International Journal of Engineering Researches and Management Studies

ANALYSIS OF SIX BLADED AXIAL FAN USING ANSYS

Jayaram Thumbe[*1] **& Jyothish VM**[2]

[*1]Associate Professor, Department of Mechanical Engineering, Srinivas Institute of Technology, Mangalore, India

[2]UG Student, Department of Mechanical Engineering, Srinivas Institute of Technology, Mangalore, India

ABSTRACT

The objective of the present study is to carry out a computational analysis to observe the physics of fluid flow through a six bladed axial fan. The axial fan is modelled based on standard specifications. The analysis is carried out using commercially available CFD code. The results were obtained for standard boundary conditions at the inlet with turbulent flow conditions. The results gave a maximum pressure of 105.7 Pa and the maximum velocity of 14.12 m/s at the outlet at these conditions. Further, these results can be used to compare the performance of six bladed axial fan with nine and twelve blades of an axial fan.

Keywords: Axial Fan, Computational fluid dynamics (CFD), Pressure, Velocity streamline.

I. INTRODUCTION

The axial flow fan is extensively used in many engineering applications. This type of fan is used in a wide variety of applications, ranging from small cooling fans for electronics to the giant fans used in wind tunnels. Axial flow fans are applied for air conditioning and industrial process applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers to automotive engine cooling and in-cabin air recirculation systems [1], [2]. The extended use of axial flow fans for fluid movement and heat transfer has resulted in detailed research into the performance attributes of many designs. Numerical investigations have been performed to quantify the performance of axial fans and their flow characteristics [3]. Axial fans blow air along the axis of the fan, linearly, hence their name. The axial-flow fans have blades that force air to move parallel to the shaft about which the blades rotate [4].

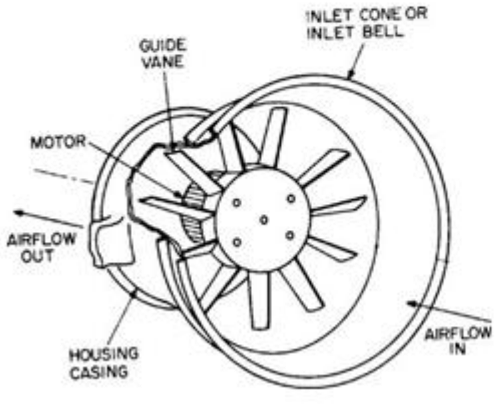

Figure 1 Simple construction of an axial fan

With the expressive computer capability and extensive development in the simulation field, CFD has drawn attention in recent years. With the help of CFD, the complex 3-D geometries of equipment can now be modeled with only minor simplifications. CFD models, if created correctly, account for the complex flows in equipment [5]. CFD models for axial fans have been used to evaluate the flow behavior and characteristics. The models provide sufficiently accurate predictions over a range of operating conditions, which are not possible using other methods. In this paper, CFD was used to model the flow passing through an axial fan. The objective was to determine ways to increase the efficiency. Axial flow fans, while incapable of developing high pressures, they are well suitable for handling large volumes of air at relatively low pressures [6].

IJERMS

International Journal of Engineering Researches and Management Studies

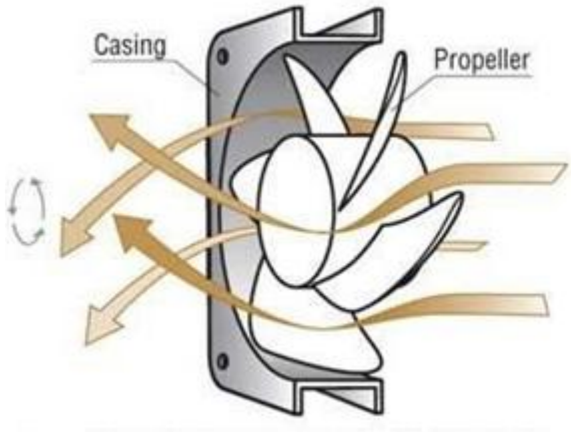

Figure 2Airflow mechanism in an axial fan

In general, they are low in cost and possess good efficiency, and can have blades of the airfoil shape. The operating principle of axial-flow fans is simply deflection of air. Flow can be decomposed into two components: axial velocity and tangential or circumferential velocity. Axial velocity is the desired velocity since it moves air from/to the desired spaces and tangential velocity is an energy loss in axial fans or it can be converted into static pressure as in the case of vane axial fans [7].

Modeling and analysis are done using CFD software. The design of the axial fan blades is carried out in commercially available software and is imported to the latter in IGES format.

The present work is carried out to find the various contours (such as pressure contour and velocity contour), Streamlines under steady conditions for six bladed axial fan.

II. LITERATURE REVIEW

S. Jain, et al [1] provides an insight of CFD modeling of a radiator axial fan for air flow distribution. The axial flow fan is extensively used in many engineering applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers and air conditioning units to automotive engine cooling and in-cabin air recirculation systems. An APT T4 repower radiator fan and fan shroud play a crucial role in complicated engine cooling systems. A radiator is a type of heat exchanger designed to transfer thermal energy from the coolant to the surrounding air by means of a mechanism known as natural or forced convection. The latter case concerns the use of a radiator fan to pull the air through the radiator core. The fan provides airflow through the radiator. The orientation of the blade also plays an important role in understanding the flow of air across the radiator and fan. For a right oriented blade, the direction of fan rotation is clockwise; for a left oriented blade, the direction of fan rotation is counter clockwise.

As far as the simulation is considered, it is preceded with the CFD code processing the data, applying the basic theory of fluid mechanics by balancing the mass continuity and momentum equations in numerical form and thereafter producing numerical predictions of the flow variables. The problem setup process is completed by defining the boundary conditions, solver controls, and convergence monitors. Assuming the flow to be ideal and dry air at a standard atmospheric pressure, the boundary conditions include fixed wall, moving internal zone at 1680 rev/min, zero pressure at the outlet, and variable mass flow rate at the inlet. The residual values of all the variables solved are monitored during the iteration process, with the mass balance set to less than 1.0E-4. This iteration process needs to be monitored for convergence and repeated if the numerical error conditions are not satisfied. The final step is to analyze the output data and present them in the form of velocity vector distribution and contour plots.

The key and important outcomes of this study are as follows:

1. The CFD modeling shown in this study proved to be very helpful in initiating further and more comprehensive numerical study of the off-road engine cooling system.

IJERMS
International Journal of Engineering Researches and Management Studies

2. CFD results were presented in the form of velocity vectors and path lines, which provided actual flow characteristics of air around the fan for different blade orientations.
3. Detection of high and low air flow regions with recirculation and vortices immensely improved understanding of flow in the complex system studied, which helped to understand the complications involved in hot air recirculation.
4. This study showed how the flow of air was interrupted by the hub obstruction, thereby resulting in unwanted reverse flow regions.
5. The different orientation of blades was also considered while performing CFD analysis. The study revealed that a left oriented blade fan with counter clockwise rotation performed the same as a right oriented blade fan with clockwise rotation.
6. The CFD results were in agreement with the experimental data measured during physical testing. Any error was probably a result of the experimental conditions, fluctuations of fan rev/min, or the considered ideal condition while simulating the analysis.

Mahajan Vandana N, et al [2] presents some of their aims regarding CFD analysis, which is as follows,

1. Modeling of blades of axial flow fan.
2. Analysis of mass flow of fluid through blades by considering various angular velocities using Ansys.
3. Analysis of velocity on the periodic surface by considering various angular velocities using Ansys.
4. Analysis of rotor pressure considering various angular velocities using Ansys.
5. Analysis of stator pressure considering various angular velocities using Ansys.
6. Analysis of mid surface velocity by considering various angular velocities using Ansys.

In this case speed of the fan is varied at 1000 rpm, 1800 rpm, and 2500 rpm, the corresponding changes in pressure and velocity at blades section is analyzed using Ansys software. Also,modeling of the fan is done using modeling software CATIA V5.

Three-dimensional air flow through an axial, multistage fan will be solved in this analysis. There are two sections of flow created by this multistage fan. In the first stage, flow is created by a 9 bladed rotor, which rotates at 1800 rpm. In the second stage, the flow is "straightened out" by means of a 12 bladed stator section located just downstream of the rotor section. At the inlet to the rotor, air at 0 Pa (gauge) is drawn in. At the outlet, air is exhausted at 0 Pa. In this investigation, a k-ε model with standard wall treatment is used in FLUENT .Air is taken through the fluid flowing through stator and rotor section. Mixing plane model is used. CFD-POST is used to calculate velocity and pressure at rotor and stator section at 1000 rpm, 1800 rpm, and 2500 rpm. In CFD-POST, fluid is considered as moving from rotor to stator while stator and rotor are kept stationary.

The results obtained by varying speeds are as follows.
1. For 1000 rpm =0.01Kg/s
2. For 1800 rpm =0.018Kg/s
3. For 2500 rpm = 0.20kg/s

As the rpm of the rotor increase, the mass flow at outlet get increase the increment in output is not linear. Also as the rpm of the rotor increases, the pressure on stator blade inlet increases at a higher rate. This result is obtained from the analysis using ANSYS for the different rpm.

1. For further optimization, one can change the angle of the blade (in the range of 20° to 30°).
2. One can also change the shape of the blade for the gating higher output of the air flow.

H. Kumawat, et al [7] describes that the axial fan is used in many engineering application, they are used for air conditioning and industrial process applications. The extended use of an axial fan includes fluid movement and heat transfer fields. Axial fans blow air along the axis of the fan, linearly and hence called has an axial fan.

In this paper, CFD was used to analyze the axial flow fans, which is incapable of developing high pressure; they are suitable for handling large volumes at low pressure. Using CFD first step is to create a axial flow fan that

IJERMS
International Journal of Engineering Researches and Management Studies

can be produced by 3D CAD SolidWorks software, after creating the 3D models are imported to CFD software, which are remodelled into different sections and refined with the finite volume meshing, where detail of geometrical shape are to be defined clearly and precisely, thus final meshing are to be accurate. After meshing import, these files to CFD code processor, which will solve based on the flow equations. Boundary conditions are needed to be set, which includes flow rate and fluid properties and next step is a simulation by applying basic theory of fluid mechanics and momentum equations. On post- processing, the numerical CFD results, the observation of temperature contour, velocity streamlines, and pressure contour are studied. Optimized design is compared with the initial design. The initial design had 7 blades, with flow rate 22m/s. By observing pressure contour the pressure changes from positive to negative, and pressure zone at the outlet. Temperature changes occur mainly due to room temperature and frictional heating, temperature variation is not uniform on the blade surface, where thermal cracks are more and efficiency is less. By observing the velocity streamlines thus velocity is also not uniform over the blade, which leads to noise and decrease in efficiency.

For optimized design fan with 11 blades with a flow rate of 22m/s. Thus temperature variation is almost uniform on blade surface and chances of forming cracks are less and efficiency is increased. Velocity streamlines are almost uniform and lesser noise and efficiency is increased.

By considering different parameters like temperature, Pressure, Velocity streamlines and number of blades. The study revealed that an optimum number of blades gave good results compared to the fan with less number of blades, as a compromise with efficiency and cost. It is found that 5 to 12 blades are good for every application.

III. NUMERICAL EXPERIMENTATION

Analysis of six bladed axial fan
The Analysis is done for six Bladed Axial fan setup. The 3D CAD model is carried out in commercial software and then exported it in IGES format. Figure 3 shows the six bladed axial fan used for the analysis.

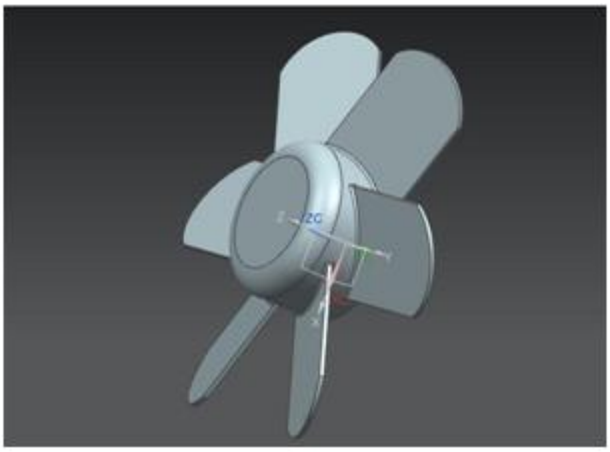

Figure 3 Six bladed axial fan blade

The exported file is imported to Design Modeler in Ansys Fluent software. The CAD model is provided with an enclosure of 0.01 m cushion radius, similarly cushion +ve direction and –ve direction of 0.5 m and 1 m is given respectively. The required solid is suppressed and meshing is carried out for specific user preference. Figure 4 shows the meshing after all the considerations being followed. It exactly shows the element quality, which helps understand the quality of the mesh, thus helping us determine whether meshing process is acceptable or not

ISSN: 2394-7659
IMPACT FACTOR- 2.789

IJERMS

International Journal of Engineering Researches and Management Studies

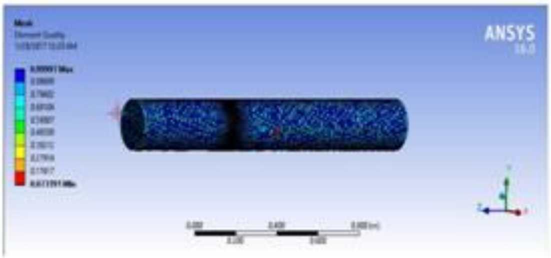

Figure 4 Meshing process

The vital part before mesh generation is to name certain part of the model as Inlet, Outlet, Wall Blade, and Outer wall. This enables to apply basic laws and boundary conditions in Fluent.

The Meshed model is then imported in Fluent Setup. To check the worth of the mesh, Check option is clicked and the mesh quality can be determined. The next part is to select the solver, Here, Steady state analysis is preferred and the solver is pressure based as we need pressure contour mainly. Absolute velocity formulation is select in the option provided below. The flow is assumed to be ideal and dry air at standard atmospheric pressure. The model of the air flow is K-ε turbulence model with standard wall function. The temperature is set as default at 298 K with pressure being atmospheric. The solver is directed for a 3D, pressure-based implicit with steady conditions. The solution method incorporated is SIMPLE, first order upwind. The idea of convergence criteria makes sense when the problem at hand is non-linear thus making it impossible to find the exact solution to it, thus the convergence criteria here provided is, $1e^{-4}$, which is the default value.

The boundary conditions are selected for the named selections. After this, Solution has to be initialized, here standard initialization is chosen and the calculation is run. The number of iterations is chosen. As the model and result expected is a bit complicated, the iterations are set in 1000s and so on. Depending on the processor the analysis is being done, it will take a while to complete the calculation.

Validation of numerical analysis

The pressure distribution obtained from the numerical analysis is compared with the results obtained from H. Kumawat, et al [7]. However, the number of blades used by H. Kumawat, et al, was seven whereas the present analysis is for six blades.

IV. RESULTS AND DISCUSSION

On post-processing the numerical CFD results, the observations are presented as pressure contour, vector (Velocity at stationary frame), and velocity streamlines.

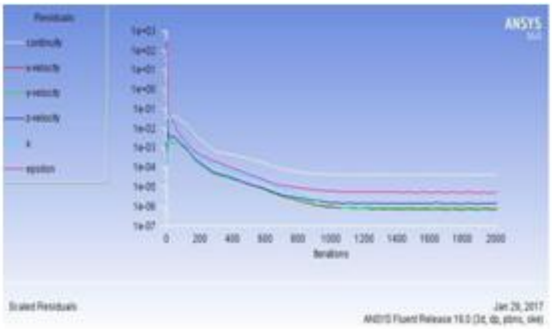

Figure 5 Residuals v/s No. of iterations

Figure 5 represents the various velocities in x, y, and z direction, with continuity, k, and epsilon plots against the number of iterations set. The number of iterations set is purely based on the convenience as well as the intensity of the results to be obtained. Although the result is entirely unknown, theoretical calculations can make it easier to determine the range of value to be accepted. Here, 2000 was set as the maximum iterations, due the hardware and time constraints

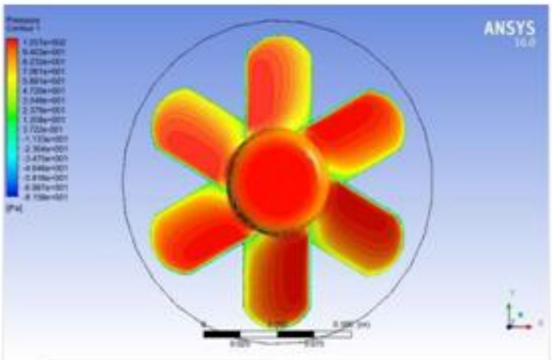

Figure 6 Pressure contour

Figure 6 shows the pressure contour of the blades when it is held stationary. The reason for the showcasing of pressure exactly is to understand the pressure acting upon fan blades and the hub under turbulent. This can be observed in any number of blades. But the maximum value of pressure will be higher as the gap allowed for air passage becomes less. As directed, the optimum number of blades to be used is between 5 and 12. More the number of blades, difficult is air passage. Also, the FD1 provided when including the enclosure was 0.01m.

The maximum value of pressure acting on the setup is $1.057e^{+02}$ Pa whereas the minimum value is - $8.158e^{+01}$ Pa. It can be observed that the minimum pressure acting on blades is very less. The blade

IJERMS
International Journal of Engineering Researches and Management Studies

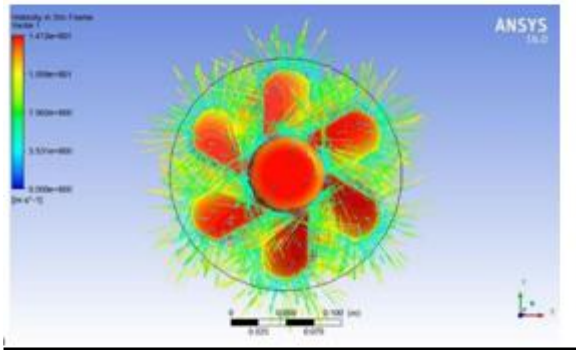

Figure 7 Velocity vector (front view)

Figure 7 shows the front view of the velocity vector. This enables an understanding of the air movement after coming in contact with the setup.

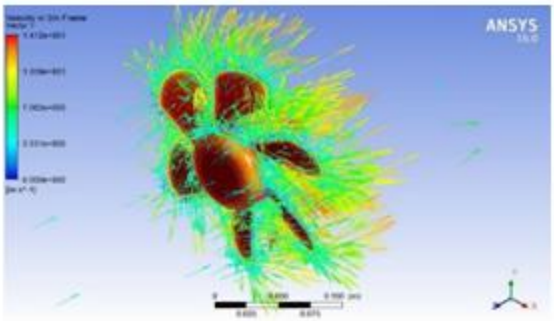

Figure 8 Velocity vector (isometric view)

Figure 8 shows the isometric view of the velocity vector. The left side of the figure is inlet and right side is the outlet. As seen movement is clearly visible movement after coming in contact with the blades.

As the analysis is done considering an enclosure, the air comes in contact with the enclosure as well. The maximum velocity is at 1.412e+01 m/s whereas th minimum value is at 0 m/s. The maximum velocity is observed after the air has hit the blades. The minimum velocity is observed at the blade-hub junction.

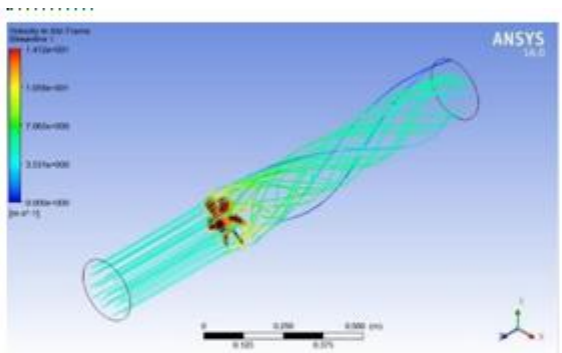

Figure 9 Velocity streamline

Figure 9 shows the velocity streamline of the present analysis. As it's clearly understandable that the flow is swirling. The streamline obtained focusses on the velocity at a stationary frame. As with the velocity vector, the result obtained is not very much different.

IJERMS
International Journal of Engineering Researches and Management Studies

The maximum velocity of the streamline is 1.412e+01 m/s and minimum value is 0 m/s. But what's interesting to see is that the movement of air after coming in contact with the blade-hub setup. The fan placed 0.5m from the inlet and 1m from outlet. This enables us to view the movement of air clearly. The inlet velocity is in the range of 1.5 m/s to 5m/s. After coming in contact, the velocity jumps up to 7 m/s. The minimum velocity is observed only close to the enclosure that is 0 m/s.

V. CONCLUSION

The analysis was performed to obtain pressure and velocity contour. The maximum pressure is observed at the centre of the hub and at the edge of every blade. The maximum velocity is observed at the blade-hub junction. The results from the numerical simulations provided an insightful understanding of the fluid flow under given conditions.

The key outcomes of this study are as follows

1. The pressure contour has a maximum value of 105.7 Pa and minimum value is -81.58 Pa.
2. Velocity vector and velocity streamline helps understand the movement of the fluid flow in an enclosed cylinder.
3. The velocity in stationary frame for both vector and streamline has maximum value of 14.12 m/s.
4. The results obtained are compared with different number of blades with minor changes in the outcome [7].

REFERENCES

[1] S. Jain, and Y. Deshpande (2012), "CFD Modelling of a Radiator Axial Fan for Air Flow Distribution," World Academy of Science, Engineering and Technology, Vol: 6

[2] Mahajan Vandana N., Shekhawat Sanjay P.(2011), Axial Flow Fan using Ansys," Engineering and Technology, E- ISSN 0976

[3] Prof. Jigar S. Patel Prof. Shailesh M. Patel (2012), "Parameter Affecting the Performance of Axial Fan Performance," International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 3, ISSN: 2278-0181.

[4] Konrad Bamberger, Thomas Carolus, "Optimization of Axial Fans with Highly Swept Blades with Respect to Losses and Noise Reduction," University of Siegen, Paul-Bonatz-Strasse 9-11, 57223 Siegen, Germany.

[5] Park, J. (2005), "A Sound Method for Fan Modelling", Fluent News, US.

[6] A report on fan design by Moore, 800 s. Missouri avenue Marceline, Missouri.

[7] H. Kumawat (2014), "Modelling and Simulation of Axial Fan Using CFD," World Academy of Science, Engineering and Technology, Vol: 8, No: 11