International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 10, Issue 5, pp 200-203, May-2026

https://doi.org/10.47001/IRJIET/2026.105026

Design and Analysis of Centrifugal Fan Blade Inspired By Teal Bird Wings

[1]Vasudha Shinde, [2]Mehak Shaikh, [3]Ruchi Yadav, [4]M. U. Gan

[1,2,3]Student, Department of Mechanical Engineering, AISSMS College of Engineering, Pune, India

[4]Professor, Department of Mechanical Engineering, AISSMS College of Engineering, Pune, India

***Abstract* - In recent years, the increasing demand for energy efficient systems has emphasized the need to enhance the performance of centrifugal fans. This paper presents the design and analysis of a centrifugal fan blade inspired by the natural structure of avian wings. Bird wings exhibit unique aerodynamic characteristics such as varying curvature, thickness, and twist along their span, which contribute to efficient airflow management. These biomimetic features are adapted and implemented in the design of a centrifugal fan blade using SolidWorks.**

The developed model is further analyzed using ANSYS Fluent to evaluate airflow behavior and pressure distribution. The simulation results indicate improved airflow uniformity and enhanced aerodynamic performance compared to conventional fan blade designs. The study demonstrates that nature-inspired design approaches can provide simple, efficient, and innovative solutions for improving the performance of engineering systems.

Keywords: Centrifugal fan, Biomimicry, Teal bird wing, Blade design, Computational Fluid Dynamics (CFD), ANSYS, SolidWorks, Aerodynamic performance, Flow analysis, Efficiency improvement.

I. INTRODUCTION

Centrifugal fans are widely used in both industrial and domestic applications such as cooling systems, ventilation, and air conditioning. These devices function by converting mechanical energy into airflow. However, conventional fan designs often suffer from performance limitations due to turbulence and flow separation, which lead to energy losses and reduced efficiency. These issues are primarily associated with suboptimal blade geometry.

To overcome these limitations, engineers are increasingly exploring the concept of bio mimicry, where design inspiration is drawn from natural systems. Nature has optimized structures over millions of years, resulting in highly efficient and robust designs. One such example is bird wings, which are naturally adapted to ensure smooth airflow and minimize drag.

Among various bird species, teal birds exhibit a unique wing shape that promotes stable aerodynamic performance and reduced drag during flight. Inspired by these characteristics, this study focuses on incorporating teal bird wing geometry into the design of a centrifugal fan blade.

In this work, a bio mimetic centrifugal fan blade was designed by integrating aerodynamic features inspired by teal bird wings. In addition, optimization techniques such as genetic algorithms were considered to refine the geometry of the blades for improved performance. A prototype of the modified fan was manufactured and its performance was evaluated against a conventional fan.

Key performance parameters such as airflow rate, power consumption, and efficiency were analyzed using multiple prototype models. The results provide valuable insight into the feasibility and effectiveness of bio mimetic design in improving centrifugal fan performance.

II. METHODOLOGY

This research aims to enhance the efficiency of a centrifugal fan through the application of biomimicry principles derived from avian wing geometries. The methodology follows a structured sequence involving design, modelling, simulation, and analysis.

2.1 Problem Statement

Conventional centrifugal fans often experience performance degradation due to turbulence, flow separation, and inefficient blade geometry. These factors contribute to reduced aerodynamic efficiency and increased energy consumption. Therefore, there is a need to develop an improved blade design by incorporating biomimetic principles inspired by bird wings, with the aim of enhancing airflow characteristics and overall fan performance.

2.2 Bio-mimetic Inspiration

Bio-mimetic design principles are derived from the examination of natural wing structures that exhibit exceptional airflow efficiency. Bird wings, in particular, have optimized shapes that reduce drag and maintain steady flow even under

© 2026 IRJIET All Rights Reserved

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 10, Issue 5, pp 200-203, May-2026

https://doi.org/10.47001/IRJIET/2026.105026

varying conditions. The curvature and tapered, streamlined shape of the wing play a crucial role in guiding airflow smoothly and preventing flow separation.

This study identifies and incorporates essential geometric characteristics, including camber, leading-edge contour, and surface smoothness, into the design of the centrifugal fan blade. These features help control the flow direction and ensure a more uniform pressure distribution across the blade surface. Additionally, the gradual curvature observed in natural wings is implemented to enhance flow attachment and minimize turbulence.

2.3 CAD Modelling

SolidWorks software is used to make a geometric model of the centrifugal fan blade. To get an efficient aerodynamic profile, the blade design is based on the coordinates of the airfoil. The chosen airfoil shape is brought in and scaled to fit the design's needs.

Using these coordinates, the blade profile is made and then improved by setting the right curvature and thickness distribution. Then, the three-dimensional model of the blade is made by using the right features, like extrusion and lofting. The blade is properly aligned and oriented to meet the design needs of the centrifugal fan.

To keep the geometry consistent, the whole fan, including the impeller and casing, is also modelled. The finished CAD model is ready for more testing and analysis.

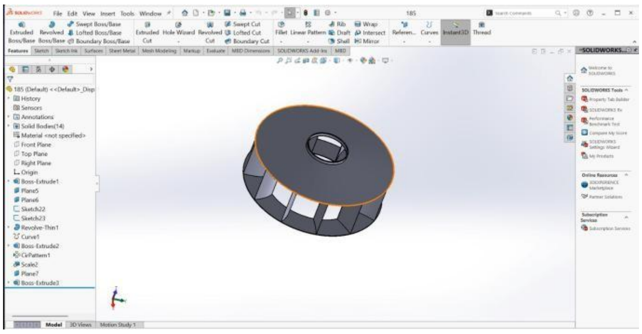

Figure 1: Design of centrifugal fan impeller

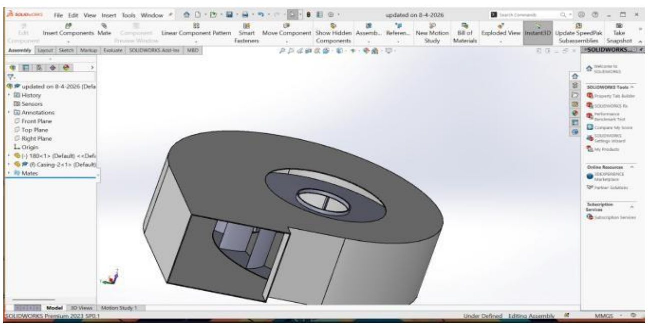

Figure 2: Assembly of centrifugal fan

2.4 CFD Analysis

Computational Fluid Dynamics (CFD) analysis was carried out using ANSYS Fluent to evaluate the aerodynamic performance of the proposed centrifugal fan. The three-dimensional CAD model developed in SolidWorks was imported into ANSYS Workbench for simulation.

A refined mesh was generated to discretize the fluid domain, ensuring accurate capture of flow characteristics, particularly in the blade and volute regions. Appropriate boundary conditions were applied, including a velocity inlet and a pressure outlet. Air was considered as the working fluid under steadystate conditions. A suitable turbulence model was employed to account for the effects of turbulent flow.

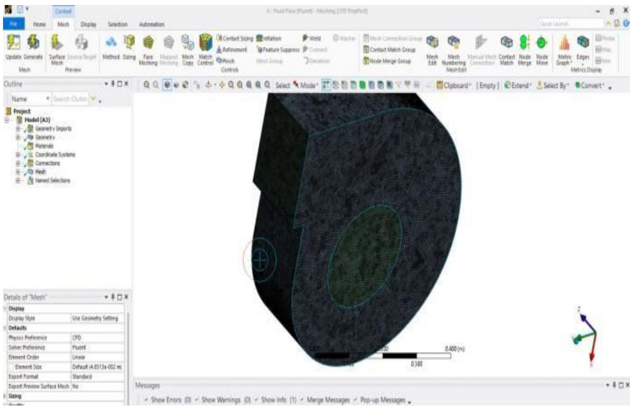

Figure 3: Meshing

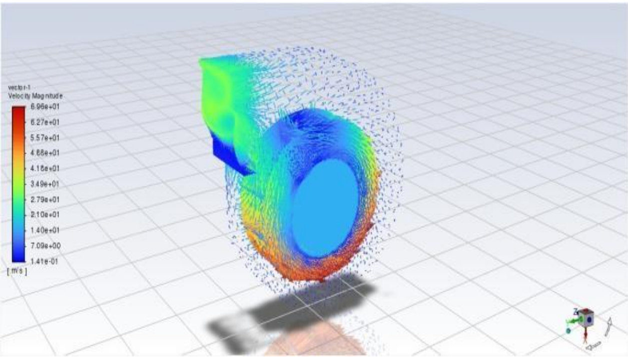

Figure 4: Post Processing on CFD 1

The CAD modeling of the centrifugal fan includes the design of the impeller and the complete assembly. The impeller is developed by defining blade geometry such as angle, curvature, and thickness, while the assembly integrates all components to ensure proper alignment and functionality.

The simulation results indicate that the velocity distribution within the fan ranges from approximately 0.14 m/s to 69.6 m/s, with maximum velocities observed near the blade tips and outlet region. The static pressure varies from

© 2026 IRJIET All Rights Reserved

International Research Journal of Innovations in Engineering and Technology (IRJIET)

ISSN (online): 2581-3048

Volume 10, Issue 5, pp 200-203, May-2026

https://doi.org/10.47001/IRJIET/2026.105026

approximately 453 Pa to 1690 Pa, indicating a significant pressure rise across the impeller.

Streamline analysis demonstrates smooth airflow with minimal flow separation within the volute casing. The results confirm that the bio-mimetic blade design improves flow behavior, enhances aerodynamic efficiency, and reduces energy losses when compared to conventional centrifugal fan designs.

2.5 Applications

The proposed bio-mimetic centrifugal fan design can be effectively utilized in various applications where efficient airflow and energy conservation are critical.

The design is particularly suitable for Heating, Ventilation, and Air Conditioning (HVAC) systems, where it ensures improved air circulation while reducing power consumption. It can also be used in cooling applications such as electronic devices, automotive radiators, and industrial machinery, where maintaining optimal temperature is essential for proper functioning.

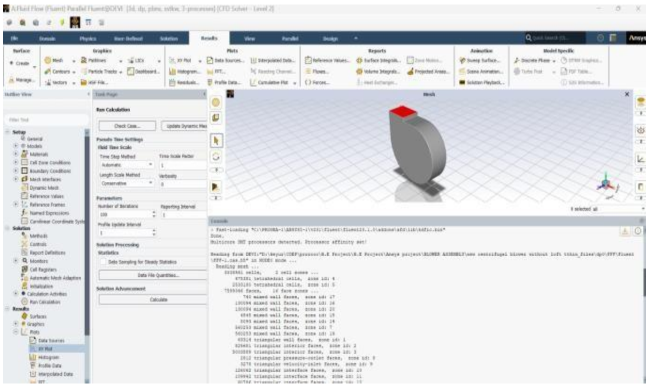

Figure 5: Iteration of CFD

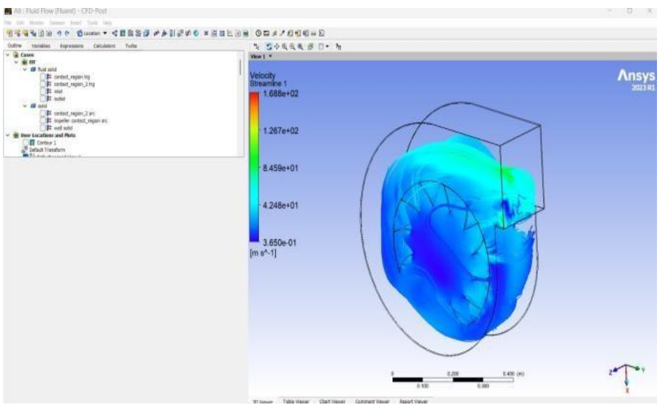

Figure 6: Post Processing on CFD 2

Furthermore, the fan design is applicable in large-scale ventilation systems, including:

1) Tunnel ventilation system
2) Mine ventilation system
3) Building ventilation system

In addition, the unique blade geometry inspired by teal bird wings contributes to reduced noise levels. This makes the fan highly suitable for use in residential and office environments where quiet operation is important.

III. FUTURE SCOPE

The proposed biomimetic centrifugal fan design can be further enhanced using advanced optimization techniques such as genetic algorithms and machine learning methods. Experimental testing of physical prototypes can be conducted to validate and improve the accuracy of simulation results.

Future research may explore different biological inspirations to develop innovative blade geometries with improved aerodynamic performance. The analysis can also be extended to various operating conditions, including variations in rotational speed and pressure.

The proposed design has potential applications in HVAC systems, cooling technologies, and renewable energy systems. Overall, this approach offers significant potential for developing energy-efficient and eco-friendly solutions for industrial applications.

IV. RESULTS AND DISCUSSIONS

This section presents the outcomes of the Computational Fluid Dynamics (CFD) analysis conducted on the biomimetic centrifugal fan blade designed using SolidWorks and simulated in ANSYS Fluent. The results are discussed in terms of velocity distribution, pressure distribution, streamline behavior, and overall aerodynamic performance, followed by a comparative evaluation against conventional fan blade designs.

V. CONCLUSION

This study showed the design and performance evaluation of a biomimetic centrifugal fan inspired by bird wing shapes. The CAD model created in SolidWorks was analyzed with ANSYS Fluent to examine airflow behavior and performance traits.

The CFD results revealed better flow uniformity, lower turbulence intensity, and improved pressure distribution compared to standard designs. The biomimetic blade shape led to greater aerodynamic efficiency by reducing energy losses and optimizing air movement in the system.

© 2026 IRJIET All Rights Reserved

ISSN (online): 2581-3048
Volume 10, Issue 5, pp 200-203, May-2026
https://doi.org/10.47001/IRJIET/2026.105026

Overall, using biomimicry along with computational analysis proved to be an effective way to improve fan performance. The findings confirm that nature-inspired design changes can greatly enhance the efficiency and operational characteristics of centrifugal fans.

This work lays a solid foundation for further research and practical use in industrial ventilation and cooling applications.

REFERENCES

[1] Tin Min Soe, Lwin Myo (2018); Impeller Design of Centrifugal Blower for 40 kW Wood Chips Gasifier; *International Journal of Scientific and Research Publications (IJSRP)*, 8(7), ISSN: 2250-3153.

[2] Neise, W. (1976). Noise reduction in centrifugal fans: a literature survey. *Journal of Sound and Vibration*, 45(3), 375-403.

[3] Bhore, N., Lohani, U. C., Shahi, N. C., & Singh, R. P. (2023). Design and development of throttle valve-based hydrodynamic cavitator for food. *Journal of Agricultural Engineering*, 60(2), 178-187.

[4] Le, T. L., Nghia, T. T., Thong, H. D., & Son, M. H. K. (2022). Numerical study of aerodynamic performance and flow characteristics of a centrifugal blower. *International Journal of Intelligent Unmanned Systems*.

[5] Nilugal, M. L., Karanth, K. V., & Madhwesh, N. (2020, February). A critical review on the application of computational fluid dynamics in centrifugal turbomachines. *Journal of Physics: Conference Series*, 1473(1), 012012. IOP Publishing.

[6] Patil, S. R., Chavan, S. T., Jadhav, N. S., & Vadgeri, S. S. (2018). Effect of volute tongue clearance variation on performance of centrifugal blower by numerical and experimental analysis. *Materials Today: Proceedings*, 5(2), 3883-3894.

[7] Wan, J., Li, L., Qiu, H., Zhang, B., & Zhou, Z. (2022). Numerical simulation study on optimization design of high efficiency and low noise centrifugal fan. *Journal of Precision Instrument and Machinery*, 2(1), 2537.

[8] Wang, J., Liu, X., Tian, C., & Xi, G. (2023). Aerodynamic performance improvement and noise control for the multi-blade centrifugal fan by using bio-inspired blades. *Energy*, 263, 125829.

[9] Wang, K., Ju, Y. P., & Zhang, C. H. (2017). Design of multi-blade centrifugal fan based on grouping model and bionic volute tongue. *Journal of Engineering Thermophysics*, 38(8), 1671-1675.

[10] Xu, C., & Mao, Y. (2016). Experimental investigation of metal foam for controlling centrifugal fan noise. *Applied Acoustics*, 104, 182-192.

AUTHORS BIOGRAPHY

Prof. M. U. Gan,
Department of Mechanical Engineering AISSMS College of Engineering Pune, India.

Vasudha Shinde,
Department of Mechanical Engineering AISSMS College of Engineering Pune, India.

Mehak Shaikh,
Department of Mechanical Engineering AISSMS College of Engineering Pune, India.

Ruchi Yadav,
Department of Mechanical Engineering AISSMS College of Engineering Pune, India.

Citation of this Article:

Vasudha Shinde, Mehak Shaikh, Ruchi Yadav, & M. U. Gan. (2026). Design and Analysis of Centrifugal Fan Blade Inspired By Teal Bird Wings. *International Research Journal of Innovations in Engineering and Technology - IRJIET*, 10(5), 200-203. Article DOI https://doi.org/10.47001/IRJIET/2026.105026

****

© 2026 IRJIET All Rights Reserved