Journal of Complex Flow, Vol. 3 No. 2 (2021) p. 28-32

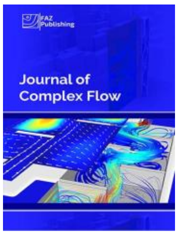
FAZ Publishing

JCF

Journal of Complex Flow

Journal homepage : www.fazpublishing.com/jcf

e-ISSN : 2672-7374



Aerodynamic Analysis of Quadrotor UAV Propeller using Computational Fluid Dynamic

Mohamad Zulfaqar Madzni[1], **Syariful Syafiq Shamsudin**[1,*]

[1] Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400, MALAYSIA

*Corresponding Author

Received 12 May 2021;
Accepted 8 August 2021;
Available online 30 Oct. 2021

Abstract: A propeller is a device with a rotating hub and blades that form a helical spiral at a specific pitch angle. It acts on a moving fluid, such as air, to convert rotational power into linear thrust. Designers of drones typically rely on vendors and layman recommendations while selecting propellers parameters for their UAVs. Several factors should be considered when designing a new drone propeller blade. However, not much research has been published on the effect of rotor configuration on propulsion system efficiency in hover. Thus, additional influence of rotor configuration such as the coaxial propeller configuration investigated in this work to supplement findings in factors that affect a drone system's propulsion efficiency compares along with the single propulsion system. Using SolidWorks and Ansys Fluent, this project will design and analyse the aerodynamic performance of a quadrotor drone propeller blade. For the computational fluid dynamics (CFD) investigation, 3D CAD models were produced in SOLIDWORKS and imported into Ansys Fluent. The shape of the model is inspired by the original APC drone propeller. The use of design tools like SolidWorks and a Computational Fluid Dynamics (CFD) solution can generate the necessary analysis to estimate the propeller blade's aerodynamic efficiency, allowing for the creation of a nearly optimal design. Several simulation examples are used in the research to find propeller shapes and configurations that improve aerodynamic and propulsion efficiency.

Keywords: ANSYS, Computational Fluid Dynamic, Computer-aided Design, Coaxial rotor

1. Introduction

Aerodynamic drone performance depends on their rotor's design, influencing the natural dynamics and propulsion power efficiency. Several factors should be considered when designing a new drone propeller blade. Primary geometry factors such as the number of blades, rotor diameters, and pitch were the main contributors to the propulsion efficiency and were extensively studied in the past [1]. However, not much research has been published on the effect of rotor configuration on propulsion system efficiency in hover. Thus, additional influence of rotor configuration such as the coaxial propeller configuration investigated in this work to supplement findings in factors that affect a drone system's propulsion efficiency.

Coaxial rotors are a pair of rotors positioned one above the other on concentric shafts, rotating in opposite directions but with the same axis of rotation. There is interference between the flows of the two propellers in a coaxial propulsion system, resulting in a lesser total thrust than when the two propellers are isolated. This propulsion system is commonly employed because it has an excellent thrust-to-volume ratio. After all, the two engines are close together and the overall output thrust is greater than a single engine [2-4]. Aircraft come in a variety of forms, but this study will focus on coaxial arrangements.

The geometry influence on rotors is critical for improving a drone's lift and drag performance. The generation of lift is affected by the form of an airfoil as well as variations in the angle of attack. The amount of an object's drag is determined by the form of the blade and how it travels through the air. An intelligent discussion of the factors affecting the magnitude of rotor blade lift and drag requires knowledge of blade section geometry [5,6]. It was shown that the equivalent single rotor performs better than the coaxial rotor at moderately high advance ratios, while the coaxial rotor performs better in hover [7-9]. Recent research suggests that the performance and other characteristics of a coaxial rotor system may have been overestimated, and they are certainly not as well understood as single rotors.

---

*Corresponding author: syafiq@uthm.edu.my

The thrust generated by a motor-propeller combination is critical in the design and simulation of an unmanned aerial vehicle (UAV). As a result, it must rely on the correct geometrical characteristics to attain the best efficiency in performance [10,11]. Aerodynamics is the dynamic division that deals with the motion of air and other gases that allows us to fly [12]. It can refer to forces acting via the air on a moving item or a stationary object in an air stream. Aerodynamic efficiency is a metric that analyses a flight parameter's efficiency in generating aerodynamic forces [13-15]. The objectives of this project are: (1) To study the effect of the chord length, pitch, number of the blade toward coaxial blade thrust generation. (2) To validate the accuracy of the simulation of the single rotor with experimental data. (3) To propose a near-optimal design blade design of the coaxial rotor and compare the aerodynamic performance with a single rotor.

2. Designs and Methodology

2.1 Modelling and Setup

In this study, the scope is focused on simulating and comparing the airflow behaviour of both single propellers and coaxial configuration propellers. Fig. 1 shows preparation for a simulation work where the rotating region is the place for the propeller and inlet and outlet ducts are showing the flow of air from the upper (inlet) to the lower (outlet) then all this happens in a stationary region. The model was created in SolidWorks by referencing the parameters as shown in Table 1

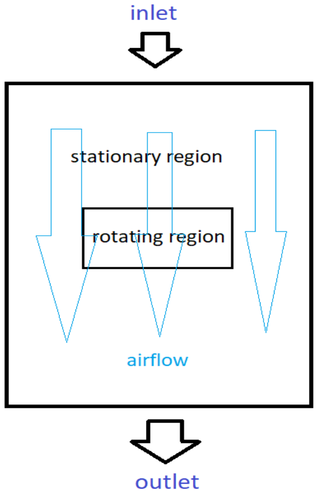


Fig. 1 – Setup for simulation work

Table 1 – Parameter for propeller design

Parameter	Propeller
Hub thickness (mm)	7.6
Hub diameter (mm)	12.7
Tip radius (mm)	14.7
Hub radius (mm)	33.6
Propeller diameter (mm)	24
Rotational speed, (rpm)	6000
Blade number	2

A total of two different propeller setups are used for this study to observe and differentiate the airflow properties between different setups. In this setup, there are variations of inlet velocity 15 m/s were used to observe the airflow pattern of the setup. The parameters of the fan setups are shown in Table 2. The setups were coaxial rotor and single rotor. For the coaxial setup, 2 blades fans as the front rotor rotating in a clockwise direction while 2 blades fans as the rear rotor rotating an anti-clockwise direction. Lastly, for the single setup, 2 blades fan rotating in the clockwise direction. In this setup, there are variations of inlet velocity 15 m/s were used to observe the airflow pattern of the setup. A positive rotational speed indicates that the fan rotates in the clockwise direction, while a negative rotational speed means that it rotates in the anti-clockwise direction.

Table 2 – Parameter for configuration setup

Name	No. of fans	No. of blades	Inlet velocity (m/s)	(RPM)
Single rotor	1	2	15.0	(+6000)
Coaxial rotor	2	2	15.0	(+6000), (-6000)

2.2 Geometry

The model was created in SolidWorks by referencing the parameters previously mentioned in Table 1 and the initial sketch in Fig. 2. The model was created in SolidWorks by referencing the parameters previously mentioned in Table 1 and the initial sketch in Fig. 2. The single propeller will use only one propeller while for coaxial configuration system will use two-propeller that will be set on top of each other with a 148 mm gap between propellers as shown in Fig. 3.

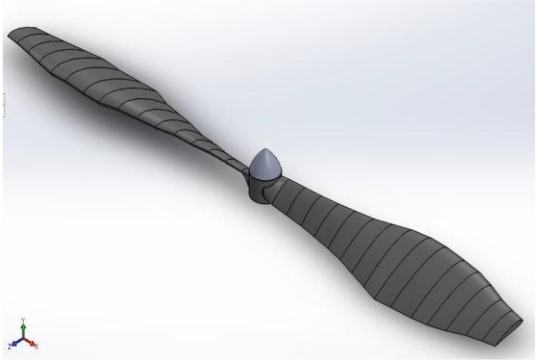


Fig. 1 - Geometry of single blade

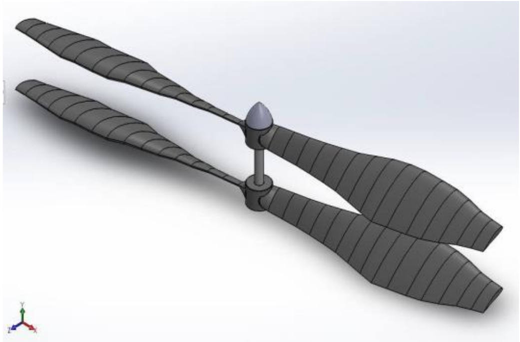


Fig. 2 - Geometry of blade for a coaxial rotor

2.3 Meshing

Mesh is described as a particle size measurement that is frequently used to determine the particle-size distribution of a granular substance. Meshing is one of the most essential processes in the fluid simulation since it influences the correctness of the computation result. The more precise the output, the smaller and finer the mesh size. However, even with a powerful computer, excessively fine mesh might result in extremely long calculation times. It may potentially cause instabilities or software crashes. For this study, a mesh size of 30 mm was used to maintain a balance between mesh size and calculation time.

The fan blades, rotating boundary, domain, inlet, outlet, and walls are selected and added as named selections to ease the future process. To control the mesh refinement, face sizing, body sizing, and inflation were added to the selected geometry. Figure 3 shows the Grid structure of the mesh for the full stationary region.

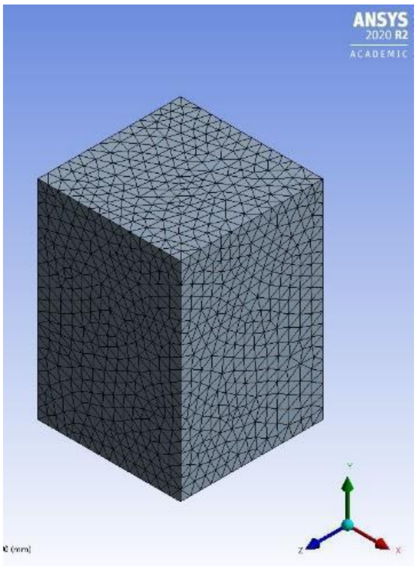


Fig. 3 - Grid structure of the mesh created

A grid-independent test was performed to evaluate the correct mesh size and number of elements that did not substantially affect the grid sensitivity of the analysis. It needs to be remeshing into a smaller element size to reach the acceptable mesh size level of adaptive. The size which has been re-meshing focused only on the blade part as it is an essential element that needs to be simulated and the performance result analyzed. The experimental tests on the inlet-velocity of 15 m/s were shown in Table 3.

Table 3 – Parameter for rotating fan

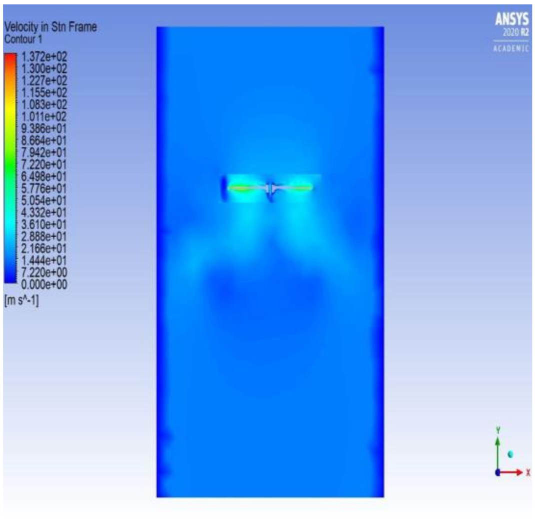
Element size (mm)	Element size (mm)	Element size (mm)	Element size (mm)
80	80	80	80
55023	55023	55023	55023
350442	350442	350442	350442

According to the grid-independent test, the 20 mm element size was chosen due to the most elements compared to the others. It was therefore considered to be the most reliable and appropriate element size.

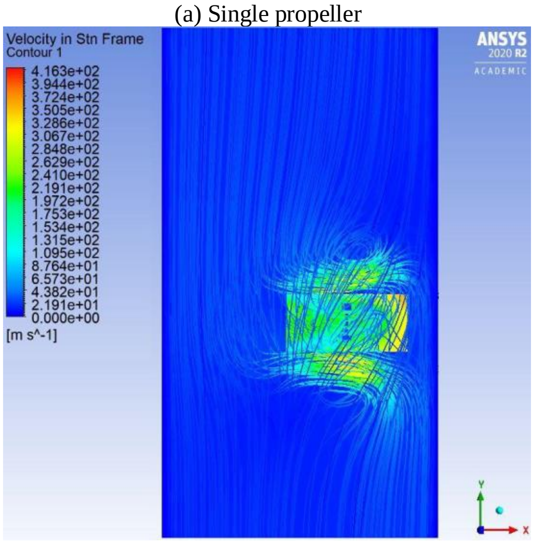
3. Results and Discussion

3.1 Velocity Contour

Two setups then had the inlet velocity set at 15 m/s to simulate the airflow created by spinning fans. The contour maps are taken from the last frame of the 100 times steps as before, and the results are as shown in Fig. 4 below.



(a) Single propeller



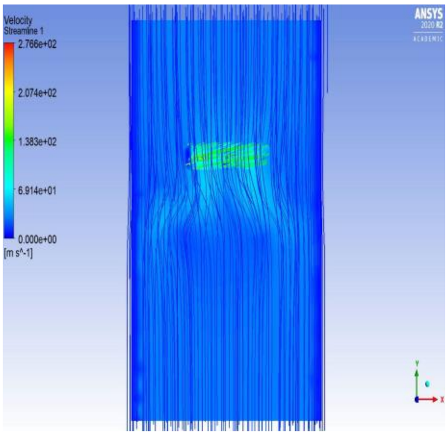
(b) Coaxial configuration propeller

Fig. 4 - Contour maps of the two setups for 15 m/s

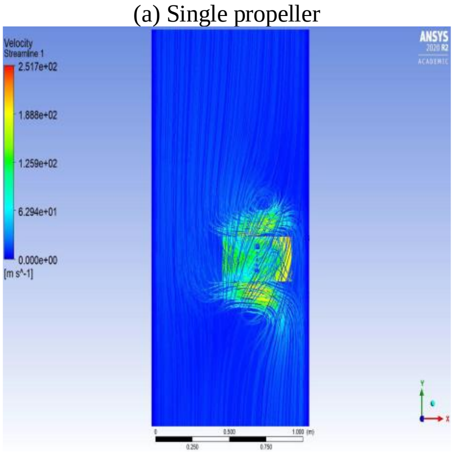
Based on the two setups, the single setup had the uniform contour compare to the other setups, but even though the coaxial setup had the less uniform contour compare to the single setup, the coaxial setup had a larger maximum velocity ranging from 394.4 to 416.3 m/s. while the maximum velocity for a single setup ranging 130.0 to 137.2 m/s.

3.2 Streamline

A streamline is a direction through the fluid domain that a particle with zero mass can follow. Streamlines start on a given locator at each node; in which case the streamline begins at the inlet. Another way to visualize the behaviour of airflow required for this study to be analysed is to Streamline. The procedure is the same as before, using the last frame of the 100-time measurement of measures as the analysis subject.



(a) Single propeller



(b) Coaxial configuration propeller

Fig. 5 - Streamline results of the two setups for 15 m/s

From 5(a) and 5(b), single setup produced a straight and wide with traverse airflow occur at the center. Single and coaxial setups produced an airflow with similar behavior, but the coaxial setup's airflow less focusing on the center. Note that in coaxial setup the flow has turbulence before entering the rotating region and has another turbulence when out from the region. This is likely caused by the rear fan disrupting the flow of the air.

4. Conclusion

Throughout this analysis, the approached computational fluid dynamics was used to numerically evaluate the performance analysis of the propeller. The simulation research was carried out using the steady procedure used to calculate the quantity. *k*-epsilon turbulence had been used for this solution as it can be expected well below the boundary wall. The results from the numerical simulations provided a strong understanding of the internal flow around the rotating region, and an aerodynamic airflow. The first objective of this research was to study the factors affecting propeller efficiency which is the propeller chord length, pitch, and the number of blades towards aerodynamic performance which shows that these three factors are the main contribution for the drone propeller to hover. Next, the objective was to propose the optimal or near-optimal blade design and rotor configuration to improve aerodynamic performance and propulsion efficiency. To propose the optimal blade the experimental data from the previous study help to provide specific data during propeller blade designing. This objective has been achieved after the finished design of the new propeller blade to be utilized for both configuration setups. The final objective is to validate the performance of propeller configuration CFD simulation. The overall aim is to demonstrate the accuracy of the CFD solution to be used for aerodynamic simulation with confidence and that the results are considered reliable for design decision-making. CFD may provide corrections to experimental observations at a higher precision than the existing CFD method.

Acknowledgement

The author would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, and Research Centre for Unmanned Vehicles (RECUV) for the supports.

References

[1] Yoon, S., Lee, H.C., and Pulliam, T.H., "Computational Analysis of Multi-Rotor Flows," *AIAA Journal*, (2016): 1-11.

[2] Quan, Q., "Introduction to Multicopter Design and Control." 2017.

[3] Yuvaraj, S., Renald, C.J.T., Jukna, A., Rathnaraj, J.D., Nallamani, M., and Kaviyarasan, P., "Design and Numerical Analysis of Five Blade Propeller for a Drone," *Int. J. Eng. Adv. Technol.*, 8(2) (2019: 116–119

[4] Norouzi Ghazbi, S., Aghli, Y., Alimohammadi, M., and Akbari, A.A., "Quadrotors Unmanned Aerial Vehicles: A Review," *Int. J. Smart Sens. Intell. Syst.*, 9(1) (2016): 309–333.

[5] Yilmaz, E., and Hu, J., "CFD Study of Quadcopter Aerodynamics at Static Thrust Conditions," *Proc. ASEE Northeast 2018 Annu. Conf. West Hartford, CT, USA*, no. April 2018: 27–28

[6] Intaratep, N., Alexander, W.N., Devenport, W.J., Grace, S.M., and A. Dropkin, "Experimental Study of Quadcopter Acoustics and Performance at Static Thrust Conditions," in *22nd AIAA/CEAS Aeroacoustics Conference*, 2016, p. 2873.

[7] W. F. Phillips, S. R. Fugal, and R. E. Spall, "Aerodynamic Twist: CFD Validation," *Journal of Aircraft*, 43(2) (2005): 1–12.

[8] Cho, T., Kim, Y., Chang, I., Kim, C., and Kim, C., "Experimental Study on Aerodynamic Characteristics for Side-Furling Control System." *J. Wind Energy*. 2 (2017): 39–44.

[9] Phillips, W.F., "Lifting-Line Analysis for Twisted Wings and Washout-Optimized Wings," *J. Aircraft.*, 41(1) (2004): 128–136.

[10] Bondyra, A., Gardecki, S., Gąsior, P., and Giernacki, W., "Performance of Coaxial Propulsion in Design of Multi-Rotor UAVs," *Adv. Intell. Syst. Comput.*, 440(3) (2016): 523–531.

[11] Sinha, P., Esden-Tempski, P., Forrette, C.A., Gibboney, J.K., Horn, G.M., "Versatile, Modular, Extensible VTOL Aerial Platform with Autonomous Flight Mode Transitions." In: *Aerospace Conference* (2012)

[12] Panagiotou, P., Kaparos, P., and Yakinthos, K., "Winglet Design and Optimization for a MALE UAV using CFD," *Aerospace Science and Technology*, 39 (2014):190-205

[13] Turanoğuz, E., and Alemdaroğlu, N., "Design of a Medium Range Tactical UAV and Improvement of its Performance by sing Winglets in Unmanned Aircraft Systems (ICUAS)," 2015 International Conference IEEE, 2015, 1074-1083

[14] Gill, R., and D'Andrea, R., "Propeller Thrust and Drag in Forward Flight," in Control Technology and Applications (CCTA), 2017 *IEEE Conference* on, 2017, 73-79: IEEE.

[15] Zabidin, Y.A.A., Pairan, M.F., and Shamsudin, S.S., "Dynamic Modelling and Control for Quadcopter UAV with LabVIEW and X-Plane Flight Simulator." *Journal of Complex Flow*, 2(2) (2020): 19-26