International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 664-671 www.ijaem.net ISSN: 2395-5252

Research Paper on Evaluation of Ceiling Fan Blade Angle Performance Using CFD

Akash Ramchandra Khedekar[1], Prof. K.R. Sontakke[2], Prof.F.R.Sheikh
Department of Mechanical Engineering, PG Students

Submitted: 25-05-2021 Revised: 01-06-2021 Accepted: 05-06-2021

ABSTRACT- This analysis have been conducted to analyze the different blade angles by computational fluid dynamics (CFD) in ANSYS software to find the maximum air delivery. By finding maximum air delivery with their blade angle the optimum design is carried out by comparison of energy consumption i.e. power with different number of blades of ceiling fan. The experiments were conducted based on three different number of blades, having different blade angles. With constant speed and blade length and mathematical model was developed based on this analysis optimum design is achieved.

.**Key words:** Ceiling fan, blade angle, air delivery, CFD.

I. INTRODUCTION

In the early days of aeronautics the flight of birds has stimulated scientists and engineers in the research and development of aircraft. Much has been learnt from nature and pioneers in the aeronautical field have studied the flight of birds. Birds move from one place to another by driving their wings in air. The motions of the bird in the air consist of flapping flight as well as gliding and soaring flight. Investigations on the aerodynamic characteristics of birds have been conducted for a variety of purposes. For example, an issue is formation flight of the birds as an energy saving mechanism during migration. Other investigations deal with aerodynamic characteristics of bird wings in steady flow conditions. In this simulation, the bird configuration is modelled at a fixed position and the flow does not change with time. However, there are also unsteady flight conditions, like yawing and rolling motions. Such movements produce a response of the airflow at the wing changing with time. This unsteady airflow affects the lateral-directional stability characteristics of birds.

II CAD, CFD THEORY AND WORKING

COMPUTER AIDED DESIGN-

- **CREO Introduction**

CAD technology is very important while designing any Product.

Following are advantages of CAD technology:

- **To increase the productivity of the designer**
  - Helping designer to conceptualize the product
  - Reducing time required to design and analyze
- **To improve the quality of the design**
  - Allows the engineer to do a more complete engineering analysis and to consider a variety of design alternatives, therefore increasing quality.

Creo is a parametric, feature based, solid modeling System. It is the only menu driven higher end software. Creo provides mechanical engineers with an approach to mechanical design automation based on solid modeling technology and the following features.

Normal fan blade @ 10 deg

Fan blade wireframe model

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 664-671 www.ijaem.net ISSN: 2395-5252

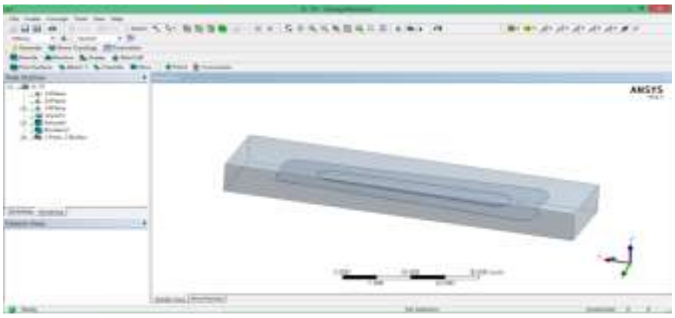

Fig : Fan blade wireframe model

Normal fan blade @ 11 deg
Fan blade wireframe model

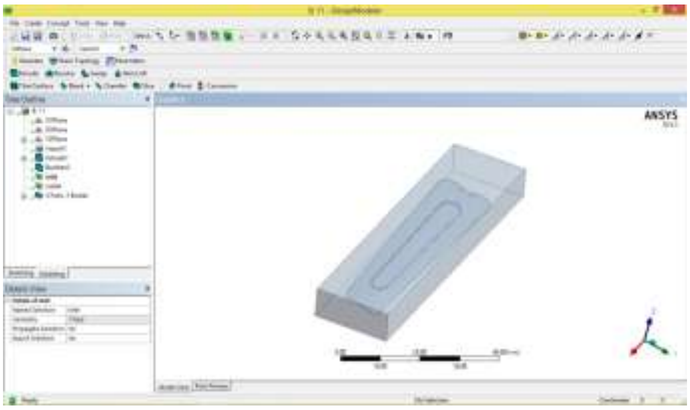

Fig : Fan blade wireframe model

Normal fan blade @ 12 deg
Fan blade wireframe model

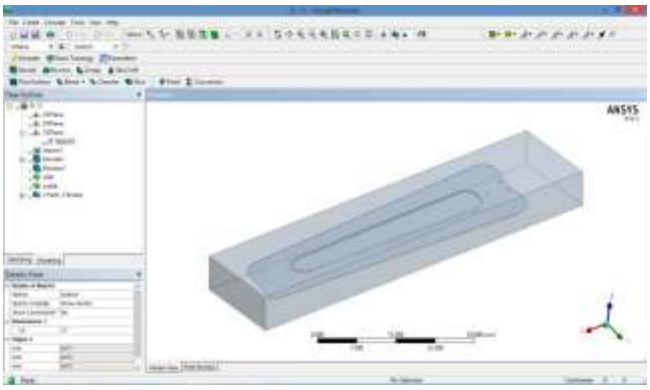

Fig : Fan blade wireframe model

Wing 1 @ 10 degree
Fan blade wireframe model

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 664-671 www.ijaem.net ISSN: 2395-5252

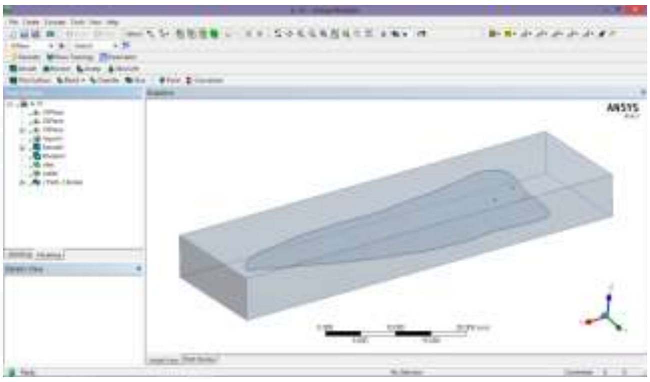

Fig : Fan blade wireframe model

Wing 1 @ 11 degree
Fan blade wireframe model

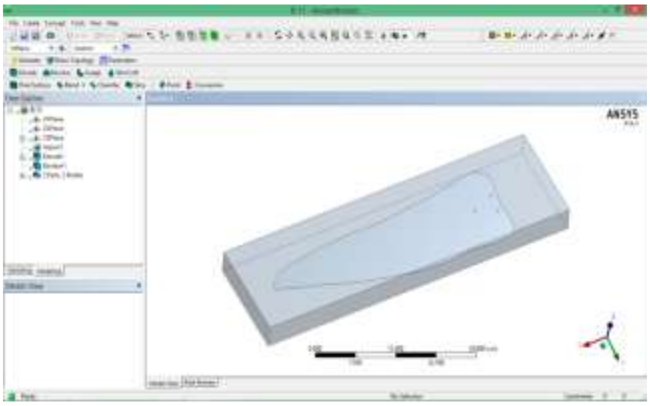

Fig : Fan blade wireframe model

Wing 1 @ 12 degree
Fan blade wireframe model

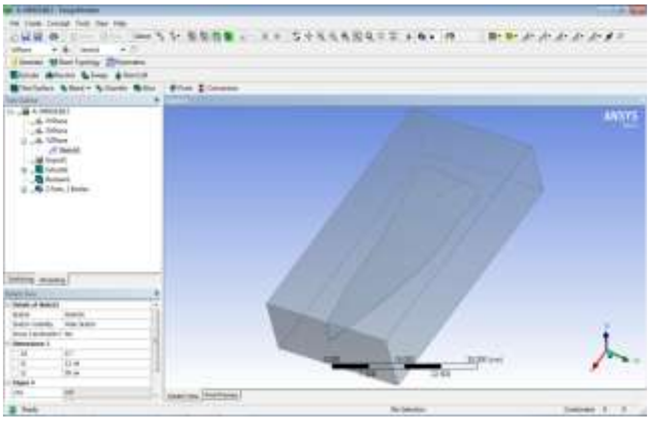

Fig 3.7 : Fan blade wireframe model

Wing 2 @ 10 deg
Fan blade Wireframe model

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 664-671 www.ijaem.net ISSN: 2395-5252

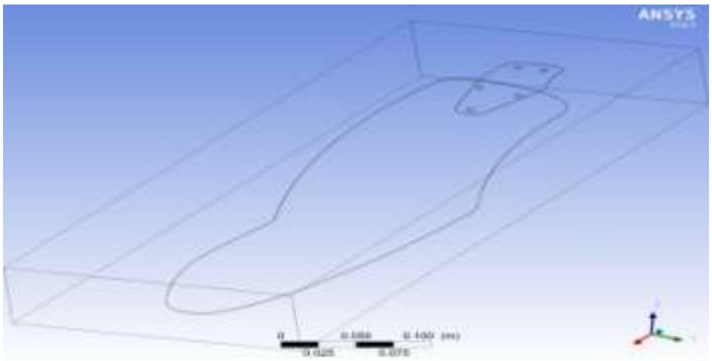

Fig : Fan blade Wireframe model

Wing 2 @ 11 deg
Fan blade Wireframe model

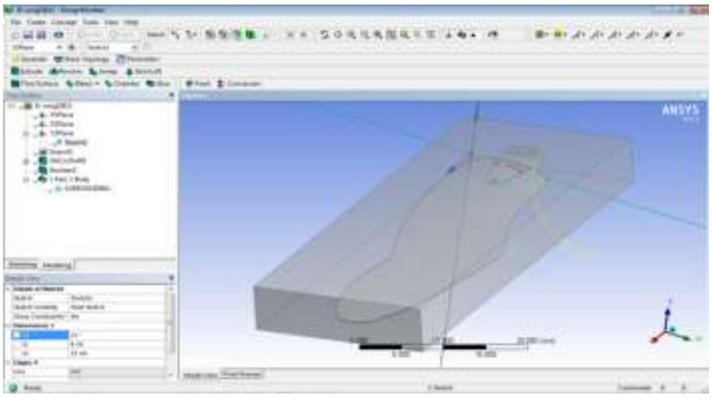

Fig : Fan blade Wireframe model

Wing 2 @ 12 deg
Fan blade Wireframe model

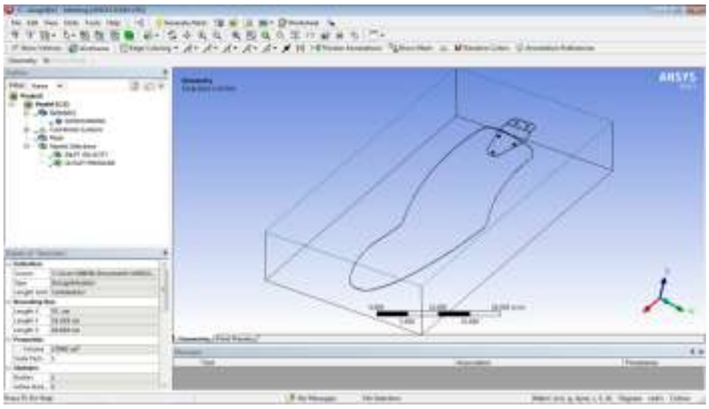

Fig : Fan blade Wireframe model

COMPUTATIONAL FLUID DYNAMICS-
➢ INTRODUCTION

The need to control and predict the movement of fluids is a common problem. The study of this area is called fluid dynamics and the systems that are studied range from global weather patterns, through aircraft aerodynamics to the way blood circulates. Computational Fluid Dynamics (CFD) takes these problems and solves them using a computer.

CFD and its application is a rapidly developing discipline due to the continuous development in the capabilities of commercial software and the growth of computer power. CFD is already widely used in industry and its application is set to spread. This guide aims to provide an introduction to CFD and an overview of current software and techniques, including ways in which smart IT based methods can increase productivity

Start Project

Start ANSYS Workbench by clicking the Windows Start menu, then selecting the Workbench 14.5 option in the ANSYS 14.5 program group.

Start → All Programs → ANSYS 14.5 → Workbench 14.5

Create a new FLUENT fluid flow analysis system by double-clicking the Fluid Flow (FLUENT) option under Analysis Systems in the Toolbox.

Name the analysis by Double-click the Fluid Flow (FLUENT) label underneath the analysis system

Import Geometry

The geometry of the structure can be imported from any external design software like AutoCAD, Creo or simply created with ANSYS-Design Modeler. By right clicking on 'Geometry' in the ANSYS workbench project, the ANSYS-Design Modeler can be initiated, as shown in Figure 4. The interface of ANSYS Design Modeler is shown in Figure . Generation of geometry can be carried out by creating points, edges, faces and volumes of the geometry in three dimensions. After creating/importing the geometry of the structure into the Design Modeler, the fluid flow regions (liquid) and solids regions (solid) of the structure have to be defined. The geometry needs to be divided into separate regions in order to apply constraints for the resulting mesh. Boundary regions are need to be specified for finer or denser meshing.

Apply Boundary Condition

Create named selections for the geometry boundaries. In order to simplify your work later on in ANSYS FLUENT, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by ANSYS FLUENT).

Generate Mesh

The Meshing tool can be invoked by right

Normal fan blade @ 10de

Velocity Streamline

clicking on 'Mesh' at the workbench project and selecting 'Edit', similar to ANSYS-Design Modeler. The home interface of ANSYS Meshing tool is shown in Figure. For creating the mesh, the 'Objects' have to be identified. An object is generally a set of face zones and edge zones. Objects are generally closed solid volumes or closed fluid volumes. Different types of mesh can be created in ANSYS Meshing tool through 'Insert' option. The mesh settings can be adjusted according to the model requirement

Pre-Process

After the mesh generation, the geometry should be exported as a mesh file for use in ANSYS Fluent. The ANSYS Fluent can be launched from the workbench either by double clicking or right clicking and selecting 'Edit' in the 'Setup' menu.

Apply Material

After selecting the appropriate model for the problem, the materials involved in problem have to be defined. ANSYS Fluent has its own database of different "materials" with their properties.

Run Calculation

Finally, the number of iterations and reporting intervals should be set before starting the calculations.

Results and Post Processing

ANSYS has separate tool for analyzing and post processing the results. User can also do limited processing within the ANSYS Fluent itself. Using the post processing tool, iso-surfaces of grid are created studying the flow features such as velocity, pressure and air profiles in longitudinal direction as well as across the depth. Contours and XY Plots of different quantities such as pressure, velocity and phases can also be generated. Capture Result

III RESULTS

The analysis was done for $\Theta = 10^0, 11^0, 12^0$, where Θ is the angles of fan blade with horizontal;.

International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 6 June 2021, pp: 664-671 www.ijaem.net ISSN: 2395-5252

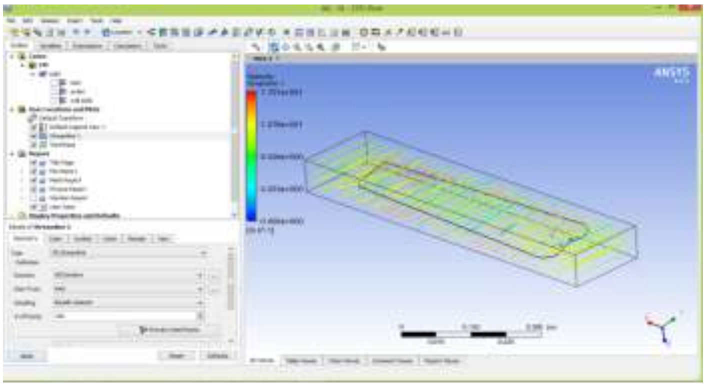

Fig : Velocity streamline

Pressure counter

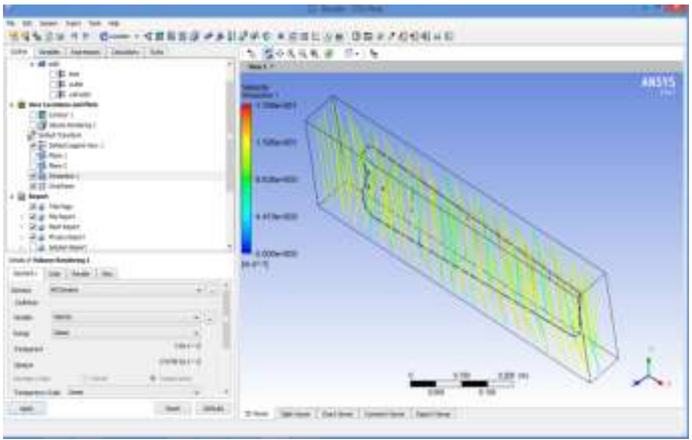

Fig : Pressure counter

Normal fan blade @ 11deg

Pressure Counter

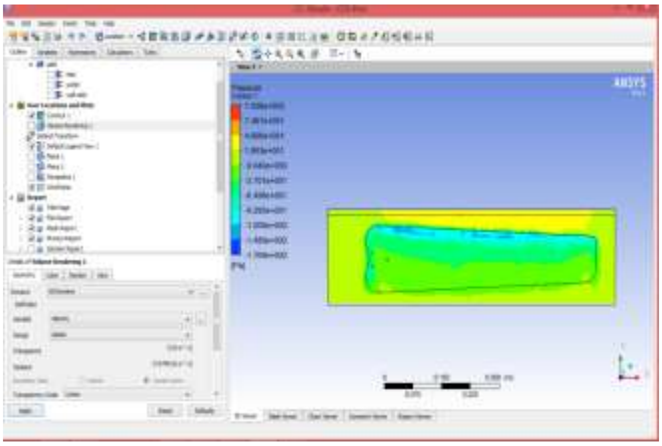

Fig 4: Pressure counter

Velocity

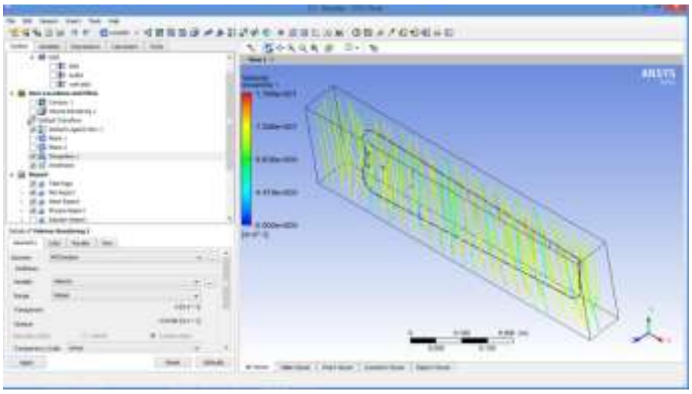

Fig : Velocity

Normal fan blade @ 12 deg

Velocity Streamline

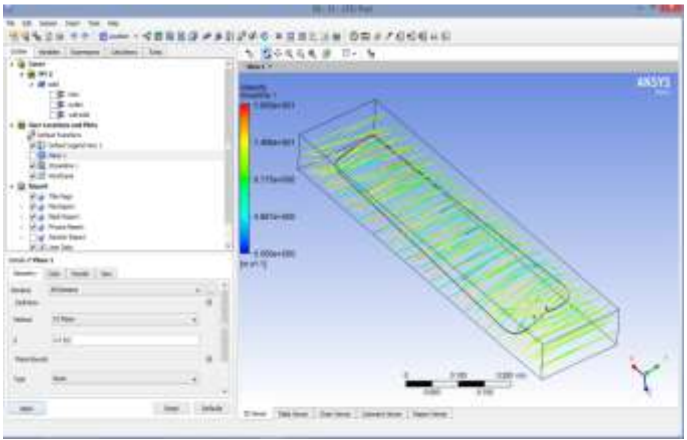

Fig : Velocity streamline

Pressure Counter

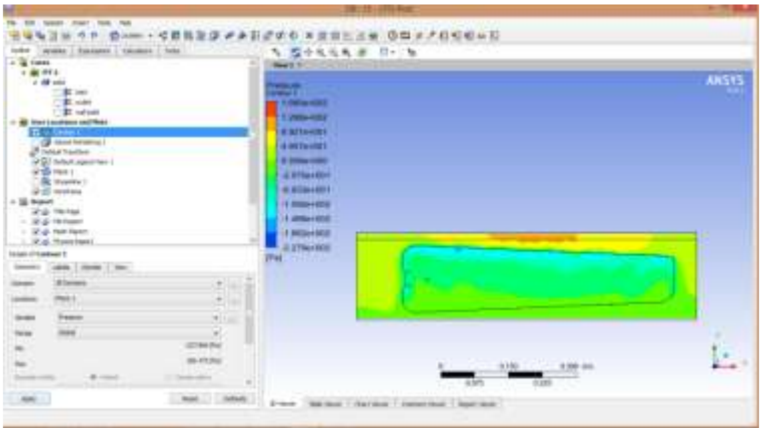

Fig : Pressure counter

Results

Table : Results for Normal fan blade

Sr. No.	Blade angle	Velocity (m/s)max	Pressure(Pa)

1	10°	17.01	109.95
2	11°	17.68	102.8
3	12°	19.55	168.5

Average Velocity for Normal Blade- 18.08 m/s
Average Pressure for Normal Blade- 127.08 m/s^2

IV CONCLUSION

From the above we have studied about Evaluation of Ceiling Fan Blade Angle Performance Using CFD by considering different fan blade design at different angle and we conclude from our analysis that wing 2 has more speed at various angles we used.

We have taken different readings of Velocity and Pressure for different fan blade angles and we can say that Wing 2 has more average velocity and less average pressure. So,It is more efficient. Specially, Wing 1 with 11 deg blade angle gives highest velocity i.e. 21.71 m/s.

This Analysis have been conducted to analyse the different blade angle by Computational Fluid Dynamics (CFD) in ANSYS software to finding the maximum air delivery. By finding maximum air delivery with their blade angle the optimum design is carried out by comparison of energy consumption i.e. power with different number of blades of ceiling fan. The experiments were conducted based on three different number of blades, have different blade angles, with constant speed and blade length and mathematical model was developed. Based on this analysis optimum design is achieved

REFERENCES

[1]. Afaq, M. A., A. Maqsood, K. Parvez and A. Mushtaq(2014). Study on the design improvement of an indoor ceiling fan. 11th International Bhurban Conference on Applied Sciences and Technology, IEEE.
[2]. International Journal of Emerging Technology and Advanced Engineering 4(6), 247-251.
[3]. Fundamentals of Compressible Flow with Aircraft and Rocket propulsion by S.M. Yahaya
[4]. Auto Design by R.B. Gupta
[5]. ANSYS. (2013). ANSYS Fluent Theory Guide Canonsburg, PA.
[6]. ANSYS. (2016). ANSYS® Academic Research, Release 16.0, ANSYS Fluent in ANSYS WorkbenchUser's Guide. ANSYS, Inc., Canonsburg, PA.
[7]. Gambit. GAMBIT 5 User Guide, 1999.
[8]. Parker, D. S., M. P. Challahan, J. K. Sonne, G. H. Su and B. D. Hibbs (2000). Development of a High Efficiency Ceiling Fan. Improving Building Systems in Hot and Humid Climates, Texas A&M University.
[9]. Efficiency improvement opportunities for ceiling fans Nihar Shah & Nakul Sathaye & Amol Phadke & Virginie Letschert
[10]. Technical Data Book, Prepared by Gulf Research and Development Company, Pittsburgh, PA, 1962.
[11]. REDESIGN OF CEILING FAN adapted to the Scandinavian market by Anna Eliasson & Martina Westman ● 07 05 14