ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

COMPUTATIONAL FLUID DYNAMICS SIMULATION OF PROPELLER FAN

Pradeep Shetty*, Trupti P.Wani
*M-Tech student, Aeronautical Engineering Department, New Horizon College of Engineering, India
Assistant professor, Mechanical Department, New Horizon College of Engineering, India
DOI: 10.5281/zenodo.160899

ABSTRACT
Cooling appliances growing demand to cool the ambience with high efficiency requires robust condenser unit. The objective of this work is to predict and correlate the mass flow rate of propeller type axial fan used in condenser unit using Computational Fluid Dynamics (CFD) technique. The flow field is simulated with the finite element Computational Fluid Dynamics CFD solver Altair HyperWorks. The three-dimensional computational domain with Spalart-Allmaras turbulence model is considered to predict the mass flow rate. The present computation is carried out for the axial fan speed of 820 rpm for the steady state condition using moving reference frame approach. The flow rate is correlated with the test results to validate the CFD modeling approach. The correlation level found closer with tested results, hence which will help to improve the futuristic model during conceptual design itself.
KEYWORDS: Propeller type Axial Fan, Computational Fluid Dynamics, mass flow-rate, Spalart-Allmaras model.

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. The schematic diagram of condenser cooling unit [1] is as shown in fig1.

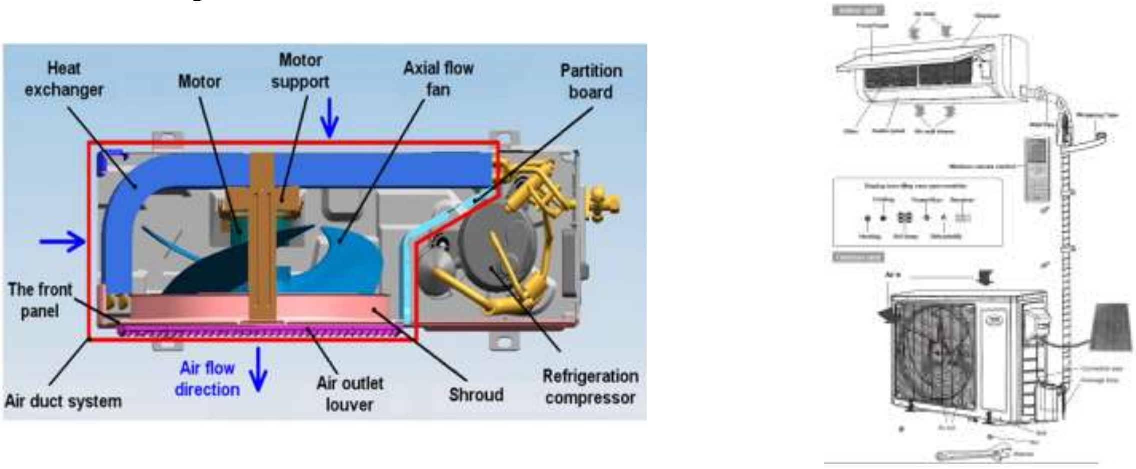


Fig1: Schematic view of condenser cooling unit

OBJECTIVE
- The main objective of the study is to predict and correlate the mass flow rate of propeller type axial fan through numerical approach. The detailed analysis using numerical simulations to enhance the performance of fan before making costly prototypes. Visualizing the distribution of pressure and the variation of velocity filed provides clear insight to understand the flow physics.

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

METHODOLOGY

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 fig 3 shows the brief methodology which we are going to proceed with the CFD modelling

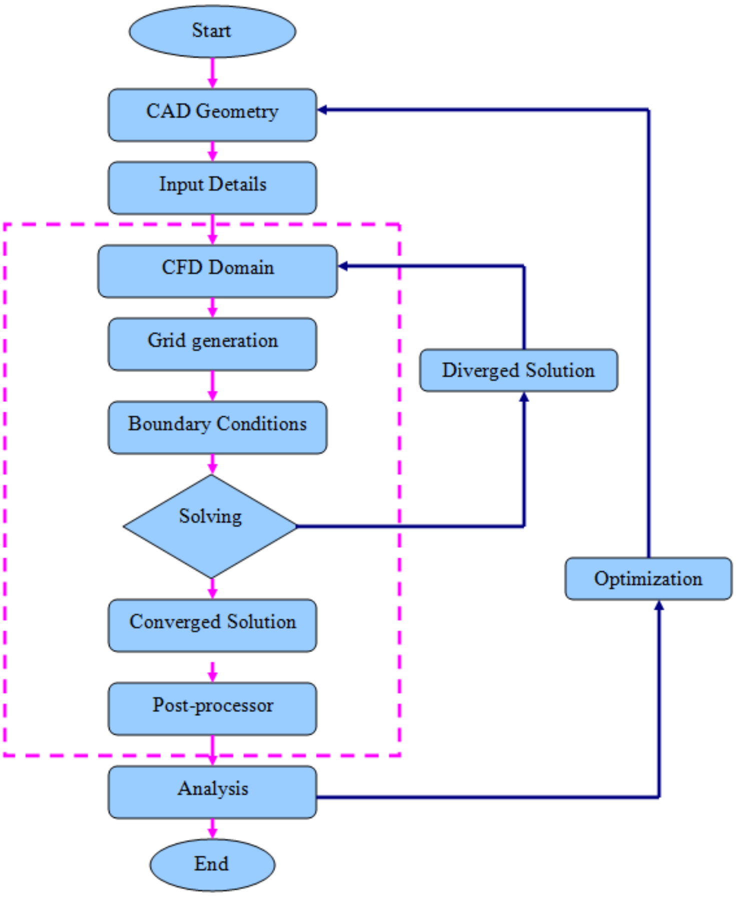


Fig2: Flowchart of Numerical Modeling

Numerical Modeling:

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.

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

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, 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

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

Geometry:

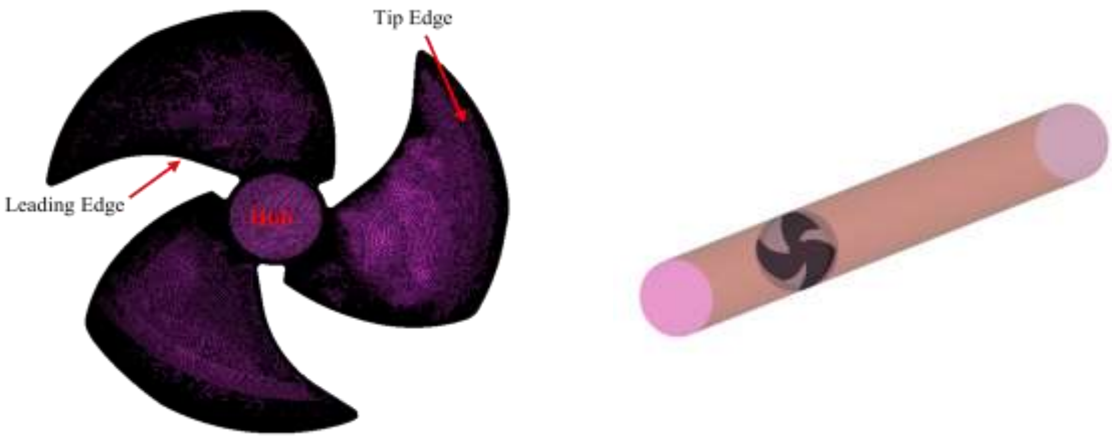


Fig3: Propeller type axial fan model [1] and computational domain

The various parts of condenser cooling unit and cooling fan model are as shown in fig 2. The diameter of fan design is 402 mm and the width of 122 mm. The motor is mounted onto the hub of 80 mm diameter. The fan consists of 3 swept blades.

Grid generation:

Fig4: Computational mesh on the fan blade and fluid domain

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. The domain of interest to compute the flow field is split into separate domain with respect to the rotational and stationary region. The discrete finite element mesh has been generated corresponding to the fan region, atmosphere and casing. The moving reference frame created around the fan region to model the rotating motion. The interface has been provided to separate the domain to maintain the relative motion between the fan and stationary region. Triangular elements for the surface modeling and tetrahedral elements for the fluid volumes are used to discretize the computational domain. The mesh distributions at the fan blade and near to boundary are fine enough to capture the flow gradients by ensuring the dimensionless wall distance Y^+ value below 4. The aspect ratio limit and mesh skewness for the mesh quality maintained below 5 and 60 on the fan blade respectively.

Boundary Conditions:

The uniform pressure condition specified at inlet and the flow is normal to the inlet plane. The atmospheric condition specified at the outlet where the diffusion flux for the variable is zero at the plane normal to the outlet boundary face. The slip wall condition for the casing by assuming the shear stress at the wall is zero. Moving reference frame assumed that the constant of revolution assigned with the enclosed volume with no explicit boundary surface.

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

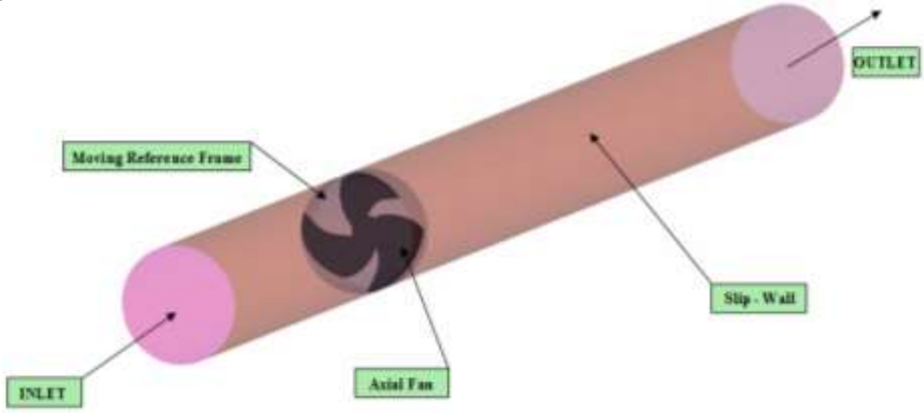


Fig5: Boundary condition details

Solver Setup:

Table 1: Solver set up details

Boundary conditions	
Inlet	Pressure inlet
Outlet	Pressure outlet
Turbulence settings	
Turbulence model	Spalart-Allmaras
Near wall treatment	Standard wall function
Air properties	
Density	1.225 [kg/m3]
Dynamic viscosity	1.75 *E-5[Pa-s]

RESULTS AND DISCUSSION

Predicting the mass flow rate:

The flow characteristics around the fan blades are studied for the speed of 820 rpm. Incompressible flow governing equations with prescribed boundary conditions predicted the mass flow rate. We have tried with various downstream domains to check the effect of sucking the air from the inlet boundary. The extended the downstream domain been finalized to make sense of flow physics. The mass flow rate and static pressure predicted numerically for the stagnation pressure inlet condition.

Table 2: Axial fan performances curve

Static Pressure, Pa	Flow rate, kg/s	Inlet Velocity, m/s
4	1.063	6.330
5	1.056	6.286
8	1.030	6.120
10	1.013	6.025
12	0.995	5.920
15	0.970	5.760
16	0.960	5.714
20	0.925	5.504
24	0.888	5.285

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

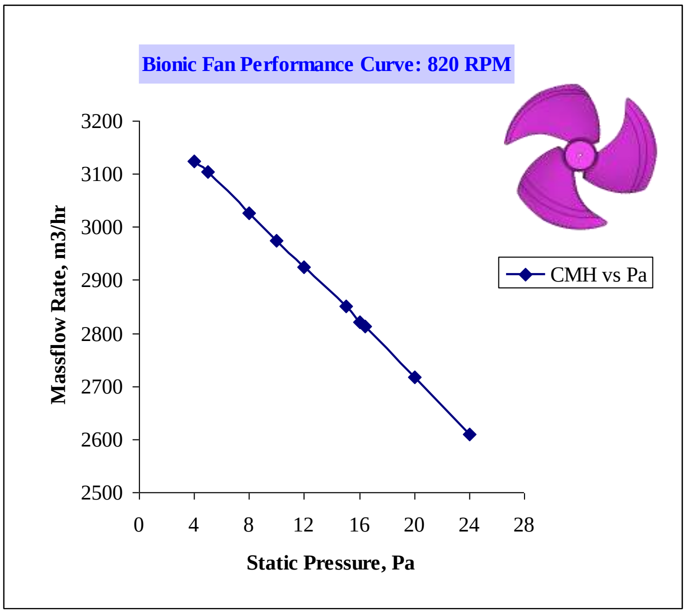
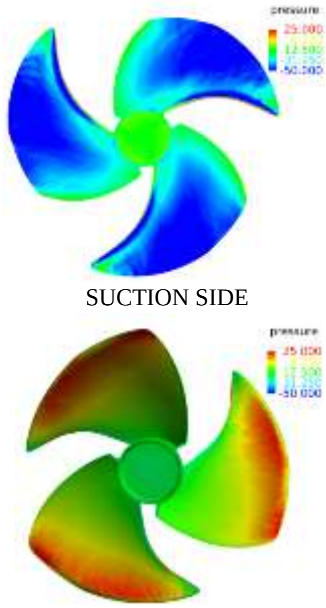


Fig 6: Mass flow rate vs Static Pressure graph



PRESSURE SIDE

Fig 7: Pressure contour on the fan blades

The uniform distribution of pressure on the axial fan blades 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 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' = $2\pi*820/60$,

$$w = 85.82 \text{ rad/s}$$

Tip velocity v = r w

$$v = 0.21 \text{ m} * 85.82 \text{ rad/s}$$
$$v = 18 \text{ m/s}$$

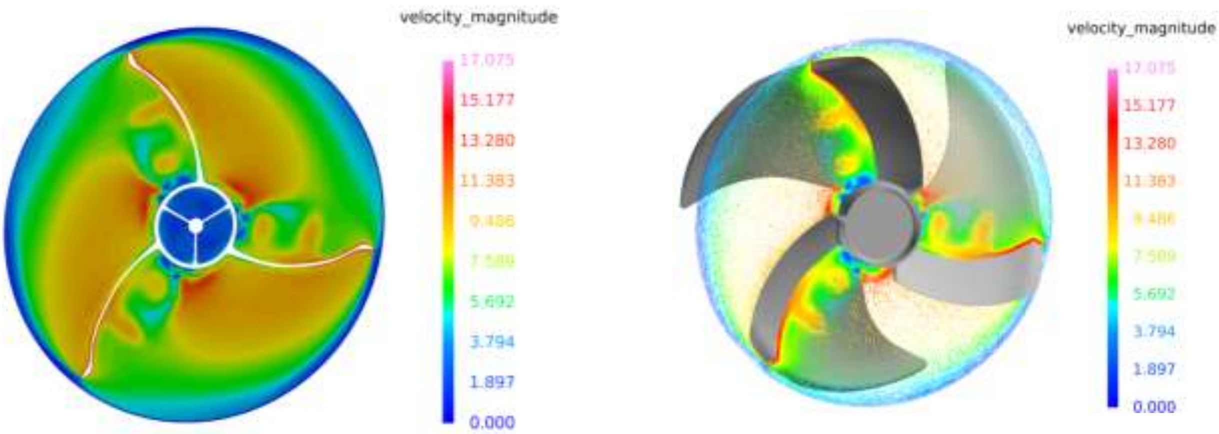


Fig 8: Cross flow pattern of axial fan velocity magnitude contour

Validation of CFD simulation by experiment:

ISSN: 2277-9655
Impact Factor: 4.116
CODEN: IJESS7

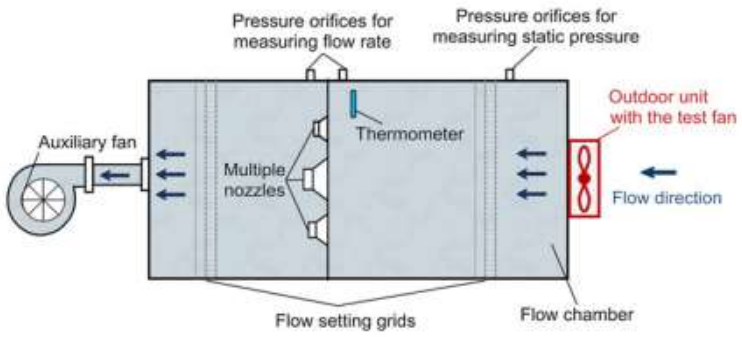


Fig 9: Typical test rig [1] for the measurement of flow rate

The result from the CFD simulation is correlated with the experimental data in-order to validate the analysis approach for the futuristic design modification. The speed vs. flow rate data has been compared with test result as listed in table 3.

Table 3: Comparison of mass flow rate between CFD and Experiment

	Static Pressure in Pa	Speed in rpm	flow rate in m3/hr
CFD	25	820	2057
Experiment	25	820	1950

Additional work was done to visualize the flow field across a single blade surface. 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.

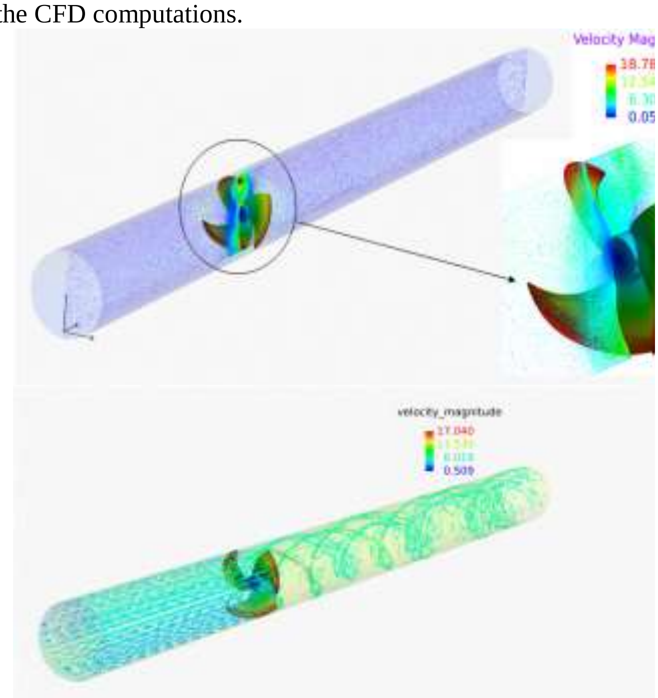


Fig 10: Flow streamline across the axial fan

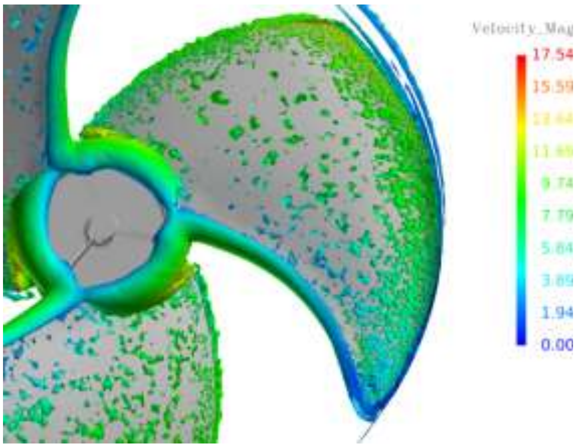


Fig 11: Iso surface plot of velocity magnitude

The flow streamline in the domain shows the representation of the flow-field produced by the axial fan as shown in fig 10. The vortices generated from the fan blade are as shown in fig 11. The reduction in vortices leads to reduce the noise level.

CONCLUSION

The aerodynamic parameters of axial fan model have been predicted virtually through Computational Fluid Dynamic approach. Mass flow rate of propeller type axial fan obtained by solving the three dimensional steady state flow governing equation through numerical technique. The moving reference frame (MRF) model has been successfully used to understand the rotational flow physics of fan. The predicted mass flow rate has been compared with test results and it found to be almost 95%. This correlation level will help to improve the performance by modifying the model during conceptual design phase.

REFERENCES

[1] 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.
[2] 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
[3] N. Gulhane, S. Patil, K. Singh,”Acoustic analysis of condenser fan of split air conditioner using numerical and experimental method,” International Journal of Air Conditioning and Refrigeration, vol. 23 (2), 2015.
[4] C. Melo, G. Pottker, RH Pereira, “Aerodynamic performance of air cooled condensing units”, International Refrigeration and Air Conditioning conference, paper 766, 2006.
[5] 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.
[6] 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.
[7] 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.
[8] 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.
[9] Dushyant Dwivedi, Devendra Singh Dandotiya, "CFD analysis of axial flow fan with skewed blades", International Journal of Emerging Technology and Advanced Engineering, Vol.3, Issue 10, 2013.