iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 12 Issue: IX Month of publication: September 2024

DOI: https://doi.org/10.22214/ijraset.2024.64425

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

Numerical Investigation of Bionic Axial Fan used in Split Air Conditioner

Aadit Saha[1], Pradeep Shetty[2], Ramalingam Muthusamy[3]
[1]*United World College of South East Asia, Singapore*
[2]*IFB Industries Ltd. Bengaluru, India*

Abstract: Bionic axial fan is the main component in the outdoor unit split type room air-conditioner (AC) which impact the cooling and comfort. The present work aims to improve the aerodynamic performances of bionic axial fan by numerical technique. The detailed simulation has been carried out to investigate the aerodynamic performance of bionic axial fan with various design proposals for a detailed investigation and development of a prospective fan design prior to the manufacture and testing of costly prototypes. According to the biological fact that the serrations at the wing improve the acoustics as well the aerodynamic performances, various design modifications performed on the bionic axial fan blade. As the serrations at the trailing edge help to improve the aerodynamic performance, the fan blade design modified to investigate the change in mass flow rate.

The flow field is simulated with the finite element Computational Fluid Dynamics (CFD) solver Altair-AcuSolve to analyse the influence of trailing edge serrations on the air flow rate of fan. The three-dimensional (3D) computational domain with Spalart-Allmaras (SA) turbulence model is considered to predict the air flow rate. The present computation is carried out for the axial fan speed of 820 rpm. The flow rate is correlated with the test results to validate the CFD modeling approach. The result shows that the trailing edge serrations at the tip of the fan has predicted improvement in flow rate since it alter the length of separation bubble near the trailing edge which also helps to reduce the noise level.

Keywords: Bionic Fan, Split Air Conditioner, CFD, Trailing edge serration

I. INTRODUCTION

Demand for cooler and comfortable ambience is increasing as there is a rapid change in atmospheric temperature and humidity. Air conditioning is a desired commodity in a globe to feel comfortable during our daily activities. Generally, air conditioner is designed to control both the humidity and temperature in a room. Basically it is a technology that modifies the air quality inside the room to keep the ambience more tranquil. The current study belongs to one type of air conditioner technology which works by supplying air from wall mounted vents, since it has an indoor unit and outdoor unit; it is called as a Split Air Conditioner.

A. Introduction to Split Air Conditioner

This type of air conditioner is used widely in the current market mainly because of its high level performance. The major fact about the popularity of split air conditioner because of its silent operation and graceful looks. One more advantage is that we don't have to create hole in the wall to install the setup which used to destroy the elegant look of the room.

There are mainly two parts of the split type air conditioner. The outdoor unit and indoor unit are shown in Fig 1 and Fig 2. As the name indicates, the indoor unit of air conditioner is kept inside the room. The indoor unit installed inside to cool the room. The outdoor unit kept outside the specified room. The outdoor unit installed in open space which can be easily maintained.

Fig 1: Indoor Unit of Split AC

Fig 2: Outdoor Unit of Split AC

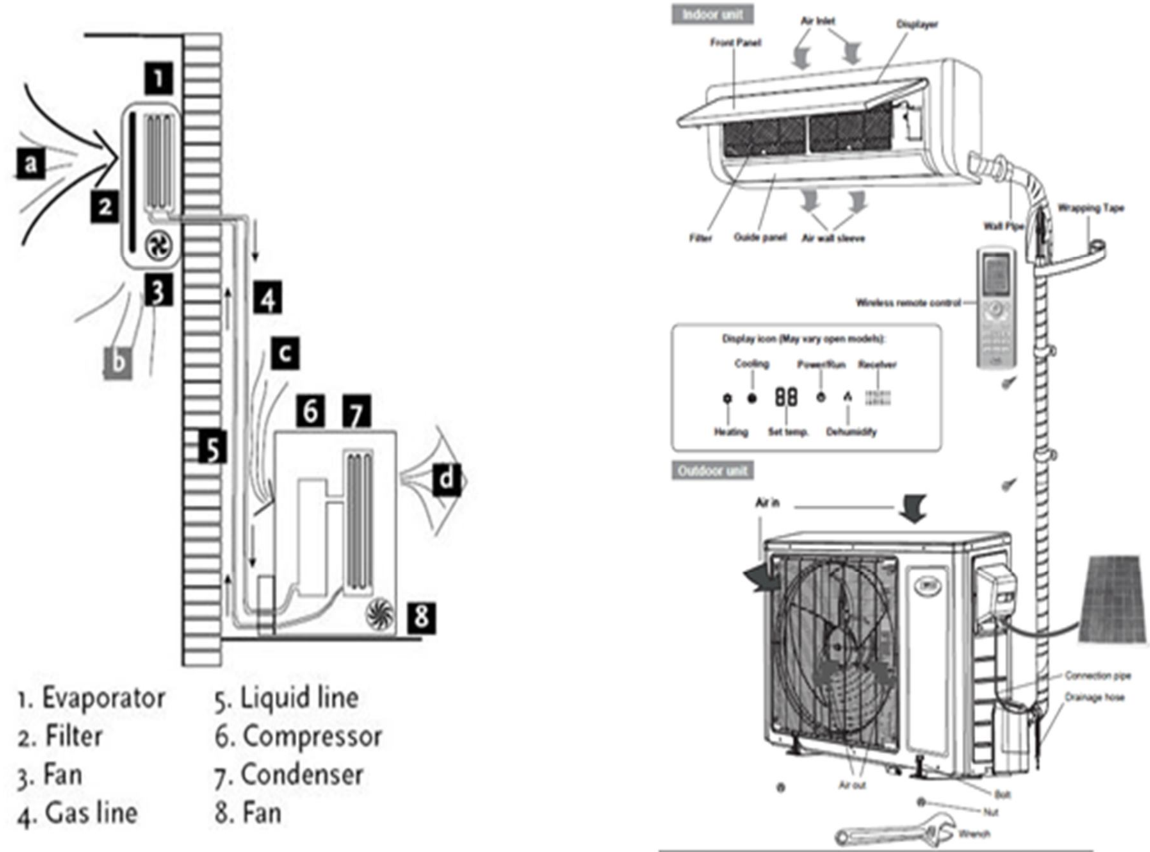


Fig 3: Schematic of Split AC components

The schematic picture of both the indoor and outdoor unit of split air conditioner is as shown in fig 3. Mainly the condenser unit, compressor and fan are outside the room in contained unit whereas the blower and evaporator coils are inside the room. These two units are connected through pipes or tubing.

B. Outdoor Unit

As our focus is to improve the aerodynamic performance of axial fan used in self-contained outdoor unit, the various parts of outdoor unit listed like compressor, condenser, expansion valve etc.

1) Compressor

Compressor is the key part of any of the air conditioner. It compresses the gas or refrigerant and increases refrigerant pressure sending prior to the condenser unit. The compressor size varies depending upon the air conditioning load. External power will be supplied to the unit through motor, which in turn drive the motor shaft. During the process of compressing the refrigerant, heat will be generated which has to be removed properly.

2) Condenser

The condenser unit of the split air conditioner consists of coiled copper tube with many rows. The number of rows depends on the size of the air conditioner load and the size of the compressor. The heat from the refrigerant are to be removed which is coming from compressor with high pressure and high temperature. Generally the tubes are made up of copper because of its high thermal conductivity.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

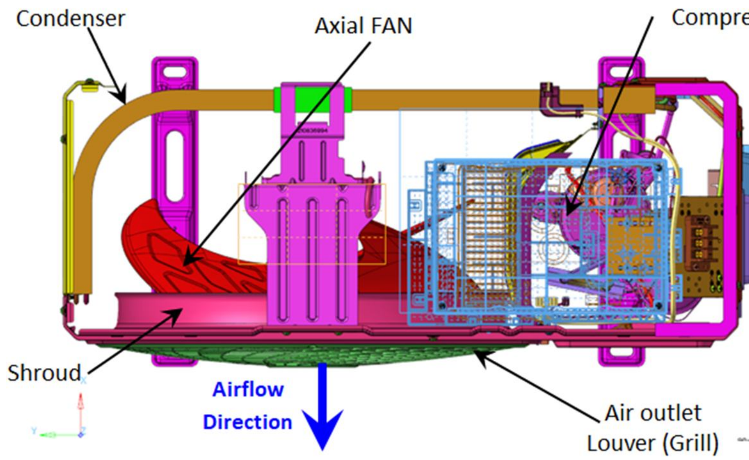


Fig 4: Condenser unit with cooling fan

3) Condenser Cooling Fan

This cooling fan is our main area of focus. We must cool down the high temperature refrigerant which is coming from the compressor, else motor coils and compressor will get heated up leading to breakdown of compressor in a long run which will reduce the life of the air conditioner. Hence, the condenser coil needs to be cooled effectively in a faster rate so as to provide more cooling effect. Since axial fan is the one which helps to cool down the condenser coils, we must ensure the mass flow rate of axial fan is good enough to provide the cooling effect. The cooling fan with three blades is rotated by motor. As the blades rotate, the air from the open surrounding space will be sucked and fed to the condenser unit. As we improve the mass flow rate of fan design, there will be more energy saving and cost as well.

C. Working of Split Air Conditioner

The system works on the basic principle of heat transfer using refrigerant. Refrigerant is fed to the compressor to increase the gas pressure. Hence, there will be rise in gas temperature.

1) The high pressure refrigerant is condensed into a liquid as it passes through succession of tubes.
2) The high pressure liquid refrigerant still travel through condenser tubes until it reaches to an expansion joint.
3) Then the pressurized liquid refrigerant becomes gas as the pressure is reduced.
4) When the pressure of the gas reduced which also release heat and becomes cooler.
5) The gas then flow through compressor again to repeat the process.
6) Air from the room is drawn and passes to the evaporator coils.
7) This fluid actions cools the air significantly which is then forced back to room via indoor unit blower.
8) Air circulation continues until a set temperature is achieved.

In a nutshell, split air conditioner draws the heat from indoor unit and flows it to the outdoor unit and then the quality of air is fed back to the room.

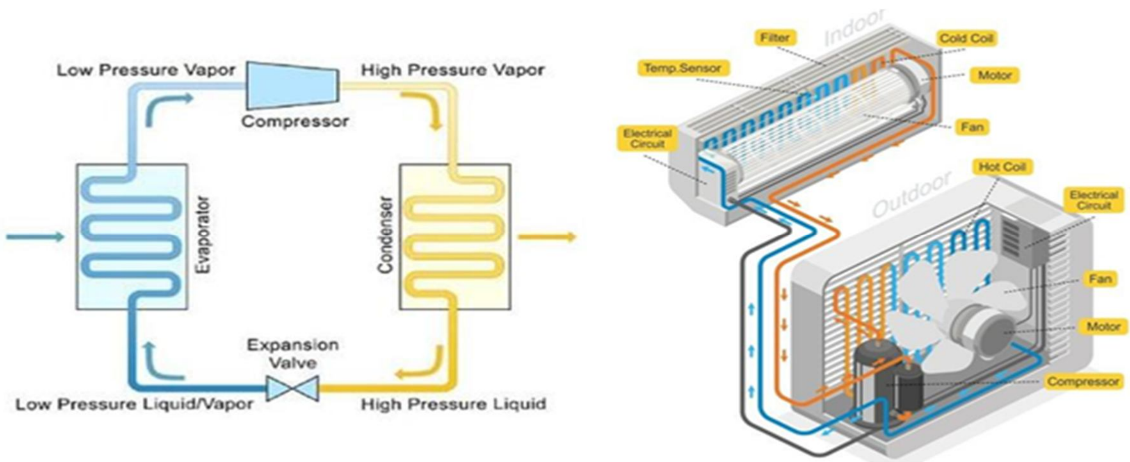


Fig 5: Representation of Working Cycle

D. Benefits of Split Air Conditioner

1) Less Energy Loss: The unit is compact and isolated; hence there is a less opportunity for the energy to escape from the unit. Waste from the system very less when compared to centralized air conditioning because of robust design.
2) Less heat loss
3) Targeted heating and cooling
4) Affordable installation

II. LITERATURE REVIEW

The virtual simulations to analyze the performance of the product yields to reduction in design cycle time and prototype cost. The CFD technique is used to analyze the axial fan aerodynamic as well aero-acoustics performance. A number of previous studies have been carried out on numerical simulation and experimental study on condenser cooling axial fan.

Nitin Gulhane et al. (2015) [1] Investigated the flow performance of fan blade with the different geometric parameters like blade width, blade depth and blade angle. As blade angle increases, the flow rate increases but at the same time noise is also increased. The fan blade serrations at the tip of the blade reduced the noise level.

Xifeng Zhao et al. (2013) [2] The study mainly focused on design modification by flanging outer edge blade and concaved trailing edge blade since the blade tip and trailing edges of the axial fan are the regions of the main sound sources. It is demonstrated that the flanging outer-edge fan tends to be more effective.

Franck Perot et al. (2010) [3] simulated the flow induced noise of low speed axial fan through numerical techniques. The direct prediction of sound pressure level by coupling the CFD and CAA simulations are investigated. The fan performance curve obtained is compared with experimental results. This study suggested that the passage between blades and the hub/blade tip interactions are major sources of induced noise.

Jie Tian et al. (2009) [4] analyzed the condenser fan unit to understand both the aerodynamic and acoustics performance of air conditioner. The importance aspects of downstream outer grille in creating noise have been investigated. The circular type grill showed improvement in flow rate and reduction in flow induced noise when compared with rectangular type grille.

Jiang Cai-ling et al. (2007) [5] carried out both experimental and numerical study on finding out the noise level from axial fan of air conditioner unit. This study has mainly focused on mechanism of sound generation and to predict the sound pressure level through CFD approach using commercial code. The sound pressure level for with grille and without grille has been studied in detail. The noise level is significant when considered with grille. The study demonstrated the role of tip vortex in the flow filed mainly at the blade tip.

Junwei Hu et al. (2006) [6] analyzed the effect of the air outlet grille or louver part on the generation of noise in the air conditioner exhaust unit. The study showed there is benefit in terms of noise by using circular shape of grille part or louver. The circular shape improves the aerodynamic performance and aero-acoustics performance as well.

Liang Gui-qiang (2010) [7] Study revealed that the non-smooth shape of the fan blade is good for preventing formation of off-body vortex. The saw tooth shaped serration improved the airflow reduced the fan noise. Experimental study has been carried out to understand the influence of serrations on the fan blade.

The detailed survey has been made on carrying out virtual simulations using CFD technique. Most of the studies mainly focused on noise prediction from condenser fan with experimental study, axial fan geometry modification with flanging outer edge, changing the passage between the blades, interaction between the hub/blade tip, geometry modification of outdoor unit grille, influence of tip vortex and the experimental study on the fan blade serrations. But very limited carried on aerodynamic performance with design proposals using CFD technique. The design proposals with detail flow simulations are discussed in detail.

III. OBJECTIVE & MOTIVATION

The main objective of the current study is to improve the air flow rate of bionic axial fan used in outdoor unit of split type air conditioner. The design modification of fan blade to ensures the improvement in capacity of fan to deliver more cooling effort. The growing demand for fast cooling design, the axial fan of condenser unit should have high mass flow rate capacity. Thus, a detailed analysis and development of a prospective fan design to be performed prior to the manufacturing and testing costly prototypes.

A. Motivation

1) Mechanism of Owl's Silent Flight

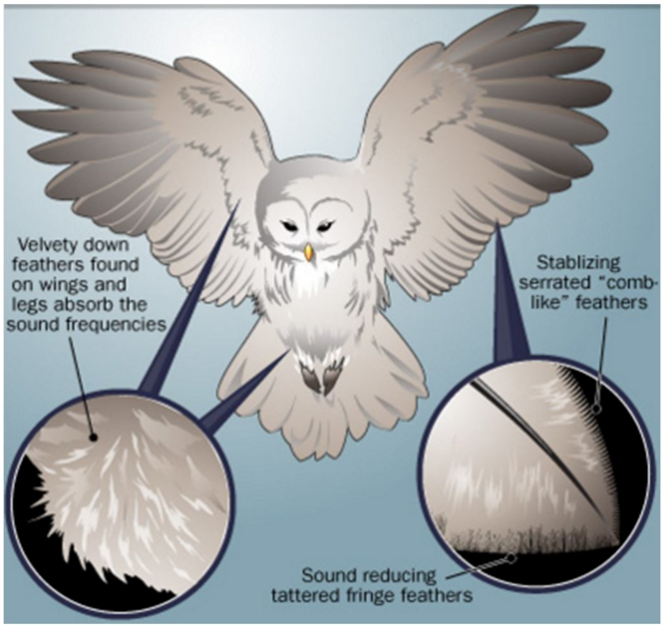


Fig 6: Biological mechanism of owl's silent flight

Fig 7: Serrated wings of bird

Owls are the silent predators of the night which can capable of hunting their prey within no time and without being detected. Owls are known as natural stealth craft. The silent flight is because of their naturally well designed feathers as shown in fig 6 and fig 7. When the flow past owl wing, creation of gushing noise is less as the owl wing can alter its turbulence. Hence the noise is reduced. This biological mechanism gives motivation to improve the fan design in terms of noise reduction and mass flow rate improvement. Hence, three design modifications have been proposed by keeping the serrations which improve the performances of axial fan. Prior to the design modifications, the detailed investigations of existing fan design helped to understand the flow characteristics of the rotating blades.

B. Existing Fan Design and Design Change Concepts

1) Existing fan Design

The CAD model and physical model of existing axial fan design are shown in fig 8. The diameter of existing design is 412 mm and the width of 132 mm. The motor is mounted onto the hub of 90 mm diameter. The hub-to-diameter ratio is around 0.218 with 3 blades.

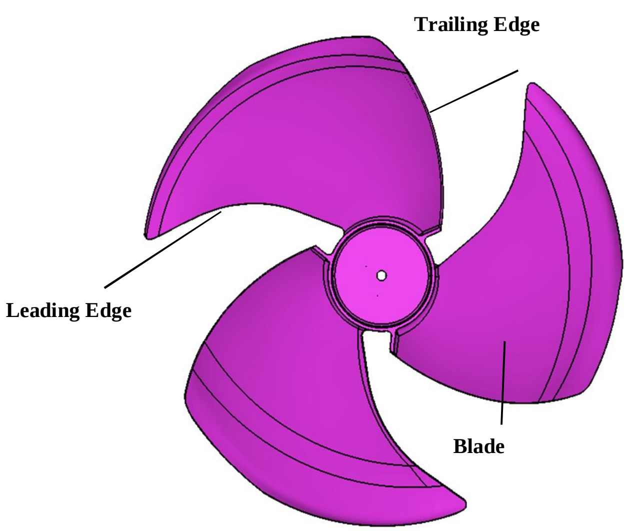


Fig 8: Existing design of fan: CAD & Physical Model

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

2) Serrated design proposals

The serration prevents the formation of off body vortex, three design modification trails have been proposed as shown in fig 9. The serration trials have been proposed based on the owl's wing structure and fig 10 shows superimposition of all the four designs. The depth D and pitch P of the trailing edge as shown in fig 11. D is 18 mm and P is 10 mm in length.

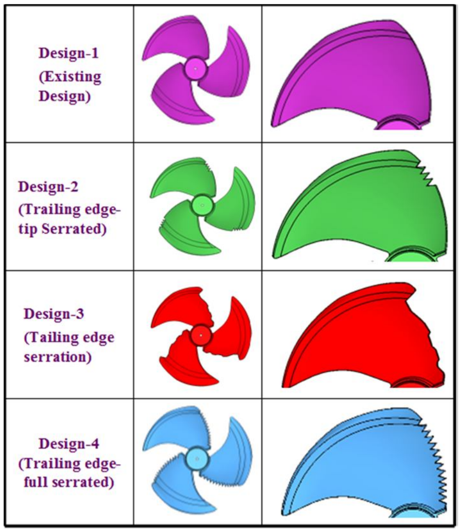


Fig 9: Axial Fan Design Proposals

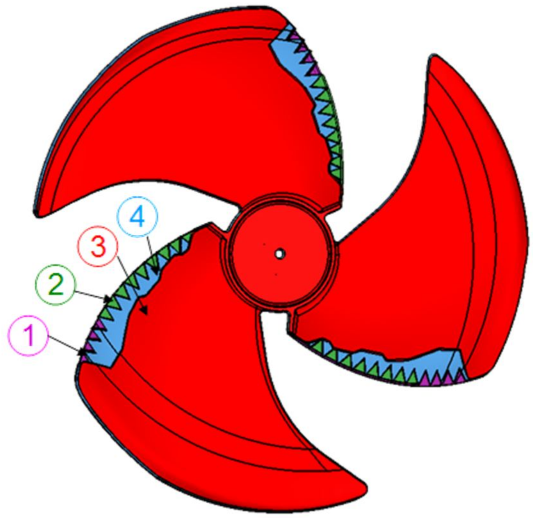


Fig 10: Superimposition of four fan design

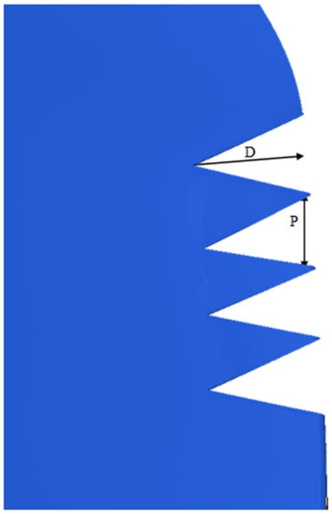


Fig 11: Specification of serrations at the trailing edge of fan blade

The motive of the analysis is to compare the outcome of the CFD analysis of an axial flow fan with the mass flow rate of axial fan obtained from fan test facility. A design improvement design has been proposed through numerical technique.

IV. METHODOLOGY

A. Simulation Process Chart

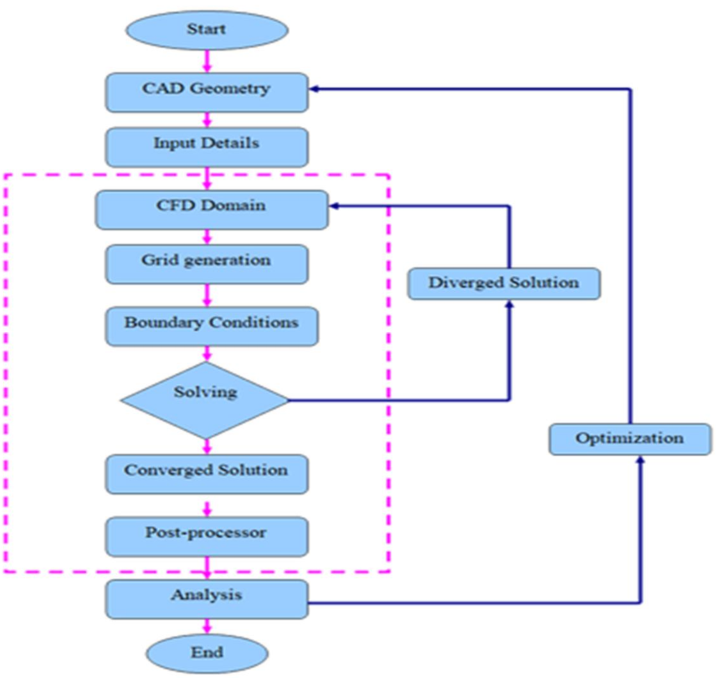


Fig 12: Flow chart of CFD simulation modeling approach

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

The detailed investigation during design phase is possible using numerical simulation. The flow characteristics have been analysed through Computational Fluid Dynamics approach by solving three dimensional unsteady fluid flow governing equations. The CAD geometry of the existing fan design is investigated prior to the design change proposals. The fig 12 shows the brief methodology which we are going to proceed with the CFD modelling.

V. NUMERICAL MODELLING

A computational method is used to predict the aerodynamic performances of axial fan used in outdoor unit of split air conditioner. In this numerical approach, basically we are going to solve the fluid flow equation in three dimensional space and time domain. In this study commercially available CFD code Altair-AcuSolve has been used successfully to investigate the flow characteristics of the axial fan. The computational domain is discretized into small domain to solve for the spatial and time domain.

A. Governing Equations

AcuSolve numerically solves the flow governing Navier-Stokes equation. It is one of the most powerful finite element flow solver based on Galerkin/Least-Square (GLS) FEM with the best robust solution for the CFD problems. This code is fully coupled pressure-velocity solver for most of the flow regimes. This solver can support with shared distributed and hybrid flux transforming modes. Second order time and spatial accuracy for all element topologies. AcuSolve is an incompressible flow solver which incorporated with RANS, DES and LES turbulence models.

1) Continuity Equation

This eq. tells about the conservation of mass flowing across a control volume. The continuity eq. can be written as:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \vec{u}) = S \qquad (1)$$

Where S = Source term, ρ = density, $\vec{u}$ = velocity

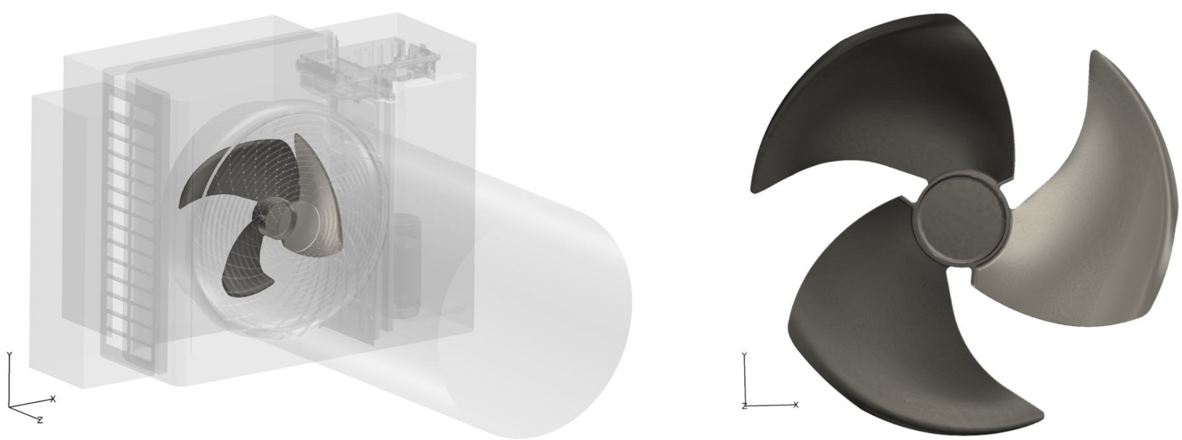
2) Momentum Equation

A momentum equation is solved for fluids and resulting velocity field is shared among all the phases. The resulting velocity and pressure fields are shared by fluid phases. Based on the local values of □q, the appropriate properties and variables are assigned to each control volume within the domain.

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla.(\rho.\vec{u}\vec{u}) = -\nabla p + \nabla\left(\mu(\nabla\vec{u} + \nabla\vec{u}^T)\right) + \rho\vec{g} + \vec{T}_\sigma \qquad (2)$$

Where p = pressure, μ = viscosity, $\vec{T}_\sigma$ = Surface Tension force

B. AC ODU and Axial Fan Model



ODU Assembly　　　　Axial Fan – Existing Design

Fig 13: AC ODU Assembly and Existing Axial fan model

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

The representation of ODU full assembly and axial fan model are highlighted in fig 13. This cooling fan is a typical axial fan with 3 blades.

C. Computational Mesh

Grid generation is a process where we discretize the computational domain into finite elements or cells where the flow variables are solved at the discretized elements. Hence dividing the physical domain into sub-domains is referred as grid generation.

Basically in present finite element simulation, the flow variables are solved at these element nodes. The surface mesh generated at the boundaries of flow domain with tria (2D) mesh. The interior domains are filled by tetrahedral (3D) type of elements as shown in fig 14.

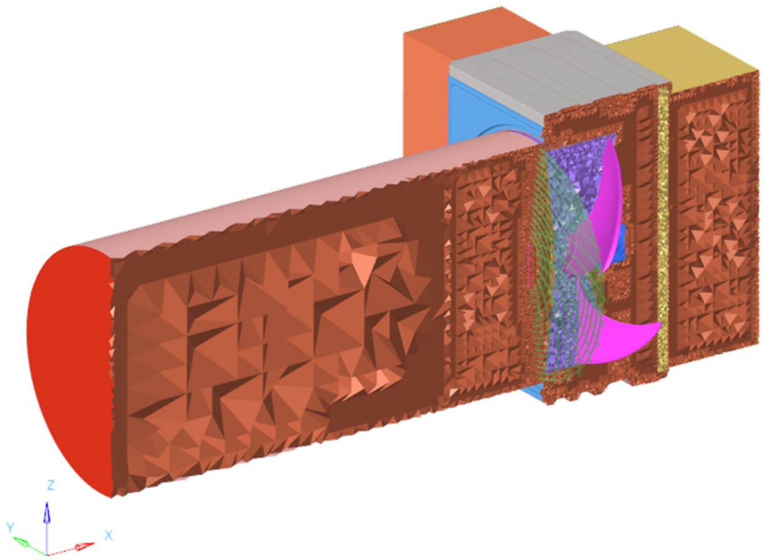


Fig 14: The computational domain and cross section of flow domain.

The boundary inputs for solving governing equations are:

1) The size and shape of the computational domain where the flow governing equations are to be solved.
2) Discretization of the domain to solve for the flow variables at the finite region.
3) Boundary conditions to mimic the experimental setup and making sense of flow physics.
4) It writes the file required for the flow solver.

The surface mesh was generated with variation in size of the element throughout the domain as shown in fig 14. A refined mesh near the fan blade been generated to capture the pressure or velocity gradients. Finer mesh gives less interpolation error near the region where the more influence of flow. Tetra mesh generated on the axial fan surface and the MRF zone to have a boundary layer mesh with 5 layers of prism elements to account for the pressure or velocity gradients.

The tetrahedral volume elements generated through surface mesh in HyperMesh at a cross section of the domain illustrating the high node density around and after the fan. The mesh count is around 7.6 million numbers of elements and 1.3 million numbers of nodes. Since the CFD flow simulations were to compare with the experimental data, the typical domain is modeled to mimic the physical condition. The quality of grid gives the best quality of results. Hence, the 2D skew is well below 59 and volume skew is less than 0.98 in 3D. The non-dimensional wall thickness Y+ is maintained around 11.

Boundary Conditions

- INLET:

The stagnation inlet boundary conditions specified in the upstream end of fan. The flow is happening normal to the boundary plane. In this boundary condition, the eddy viscosity and temperature parameters are to be specified, referred as static pressure boundary. AcuSolve allows the user to specify the gauge pressure at the inflow boundary, referred as total pressure boundary.

Static pressure = Total pressure - Dynamic pressure

- OUTLET:

In this boundary condition, we specify diffusion flux for all the flow variables is zero at the plane normal to the face of the boundary.

This boundary condition is specifically applied to zone where the flow is fully developed and there will is no change in the cross stream profile of velocity is not changing in the flow direction. The atmospheric pressure of zero gauge pressure has assigned for the outlet boundary condition.

- WALL:

Wall boundaries are specified to the region where the solid surfaces are to be modeled. The viscous stresses at the near wall region are calculated through standard wall function. This is refereed as no-slip wall. Axial fan is also considered as a no-slip wall since it is a solid body.
When we assume the shear stresses at the surface is zero, that boundary condition specified as slip-wall boundary condition. This boundary doesn't allow the variables to change across the domain. The velocity normal to the face of the boundary is zero.

- MRF:

Moving reference frame assumed that the constant of revolution assigned with the enclosed volume with non wall boundary surface. The fan blade volume considered as a moving reference frame.

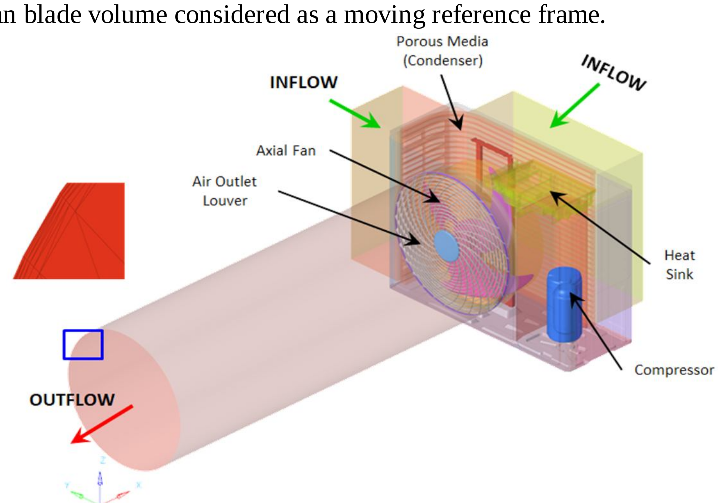


Fig 15: Boundary conditions for the flow domain

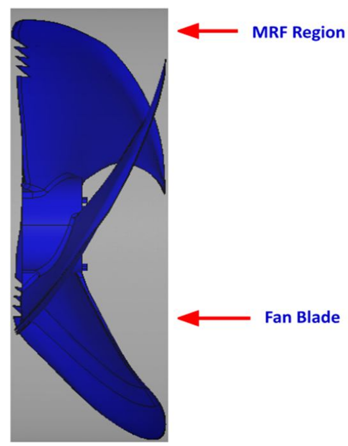


Fig 16: MRF Boundary Condition

MRF model is to specify the region where it includes revolving or rotating parts. The boundary conditions detail and the MRF zone include fan blade and hub as shown in fig 15 and fig 16 respectively.

D. Solver Setup

Table 1 show the primarily setting used for all simulations. The working temperature was set to 27°C and the corresponding density and viscosity are presented below. As solver and post processor AcuSolve and AcuFieldView were used respectively.

TABLE 1: SOLVER SET UP DETAILS

Boundary conditions	
Inlet	Mass flow inlet or Pressure inlet
Outlet	Pressure outlet
Turbulence settings	
Turbulence model	Spalart-Allmaras
Near wall treatment	Standard wall function
Air properties	
Density	1.225 [kg/m^3]
Dynamic viscosity	1.81 *E^{-5} [Pa-s]

E. *Model Assumptions*

The following assumptions were made in order to reduce the complexity in numerical simulation.

1) The flow is assumed it to be incompressible.
2) The material for the computational domain considered as Air.
3) Spalart - Allmaras eddy-viscosity based turbulence model has been used to account for flow turbulence.
4) Standard wall function algorithm is used to account for calculating the flow gradients near the wall.
5) Gravity force is neglected.

VI. RESULTS AND DISCUSSIONS

A. *CFD Analysis of Existing Fan Design*

Three-dimensional flow investigation has been carried out by solving the Reynolds Averaged Navier-Stokes equations for the existing axial fan design of outdoor condenser unit of split air conditioner unit through numerical approach. The flow characteristics around the fan blades are studied for the speed of 820 rpm. Incompressible flow governing equations with prescribed boundary conditions are being used to predict the mass flow rate.

The airflow dominance can be observed at the axial fan peripheral as shown in fig 17. The condenser unit offers resistance to flow due to its intricate structure. The porous model representation for condenser in fig 18 shows the pressure drop before the condenser unit.

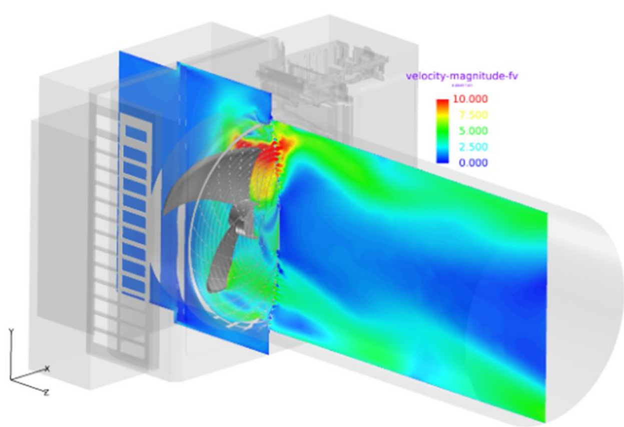


Fig 17: Cross plane section of Velocity contour at the ODU

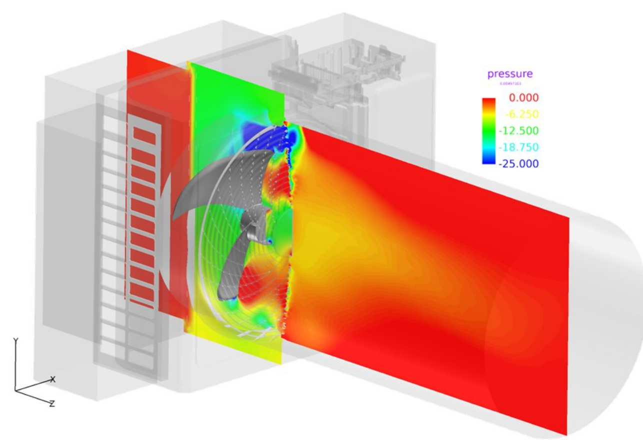


Fig 18: Cross plane section of Pressure contour at the ODU

B. *Validation of CFD simulation by experiment*

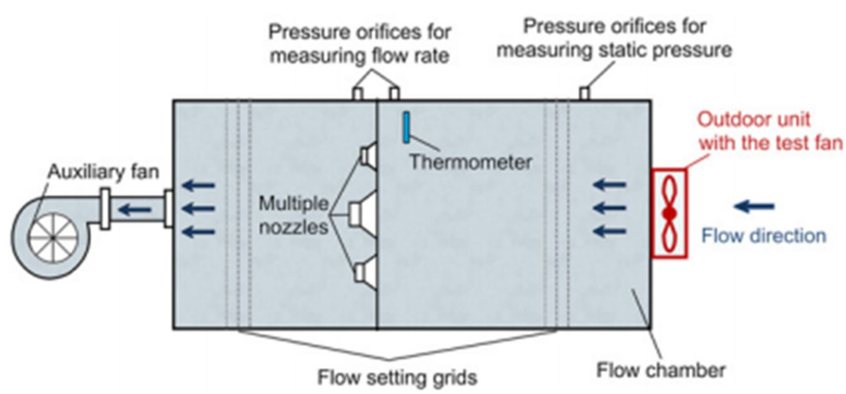
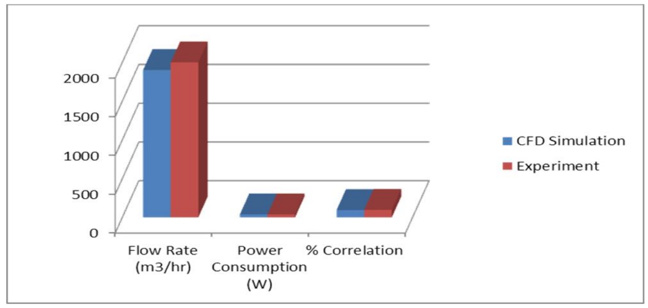


Fig 19: Typical experimental test rig to measure the flow rate of axial fan.

The result from the CFD simulation is correlated with the experimental data in-order to validate the analysis approach prior to the design modification. The airflow rate and fan power data have been compared with test result as listed in Table 2.

TABLE 2: COMPARISON OF MASS FLOW RATE BETWEEN CFD AND EXPERIMENT

	Flow Rate (m3/hr)	Power Consumption (W)
CFD Simulation	1900.1	37.88
Experiment	2000.0	39
% Correlation	94.7	97.03



The mass flow rate obtained from the fan test facility provides a means to verify and validate the accuracy and integrity of the CFD simulation. This was done firstly to produce a new data set for the newly manufactured fan blades and secondly to afford more insight into the operation of the axial flow fan in practical use, especially in terms of the manner in which the air flow exits the fan rotor. This knowledge was deemed important in analyzing the streamlines of the CFD computations.

The flow separating regions are highlighted based on the Q-criterion Fig 20. The contour shows the flow separations from the fan as well outer grill. The complex flow characteristics through the complete ODU assembly along with the condenser unit make the cycle time longer. As our interest is to investigate the aerodynamic performance of the axial fan, we have limited the focus to consider axial fan alone. Hence the axial fan model was isolated from the assembly for simplifying the approach. This assumption in the current study has helped to reduce the computation time and cycle time for design modification.

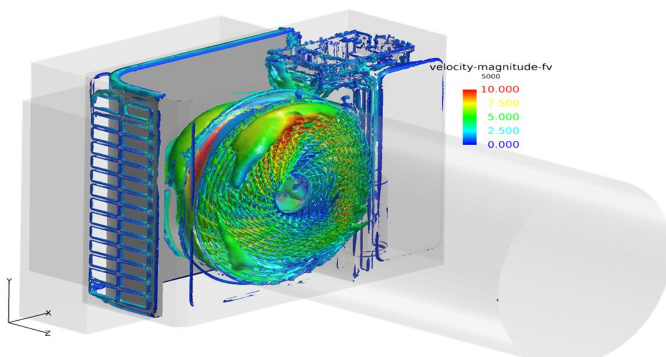
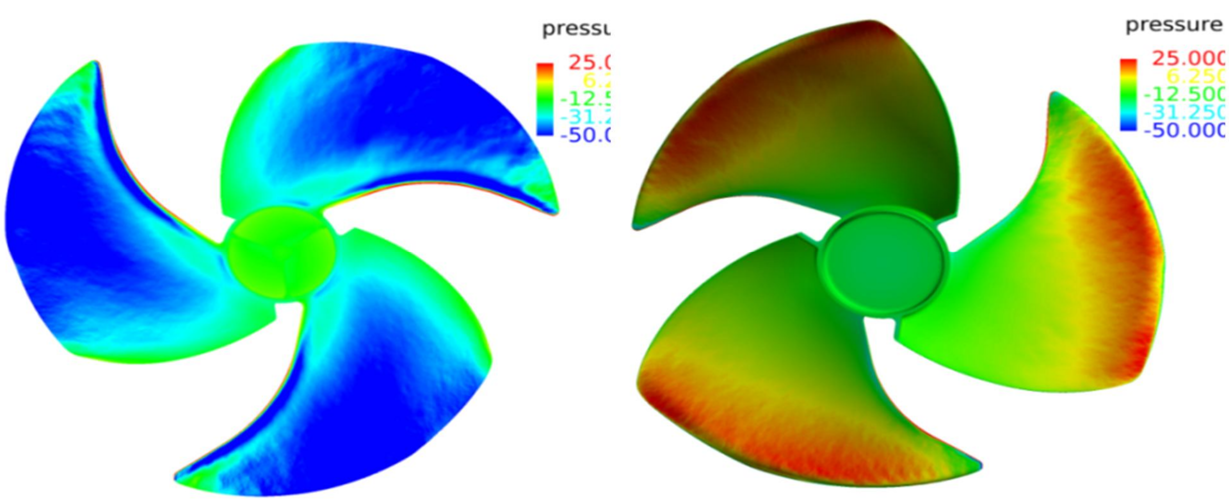


Fig 20: The pressure distribution at the upper and lower surface of the axial fan blades.

C. Design Change Iterations Based on the Simplified Domain



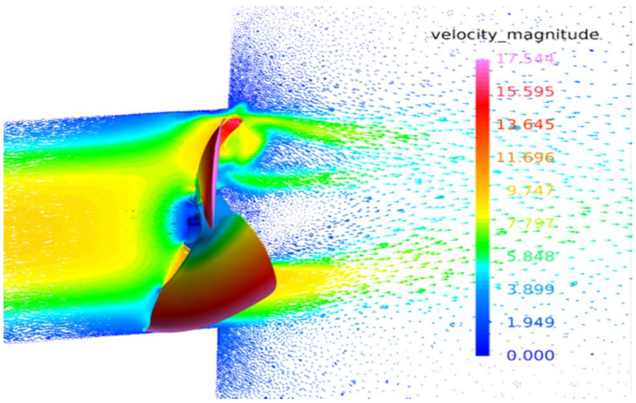
SUCTION SIDE

PRESSURE SIDE

Fig 21: The pressure distribution at the upper and lower surface of the axial fan blades.

The uniform distribution of pressure on the axial fan blades as shown in fig 21, indicate the flow regime is below stall. The flow past through leading edge of axial fan blade shows maximum pressure where the flow is chaotic and unstable. The flow separation over the fan blade generates a stalled condition where the lift force is restricted. As the Reynolds number of flow past fan increases, the flow tends to be more random and chaotic.

The flow vector in the domain shows the representation of the flow-field produced by the axial fan as shown in fig 22. The maximum radial velocity is at the tip of the blade is around 17.544 m/s. Analytically the tip velocity 'v' is equal to radius of the fan multiplied by angular velocity.



Speed 820 rpm,

Angular Velocity 'w' = 2π*820/60

w = 85.82 rad/s

Tip Velocity v = r w

v = 0.205 m * 85.82 rad/s

v = 17.5 m/s

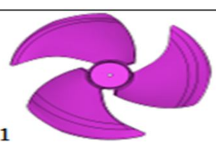
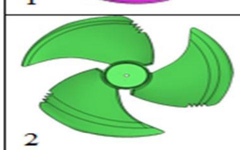
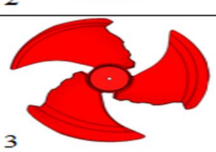
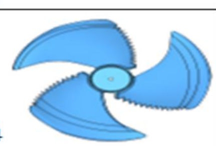
Fig 22: Velocity vector flow of existing fan design

D. CFD analysis of Design Modifications

The detailed analysis has been carried out after establishing the numerical technique. The change in flow rate when compared with the existing fan design (Design-1) is shown in Table 3. The simulation is repeated for all the modified fan design to investigate the effect in varying the blade design. There is 3.9% improvement in flow rate for the tip serrated at trailing edge fan model (Design-2) when compared with plane trailing edge (Existing Design) of fan blade.

The serrations at the trailing edge (Design-2 & Design-4) predicted improvement in aerodynamic performances. The model with only tip serrations predicted better when compared with the full serration model since the velocity filed at the tip is dominant when compared to the velocity near the hub.

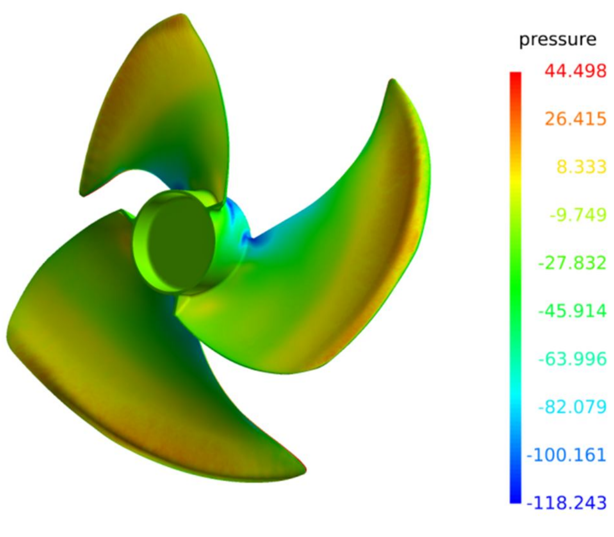
TABLE 3 : FLOW RATE VS. DESIGN PROPOSALS FOR SIMPLER DOMAIN FOR ONLY FAN

	Speed (rpm)	Static Pressure (Pa)	Flow rate (CMH)	% Improvement
 1	820	16.4	2480	-
 2	820	17.45	2577	~ 3.9 %
 3	820	16.4	2483	~ 0.1 %
 4	820	17.07	2528	~ 1.9 %

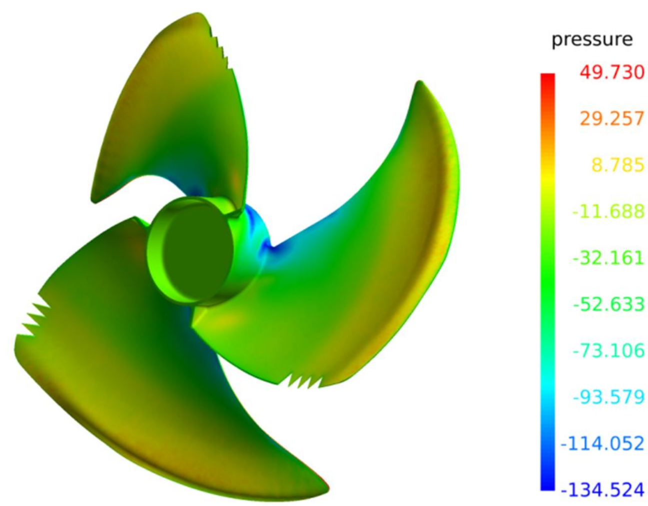
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue IX Sep 2024- Available at www.ijraset.com

The axial flow fan creates a negative pressure in the upstream of fan to have a flow across the blade. Pressure difference is the driving force for any flow to happen internally. The lower surface of the fan blade causes the pressure difference. The contours of pressure at the fan blade for all the fan models are as shown in Fig 23.

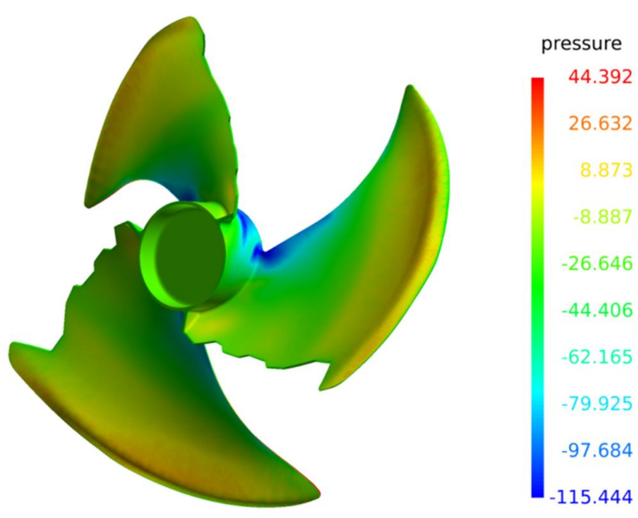
Design-2 predicts the maximum pressure of 49.73 Pa when compared with 44.49 Pa of Design-1. Increase in pressure difference generates more flow rate across the fan. The suction pressure is also more in design-2 when compared to design-1.



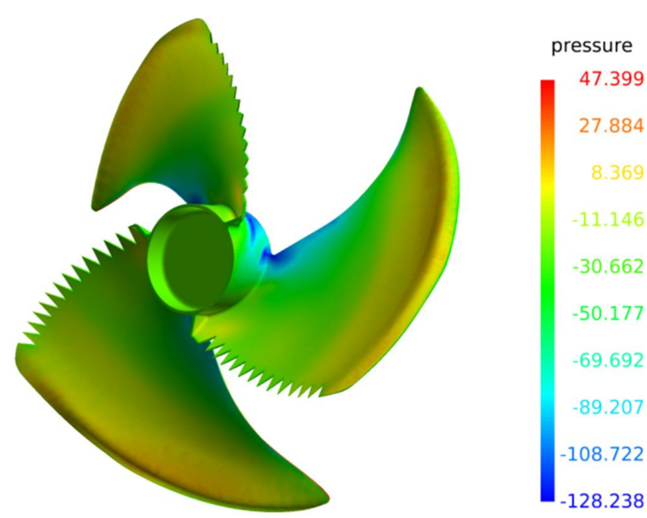
Design-1



Design-2



Design-3



Design-4

Fig 23: The pressure contour on the fan blades

The sudden interaction of flow happens at the tip of the fan blade when the fluid flows through tip of the fan blade from pressure side to suction side. This separation of flow generates kind of vortex at the tip. This formation of tip vortex induces the increase in sound pressure level. The interaction of rotating flow with the stationery wall has great impact on flow induced noise as well on aerodynamic performance.

It is well known that the serrations at the trailing edge reduce the flow induced noise by reducing the vortex strength. The main source of vortex is at the blade tip. The serrations at the tip would create local turbulence to cut down the turbulent eddies to grow. Hence, the reduction in tip vortices improves the performances of fan.

The internal fluid flow behavior of fan can be visualized by vorticity distribution as shown in Fig 24. The vorticity in the flow field determines the velocity around blade since it is derived from the velocity gradients.

The change in the velocity magnitudes occur on both suction side and pressure side. The change in velocity magnitude and the adverse pressure gradient lead to separation of flow. The vorticity refers to flow separation and flow induced noise as well. The tip vortex is generated at the tip of the blade near to leading edge. This vortex is extended to trailing edge downstream of axial fan blade exit. At the fan hub the vortex strength seems to be high as the flow is tending inward to hub direction.

The breakdown of vortex is determined by the presence of stagnation point with the recirculation region of bubble like eddies around the fan blades. The breakdown of tip vortex improved the aerodynamic performance. The flow characteristics at the fan blade create broadband noise where the trailing edge vortex and tip vortex shedding. The vortex is going to breakdown from the leading edge of blade tip in the downstream of blade, while in the trailing edge the vortex moves towards hub area and then to downstream.

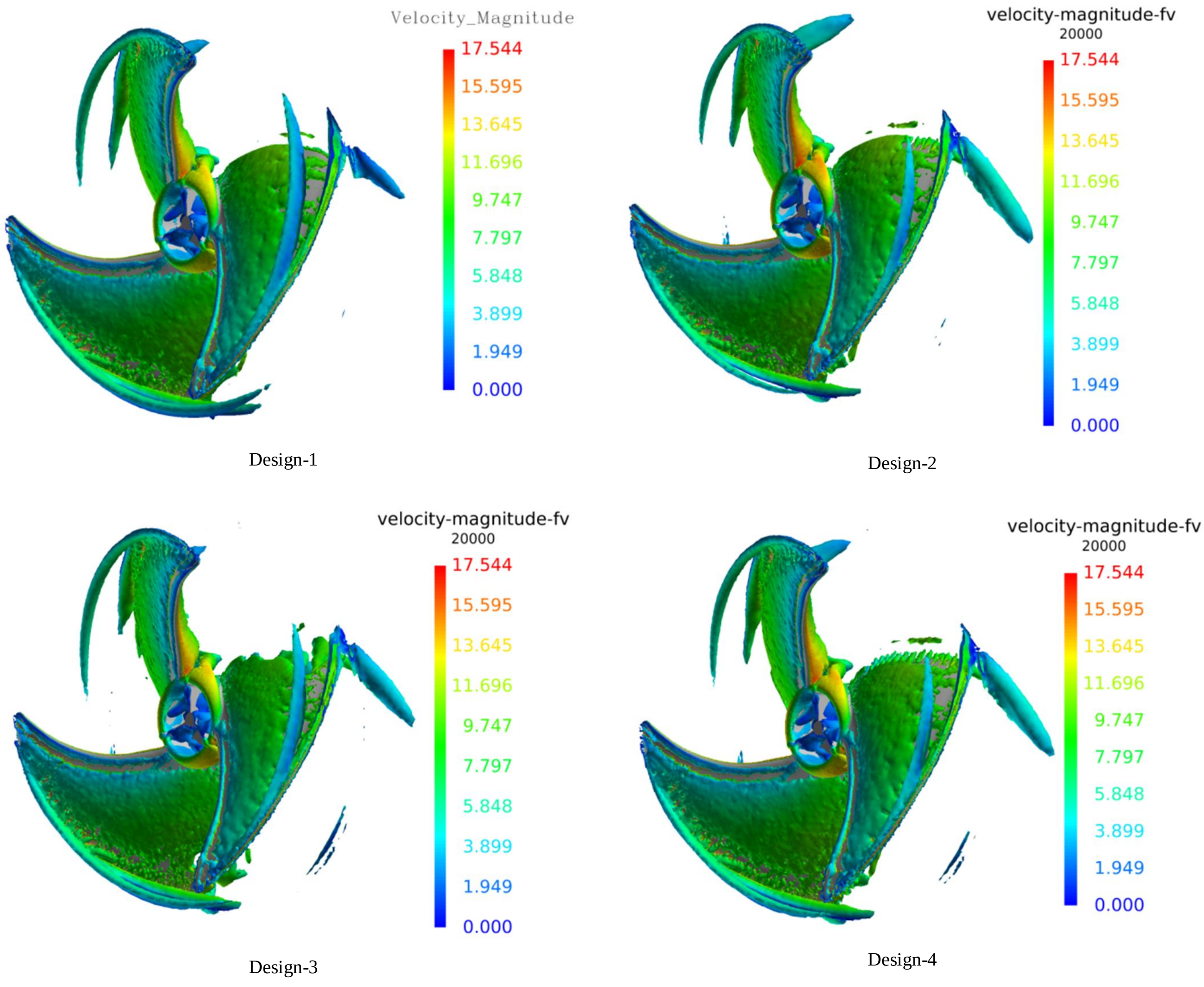


Fig 24: Iso-surface plot of velocity magnitude at the suction side view at Q-criteria = 20000

VII. CONCLUSION

The detailed flow analysis has been carried out to predict the mass flow rate of axial flow fan used in outdoor unit of split air conditioner unit. Computational Fluid Dynamics technique has been extensively used for the design change trails of bionic axial fan. The existing fan design simulated with various computational domain as a preliminary study.

Once after finalizing the simulation approach, three design modifications have been made to predict the aerodynamic performances. The mass flow rate obtained using the numerical simulation has been validated with experimental results. The reasonable agreement between the present computation and test results has been observed. Since our interest was to modify the fan design, we restricted our scope of interest to fan alone in order to reduce the cycle time of design change.

The mass flow rate investigated for all the modified fan designs and the observations are:

1) The improvement in air flow rate for Design-2 & Design-4 as the saw tooth serration at the trailing edge of bionic axial fan reduces the flow vortices.
2) Design-2 shows ~6% improvement in airflow rate when compared with Design-1.
3) The Design-1 and Design-3 results are very close in terms of airflow rate. But the noise level for the Design-3 modification could be an important area to explore.
4) The commercial code Altair-AcuSolve has been extensively used to predict fan performances. An unsteady eddy viscosity-based RANS model is used for the simulation. This tool was used successfully to simulate the flow characteristics of axial fans at a relatively low Reynolds Number regime.
5) The fan performance curve is predicted with different static pressures for the fan speed of 820 rpm to gain a flow rate relationship with static pressure.
6) The highest pressure on the fan blade happens on the tip of fan blade. The bottom surface is the pressure side which will help to create the pressure difference. Almost uniform distribution of pressure on the blades indicates that there is nearly no loss of lift from the blades, i.e., nearly no stall.
7) The vortices are observed behind the blades. The noise mainly occurs due to disturbances in the flow and separation developed over the blades. The present simulation helped to capture the wake flow and the tip vortex.

VIII. FUTURE WORK

The main focus of the current study classified to analyze the aerodynamic performance by predicting the mass flow rate and static pressure of air conditioner condenser cooling fan. Below listed are some of the important aspect where we need to give more attention in future study.

1) The URANS turbulence is used to determine the turbulence effects for the present simulation. The effect of using DES/LES model can be used for the futuristic modifications.
2) The Computational Aero Acoustics technique can be used to predict the flow induced noise

IX. ACKNOWLEDGEMENT

This research work was supported by IFB Industries Ltd. Bengaluru, India.

REFERENCES

[1] N. Gulhane, S. Patil, K. Singh,"Acoustic analysis of condesner fan of split air conditioner using numerical and experimental method," International Journal of Air Conditioning and Refrigeration, vol. 23 (2), 2015.
[2] X. Zhao, J. Sun and Z. Zhang,"Prediction and measurement of axial flow fan aerodyanmic and aeroacoustic performance in a split-type air-conditioner outdoor unit", International Journal of Refrigeration, vol. 36, 1098-1108, 2015.
[3] L. Zhang, Y. Jin,"An investigation on the effects of irregular airfoils on the aerodynamic performnace of small axial flow fans," Journal of Mechanical Science and Technology 27 (6) 1677-1685, 2013.
[4] J. Tian, H. Ouyang and Y. Wu, "Experimental and numerical study on aerodynamic noise of outdoor unit of room air conditioner with different grilles", J. Refrig. 32, III2–II22, 2009.
[5] J. Cai-ling, C. Jiang-ping, C. Zhi-jiu, T. Jie and H. O. Yang, "Experimental and numerical study on aero-acoustic sound of axial °ow fan in room air conditioner", J. Appl. Acoust. 68, 458–472, 2007.
[6] J. Hu and G. Ding, "Effect of the air outlet louver on the noise generated by the outdoor set of a split-unit air conditioner", J. Appl. Therm. Eng. 26, 1737–1745, 2006.
[7] L. Gui-qiang, W. Jing-chun, C. Yu, Z. Chang-hai, L. Jie, and R. Lu-quan, "The study of Owl's Silent Flight and Noise Reduction on Fan Vane with Bionic structure," Advances in Natural Science, vol.3, no. 2, pp.192-198, 2010.
[8] C. Melo, G. Pottker, RH Pereira, "Aerodynamic performance of air cooled condensing units", International Refrigeration and Air Conditioning conference, paper 766, 2006.
[9] S. Choi and J. Kim, "3D Modeling and Analysis Flow of Air Conditioner Outdoor Fan for Cost Reduction", Adv. Mater. Res. 264–265, 1568–1573, 2011.
[10] S. Chen, D. Wang and S. Sun, "Bionic fan optimization based on Taguchi method", J. Eng. Appl. Comput. Fluid Mech. 5, 302–314, 2011.
[11] G. V. R. Seshagiri Rao and V. V. Subba Rao, "Design of cooling fan for noise reduction using CFD", Int. J. Sci. Eng. Res. 2, 1–5, 2011.
[12] F. Perot, M. Kim, S. Moreau, M. Henner, D. Neal,"Direct acoustic prediction of a low speed axial fan," Aero- acosutics conference, AIAA 2010 - 3887.
[13] M. Gruber, M. Azarpeyvand and P. F. Joseph, "Airfoil trailing edge noise reduction by the introduction of sawtooth and slitted trailing edge geometries," in Proceedings of 20th International Congress on Acoustics, Sydney, Australia, August 2010.
[14] M. Gruber, P. Joseph, and TP. Chong. "Experimental investigation of airfoil self noise and turbulent wake reduction by the use of trailing edge serrations". In 16th AIAA/CEAS Aeroacoustics Conference, 2010.
[15] H. Y. Kim, S. H. Yoon, S. J. Moon, J. H. Lee, H. Yoo and Y. C. Im, "A study on the fan efficiency decrease on the backward flow in an axial fan with adjustable pitch blade", Int. J. Air-Cond. Refrig. 18, 101, 2010.
[16] S.M. Sun et al., "Experimental research on noise reduction of bionic axial fan blade and mechanism analysis," Journal of Jilin University, 39, pp. 382-387, 2009.

[17] Jiang CL, Tian J, Ouyang H, et. al. "Investigation of air-flow fields and aeroacoustic noise in outdoor unit for split-type air conditioner". Noise Control Eng J, 54 (3): 146-56, 2006.
[18] J.W. Lu, W.X. Wu, T.N. Chen, et al., "Fault diagnosis of the noise of a split-unit air conditioner", Machine Design and Research (China) 19 (3), 65–67, 2003.
[19] R. Parchen, W. Hoffmans, A. Gordner, and K. Braun."Reduction of airfoil self-noise at low Mach number with a serrated trailing edge". In International Congress on Sound and Vibration, 6 th, Technical Univ. of Denmark, Lyngby, Denmark, pp. 3433–3440, 1999.
[20] D.A. Quinlan, P.H. Bent, "High frequency noise generation in small axial flow fans", Journal of Sound and Vibration 218 (2), 177–204, 1998.
[21] Sato S, Kinoshita K. "Improvement in performance of propeller fans for outdoor units of air conditioners". Proceeding of the 4th Asian International Conference on Fluid Machinery, 1: 166–70, 1993.
[22] MS Howe,"Aerodynamic noise of a serrated trailing edge". Journal of Fluids and Structures, 5(1), pp. 33–45, 1991.