Terbit *online* pada laman web jurnal : https://metal.ft.unand.ac.id

METAL: Jurnal Sistem Mekanik dan Termal

| ISSN (Print) 2598-1137 | ISSN (Online) 2597-4483 |

Research article

Analysis of Computational Fluid Dynamics on 3D Printing Propeller Design using Thermoplastic Material

Nurhidayanti[1,a*], *Muhammad Ikhsan*[2,b], *Muh. Abdillah*[3,c]

[a] *Center for Material and Manufacturing, Department of Mechanical Engineering, Politeknik Negeri Ujung Pandang, Sulawesi Selatan, 90245, Indonesia*

[b] *Department of Mechanical Engineering, Politeknik Bosowa, Sulawesi Selatan, 90113, Indonesia*

ARTICLE INFORMATION

Article History:
Received : 09 May 2025
Revised : 09 September 2025
Accepted : 31 October 2025

KEYWORDS

Computational Fluid Dynamics

3D Printing

PLA Thermoplastic

Propeller

CORRESPONDENCE

E-mail: nurhidayantisamsir@poliupg.ac.id

A B S T R A C T

The propeller is essential in generating thrust for various applications, from UAVS to ship propulsion and industrial ventilation. Propeller performance is greatly influenced by the characteristics of the fluid flow generated by the interaction between the propeller blades and the surrounding medium. 3D printing technology offers design flexibility and cost efficiency in propeller manufacturing. This study aims to analyse the effect of rotational speed variations on the performance of 3D-printed propeller designs made of PLA. Simulation of Computational Fluid Dynamics (CFD) was used to evaluate the fluid flow velocity distribution, flow pattern, and flow type transition around the propeller. The results showed that the variation of rotational speed affects the fluid flow type, with low rotational speed producing laminar flow and high rotational speed producing transitional flow. This understanding is important for the optimisation of PLA 3D printing propeller design in improving efficiency and reducing noise.

1. INTRODUCTION

Propellers, as devices that produce thrust through rotation, play a very important role in various engineering applications. Their role is vital, starting from moving aerial vehicles such as Unmanned Aerial Vehicles (UAVs), which are now increasingly used in various fields[1], such as applications in the marine industry for ship propulsion, and even in industrial ventilation systems to move large volumes of air. In all these applications, propeller performance directly affects energy efficiency, productivity, and even the safety of the system as a whole.

Propeller performance itself is largely determined by the characteristics of the fluid flow generated by the interaction between the propeller blades and the surrounding medium. A deep understanding of these fluid dynamics issues is crucial to optimizing propeller design.

In recent years, 3D printing technology has emerged as a promising manufacturing method for propeller production. Compared with traditional manufacturing methods such as machining or injection moulding, 3D printing offers several significant advantages. One of its main advantages is its high design flexibility, allowing the creation of complex propeller geometries that are difficult to achieve with conventional methods[2].

https://doi.org/ 10.25077/metal.9.2.122-129.2025

This is an Open Access article distributed under the terms of the Creative Commons Attribution-NonCommercial-ShareAlike 4.0 International License

In addition, 3D printing also offers better cost efficiency, especially for prototype production and small-volume production, and allows for personalisation and customisation of designs according to specific application needs. The rapid development of 3D printing material technology also expands the possibilities of directly building products with various materials, complex structures, and integrated functions, according to CAD models, without being limited by shape[3], such as the materials commonly used in 3D printing is Polylactic Acid[4].

PLA is a thermoplastic polymer derived from renewable resources such as corn starch or sugar cane, so it is considered a biodegradable and environmentally friendly material[5]. In addition, PLA is also known for its ease in the 3D printing process, with relatively low printing temperatures and wide availability of laments. Recent studies have also shown the potential of PLA in wider applications with the development of innovative formulas and printing methods[6]. However, the mechanical characteristics of PLA, such as tensile strength, elastic modulus, and impact resistance, need to be well understood to ensure optimal performance and reliability of the resulting propeller. Each material has its unique characteristics, and these PLA characteristics will affect the design and application of the propeller.

Computational Fluid Dynamics (CFD) has become the primary medium for evaluating and optimising propeller performance. CFD simulations allow for the visualisation and calculation of the type of fluid flow that occurs around a propeller. Using numerical methods, CFD solves the equations that govern fluid motion, such as the Navier-Stokes equations, to predict flow characteristics such as velocity, pressure, and turbulence[7].

The information obtained from these CFD simulations is invaluable in understanding how propeller design affects its performance and efficiency. For example, CFD can be used to identify areas of flow that experience separation or high vorticity, which can lead to reduced efficiency and increased noise. In addition, CFD also allows for analysis of the flow transition from laminar to turbulent, which has a significant impact on the drag and lift generated by the propeller.

Recent research continues to develop more sophisticated CFD methods to improve the accuracy and efficiency of simulating complex fluid flows around a propeller[8]. Previous researchers, such as Hara et al. 2011 have studied the potential use of carbon fibre-reinforced polymer (CFRP) as an alternative material for ship propellers, replacing conventional metals such as Nickel Aluminium Bronze (NAB). CFRP is considered superior in specific strength, fatigue resistance, and elastic deformation efficiency. However, due to its different elastic properties from metals, CFRP performance cannot be evaluated only by fluid analysis. Therefore, they developed the fluid-structure interaction analysis (FSIA), which combines finite element analysis (FEA) and computational fluid dynamics (CFD). The system was validated through tank tests and demonstrated performance simulation results that were close to actual conditions, thus strengthening the potential for CFRP application in ship propellers[9].

Developments were carried out by Guilmineau et al. 2016 in their research using ISIS CFD numerical modelling based on RANS (Reynolds-Averaged Navier-Stokes) to simulate the flow around a ship's propeller under two conditions: straight flow (straight ahead flow) and oblique flow. Two propeller models were evaluated, namely the INSEAN E779A with four blades and fixed pitch geometry, and the PPTC with ve blades and a more complex and inclined geometry, showing that the hybrid RANS-LES approach is more accurate in capturing the evolution of blade tip vortices and wake instability, especially under inclined flow conditions[10]. This study uses CFD simulations to analyze how varying rotational speed impacts the performance of 3D-printed PLA propellers. It will examine changes in fluid flow velocity, patterns, and flow types around the propeller at different speeds.

 https://doi.org/ 10.25077/metal.9.2.122-129.2025

2. METHOD

This study adopts a computational simulation approach to analyse the performance of a 3D printed propeller design made of PLA. This research method comprises several structured stages, ranging from design planning to the analysis of CFD simulation results.

2.1. Experimental Design

The material used in this study is PLA filament with a diameter of 1.75 mm (Figure 1). The specifications of the PLA material used can be seen in Table 1.



Figure 1. PLA material

Table 1. Material Properties of eSUN PLA[11]

Specification	Value	Unit
Grade	Semi-crystalline	-
Printing temperature	205-225	ºC
Density	1.25	g/cm³
Melt index	4(190ºC/2.16kg)	g/10min
Tensile strength	65	MPa
Yield strength	49	N/mm²
Elastic modulus	3500	N/mm²
Poisson's ratio	0.36	N/A
Shear modulus	1287	N/mm²
Flexural strength	75	MPa
Elongation at break	12	%
IZOD Impact strength	8.5	kJ/m²
Temperature distortion	52	ºC; 0.45 MPa
Thermal conductivity	0.195	W/(mºK)
Specific heat	2060	J/(kg.ºK)

2.2. Design Geometry Propellers

The propeller design was modelled using Computer-Aided Design software[12], applied in simulation CFD, is Dassault Systèmes SolidWorks 2019. The propeller design used has 7 blades with dimensions and components shown in Figure 2.

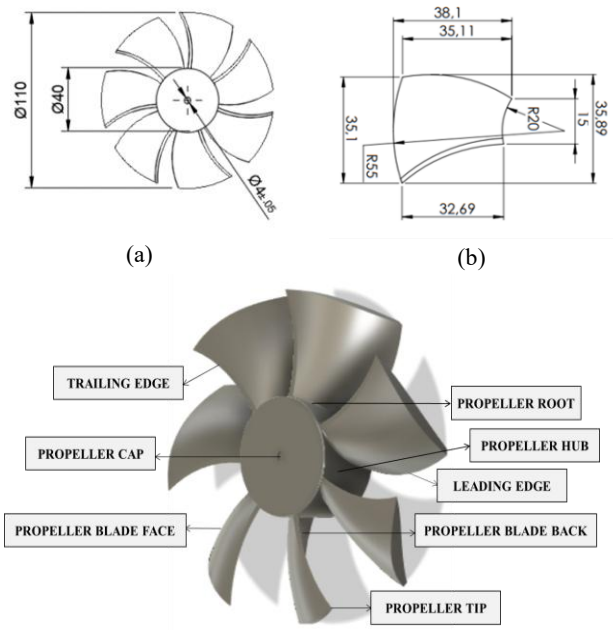


Figure 2. Propeller design with 7 blades (a) dimensional propeller; (b) dimensional blade propeller; (c) 3D interpretation of propeller design

Table 2. Fluid Condition Parameters

Parameters	Setting
Dimensions design	Hub Ø 40 mm; propeller Ø 110 mm; thickness 3.5 mm
Domain type	Fluid air
Domain movement mesh	Static Global mesh
Rotation speed	240 rpm; 5000 rpm
Axis turn	Y-axis
Flow type	Laminar and Turbulent
Pressure	101.325 kPa
Temperature	293.2 ºK

2.3. Computational Fluid Dynamics Method

Computational Fluid Dynamics (CFD) simulations were conducted using Dassault Systèmes SolidWorks 2019 software.

In this study, the software automatically adapts the flow model based on the calculated Reynolds number value. This feature enables a smooth transition from laminar to turbulent flow conditions without requiring manual selection of a specific turbulence model (e.g., k-epsilon), simplifying the simulation process while ensuring accuracy.

For low-speed flow conditions (240 rpm) that tend to be laminar, the Navier-Stokes equations are solved assuming laminar flow. The Navier-Stokes equations, as the primary foundation of computational fluid dynamics (CFD), are used to model the movement of viscous fluids. It is important to note that the choice of propeller material (whether PLA plastic, aluminium, or other) does not affect the mathematical basis of these equations. Meanwhile, for high-speed conditions (5000 rpm) that exhibit transition characteristics from laminar to turbulent, the software automatically handles the transition based on the calculated flow parameters.

The computational domain is defined as a rectangular volume enclosing the propeller, as illustrated in Figures 3 and 4. The domain dimensions are carefully designed to be large enough, ensuring that boundary effects on the flow around the propeller are minimised.

The boundary conditions applied include:

- An inlet pressure of 101.325 kPa is applied at the front of the domain.
- A static pressure condition is set at the outlet at the back of the domain. This arrangement facilitates fluid exit without significant backpressure resistance, replicating realistic conditions.
- The other sides of the domain (top, bottom, left, and right axially) are assumed to be slip walls or open boundary pressure conditions, allowing free fluid flow.
- The propeller is modelled as a rotating body with a specified rotational speed on the Y-axis.

2.4. Simulation Parameters and Rotation Speed

CFD simulations were carried out with the setup parameters detailed in Table 2. Variations in propeller rotational speed were used to analyse the performance under different operational conditions. The selection of rotational speed was based on practical application considerations and the need for comparison of results. Two main rotational speed values were selected, namely 240 rpm and 5000 rpm.

The rotational speed of 240 rpm was chosen based on the study of Indiaryanto et al. 2018[13], which simulated a similar propeller design with seven aluminium blades. This reference provides a strong validation basis for low-speed conditions. For comparison and to understand the flow characteristics at higher speeds, a rotational speed of 5000 rpm was adopted from the study of Alfaridzi et al. 2022[14], the research involved the study of a rotating polymer-aluminium composite material propeller for UAV applications.

Thus, this study applies two variations of rotational speed (240 rpm and 5000 rpm) on the propeller made of PLA thermoplastic.

3. RESULT AND DISCUSSION

3.1. Data Collection

Computational Fluid Dynamics involves a fluid flow analysis system. Based on the simulation results from the Solidworks software, the propeller design used in the CFD simulation is PLA thermoplastic material because the material is recommended.

After all, the durability and strength of the material are more optimal and worthy of use, so fluid dynamic simulations are carried out to identify the type of flow that occurs when the propeller is in a rotating condition.

 https://doi.org/ 10.25077/metal.9.2.122-129.2025

3.2. Flow Trajectories 240 rpm

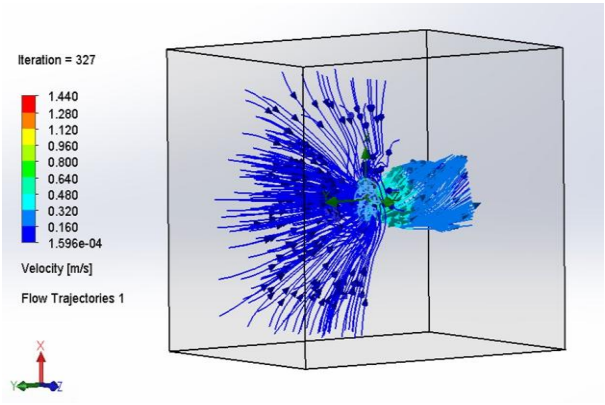


Figure 3. Result of flow trajectories 240 rpm

Based on the CFD simulation results shown in Figure 3. PLA propeller design with a rotating speed of 240 rpm or equivalent to 25.132 rad/s obtained axial speed (axial velocity) which comes out at 0.960 m/s; the highest red fluid velocity of 1.440 m/s occurs at the tip surface of the blade propeller because the fluid flow conditions pass through a complex area so that there is a change in pressure which increases acceleration.

The lowest fluid velocity is in the part near the hub, with a velocity of 0.0001596 m/s. Low-velocity values indicate that flow conditions are slow and steady, which tends to result in an even fluid distribution and no turbulence. The CFD simulation shows a relatively regular fluid pattern, characterised by axially dominated flow lines (flow trajectories) and a gradual change in velocity from low (blue) to high (green to yellow) around the propeller.

Based on the data from the CFD simulation approach, in line with the research conducted by Fadlin 2017 the largest fluid velocity is located on the outer side of the blade, while the part that receives the lowest fluid is at the back of the propeller around between the hub and propeller root[15].

3.3. Flow Trajectories 5000 rpm

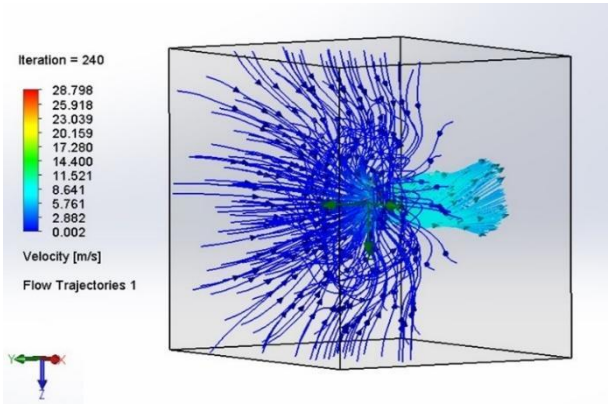


Figure 4. Result of flow trajectories 500 rpm

Meanwhile, to get a comparison, a CFD simulation was carried out with a high rpm, namely 5000 rpm. The selection of the rpm value is based on the reference from the researchers Alfadrizi et al. 2022 on the propeller type, against rotating for Unmanned Aerial Vehicle (UAV) aircraft with 2 blades using aluminium material and polymer composite[14].

The parameter rotating speed is 5000 rpm, simulated CFD on the design model of a 7-blade propeller made of PLA thermoplastic material to determine the velocity value and the type of fluid flow that occurs. Based on the simulation approach shown in Figure 4, it can be seen that with high rpm increases, fluid acceleration increases, with fluid velocity coming out to 17.280 m/s, the highest velocity of 28.798 m/s, and the lowest 0.002 m/s.

Increasing the rotational speed to 5000 rpm results in a drastic change in the flow pattern. The flow becomes turbulent, indicating a much higher fluid velocity and a more spread-out flow pattern. This flow turbulence can cause fluctuations in thrust and potentially increase noise due to the design of the sharply inclined surface, as shown in the Figure.
This difference in flow patterns underscores the importance of considering rotational speed in propeller design to optimise performance and minimise undesirable effects.

This was confirmed by researchers Baban Pawar et al. 2019 investigated 4 propellers with Fibre Reinforced Plastic material using a 300 rpm rotation

of 18.312 m/s. The chordwise distribution of the axial velocity shows different velocity levels, which occur due to the number and spacing between blades[16]. Based on the comparison of the research results above, it can be said that the smaller the number of blades, the higher the loading conditions that occur, but the greater the flow acceleration, while the greater the number of blades, the smaller the distance between them so the blade can easily cause fluid blockage[17]. Based on the results of the CFD simulation, the Reynolds number can be obtained using the equation below:

$$Re = \frac{\rho.v.D}{\mu}$$

where in value parameters are $\rho\ is\ 1.184\ kg/m^3$ (density propeller design); $v_1\ is\ 0.022\ m/s$; $v_2\ is\ 0.433\ m/s$; $diameter\ is\ 110\ mm$; and then value $\mu\ is\ 1.85\ x\ 10^{-5}$[18].

a. Reynolds number 240 rpm

$$Re = \frac{\rho.v_1.D}{\mu} = \frac{1.184\ x\ 0.022\ x\ 0.11}{1.85\ x\ 10^{-5}}$$

$$= 154.88$$

b. Reynolds number 5000 rpm

$$Re = \frac{\rho.v_1.D}{\mu} = \frac{1.184\ x\ 0.433\ x\ 0.11}{1.85\ x\ 10^{-5}}$$

$$= 3048.32$$

The Reynolds number (Re) calculation is a key indicator of the characteristics of fluid flow. However, it is emphasised that the critical Re value for the transition from laminar to turbulent flow depends on the specific geometry and boundary conditions of the flow. This is especially true for complex objects such as propellers, due to their intricate radius and shape dimensions. Therefore, there is no single universal critical Re value for flow through a propeller.

However, visual observations from our CFD simulations give a clear indication of this flow transition. At a rotational speed of 240 rpm, the resulting flow pattern tends to be regular and stable, indicating laminar conditions. In contrast, at a rotational speed of 5000 rpm, we observe a significant increase in turbulence, even a full transition to turbulent conditions. This change in flow pattern directly reflects the effect of the Reynolds number on the fluid dynamics around the propeller.

Table 3. Result simulation of fluid dynamics

Speed (rpm)	Min velocity (m/s)	Max velocity (m/s)	Re	Fluid type
240	0.021	0.023	154.88	Laminar
5000	0.421	0.445	3048.32	Transition

Table 3 presents the results of the calculation of Reynolds number (Re) which indicates the flow characteristics at various rotational speeds. At a rotational speed of 240 rpm, the Reynolds number value obtained is 154.88. This value strongly indicates laminar flow characteristics. Laminar flow is characterised by more uniform fluid velocity and minimal turbulence, resulting in quieter flow without the potential for significant noise or vibration to the propeller. Although the critical Re value for the transition from laminar to turbulent may vary depending on the geometry and orientation of the flow (especially for complex objects such as propellers), the uniform flow pattern and absence of significant turbulence in our CFD simulations confirm that the flow at 240 rpm is close to laminar conditions for this propeller design.

Increasing the rotational speed to 5000 rpm results in a drastic change in the flow pattern to transition. The calculated Reynolds number is 3048.32. This value indicates that the flow has crossed the laminar threshold and is in the transition regime, where the flow characteristics can fluctuate between laminar and turbulent. This transition is strongly influenced by the complex geometry of the propeller and its high rotational speed, which promotes the formation of vortices and greater flow instability compared to low rotational speeds. This transition flow can cause changes in dynamic conditions, such as fluctuations between laminar and turbulent.

4. CONCLUSION

This study clearly demonstrates the crucial influence of rotational speed on the fluid dynamics and performance of 3D printed PLA propellers. Through Computational Fluid Dynamics (CFD)

 https://doi.org/ 10.25077/metal.9.2.122-129.2025

simulations, it is observed that a lower rotating speed (240 rpm) results in laminar flow, characterised by a more uniform fluid velocity distribution and minimal turbulence. In contrast, higher rotational speeds (5000 rpm) lead to transitional flow with increased fluid velocity and turbulence, which can affect thrust as well as noise potential.

These findings underscore the importance of careful consideration of rotational speed in the design and application of PLA propellers to optimise performance and minimise undesirable effects. Insights from this study contribute to a deeper understanding of the relationship between operational parameters and fluid behaviour in 3D printed propeller systems, offering valuable guidance for future design improvements and applications in various fields.

REFERENCES

[1] R. Austin, Unmanned Aircraft Systems. A John Wiley and Sons, Ltd, 2010.

[2] I. Gibson, D. Rosen, and B. Stucker, IannGibsonn· DaviddRosen BrenttStucker Additive Manufacturing Technologies 3D Printing, Rapid Prototyping, and Direct Digital Manufacturing Second Edition, Second. Springer New York Heidelberg Dordrecht London, 2015.

[3] Y. Yang, B. Yang, Z. Chang, J. Duan, and W. Chen, "Research Status of and Prospects for 3D Printing for Continuous Fibre-Reinforced Thermoplastic Composites," Sep. 01, 2023, Multidisciplinary Digital Publishing Institute (MDPI).

[4] Nurhidayanti, Ishak, and A. Bayu Wicaksono, "Experimental Modelling of Gripper Arm Using CAD Fusion 360 and Fused Deposition Modelling 3D Printing Process of Polymer," Jurnal Tematis, vol. 6, no. 2, pp. 1–7, 2024.

[5] R. E. Drumright, P. R. Gruber, and D. E. Henton, "Polylactic Acid Technology," Advanced Materials, vol. 12, no. 23, 2000.

[6] S. Farah, D. G. Anderson, and R. Langer, "Physical and mechanical properties of PLA, and their functions in widespread applications — A comprehensive review," Dec. 15, 2016, Elsevier B.V.

[7] H. Wang et al., "Recent Advances on Machine Learning for Computational Fluid Dynamics: A Survey," Journal Of Latex Class Files, vol. 14, no. 8, Aug. 2024.

[8] E. Ferrer et al., "A high-order discontinuous Galerkin solver for flow simulations and multi-physics applications," Comput Phys Commun, vol. 287, Jun. 2023.

[9] Y. Hara, T. Yamatogi, H. Maruyama, K. Uzawa, and K. Kageyama, "Performance Evaluation Of Composite Marine

[10] E. Guilmineau et al., "Numerical Simulations for the Wake Prediction of a Marine Propeller in Straight Ahead Flow and Oblique Flow," 2016, doi: 10.1115/1.4037984ï.

[11] "Shenzhen Esun Industrial Co., Ltd. - PLA, ABS." Accessed: Jan. 22, 2022.

[12] N. Syamsir, R. Nur, A. Salam, and Y. Busrah, "Analyzing and Modelling Gripper Arm using Shape Optimization of Fusion 360 and 3D Printing of Polylactic Acid," in AIP Conference Proceedings, American Institute of Physics Inc., Nov. 2022. doi: 10.1063/5.0094739.

[13] M. Indiaryanto, M. R. Utina, N. Asrowibowo, and S. Sadiah, "Design and Hydrodynamic Model Test of Mini Submarine Propeller with High Efficiency and Low Cavitation," EPI International Journal of Engineering, vol. 1, no. 2, pp. 59–64, Nov. 2018, doi: 10.25042/epi-ije.082018.09.

[14] A. Y. Alfaridzi and A. Kurniawan, "Analisis Computational Fluid Dynamic Pengaruh Jarak Propeller Pada Contra Rotating Propeller Terhadap Gaya Dorong Pesawat Tanpa Awak," JTM-ITI (Jurnal Teknik Mesin ITI), vol. 6, no. 2, p. 87, Aug. 2022, doi: 10.31543/jtm.v6i2.755.

[15] D. Fadhlin, "Analysis of Passenger Ship Propeller Thrust Using Solidworks," Universitas Muhammadiyah Sumatera Utara, Medan, 2017.

[16] A. Baban Pawar, T. Santosh Petkar, P. Sanjeev Pitale, and S. Shivaputrappa

 https://doi.org/ 10.25077/metal.9.2.122-129.2025

Salagare, “Design and Simulation of Marine Propeller with Different Blade Geometry,” Int J Innov Sci Res Technol, vol. 4, no. 3, 2019.

[17] M. Felli, G. Guj, and R. Camussi, “Effect of the number of blades on propeller wake evolution,” Exp Fluids, vol. 44, no. 3, pp. 409–418, Mar. 2008, doi: 10.1007/s00348-007-0385-0.

[18] R. D. B. Blevins, “Applied Fluid Dynamics Handbook,” *Book*, p. 558, 2003.