International Journal of Advance Research in Science and Engineering
Vol. No.6, Issue No. 10, October 2017
www.ijarse.com
IJARSE
ISSN 2319 - 8354

CFD MODELING AND SIMULATION OF AXIAL FAN FOR APPLICATION IN AIR COOLED HEAT EXCHANGER

Sukesh Kumar[1], Dr.V.N Bartaria[2]

[1]*M.Tech Scholar, Dept. of M.E, LNCT, Bhopal (India)*

[2]*Prof & HOD(ME), Dept. of M.E, LNCT, Bhopal (India)*

ABSTRACT

The characteristics of heat transfer of air-cooled heat exchange depends upon the ability of the fan system to provide sufficient cooling air. Strength and Flow direction often subject peripheral fans to distorted inlet conditions at normal operating conditions with certain reduction in volumetric cooling and flow rate the capacity of cooling. This paper gives the design methodology for single-rotor axial flow fans, suitable for heat exchanger cooling system. The main aim of this work is to solve the issues of robust off-design performance, in particular, distorted inlet flow tolerance. Using this methodology, two 7-bladed and 11 bladed prototype fans were designed, tested and built in accordance with NACA 4609 standards. Axial fans flow handles large volumes of air at relatively low pressures. In general, the manufacturing cost is low and delivers good efficiency, and the shape of the blade is of aerofoil shape. Axial flow fans are operated on high static pressure if required and gives the good efficiency. The main theme is to analyze and model the flow characteristics of AXIAL FANS using CFD Software and draw inference from the obtained results. The performance simulation was estimated using CFD by varying parameters like the blade number, noise level, temperature, velocity, and pressure distribution on the blade surface. This paper aims to present a final 3D solid work model of axial flow fan. Adapting this model to the available components in the market, the first optimization was done. After this step, FLUENT solver is used to do the necessary numerical analyses on the aerodynamic performance of the model. The optimized final result is found which is presented in this article.

Keywords: ANSYS FLUENT Axial Fan, Computational Fluid Dynamics (CFD), Optimization.

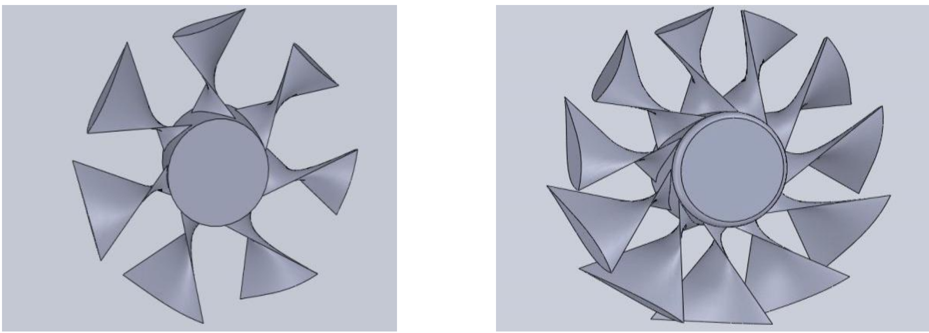
I. INTRODUCTION

The axial flow fan is used for various operations in engineering field. The variety of applications is seen, ranging from small cooling fans for electronics to the giant fans used in tunnel winds. Variety of air industrial and conditioning process application are solved using fans axial flow. The adaptability of the axial flow fan has resulted in implementation into large scale systems, from industrial dryers to in-cabin and locomotive cooling and recirculation systems [1], [2]. The performance of axial fans and their flow characteristics have been performed to quantify by Numerical investigations [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].With the expressive computer capability and extensive development in the simulation field, CFD have 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 [5]. CFD models, if created correctly, can account for the complex flows in equipment. To evaluate the flow behavior and characteristics of axial flow fan CFD models have been used. The models provide sufficiently accurate predictions over a range of operating conditions, which are not model, the flow passing through an axial fan. The main work of this dissertation is to optimized performance. Axial flow fans, while incapable of developing high pressures, they are well suitable for handling large volumes of air at relatively low pressures [6]. In general, they are low in possessing good efficiency and cost, can have blades of 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 tangential velocity and desired spaces and is an energy loss in axial fans or it can be converted into static pressure as in case of vane axial fans.

Fig. 1 Axial fan assembly

II. GEOMETRICAL MODELS

The first step is identify a typical axial flow fan that can be represented as a 3-D CAD Solid works® software engineering drawing package (Fig. 2). The 3-D models are then imported into the CFD software, remodeled into different sections, and refined to generate a finite volume meshing. This is a crucial step, where details of the geometrical shape need to be defined precisely. The flow domain is also created (Fig. 3), and the final meshing of all components needs to be accurate. Any errors in the drawings and flow area need to be corrected before continuing.



(A) Initial design **(B) Optimal design**

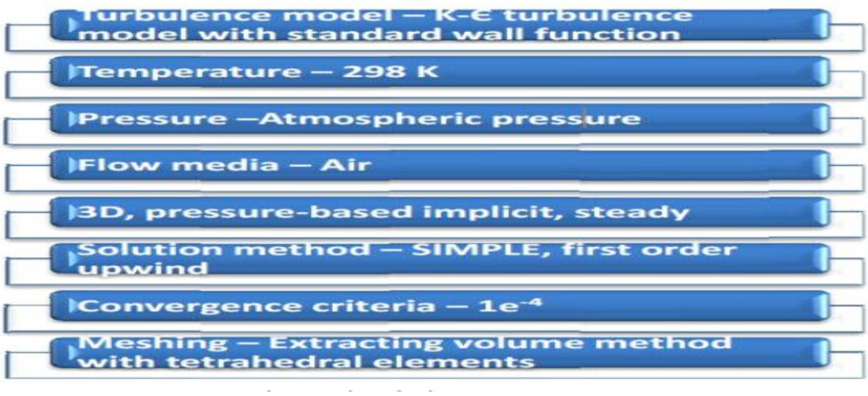
Fig. 2 Solidworks CAD model of fan

S.NO	SPECIFICATION	INITIALLY DESIGNED FAN	OPTIMISED DESIGNED FAN
1.	Blade root chord (Lr, mm)	18	18
2.	Blade end chord (Le, mm)	33	33
3.	Setting angle (θ,°)	47	47
4.	Number of blades	7	11
5.	Fan speed (ω, rpm)	2500	2500
6.	Hub diameter (DHub, mm)	40	40
7.	Fan diameter (D, mm)	110	110
8.	Hub height (Hu+Hl=H, mm)	15	15
9.	Fan gap (mm)	2	2
10	Air volume flow rate (Q, m/s)	22	22

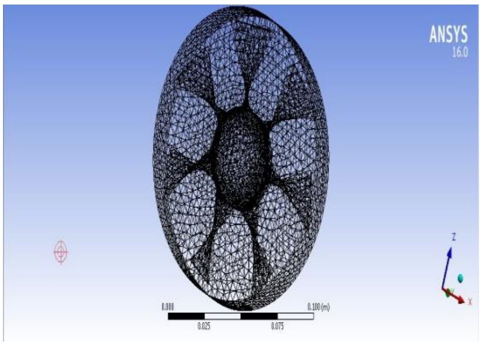
Table1: Fan Design Geometrical parameters

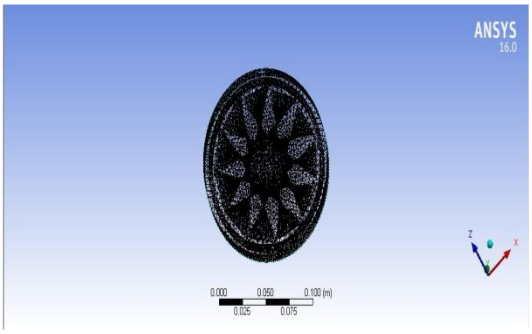
III. MESHING AND NUMERICAL SIMULATION

The second step is importing the files into the CFD code preprocessor, which will solve the flow equations. Here, the field flows boundary conditions are set. These include inlet air mass flow, outlet pressure, flow domain and fluid properties characterization, such as moving internal zone and stationary solid walls. The next step is to set the simulation process as a 3-D turbulent and steady problem (Fig. 4).The simulation 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 results of the fluid 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 standard atmospheric pressure, the boundary conditions include fixed wall, moving internal zone, zero pressure at outlet, and variable mass flow rate at inlet. The values of all residual variables solved are monitored during the iteration process. This iteration process needs to be monitored for convergence and repeated if the numerical error conditions are not satisfied. Finally it is to analyze the output data and present them in the form of velocity streamline (Fig.5) and contour plots (Fig.6).



IJARSE
ISSN 2319 - 8354





(A) Grid of 7 blade **(B) Grid of 11 blade**

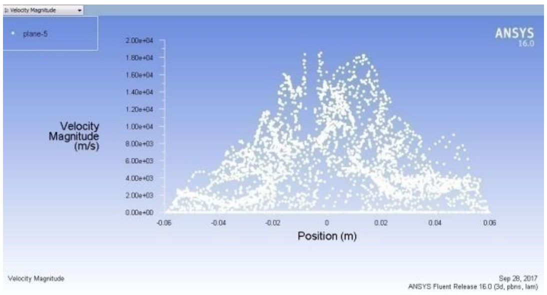
Figure 4. Computational grid

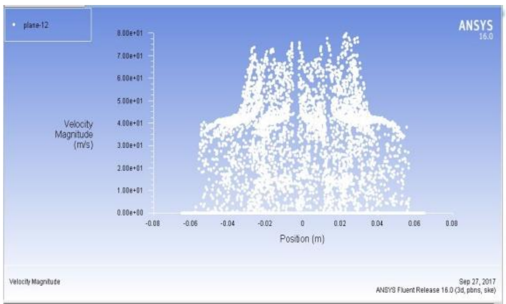
In this paper, the mass flow input is given as an input limit condition, while the output condition is a reliable output. The solid walls, such as the blade surfaces and the hub, satisfy the non-slip state in the design domain. The finite volume method is performed in numerical calculation. It is assumed that the flow of the turbine is incompressible and invisible. The stable modeling for both models is performed using RNG k-ε. Meanwhile, the second-order reversal difference scheme is adopted as a numerical discretization method of the control equation. The residuals are equal to or less than the specified norm (10^{-3}) and the relative error is 0.5%, then the calculation is convergence.

IV. RESULTS AND DISCUSSION

4.1 Velocity Characteristics of Models

Static characteristics are an important factor in analyzing the performance of a small axial fan. Meanwhile, the static characteristics are reflected in the performance curves P-Q and η-Q.



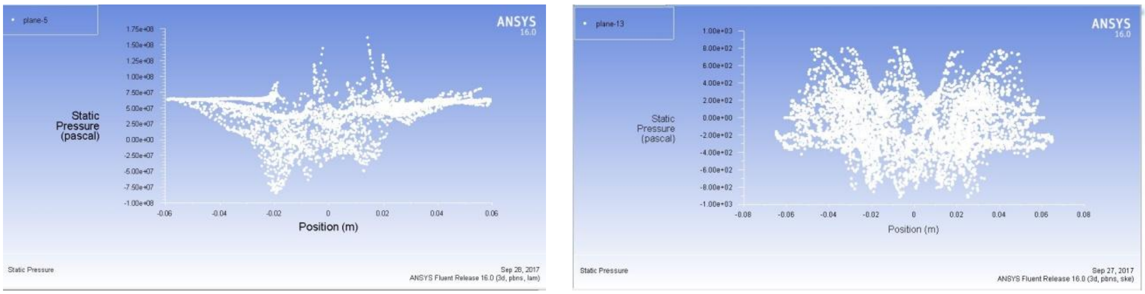


(A) Initial design **(B) Optimal design**

Fig. 5 Velocity magnitude along the radius

From the above figure 5 (A) and (B) it is concluded that the velocity magnitude is maximum at the middle portion of the fan and decreases radially and the optimal blade design have good fan characteristic.

4.2 Static Characteristics of Models



(A) Initial design **(B) Optimal design**

Fig. 6 Static pressure along the radius

Fig.6 (A) and (B) represents the static pressure along the radius of the initially and optimal design. The abscissa is radius position in this fig. After analysis, it can be seen that the pressure at the outlet is higher than the inlet position, and the maximum pressure of the 11 blade fan is higher than that of the 7 blade fan in the rotating fluid region. From observation it also concluded that the static pressure rise is higher in 11 blades than the 7 blade fan.

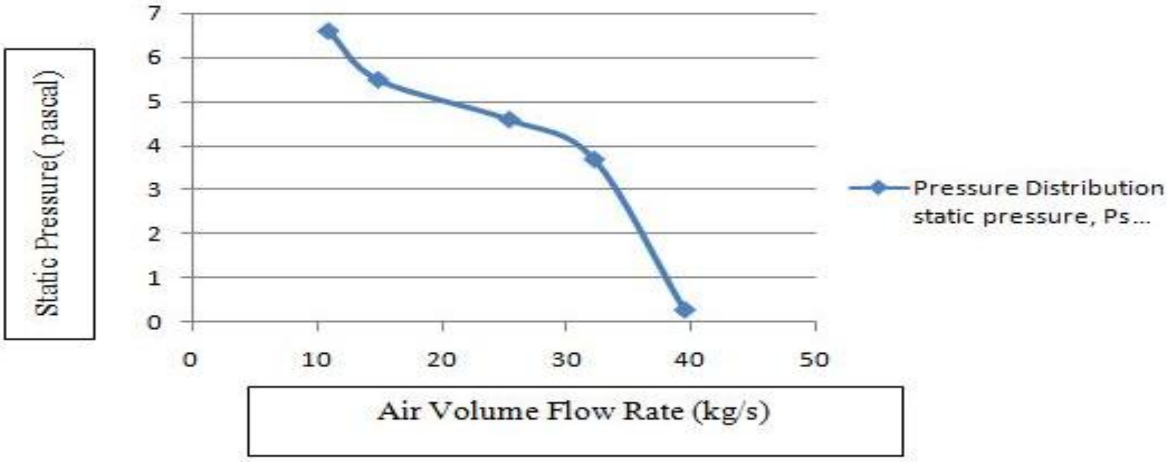


Figure 7(A): static pressure v/s air flow rate of initial design

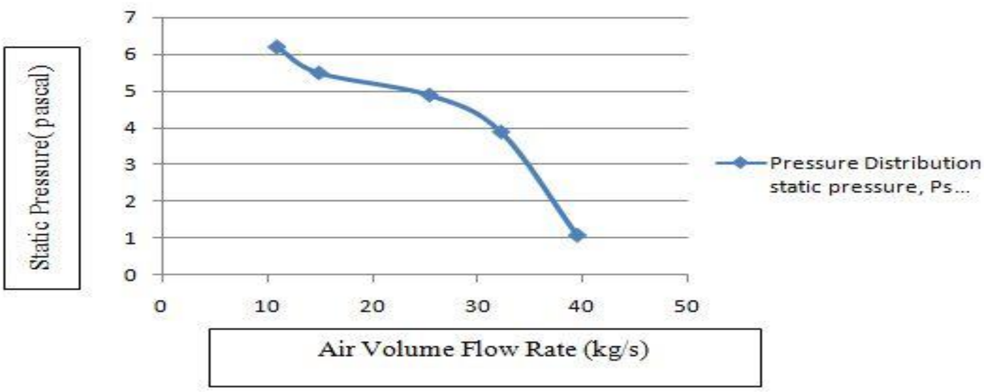


Figure 7(B): static pressure v/s air flow rate of optimal design

A comparison of the variable 7 and 11 blade fan characteristics can be seen from the fig. 7 (A) and fig. 7 (B). From these above two graphs it can be seen that the static pressure at the inlet and outlet of the two designed models decreases with the increase of inlet flow rate Q. From the figure 27 shows the sudden decrease of pressure in comparison with the figure number 7 of 11 number of blade model of optimal design which causes the noisy condition of the fan. From above graph we concluded that the 11 blade fan have the better design performances.

V. CONCLUSION

Considering new trends of special field of rotating machinery the aim of my thesis is creating a calculating model and its numerical investigation which takes the 3D and the real hydraulically efficiency into consideration during the preliminary design. Being able to numerically simulate any physical process or operation accurately holds many obvious advantages. Numerical simulation is carried out for the blade having 11 and 7 in number using ANSYS Fluent. Once the method is established importing the numerical models into ANSYS design modeler and the seamless integration with ANSYS turbo grid resulted in accelerated set-up times for simulations. ANSYS turbo grid was found to simplify and streamline the mesh generation process of the fan passage. However, ANSYS turbo grid has certain limitations with regards to the inlet and outlet's length and size. Therefore, the larger domains if required it had to be created in ANSYS Meshing, ANSYS's general meshing software package. These meshes consisted of three parts, an inlet, passage (rotor) and outlet. The inlet and outlet domains are circular sections.

Finally on the basis of it can be concluded that the results obtained using 11 blades give better results. And it can be used in air cooled heat exchanger for better future results.

REFRENCES

1. A report on fan design by Moore, 800 s. Missouri avenue Marceline, Missouri.
2. A report on axial flow fan and centrifugal fan by marathon electrical, a regal Beloit company.
3. A research report on blades for axial flow fan by Camargo Do, Amaranth Odilon Antonio (Rua Fernandes de Barros, 1455 apto. 07 -200 Curitiba, PR, CEP-80040, BR).
4. A research report on basic guidelines for plastic conversion of metal axial flow fans by BASF Corporation.
5. A Report on Basic Fan Laws - Axial Fan Blades by Air Turbine Propeller Company
6. Energy Efficient Axial Flow FRP Fans by TIFAC Department of science and technology, govt. of India.
7. CFD solutions for turbo machinery design.
8. CAD/CAM & Automation, Technical Prakashan Pune, 2003, (pp 4.143-4.144) by R.B. Patil and A.M. Chakradeo (2003)
9. Mechanical Engineering Designs Second Edition McGraw-Hill Book Company, 1987, Joseph Edward Shingle.