www.ijiarec.com

ISSN:2348-2079

Volume-5 Issue-2

International Journal of Intellectual Advancements and Research in Engineering Computations

Modeling and performance investigation of axial flow type propeller fan

[1]Praveenkumar.S, [2]Ragavan.G, [2]Rithick.R.S, [2]Sridhar.V, [2]Sudalaisuresh.M
[1]Assistant Professor, [2]UG Students,
Department of Mechanical Engineering, Shree Venkateshwara Hi-tech Engineering College, Gobi, Tamil Nadu, India.
sivapraveen09@gmail.com, kavinks66@gmail.com

***Abstract* - The current work agreements with axial flow fans that are mainly used for as extended as the mandatory airflow for heat and mass transfer courses popular the various industrial challenge and developments. It canister be recycled for freezing, aeration or for drying drives. These comprise cooling barbicans for air-conditioning and freshening, humidifiers in textile mills, air heat exchangers for several chemical processes, ventilation and exhaust as in removal business. In observation of all these application several efforts in the past need existed done to evaluate the performance of axial flow parties. In the existing work computational analysis of axial flow backer with Accelerative and Reversing Tilted blade profile carried out using ANSYS CFX software and the outcomes are linked with the untried results from poetry. The CFD examination is far-reaching by showing the axial fan in SOLIDWORKS 2013 and using Regular *k-ϵ* model with the Normal wall purpose for showing turbulence. The search is carried out with blade stagger angle of 60^0 and 80^0, and at 1500rpm. The aim is to study fan with these two types of side view and equate the static pressure, flow rate, Velocity and weight feature shaped by fan. After resounding out computational autopsy it is found that computational results holds good treaty with the trial grades taken from the poetry.**

Keywords: airflow, fan, modeling, analysis.

I. INTRODUCTION

A fan is a engine used to make flow inside a liquid, naturally a gas such as air. The fan involves of a rotating planning of blades or cutting edge which act on the fluid. The gyratory meeting of blades and hub is known as an impeller, a rotor, or a runner. Commonly, it is confined within some form of protector or case. This may shortest the airflow or increase protection by checking objects from give somebody a bell the fan knife-edges.

Most fans are motorised by electric motors, but other sources of authority may be used, including hydraulic motors and internal combustion machines. Fans produce flows with high volume and low compression (although higher than ambient pressure), as opposed to compressors which produce high pressures at a reasonably low volume.

A fan blade drive often rotate when exposed to a fluid watercourse, and devices that take advantage of this, such as anemometers and wind turbines, regularly have designs similar to that of a fan. Typical applications include climate switch and personal thermal comfort (e.g., an electric table or floor fan), automobile engine cooling systems (e.g., in front of a radiator), machinery refrigeration systems (e.g., inside computers and auditory power amplifiers), ventilation, fume extraction, sorting (e.g., separating chaff of cereal grains), confiscating dust (e.g. in a vacuum cleaner), drying (usually in amalgamation with a heat source) and to provide draft for a fire.

While fans are habitually used to cool people, they do not actually cool air (if anything, power-driven fans warm it slightly due to the baking of their motors), but work by evaporative cooling of wetness and increased heat convection into the surrounding air due to the airflow from the fans. Thus, fans may become futile at cooling the body if the immediate air is near body disease and comprehends high humidity. During stages of very high heat and sweatiness, governments actually instruct against the use of fans.

II. LITERATURE REVIEW

Young-Kyun Kim, Three dimensional flow distinctive was present by a revision of an impeller of an axial turbo fan for refining the air flow rate and the static pressure. To consider an incompressible steady three dimensional flow calculations are used as the overriding equation and the normal turbulence picture-perfect is elected. The pitch angle of 440,540,590 and 640 are grasped for the numerical model. The mathematical results displays the air flow rates of each pitch angle the alteration of the static pressure at impeller inlet and outlet are 120Pa, 214Pa, 242Pa, and 60Pa according to particular pitch angles. It means that the 590 of the impeller pitch is greatest to improve the airflow rate and the static pressure [1].MahajanVandana**,** The CFD based enquiry has been stated in order to study the effect of change in speed of fan on velocity, heaviness, and mass flow rate of axial flow fan. It has been experiential that there is a important change in mass flow rate, speed of rotor and guide or stator vanes as the speed of the fan is wide-ranging. As the performance of the fan is directly in need of on mass flow output. So there would be a modest velocity and substance profile as all these strictures are linked .In command to expect the mass flow output, velocity and pressure on stator then rotor section [2].

Dunn (1983) done experimental analysis of Reverse Act Features of main source fans. During a mine fire or other reserve, it may be looked-for to other the airflow in order to provide an escape way or to isolate a fire. Also, in colder areas, the airflow may be reversed to avoid ice build-up. When reversing main mine fans, the source operator usually does not know what operating characteristics of flow and pressure to expect [3].Yang et.al (2007) wilful the flow mechanism analysis and new study of a forward-skewed impeller and a centrifugal impeller in low force axial fan.The forward-skewed cutting edge was obtained by the optimization design of the radial blade and CFD technique. Depth of the two blades was carried out in aerodynamic and aero sound performance. Related to the circular blade, the forward-skewed blade has confirmed the developments in efficiency [4].The performance of the fan are governed by different fan laws. Hence lot of importance is given to understand the basic theory of fans, their types and their occupied. The choice of critical limitations is very essential while determining the performance of the fans [5].AlirezaFalahat (2011) In this study an attempt was made to find the greatest the best angle of attack and rotational velocity of a flat plate at a fixed hub to tip ratio for a maximum flow magnitude and turbulent conditions. The blade angles are varied from 30 to 70 degrees and the revolving rapidity is changed from 50 to 200 rad/sec for a figure of blades from 2 to 6, at a fixed hub to tip part. The arithmetic and experimental results show that, the maximum flow figure 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 geometric results show that as the rotational velocity increased, the flow constant increased but then at very high rotational velocities the flow number remained constant. We have identified the following difficulties: The velocity drop across inlet to exit plays an significant role in maintaining flow rate. Due to swiftness drop, the impact force and flow rate of air enters gets summary. In this regards, we start our project work to bargain out a solution to the above listed problem [6].

III. MODELLING COMMANDS

A. SKETCH

This knack is used to create the new sketch like circle, line, rectangle, ellipse, etc,.The Solid Works window selects the sketch image and select the plane or surface want to sketch.This command is rummage-sale to create the circle. Create circle by picking the center point and a point on the circle from Right Toolchest. Pick the origin for the circle's center - pick a point on the circle's edge- click the middle mouse switch ok. This command is used to create the ellipse. Create ellipse by preference the center point and a minor radius point and major circle point, the minor and major radius of the ellipse is vertical and plane direction depend upon the shape of ellipse we want. Excellent the ellipse icon from right toolchest- Pick the center for the ellipse – pick the minor radius of ellipse point and pick the major radius of the ellipse- click the middle mouse badge – ok. This command is used to create the line. Produce the line by start point and end point. Select the line icon from the precise Toolchest – click the start point of the line – get on the end point of the circle - enter

This grasp is used to create the arc. Create the arc by using three points. The three points are start point, end point and center point of the curve. Select the arc icon from the right Toolchest – click the start refrain of the arc – click the end point of the arc and click the middle point of the arc –enter. The dimension of the arc is revised by double click on the arc then the dimension will give the impression in the pop up box, then provide the charge of the arc.The hexagon is created by insert foreign data icon in the Right Toolchest. Insert foreign data from

Palette interested in active object - scroll down to see the hexagon -double-click hexagon- Place the hexagon on the sketch by picking a position - with the left mouse button**,** drag and drop the center of the hexagon at the foundation - modify Scale value to the required size – click enter

This command is used to create the rectangle and square. Click the frame icon in the right Toolchest – click the lower left point of the rectangle and developed right corner of the rectangle we famine to appeal. After drawing the rectangle the dimension of the border is provided by the pick the dimension command from the dimension icon in the right toolchest of the Solid Works software.This command is used to provide the width of the sketched entities the entities are circle, line, square, parabola, spline, rectangle, ellipse, etc,.. The aspect is offer to the sketch by select the facet icon from the right tool chest then select the sketched entities and press the central mouse button to finish the dimensioning.

To change the dimensions of the drawn entities by just double click the dimension line of formed sketch.This command is used to create the material (to make 3D object from 2D sketch) from the sketched entities. The entities may be circle, line or rectangle, etc, Select the extrude icon from the right toolchest then select the outlined part in the window, enter the extrude size and press the middle mouse button to finish the extrude command.This command is used to create the material from taking the one axis and sketched entities. The axis is the center of the revolved part. 0 degree to 360 is the spin angle degree. Choose the revolve icon from the right tool ribs then select the sketched part and axis of the piece in the graphical window, enter the revolve angle and press the middle mouse button to finish the extrude knack.

B. SWEEP FEATURES

The Sweep option extrudes a section along a defined trajectory. The order of operation is to first create a trajectory and then a section. A course is a path sideways which a section is swept.

The trajectory for a sweep feature jerry can be sketched or selected. The Bend option of protrusion is similar to the Extrude option. The only variance is that in the case of the Extrude option, the feature is extruded in a direction normal to the sketching plane, then again in the case of the Sweep option, the section is swept along the sketched or certain trajectory.

The trajectory can be open or closed. Standard sketching tools are used aimed at drawing the trajectory. The cross-section of the swept feature remains continuous throughout the sweep. To create a Sweep Cut feature, the technique to be followed is the same as that in Sweep Protrusion.

The Cut option can be invoked in choosing Insert > Sweep > Cut from the menu bar. A cut can be a solid swept cut or a thin swept cut.

The loft feature is an chief tool for surface modeling. The loft feature creates a shape by making transitions between multiple profiles and guide curves. This tool is very valuable for organic surface modeling.

Splines are clear both with fit ideas, or with control vertices. By default, fit ideas be in accord with the tooth, while control vertices define a regulator frame. Control frames provide a convenient system to shape the spline.

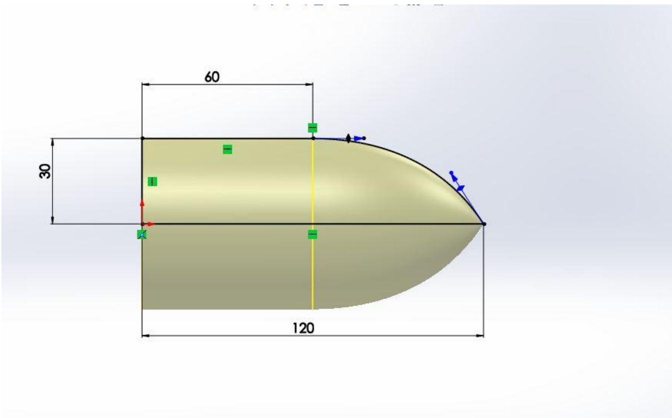

Figure No.1 Modeling Stage 1

The beginning sketch is complete with the help of line and Spline commands. The above sketch is for the boss where the blades are riding.

The Revolve command is used to make hub with the help of previous sketch. Create a new plane parallel to Top plane in order to draw sketch to create blade as exposed in below figure.2.

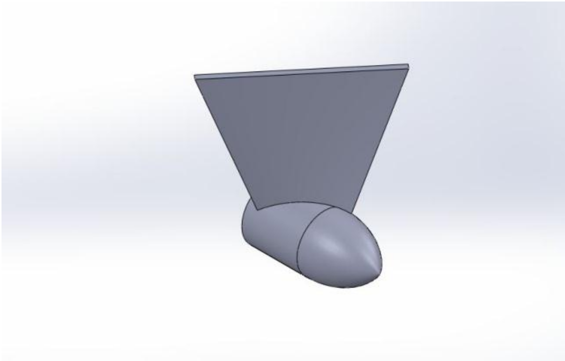

Figure No.2 Modeling Stage 2

With the help of fillet grasp, the sharp edges and corners of the blade are made as circular fillet in order to avoid design and damage to the blades.
There is an selection to make the similar parts in circular orbit called Circular pattern that helped to bourgeon the number wings to five.

The 3D model of the axial flow propeller fan is converted as IGES format over and done with the SOLID WORKS software. The IGES (International Graphic Exchange Specification) format is appropriate to significance in the ANSYS Workbench for evaluating.

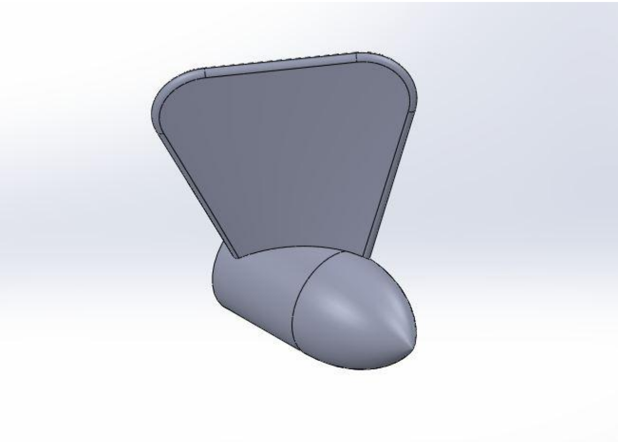

Fig.3 Modeling Stage 3

Open the ANSYS Workbench then create a new project on FLUID FLOW (CFX) by arranged create new geometry and importing the file to engender and give the shape with radius of 50mm and size of 240mm.

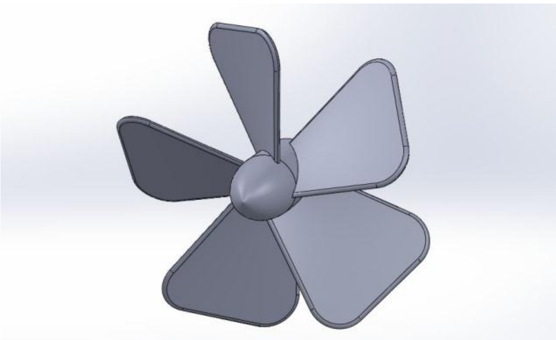

Fig.4 Modeling Stage 4

IV. FLOW ANALYSIS

A. MESHING

By giving the succeeding commands to the meshing stage the model has to be perfectly meshed for the flow analysis. Open Mesh, defaults at that time physical preference to CFD, defaults then solver liking to CFX, Advanced then relevance centre to fine, Advanced then group size to default, Right click the mesh in the tree view create mesh.

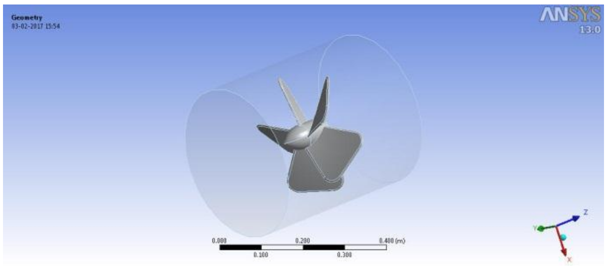

Fig.5 Propeller Fan with Envelope

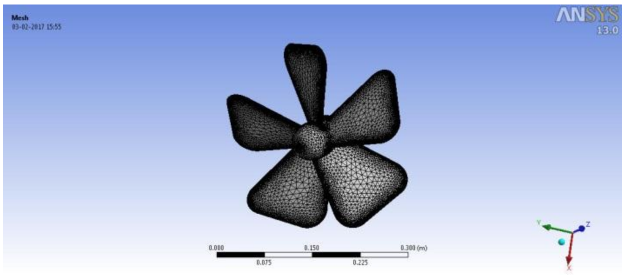

Fig.6 Propeller Fan after Meshing

B. INPUT CONDITION SETUP

The following directions describes the input situations of the input which is given to the model Open Setup of CFX, Import the mesh case, Twin click on the Default domain, Select place, Select “Fluid domain” as Area type & select material as “Air at 25^{o}c”, Create a new frontier and named as “Inlet”, Select Boundary type as “Inlet” and select the location of inlet piece under Basic setting TabClick on Edge details tab, Select “Subsonic” as Flow regime and click on possibility drop down list box to select “Mass Flow Rate” and then enter value of mass flow rate as 50 Kg/s, Click on Apply and OK.

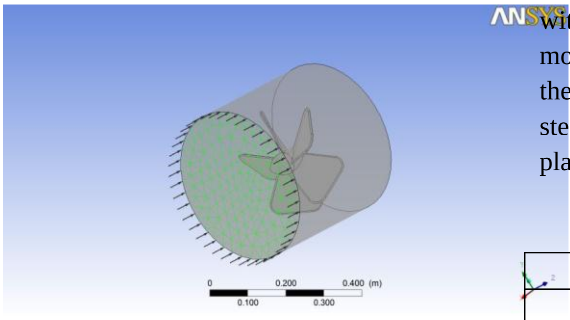

Fig.7 Propeller Fan with Inlet Condition

C. OUTPUT CONDITION SETUP

The next commands are used to set the boundary situations of the output of the model. Create a new boundary and named as “outlet”, select boundary type as “outlet” and select the place of outlet section under basic setting tab, click on borderline details tab, select “subsonic” as flow administration and click on option drop down list box to select “mass flow rate” and then enter value of mass flow rate as 50 kg/s, click on apply and ok.

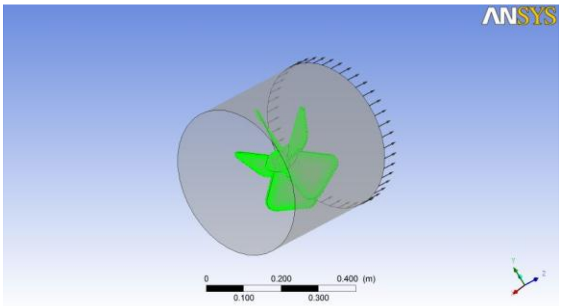

Figure No.8Propeller Fan with Outlet Condition

V. MATERIAL PROPERTIES

A. STEEL

Steel is an alloy of iron and other elements, primarily carbon, which is widely used in construction and other bids because of its high ductile strength and low cost. Steel's base metal is iron, which rests able to take on two crystalline forms (allotropic forms), body focused cubic and face centered cubic (FCC), dependent on its temperature. Carbon, other elements, and enclosures within iron act as hardening agents that foil the movement of dislocations that otherwise occur in the rock frames of iron atoms.The carbon content of steel is between 0.002% and 2.1% by weight used for plain iron–carbon alloys.

Table No.1 Material Properties

DESCRIPTION	VALUE
Density (Kg/m^3)	7800
Young’s Modulus (GPa)	200
Poison’s Ratio	0.26
Bulk Modulus (GPa)	140
Shear Modulus (GPa)	75

VI. RESULTS AND DISCUSSION

A. FLOW ANALYSISRESULT FOR 60^0 FAN DESIGN

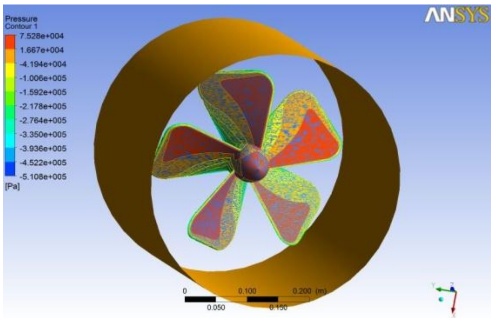

Figure No.9 Pressure Contour for 60^0

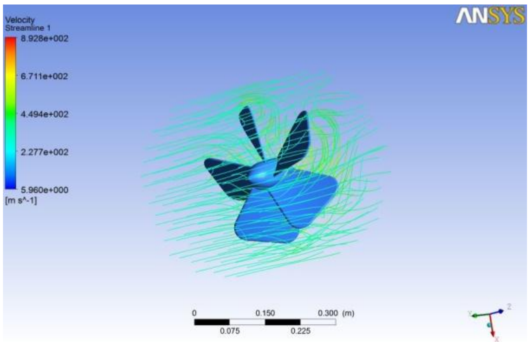

Figure No.10 Streamline of Air with velocity for 60^0

From the above statistics and table, we state the Velocity and Pressure of the air that Celtic cross from inlet to Outlet of Propeller fan having 60^0 blade angles.

Table No.2 Tabulation for 60^0 Fan Design

60^0	MIN	MAX
Velocity (m/s)	5.960	892.8
Pressure (Pa)	$- 5.108 \times 10^5$	$+ 0.753 \times 10^5$

B. FLOW ANALYSIS RESULT FOR 80^0 FAN DESIGN

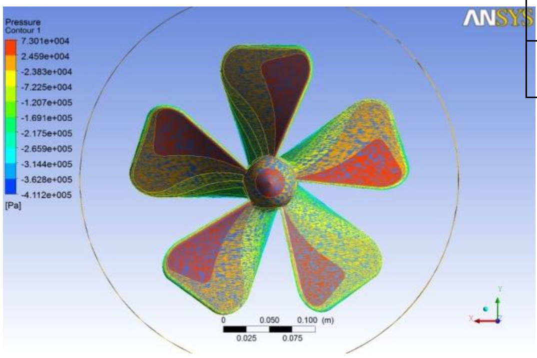

Figure No.11 Pressure Contour for 80^0

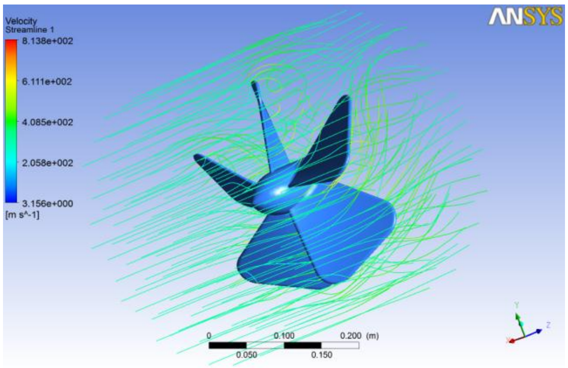

Figure No.12 Streamline of Air with velocity for 80^0

Table No.3 Tabulation for 80^0 Fan Design

80^0	MIN	MAX
Velocity (m/s)	3.156	813.8
Pressure (Pa)	$- 4.112 \times 10^5$	$+ 0.73 \times 10^5$

From the above figures and table, we state the Velocity and Weight of the air that crosses from inlet to Exit of Propeller fan having 80^0 vane angles.

VII. CONCLUSION

The presentation of axial flow type propeller fan is studied with two various angles in blade design ($60^0$and 80^0). With the best blade enterprise, the pressure drop is minimized and rapidity of air is maintained effectively. Flow through the propeller was investigated using commercial CFD raft ANSYS-CFX. The Reproduction results were obtained at the operating speed of 1500rpm with corresponding mass flow rate (50Kg/s) for carrying of Air at 25^0C. The Simulation was performed by using subsonic flow with turbulence. Pressure contour, Air particle reorganize and Velocity contour were predicted.

REFERENCES

1. Bleier FP, (1997), Fan Handbook Choice Application and Design, McGraw Hill Newspapers
2. Jang C M, Sato D, Fukano T, (2005), Tentative Analysis on Tip Escape and Wake Flow in Axial Flow Fan According to the Flow Rates, J. Fluids Engineering, Vol. 127, pp. 322-329.

3. Kokturk T (2005) Plan and Performance Analysis of Reversible Axial Flow Fan. M.S.Thesis, Branch of Mechanical Industrial, Middle East procedural University
4. Reynolds, B., and Lakshminarayana, B., (1982), Blade Charging and Span Wise Effects on Wake Characteristics of Compressor Rotor Blades, J. Aircraft, Vol. 19, pp. 97-103**.**
5. Schmidt K, and Patterson DJ, (2001) Act Results for a High Efficiency Hot Ceiling Fan and Comparison with Predictable Fans-Demand Side Running via Small Appliance Efficiency, Renewable Energy, Vol.22, pp. 169-176
6. Singh K (2005), Progress of a Test Facility to Study the Aerodynamic Routine of Axial Flow Fans, M. Tech. Thesis, Department of Power-driven Engineering, IIT Guwahati, India.
7. Wilcox D.C. (1994) Turmoil Modeling For CFD .DWC Trades, Inc. La Canada, California.