JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

INVESTIGATION OF EFFICIENCY OF FAN BLADES IN A CONDUIT SYSTEM

[1]Theegala Anjan Reddy, [2]Tadelu Pratap Kumar, [3]Govindu Lavanya, [4]K. Arun Kumar

[1]Graduate, [2]Graduate, [3]Graduate, [4]Assistant Professor
[1]Department of Aeronautical Engineering,
[1]Bharath Institute of Higher Education and Research, Chennai, India

Abstract : This project, titled "Investigation of Efficiency of Fan Blades in a Conduit System", explores the impact of varying the number of fan blades and pitch angle on the performance of a fan system. The reference paper primarily focused on changes to blade parameters such as Airfoil and blade width. In our study, we extended this investigation by modifying the number of blades and pitch angle, taking the reference design with 7 blades and considering models with 6,8 blades and 54^o,64^o pitch angle. We conducted a CFD (Computational Fluid Dynamics) analysis to simulate airflow behavior for the designed fan blades, yielding velocity contours of 1.93 m/s for the 6-blade model, 3.19 m/s for the 7-blade reference model, 3.25 m/s for the 8-blade model, 2.93 m/s for 64^o pitch angle, 2.1 m/s for 54^o pitch angle. Subsequently, we built an experimental setup to validate the CFD results, and measured the airflow velocity for the different blade configurations with Hot wire anemometer. The experimental data showed velocities of 1.87 m/s for the 6-blade model, 2.86 m/s for the 7-blade reference model, 3.09 m/s for the 8-blade model, 2.76 m/s for 64^o pitch angle, 1.89 m/s for 54^o pitch angle. Efficiency calculations and error percentage were performed, revealing notable differences in performance between the configurations. This study highlights the importance of blade count as a crucial factor in fan efficiency, offering valuable insights into the optimization of fan blade designs for improved airflow and energy performance in conduit systems.

***IndexTerms* - Axial fan blade, number of blades, Pitch angle, Volumetric flow rate, Efficiency.**

I. INTRODUCTION

Axial fans are essential components in HVAC (Heating, Ventilation, and Air Conditioning) systems, playing a crucial role in moving air efficiently through ducts, vents, and air handling units. These fans consist of blades mounted on a central hub, which rotate to create airflow along the axis of rotation. Their design and performance depend heavily on the geometry and parameters of the blades, which directly impact their efficiency, noise levels, and capacity. Key blade parameters include the blade angle, blade length, chord length, and number of blades. The blade angle influences the fan's ability to move air; a higher angle allows for greater airflow but increases the power consumption, while a smaller angle reduces energy usage but can decrease airflow capacity. The blade length also plays a critical role, as longer blades generally result in higher airflow but require more power to drive. Chord length, the distance from leading to trailing edge of a blade, affects the flow distribution across the fan's span and influences the pressure generated by the fan. The number of blades determines the fan's efficiency and noise level, with more blades offering smoother airflow but potentially leading to higher noise levels.

In HVAC systems, axial fans are chosen for their ability to handle high-volume, low- pressure applications, such as large ventilation spaces, industrial cooling, and air circulation. They are more efficient at moving large volumes of air at lower pressures, making them ideal for applications like exhaust systems and supply ventilation. However, the fan blade design must be optimized for the specific requirements of the system to ensure a balance between airflow efficiency and energy consumption. Additionally, fan blades with high solidity (more blades) tend to produce greater pressure but increase drag and reduce efficiency, while blades with lower solidity offer higher efficiency but lower pressure output. Understanding these blade parameters and their effects is vital for selecting the appropriate axial fan for an HVAC system to achieve optimal performance and energy savings.

Axial fan blades are the rotating blades on an axial fan that move air or gas in a direction parallel to the fan's shaft. The spinning motion of the blades creates a low-pressure area behind the fan and a higher-pressure area in front of it. This pressure difference causes air to be drawn in from the side and directed through the blades, resulting in the characteristic axial flow pattern. The blades are similar to those of an aircraft propeller or ship screw. The number and shape of the blades are important factors in determining the fan's performance.

By increasing the blade chord the efficiency can be improved and at high flow rate an increment in the number of blades had no effect on the produced static pressure.[1] Airfoil is used to improve the lift to drag ratio for better efficiency, by optimizing the installation angle and chord length the forward swept blade have higher total pressure efficiency.[2] FANDAS code is used for designing, optimization and analysis of fan blade.[3] Introduces the FANDAS code, a simulation tool for designing high-efficiency axial fans. The code predicts aerodynamic performance and noise levels using through-flow analysis and parametric studies.

A 10% improvement in efficiency was achieved in optimized designs compared to existing market products. The study validated FANDAS code predictions through experimental testing, showing a small percentage of relative error.[6] Inverse design method begins with a desired outcome or performance goal and works backward to determine the optimal design parameters.[7] state of the art prototyping refers to the most advanced and innovative techniques used to create prototypes, which are early models or samples of a product or system.[8] guide vanes direct the flow of the fluid in a specific direction, which helps in reducing the turbulence.[9]

Small Hub to Tip ratio helps to increase the mass flow rate and improves the efficiency of fan by minimizing the tip losses.[10] microplates are flow control device to suppress boundary layer separation on blades and thus improve the aerodynamic performance of a low speed axial flow fan.[11] The lift of the blade and performance improves in high flow rate region by thinning the blade thickness and by extending the blade chord length.[12].

II. DESIGN OF AXIAL FANS (6, 7, AND 8 BLADES AND 54^{O},64^{O}, VARYING PITCH ANGLE) IN CATIA V5

In this project, the geometric modeling of axial fan blades was carried out using CATIA V5, a parametric 3D CAD software widely used in engineering design and product development. The objective of this phase was to construct multiple axial fan blade models with varying geometric features to serve as the basis for subsequent analysis and fabrication. The two primary parameters varied during the design process were the number of blades and the pitch angle, while other parameters such as hub diameter, span length, and airfoil section were kept constant to ensure consistency across all models.

A total of five unique fan blade configurations were created with three different blade counts 6, 7, and 8 blades and with two pitch angles 54° and 64°. For each configuration, a common base structure was established, starting with the design of the hub using standard dimensions, followed by the placement of airfoil profiles at the root and tip of each blade. The airfoil geometry was imported from an external dataset and projected into CATIA using the Generative Shape Design (GSD) module.

To give the blade a realistic aerodynamic form, a twist angle was applied along the span from root to tip. This was done by rotating the airfoil sketches at different radial locations based on the required blade setting. The multi-section surface tool was then used to generate a smooth and continuous blade surface by connecting the root and tip airfoils along a guide curve. This ensured proper aerodynamic shape and manufacturability.

After designing the blade around the hub, with the use of circular pattern tool 6 blade axial fan is produced and again repeating the procedure of design but with varying number of blades and varying in the pitch angles with the reference model. The blade parameters are shown in the table 2.1 and table 2.2.

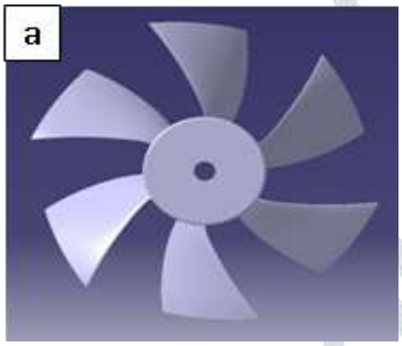

Figure:2.1 6 Blade

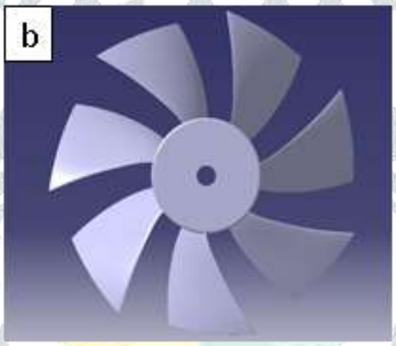

Figure: 2.2 7 Blade

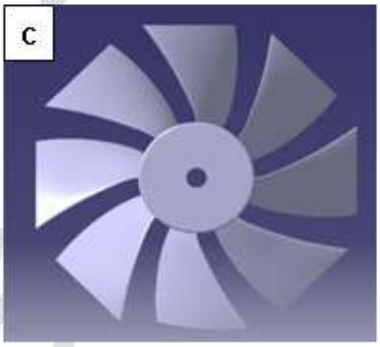

Figure:2.3 8 Blade

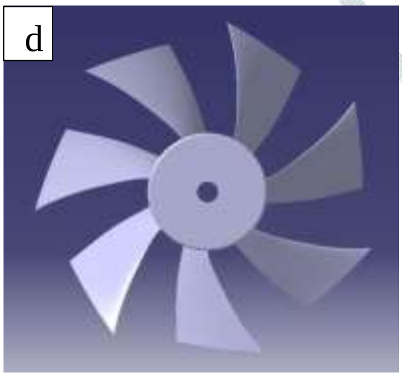

Figure: 2.4 54º Pitch Angle Fan Blade

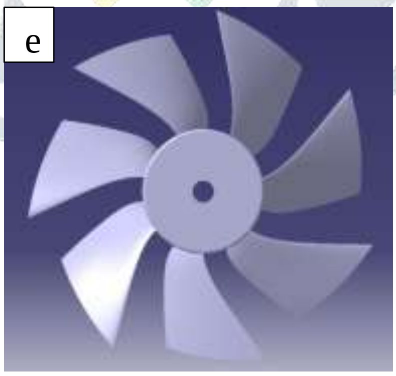

Figure: 2.5 64º Pitch Angle Fan Blade

Table 2.1 Design parameters of axial fan blade with varying number of blades

Table 2.2 Design parameters of axial fan blade with varying pitch angle

Parameters	Dimensions
Airfoil	NACA 5608
Blade root chord (Lr)	19.38mm
Blade tip chord (Lt)	37.86mm
Pitch angle	59^o
Twist angle	26.01^o
Number of Blades	6,7,8
Hub Diameter (D_{hub})	40mm
Fan diameter(D)	110mm
Hub height(H)	15mm

Parameters	DI
Airfoil	NACA 5608
Blade root chord (Lr)	19.38mm
Blade Tip chord (Lt)	37.86mm
Pitch angle	54^o,64^o
Twist angle	26.01^o
Number of Blades	7
Hub Diameter (D_{hub})	40mm
Fan diameter(D)	110mm
Hub height(H)	15mm

III. COMPUTATIONAL METHODOLOGY

The computational methodology for the design of the axial fans with 6, 7, 8 blades and 54^o,64^o Pitch Angle involved using CATIA V5 for modeling. After the fan designs were created, CFD analysis was performed in ANSYS Fluent within ANSYS Workbench. The analysis was conducted at a rotational speed of 2500 RPM, and the velocity distribution at various locations within a conduit pipe was evaluated. This helped in assessing the performance of the axial fan models and understanding how each configuration affected the airflow within the system. The CFD analysis for the designed axial fan blades is carried out through fluid flow(fluent). The analysis is conducted in a conduit system which is a 6 inch in diameter. The fan blade is placed 50mm from the inlet of the conduit system. A rotating region (or sliding mesh zone) is created around the fan instead of directly assigning rotation to the blade is because of how Fluent handles rotating frames of reference and ensures proper momentum transfer to the surrounding fluid. The Shear Stress Transport (SST) k-ω turbulence model was employed to simulate the airflow through axial fan blades. This model was selected due to its superior capability in accurately capturing boundary layer behavior and flow separation phenomena, which are critical for predicting the performance of axial fans.

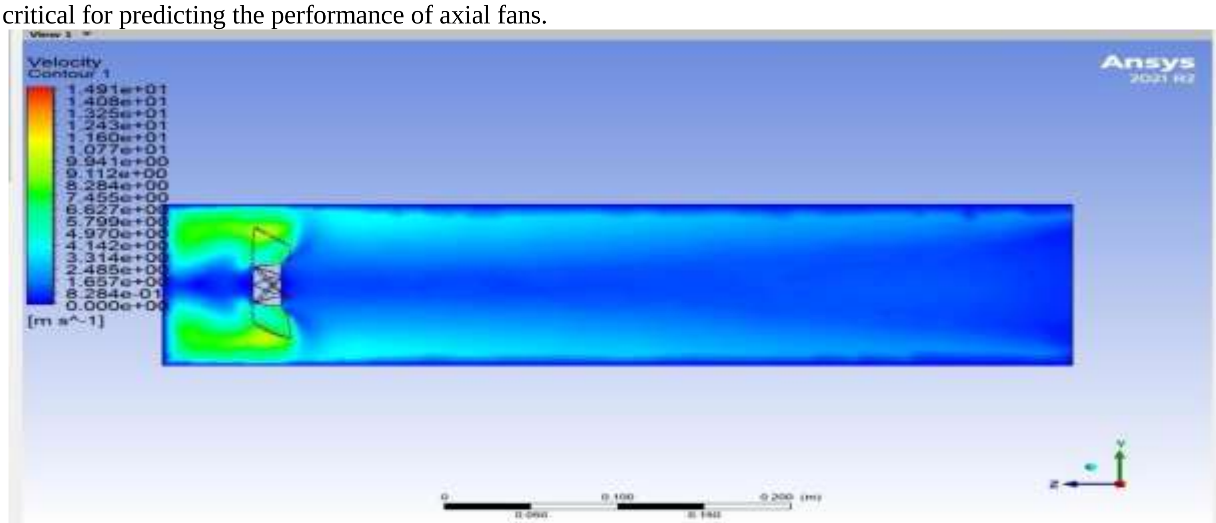

Figure: 3.1 velocity contour of 6 blade axial fan

Figure (3.1) shows the velocity contour of 6 blade axial fan where the Tip velocity is 14.91m/s. The velocity is decreasing from the upstream to the downstream because of the turbulence which can cause a temporary reduction in axial velocity as energy spreads in radial and swirl directions. The highest velocity achieved in the conduit after the placement of the fan blade is 1.93m/s and the lowest is 0.7m/s.

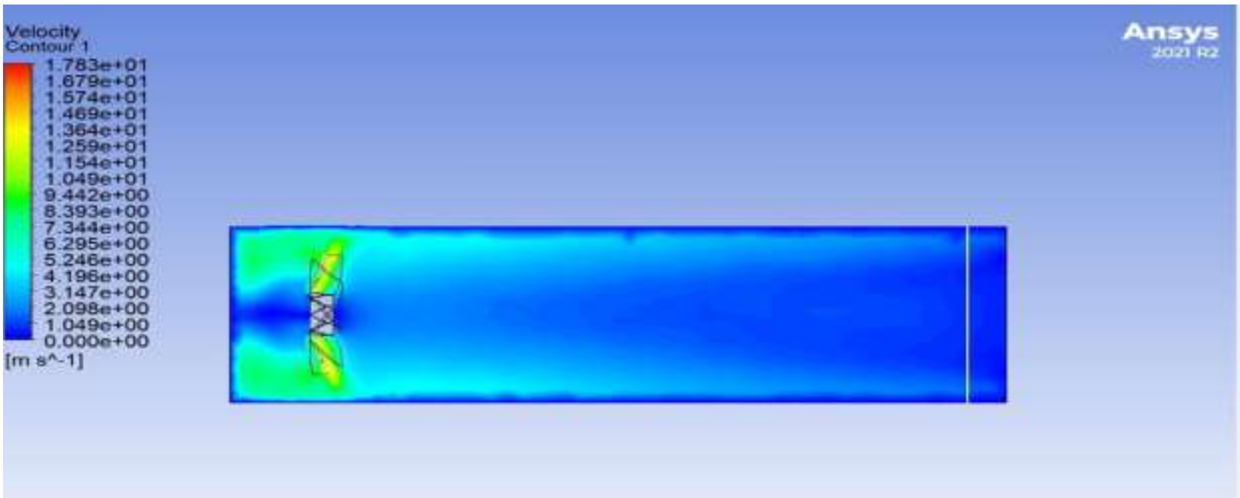

Figure: 3.2 Velocity contour of 8 blade Axial fan

Figure (3.2) shows the velocity contour of 8 blade axial fan where the Tip velocity is 17.83m/s. The velocity is decreasing from the upstream to the downstream because of the turbulence which can cause a temporary reduction in axial velocity as energy spreads in radial and swirl directions. The highest velocity achieved in the conduit after the placement of the fan blade is 3.25m/s and the lowest is 1.05m/s.

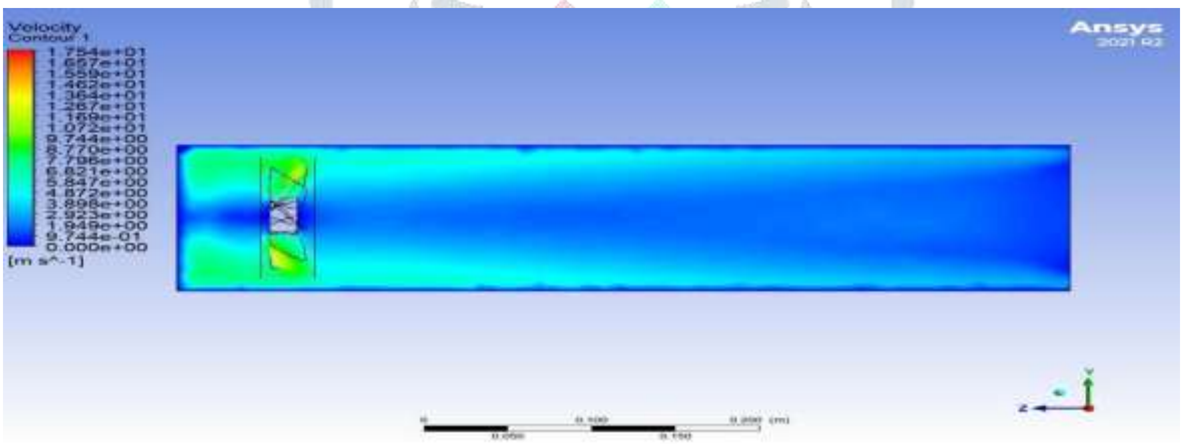

Figure: 3.3 velocity Contour of 7 Blade axial fan

Figure (3.3) shows the velocity contour of 7 blade axial fan where the Tip velocity is 17.54m/s. The velocity is decreasing from the upstream to the downstream because of the turbulence which can cause a temporary reduction in axial velocity as energy spreads in radial and swirl directions. The highest velocity achieved in the conduit after the placement of the fan blade is 3.19m/s and the lowest is 1.13m/s.

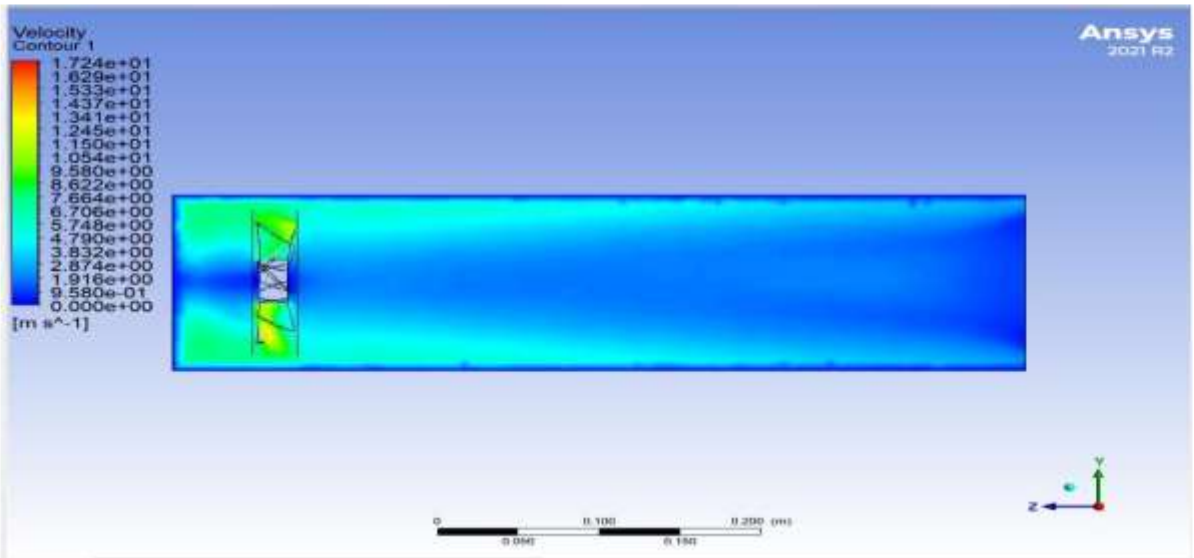

Figure: 3.4 velocity Contour of 54° pitch angle

Figure (3.4) shows the velocity contour of 54° Pitch angle axial fan where the Tip velocity is 17.24m/s. The velocity is decreasing from the upstream to the downstream because of the turbulence which can cause a temporary reduction in axial velocity as energy spreads in radial and swirl directions. The highest velocity achieved in the conduit after the placement of the fan blade is 2.1m/s and the lowest is 0.9m/s.

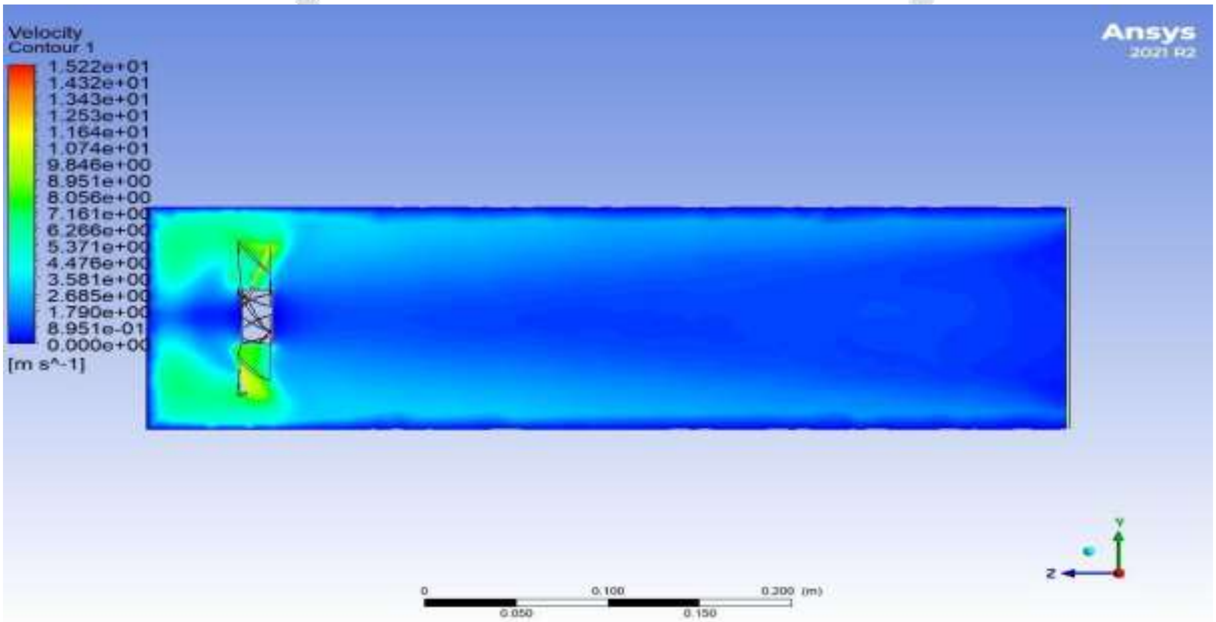

Figure 3.5 velocity Contour of 64° pitch angle

Figure (3.5) shows the velocity contour of 64° Pitch angle axial fan where the Tip velocity is 15.22m/s. The velocity is decreasing from the upstream to the downstream because of the turbulence which can cause a temporary reduction in axial velocity as energy spreads in radial and swirl directions. The highest velocity achieved in the conduit after the placement of the fan blade is 2.93m/s and the lowest is 0.97m/s.

IV. CFD RESULTS

The results from the CFD analysis were obtained by analyzing the velocity contours of the 6, 7, 8 blade and 54°,64° Pitch Angle axial fan models. These contours provided a detailed view of the velocity distribution within the system. To further understand the performance at different locations in the conduit pipe, we plotted graphs showing the velocity at various positions along the pipe, specifically at distances of 100 mm, 200 mm, 300 mm, 400 mm, and 500 mm from the fan inlet.

Table 4.1 velocity Calculation using CFD

	Location				
Blade Count	Velocity at 100mm (m/s)	Velocity at 200mm (m/s)	Velocity at 300mm (m/s)	Velocity at 400mm (m/s)	Velocity at 500mm (m/s)
6 Blade	1.93	1.77	1.47	1.23	0.7
7 Blade	3.19	2.61	2.15	1.79	1.13
8 Blade	3.25	2.71	2.23	1.83	1.05
54°	2.1	1.7	1.42	1.2	0.9
64°	2.93	2.33	1.67	1.26	0.97

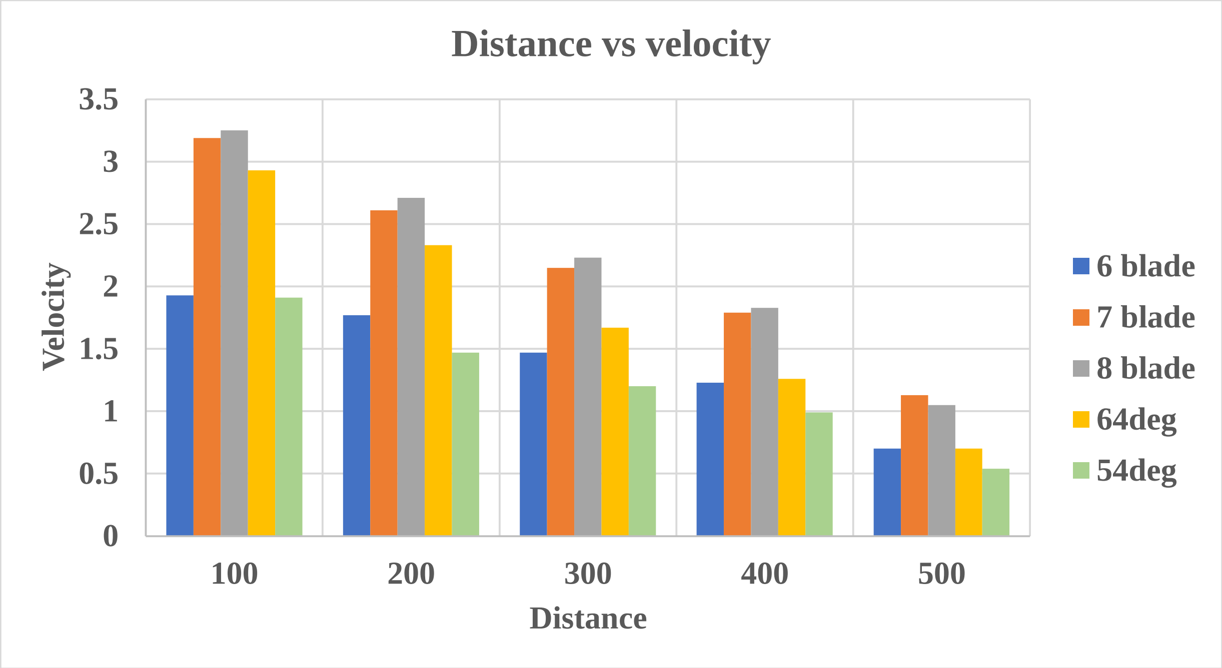

Figure: 4.1 Distance vs Velocity

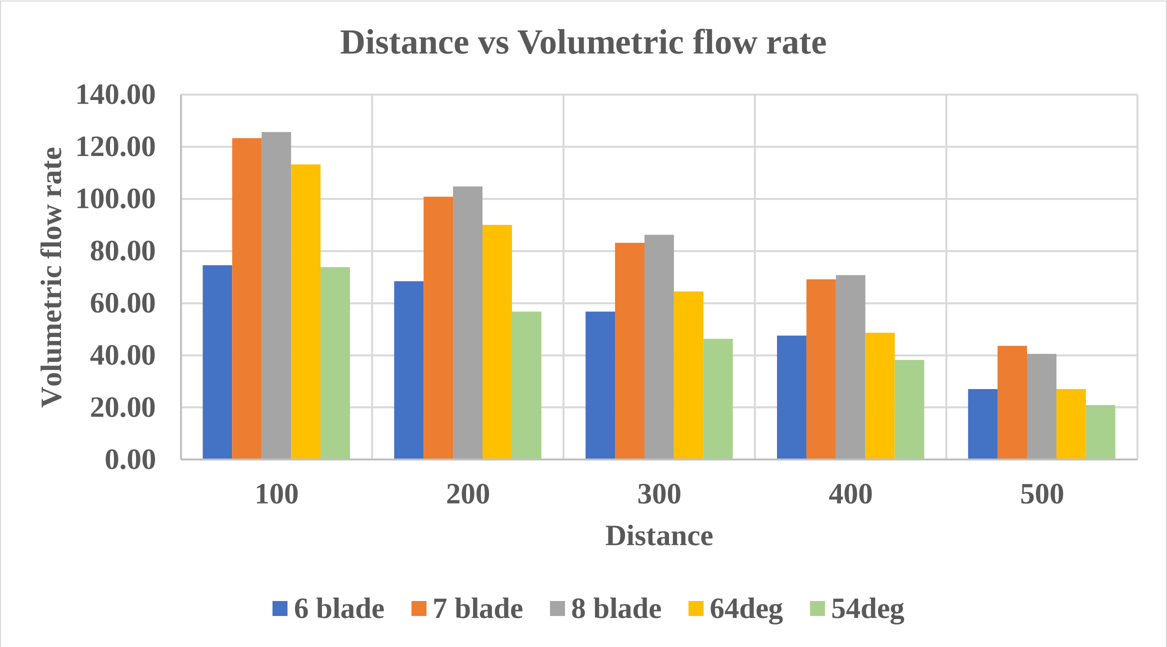

Figure:4.2 Distance vs Volumetric flow rate

V. EXPERIMENTAL ANALYSIS

Figure:5.1 Flycat 5010- 750 KV

Figure:5.2 ESC

Figure:5.3 Battery

The experimental setup was designed to ensure precise control over the operating conditions of the axial fan system. A Flycat 5010–750 KV brushless DC motor (Figure 5.1) was selected as the prime mover because of its ability to deliver high torque at a moderate KV rating, making it well-suited for driving large-diameter fan blades at controlled rotational speeds without excessive power losses. The motor was operated through an Electronic Speed Controller (ESC) (Figure 5.2), which provided smooth regulation of the motor speed by adjusting the supplied voltage and current in response to throttle input. Power for the motor-ESC system was supplied by a 3-cell, 1500 mAh Lithium-Polymer (Li-Po) battery (Figure 5.3), chosen for its high energy density and ability to deliver stable current output, thereby ensuring uninterrupted operation throughout the testing period. Together, this configuration provided reliable and consistent fan blade rotation for both experimental validation and performance assessment.

To monitor and control the fan speed, a digital tachometer was used, allowing the fan to be set precisely at 2500 RPM. This reference speed was maintained consistently across all experimental trials to ensure uniformity in data collection. To evaluate the airflow behavior within the conduit, a hot-wire anemometer was employed for velocity measurements. The probe was positioned at axial locations of 100 mm, 200 mm, 300 mm, 400 mm, and 500 mm from the fan inlet along the conduit centerline. The measurement points were chosen to record how the airflow velocity changed as it moved through the conduit. The data from the hot-wire anemometer was then compared with the CFD results to check whether the simulations matched the actual experimental performance.

Figure:5.4 Hot wire anemometer

Figure:5.5 Tachometer

Figure: 5.6 6 Blade

Figure:5.7 7 Blade

Figure:5.8 8 Blade

Figure:5.9 54^{o} Pitch Angle

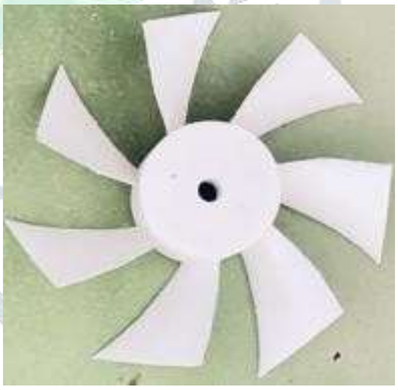

Figure:5.10 64^{o} Pitch Angle

The above figures show the physically fabricated models of axial fan blades created using 3D printing technology with PLA White (Polylactic Acid) filament. After conducting the CFD analysis of the 6, 7, 8blade and 54^{o},64^{o} axial fan models using ANSYS Fluent, an experimental setup is constructed to validate the simulation results. The experimental analysis was carried out in a conduit system, specifically a pipe, where each fan model was tested under controlled conditions. During the tests, the velocity is measured at different locations within the pipe using a hot wire anemometer, allowing to capture detailed airflow data. Additionally, the rotational speed (RPM) of the fan was measured using a laser tachometer to ensure consistency with the CFD simulations, which were conducted at 2500 RPM. The results from the experimental analysis were then compared with the CFD results.

Figure:5.11 Experimental setup

Table 5.1 Validation of CFD results & Experimental results.

No. of Blades	Location (mm)	CFD Velocity(m/s)	Exp Velocity(m/s)	Error%
	100	3.19	2.86	10.34
	200	2.61	2.33	10.73
7 Blade	300	2.15	1.92	10.70
	400	1.79	1.54	13.97
	500	1.13	1.26	11.50
	100	1.93	1.87	3.11
	200	1.77	1.59	10.17
6 Blade	300	1.47	1.36	7.48
	400	1.23	1.11	9.76
	500	0.7	0.75	7.14
	100	3.25	3.09	4.92
	200	2.71	2.53	6.64
8 Blade	300	2.23	2.09	6.28
	400	1.83	1.69	7.65
	500	1.05	1.12	6.67
	100	2.1	1.89	10.00
	200	1.7	1.62	4.71
54deg	300	1.42	1.35	4.93
	400	1.2	1.15	4.17
	500	0.9	1.04	13.46
	100	2.93	2.76	5.80
	200	2.33	2.21	5.15
64deg	300	1.67	1.59	4.79
	400	1.26	1.23	2.38
	500	0.97	1.12	15.46

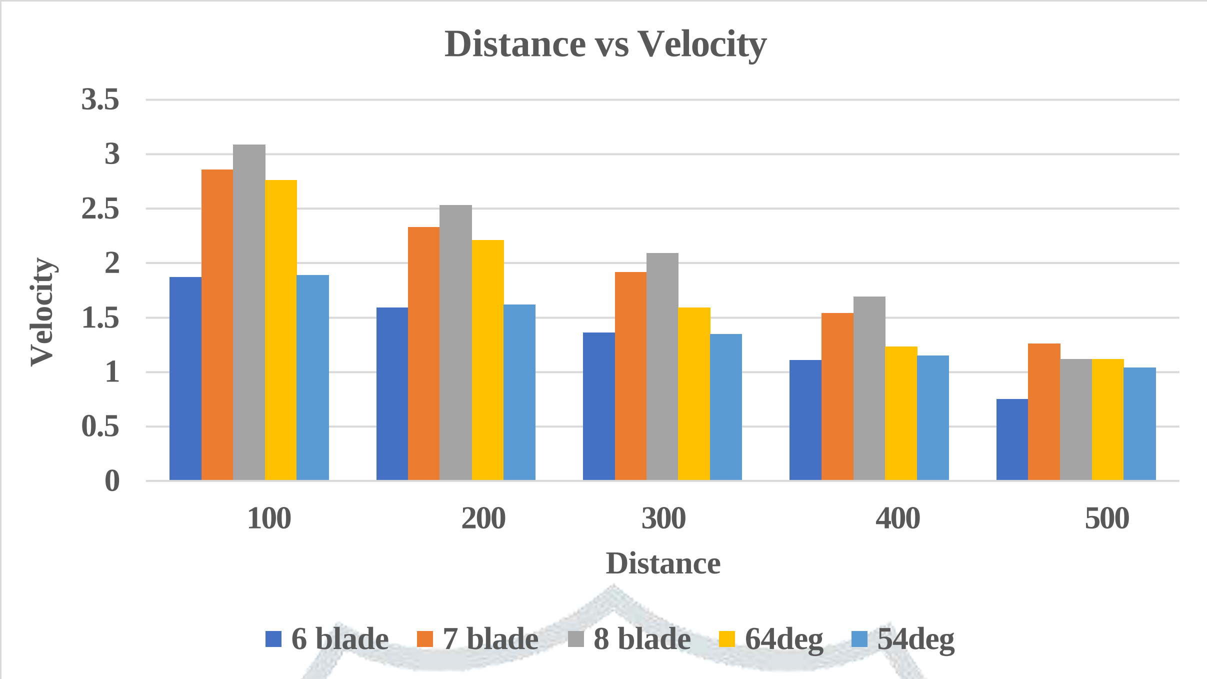

Figure:5.11 Distance vs Velocity

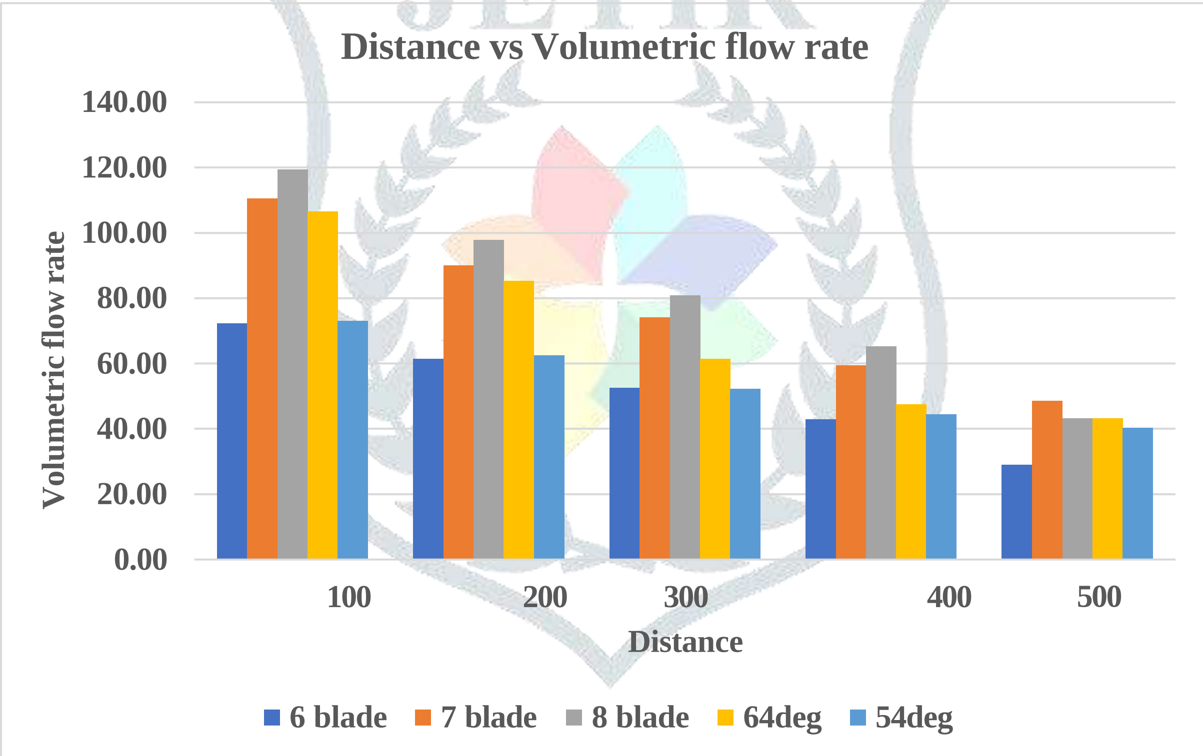

Figure:5.12 Distance vs Volumetric flow rate

VI. conclusion

This project, Investigation of Efficiency of Fan Blades in a Conduit System, comprehensively explored how variations in the number of fan blades and pitch angles influence the efficiency and airflow characteristics of axial fans. By utilizing a combination of Computational Fluid Dynamics (CFD) simulations and experimental validation, the study successfully identified key performance trends and validated the reliability of simulated results. The CFD analysis revealed that increasing the number of blades generally leads to improved airflow velocity and static pressure. Among the tested configurations, the 8-blade fan exhibited the highest velocity contours in simulations, peaking at 3.25 m/s, followed by the 7-blade reference model at 3.19 m/s, and the 6-blade model at 1.93 m/s. Experimental results closely supported these findings, with the 8-blade fan achieving 3.09 m/s, further validating the accuracy of the CFD approach. Additionally, pitch angle variations showed a clear correlation with airflow efficiency, where an angle of 64° performed better than 54°, emphasizing the need for optimal angle tuning. The fabrication of the fan blades using PLA and 3D printing techniques demonstrated the practical feasibility of rapid prototyping for aerodynamic components. The use of reliable measuring tools such as a hot wire anemometer and laser tachometer ensured accurate velocity and RPM measurements, strengthening the validity of experimental observations. A significant outcome of this project is the confirmation that both blade count and pitch angle are interdependent variables that critically influence the fan's aerodynamic efficiency. While an increase in blade number enhances pressure generation, it may also introduce challenges such as increased drag or power consumption if not optimized properly. Therefore, a balance must be struck between aerodynamic performance and energy efficiency.

VII. Acknowledgment

The author expresses sincere gratitude to the Department of Aeronautical Engineering, Bharath Institute of Higher Education and Research, Chennai, for providing the facilities and resources necessary to carry out this research work. Special appreciation is extended to K. Arun Kumar, Assistant Professor, for valuable guidance, encouragement, and continuous support throughout the course of this project.

REFERENCE

[1] N. Rajabi et al., "Effect of blade design parameters on air flow through an axial fan," International Journal of Engineering, vol. 10, no. 30, pp. 1583–1591, 2017.
[2] Y. Wu and D. Huang, "Optimization design of axial fan blade," Journal of the Chinese Institute of Engineers, vol. 42, no. 6, pp. 473–478, 2019.
[3] C. Lee et al., "Optimal design and the CFD verification of a high efficiency axial flow fan," International Journal of Recent Advances in Mechanical Engineering (IJMECH), vol. 12, no. 2/3, pp. 35–47, 2023.
[4] J. J. Corona Jr et al., "The best efficiency point of an axial fan at low-pressure conditions," Advances in Mechanical Engineering, vol. 13, no. 3, pp. 1–17, 2021.
[5] C.-H. Huang and C.-W. Gau, "An optimal design for axial-flow fan blade: theoretical and experimental studies," Journal of Mechanical Science and Technology, vol. 26, no. 2, pp. 427–436, 2012.
[6] C. Lee et al., "Development of high-efficiency axial flow fan using the FANDAS code," International Journal of Smart Grid and Clean Energy, vol. 7, no. 4, pp. 255–270, 2018.
[7] K.-Y. Lee et al., "Design of axial fan using inverse design method," Journal of Mechanical Science and Technology, vol. 22, pp. 1883–1888, 2008.
[8] M. Szelka et al., "Study of the blade shape impact on the improvement of fan efficiency based on state-of-the-art prototyping methods," Energies, vol. 16, no. 542, pp. 1–15, 2023.
[9] K. M. Munisamy et al., "Experimental investigation on design enhancement of axial fan using fixed guide vane," IOP Conference Series: Materials Science and Engineering, vol. 88, no. 012026, pp. 1–7, 2015.
[10] J. Wang and N. P. Kruyt, "Design for high efficiency of low-pressure axial fans with small hub-to-tip diameter ratio by the vortex distribution method," Journal of Fluids Engineering, vol. 144, no. 081201, pp. 1–15, 2022.
[11] D. Luo et al., "A computational study on the performance improvement of low-speed axial flow fans with microplates," Journal of Applied Fluid Mechanics, vol. 10, no. 6, pp. 1537–1546, 2017.
[12] T. Hirano et al., "Study on performance evaluation of small axial fan," Open Journal of Fluid Dynamics, vol. 7, pp. 546–556, 2017.
[13] L. H. K. Cheteu et al., "Analysis and simulation of axial fan using CFD," International Journal of Mechanical and Industrial Technology, vol. 9, no. 2, pp. 11–17, 2022.