the energy efficiency of ducted fan systems. A reduction or elimination of the momentum deficit in tip vortices is also essential to reduce the adverse performance effects originating from the unsteady and highly turbulent tip leakage flows rotating against a stationary casing.

Laszlo MOLNÁR1, Dr. JánosVad [9] Geometrical versions of an axial flow fan rotor of high specific performance was designed using various controlled vortex design methods. In addition to the formerly established method prescribing a power function circulation distribution along the blade span, a new design concept was applied and compared to the power function method. The new concept prescribes a span wise circulation distribution corresponding to a segment of a Gaussian function. This method enables the unloading of the blade sections near the hub and tip, whereas guarantees reasonably high loading along the dominant part of span. The Gaussian distribution provides a convenient mathematical tool for the designer. A computer-aided tool has been established for design and study of the blade geometrical versions.

D. N. So. Rensen , M. C. Thompson et al.[10]Numerical design optimization of the aerodynamic performance of axial fans is carried out, maximizing the efficiency in a design interval of flow rates. Tip radius, number of blades, and angular velocity of the rotor are fixed, whereas the hub radius and span wise distributions of chord length, stagger angle, and camber angle are varied to find the optimum rotor geometry. Constraints ensure a pressure rise above a specified target and an angle of attack on the blades below stall. The optimization scheme is used to investigate the dependence of maximum efficiency on the width of the design interval and on the hub radius.

Eimad E. Elhadi& Wu Keqi et al [11] This work aims to study and analyze the behavior of flow in an axial flow fan using numerical simulation based on solving Reynolds-averaged Navier-Stokes equations coupled with a modified Spalart-Allaras turbulence model. In the present work, different flow phenomena occurring in an axial flow fan were presented and interpreted at different design conditions and at different regions, with special concern to the rotor exit and stator regions. Large vortices and reverse flow at the stator suction side were observed under high loading conditions. These are mainly due to radial and centrifugal effects in the fluid due to a decreasing flow rate. To avoid this type of flow, the stator flow pattern has been modified using a new stator blade. This blade was designed by changing the stator stagger angle by 10°in the radial direction. This study indicates that the flow in the modified stator region behaves well and gives better performance than that obtained when using a baseline stator, especially under high loading conditions. This study also indicates that the stator stagger angle has a significant effect on modifying the fan performance.

Young-Kyun Kim, Tae-Gu Lee, Jin-HuekHur et al [12]In this paper, three dimensional flow characteristic was presented by a revision of an impeller of an axial turbo fan for improving the airflow rate and the static pressure. TO consider an incompressible steady three-dimensional flow, the RANS equations are used as the governing equations, and the standard k-ε turbulence model is chosen. The pitch angles of 44°, 54°, 59°, and 64° are implemented for the numerical model. The numerical results show that airflow rates of each pitch angle are 1,175 CMH, 1,270 CMH, 1,340 CMH, and 800 CMH, respectively. The difference of the static pressure at impeller inlet and outlet are 120 Pa, 214 Pa, 242 Pa, and 60 Pa according to respective pitch angles. It means that the 59° of the impeller pitch angle is optimal to improve the airflow rate and the static pressure.

S.J. van der Spryt T.W. von Backstrdml [13] The major loss source in rotor only axial fan is the kinetic energy associated with the axial with circumferential component of exit flow. This paper investigated aerodynamics and acoustics performance of two fans were tested foe fan noise and performance in accordance BS 848. The low fan noise are quieter and more of the two fan, whist both fan are quieter efficient than an existing typical general application fan, of unknown design. The underlined the importance of following a well defined design methodology when design rotor only axial flow fan.

AlirezaFalahat[ISSN: 2045-7057][14] In this study an attempt was made to find the best the best angle of attack and rotational velocity of a flat blade at a fixed hub to tip ratio for a maximum flow coefficient in an axial fan in a steady and turbulent conditions. In this study the blade angles are varied from 30 to 70 degrees and the rotational velocity is varied from 50 to 200 rad/sec for a number of blades from 2 to 6, at a fixed hub to tip ratio. The numerical and experimental results show that, the maximum flow coefficient is achieved at the blade angle of attack of between 45 to 55 degrees when the number of blades was equal to 4 at most rotational velocities. The numerical results show that as the rotational velocity increased, the flow coefficient increased but at very high rotational velocities the flow coefficient remained constant.

Subrata Roy, Phillip Cho[ISSN 0148-7191I] [15] A comprehensive finite element methodology is developed to predict the compressible flow performance of a non-symmetric 7-blade axial flow fan, and to quantify the source strength and sound pressure levels at any location in the system. The acoustic and flow performances of the fan are predicted simultaneously using a computational aero-acoustic technique combining transient flow analysis and noise propagation. The calculated sound power levels compare favorably with the measured sound power data per AMCA 300-96 code.

Oday I. Abdullah, Josef Schlattmann, [16] In this paper the finite element method has been used to determine the stresses and deformations of an axial fan blade. Three dimensional, finite element programs have been developed using eight-node super parametric shell element as a discretization element for the blade structure. All the formulations and computations are coded in Fortran-77. This work was achieved by modeling the fan blade as a rotating shell. The investigation covers the effect of centrifugal forces on stresses and deformations of rotating fan blades. Extensive analysis has been done for various values, speed of rotation, thickness, skew angle, and the effect of the curvature on the stress and deformation. The numerical results have shown a good agreement compared with the available investigations using other methods.

MahajanVandana N.,* Shekhawat Sanjay P [17] In the present paper CFD based investigation has been reported in order to study the effect of change in speed of fan on velocity, pressure, and mass flow rate of axial flow fan. It has been observed that there is a significant change in mass flow rate, velocity of rotor and guide or stator vanes as the speed of fan is varied. As the performance of fan is directly dependent on mass flow output. So there should be a moderate velocity and pressure profile as all these parameters are co-related. In order to predict the mass flow output, velocity and pressure on stator and rotor section ,analysis is done using a software ansys12.and to create a general idea about an axial flow fan a model is created using a modeling software catiav5.

Gifford N. L., Hunt A. G.et al [18] The present research examines the effect of a downstream-mounted blockage plate on the performance of low-pressure, axial-flow, automotive cooling fans (LPF). Measurements conducted in a plenum chamber quantify performance changes as a function of blockage distance. Three-component Laser Doppler Velocity meter (LDV) measurements were made of the downstream flow with and without blockage. Experiments were performed for two fan designs, one optimized for low flow rate, high-pressure operation, and a second optimized for high flow rate, low-pressure operation. The results show that the pressure loss caused by the blockage plate increases with increasing flow rate and decreasing blockage distance. LDV measurements show that the blockage plate causes a reduction in the flow rate, an increase in the reverse flow near the fan hub, and a dramatic increase in the radial flow. The relation between blockage to fan proximity and fan performance was established. It is found that the pressure change follows a quadratic function, but the coefficients are fan specific.

Tolga Kokturk [19], Reversible axial flow fans are used as emergency ventilation fans to discharge the smoke generated on the probable fires occurring in the underground transportation systems and mines as quickly as possible, without causing any harm to people exposed to it. The fans which are placed in different configurations according to the location of fire must be able to work bi-directionally, namely reversible. Due to this fact, the blade profiles of the fan must possess the same aerodynamic performance while working on either discharge or suction condition of the fan, dictated by direction of the rotation.