Jurnal Media Mesin, Vol. 26 No. 2
Printed ISSN: 1411-4348
Online ISSN: 2541-4577

DEVELOPMENT AND ANALYSIS OF A 7040 PROPELLER AIRSCREW TEST BENCH USING EXPERIMENTATION AND CFD

Anggit Bayu Prabowo
Master of Mechanical Engineering Program, Faculty of Engineering
Universitas Muhammadiyah Surakarta, Indonesia
Email: u100230009@student.ums.ac.id

Sarjito
Master of Mechanical Engineering Program, Faculty of Engineering
Universitas Muhammadiyah Surakarta, Indonesia
Email Corresponding author: sarjito@ums.ac.id

Wijianto
Master of Mechanical Engineering Program, Faculty of Engineering
Universitas Muhammadiyah Surakarta, Indonesia
Email: wij109@ums.ac.id

Buyung Junaidin
Department of Aerospace Engineering, Faculty of Aerospace Technology
Institute Teknologi Dirgantara Adisutjipto, Indonesia
Email: buyung112011@gmail.com

ABSTRAK

Penelitian ini menyajikan evaluasi performa propeller 7×4 melalui pendekatan eksperimental dan komputasi. Alat uji (test bench) dirancang untuk mengukur gaya dorong statis (thrust) dan konsumsi daya pada berbagai kecepatan putaran (RPM), sementara simulasi Computational Fluid Dynamics (CFD) dilakukan menggunakan ANSYS CFX untuk menganalisis perilaku aerodinamika dan gaya dorong yang dihasilkan pada kondisi yang sama. Sistem eksperimen menggunakan Arduino yang terintegrasi dengan sensor untuk mengukur thrust, tegangan, arus, dan RPM. Hasil menunjukkan nilai thrust meningkat seiring bertambahnya RPM, dengan maksimum 2,7468 N pada 9000 RPM. Sebagai perbandingan, simulasi CFD memprediksi gaya dorong sebesar 3,6358 N pada kecepatan yang sama. Meskipun terdapat selisih pada nilai tertentu—terutama pada RPM tinggi—tren yang dihasilkan dari pengujian eksperimen dan simulasi menunjukkan kesesuaian. Hasil ini menunjukkan bahwa alat uji yang dikembangkan mampu memberikan pengukuran yang andal pada kondisi statis, serta bahwa CFD dapat digunakan sebagai alat prediksi performa propeller. Pengujian lanjutan dalam kondisi dinamis serta penyempurnaan model simulasi disarankan untuk meningkatkan akurasi dan relevansi terhadap kondisi penerbangan nyata.
Keywords: Performa propeler 7040 , static thrust, simulasi CFD, ANSYS CFX, validasi eksperimen, test bench

ABSTRACT

This study presents a performance evaluation of a 7×4 propeller using both experimental and computational approaches. A test bench was developed to measure static thrust and power consumption across various rotational speeds (RPM), while Computational Fluid Dynamics (CFD) simulations were performed using ANSYS CFX to analyze the aerodynamic behavior and *thrust generation under identical condition. The experimental setup employed an Arduino-based system integrated with sensors for thrust, voltage, current, and RPM measurements. Thrust values increased proportionally with RPM, reaching a maximum of 2.7468 N at 9000 RPM. In comparison, the CFD simulation predicted a higher thrust of 3.6358 N at the same speed. Although some deviations were observed—particularly at higher RPMs—the overall trends between experimental and CFD results were consistent, indicating that both methods effectively capture the propeller's performance characteristics.The findings confirm that the test bench provides reliable measurements under static conditions and that CFD can serve as a predictive tool for propeller performance. Further improvements through dynamic testing and simulation refinement are recommended to enhance accuracy and represent real-world flight conditions.*

Keywords: *7040 Propeller performance, static thrust, CFD simulation, ANSYS CFX, experimental validation, test bench*

1. INTRODUCTION

The analysis of propeller performance plays a critical role in the aerospace and aviation industries, as it directly affects thrust generation, energy efficiency, and overall system stability. Traditional experimental methods for evaluating propeller behavior often face challenges in achieving accuracy, consistency, and scalability. As a result, Computational Fluid Dynamics (CFD) has emerged as a valuable complementary tool for performance evaluation and design validation.

Among various types of small-scale propellers, the 7040 airscrew propeller has gained popularity due to its widespread use in lightweight Unmanned Aerial Vehicles (UAVs), especially in applications requiring high maneuverability and efficient static thrust generation. Its compact size, compatibility with brushless DC motors, and availability in commercial markets make it a preferred choice for research and prototyping in drone systems. However, despite its frequent usage, there remains a lack of detailed empirical and simulation-based validation of its performance under varying operational conditions.

This research therefore addresses this gap by developing a dedicated test bench capable of accurately measuring the thrust, torque, and electrical power consumption of the 7040 propeller. Furthermore, the results are validated through CFD simulations to enhance the understanding of its aerodynamic characteristics. This dual-approach is expected to support further design optimization and practical application in UAV propulsion systems

Previous studies have contributed to the development of thrust measurement systems. Fahrureza & Priangkoso designed a test bench equipped with thrust and rolling force measurement systems, along with motor temperature monitoring for an Electric Ducted Fan (EDF). Their findings highlighted the importance of considering both thrust and rotational torque in EDF-based propulsion systems. High operating temperatures were also identified as a potential cause of motor damage, impacting system reliability [1].

Maulana developed a propeller thrust test rig to characterize UAV propulsion systems. Their study focused on assembling a bench capable of measuring thrust, RPM, voltage, and current to determine motor characteristics under static conditions. This system provided reliable performance data for various UAV motors [2].

Other experiment as further designed and built a thrust measurement tool for brushless motors using different propeller types (6×4, 9×6, 10×5). Load sensors and current sensors were integrated using an HX711 ADC module, demonstrating the system's ability to collect accurate experimental data for drone propulsion characterization [3].

Unnikrishnan et. al. conducted experimental and analytical performance analysis of a Master Airscrew 11×10 propeller. Using wind tunnel testing and theoretical modeling, they evaluated thrust behavior at various advance ratios and RPMs. Their results were validated against the UIUC propeller database, confirming minimal deviation and consistency with established trends [4].

Study by Sasmito designed an airscrew propeller with RAF-6 airfoil profile for Wing-in-Surface Effect (WiSE) A2C vehicles, using a simplified design methodology. The RAF-6 profile was selected for its high lift-to-drag ratio, offering improved aerodynamic efficiency for alternative propulsion systems [5].

Further work conducted CFD simulations of a propeller blade modeled in SolidWorks and analyzed in ANSYS Fluent. By varying the RPM (2000, 2500, and 3000), they examined the relationship between pitch angle and thrust generation. The propeller at 2000 RPM showed the best match with manufacturer data, suggesting that higher RPMs do not necessarily equate to higher thrust output [6].

[7] An experiment in generating airflow, the propellers give drones or unmanned aerial vehicles (UAV) a lift force or thrust. This work examines the method to calculate the thrust force generated by propellers using the Blade Element Momentum Analysis which is programmed in MATLAB. This program is developed to perform the calculation of thrust and torque for a given propeller blade geometry. This investigation compares the thrust coefficient produced by different propeller designs a various rotation speeds and parameters using the extended blade element momentum theory.

Study in different area investigated the feasibility of CFD simulation for predicting propeller performance across various disc angles and rotational speeds (1000–8000 RPM). Using the Multiple Reference Frame (MRF) model in ANSYS Fluent, they compared CFD results with wind tunnel data. The trends observed in thrust coefficient and efficiency were consistent with empirical findings, with thrust estimation errors below 12% [8].

Rajendran presented a 3D CFD simulation and experimental validation of APC Slow Flyer propeller blades using the Multiple Reference Frame (MRF) approach in Fluent at 3008 RPM. Their study showed good agreement between simulation and experimental results, particularly at low advance ratios. However, the scope was limited to a specific RPM and did not explore varying blade geometries or load conditions, which are important for broader UAV applications. Additionally, the study focused more on aerodynamic coefficients rather than integrating electrical power characteristics into the analysis, which differs from the present study that considers motor performance holistically [9].

Recent study investigated the performance differences between ducted and free propellers for UAV applications under static conditions. They employed a deformation-based load cell to measure thrust and compared the results with manufacturer data for the Master Airscrew E9x6 propeller. While their findings highlighted the efficiency advantages of ducted configurations, the analysis was constrained to static testing without dynamic flow variations or CFD validation. This contrasts with our study, which not only applies CFD simulation but also focuses on the performance of a non-ducted 7040 propeller commonly used in lightweight UAVs—an area that remains underexplored in the literature [10].

Lastly, Previous study was conducted a combined CFD and experimental study on the Master Airscrew G/F 39x6 propeller. Using 3D scanning to extract geometry, CFD simulations were carried out in ANSYS CFX with varying flow velocities and RPMs. Experimental validation was conducted in a closed wind tunnel. The research focused on the 7×4 propeller model under static testing conditions, aiming to investigate performance characteristics not yet explicitly addressed in previous studies [11].

In summary, while numerous studies have investigated propeller performance using experimental testing, CFD validation, or analytical approaches, most have focused on general-purpose or ducted propellers under ideal or simplified conditions. Very few have specifically targeted the 7040 propellers, despite its widespread use in small-scale UAVs for missions requiring high manoeuvrability and efficient static thrust.

To date, no previous study has developed and validated a dedicated test bench for the 7040 propellers while integrating both experimental and CFD-based performance evaluations.

Furthermore, prior works often examined aerodynamic coefficients without linking them to real-world electrical power consumption or motor behaviour.

This study fills that gap by developing a fully instrumented test bench and validating it with CFD simulations under static conditions. The integration of physical measurements with numerical modeling offers a comprehensive understanding of thrust generation and energy efficiency.

Moreover, the CFD-based approach provides a faster, more cost-effective, and flexible way to visualize flow characteristics and predict propeller performance—something that has not been demonstrated in previous studies involving the 7040 model. The resulting methodology serves not only to validate current performance but also as a platform for future design optimization in UAV propulsion systems.

2. METHODOLOGY

This study applies a combination of experimental and numerical (CFD) approaches to evaluate the performance of a propeller. Through the bench test, parameters such as rotational speed (RPM), thrust force, voltage, current, and power consumption are measured directly using sensors and Arduino-based data acquisition. Simultaneously, CFD simulations are performed to visualize and analyze airflow characteristics and aerodynamic behavior, which complements the experimental findings.

2.1 Experimental Setup

This study utilizes a bench test integrated with sensors and Arduino-based data acquisition to evaluate propeller performance. The 7×4 propeller is mounted on a brushless motor shaft, ensuring optimal alignment and mechanical stability. A calibrated load cell is attached to measure the generated thrust force. Additional sensors for voltage, current, and RPM are connected to an Arduino UNO, which processes and sends real-time data to an LCD monitor.

The experimental procedure involves:

- Charging the LiPo battery,
- Ensuring all control elements are in the correct initial positions,
- Zeroing all digital measuring instruments (tare),
- Calibrating RPM, voltage, current, and thrust sensors.

During static testing, the propeller is rotated at varying speeds, and the corresponding thrust, torque, and electrical power consumption are recorded. The system is designed to ensure synchronized data logging and accurate measurement.

The test setup enables the real-time acquisition of propeller RPM, motor power consumption, current (A), voltage (V), and thrust (N), all visualized via an LCD monitor. **Figure 2.1** illustrates the overall measurement and data acquisition configuration of the test bench.

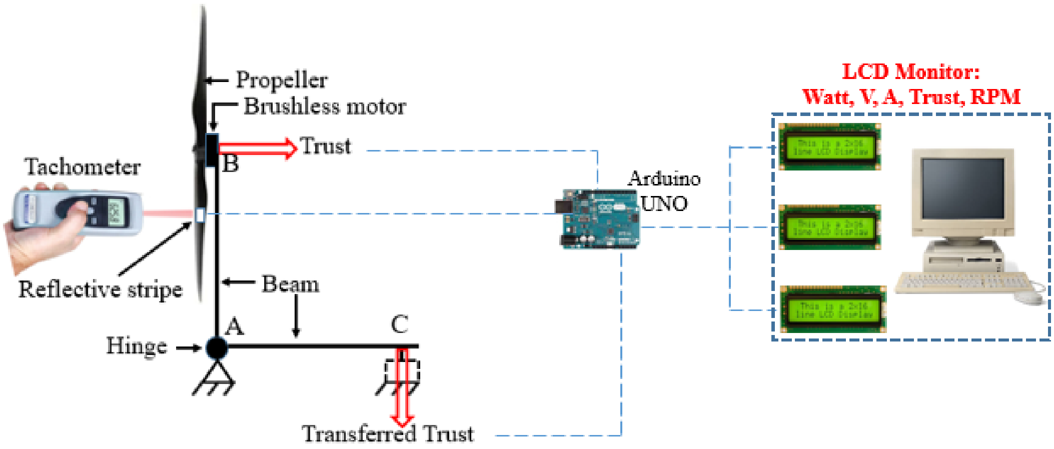

Figure 2.1 Kinematic and measurement schematic of the brushless motor test bench.

2.1.1 Thrust Measurement

The thrust measurement setup utilizes a horizontal beam mechanism equipped with a digital load cell. The propeller and brushless motor assembly is mounted at one end of the beam (point B), while the beam is hinged at point A and supported by a load cell at point C. The configuration forms a balanced lever system, where distances AB and AC are kept equal to ensure symmetrical load distribution.

As the propeller generates thrust during rotation, the force at point B causes a downward reaction at point C. This reaction is measured by the digital scale in the form of mass (grams), which is then converted into thrust force (Newtons) using the following equation:

$$F_{thrust} = m \cdot g \qquad 2.1$$

where:

- F_{thrust} = thrust force (N),
- m = measured mass (kg),
- g = gravitational acceleration (9.81 m/s²).

This method provides a simple yet effective way to translate dynamic thrust into a measurable static load using mechanical leverage. To enhance clarity, a dedicated schematic illustrating the thrust transfer mechanism is recommended, complementing the system overview provided earlier.

2.2 Simulation Setup

The CFD simulation was conducted using ANSYS Workbench 2021 R2 and consisted of three main stages: geometry modeling, meshing, and setup in ANSYS CFX.

2.2.1 Geometry Modeling

The propeller geometry was obtained from a 3D scan of a commercial propeller, then refined and reconstructed in SolidWorks. The final CAD model was exported in IGES format and imported into ANSYS Geometry. Two enclosures were created to represent the fluid domain: a rotating cylindrical domain surrounding the propeller, and a stationary rectangular domain representing the ambient air volume. Boolean operations were used to subtract and define the interaction between the rotating and stationary domains. Named selections were assigned to critical surfaces such as propeller, inlet, outlet, and opening to facilitate boundary condition definitions in CFX. Shown on **Figure 2.1.**

Figure 2. 2. Resul of 3D Scanned

Figure 2.3. Conversion 3D to Solidwork

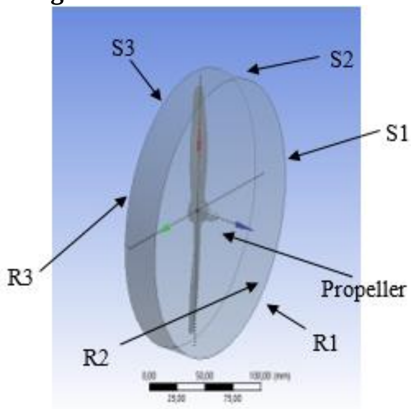

Figure 2.4. surface identifications

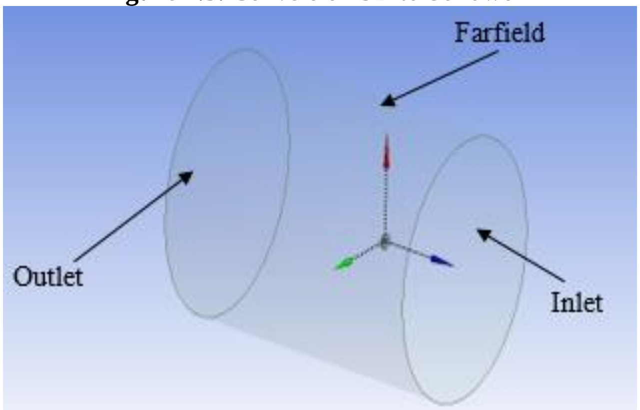

Figure 2.5. Computational domain

2.2.2 Computational Domain and Mesh Structure

The computational domain (as shown in Figure 2.5) consists of two primary regions:

- A rotating cylindrical domain, which includes the propeller geometry, and
- A stationary rectangular domain, which represents the surrounding free-stream environment.

To prepare the model, a *Boolean subtract* operation is applied so that the fluid region excludes the volume occupied by the propeller, allowing the software to treat the blades as solid walls through which no flow passes. The rotating domain is assigned an angular velocity corresponding to the propeller's RPM, while the stationary domain facilitates fluid inflow and outflow.

The mesh was generated using unstructured tetrahedral elements to accommodate the complex blade geometry. A Body of Influence (BoI) technique was applied near the propeller region to increase mesh resolution in areas of high flow gradient. The total number of mesh elements was approximately 1.2 million, which ensured both computational efficiency and accuracy.

To ensure mesh quality and independence, a mesh refinement study was conducted. Three mesh sizes (coarse, medium, and fine) were tested, and the results were compared based on thrust output and residual convergence. The final mesh (medium resolution) was selected as it provided <2% variation in thrust prediction compared to the finer mesh, indicating mesh independence and validating the reliability of the simulation results.

2.2.3 Mesh Generation

Meshing was performed using ANSYS Meshing with an unstructured tetrahedral grid, chosen for its ability to discretize complex blade geometries effectively (see *Figure 2.6*). A Body of Influence (BoI) technique was applied around the propeller to refine the mesh locally and improve resolution in high-gradient zones, especially near the blade surfaces.
Mesh characteristics used in the final simulation are summarized as follows:

Mesh Parameter	Value
Mesh type	Unstructured (Tetrahedral)
Total number of elements	~1,236,000
Total number of nodes	~290,000
BoI refinement zone	Cylindrical region around propeller
Average skewness	< 0.25 (good quality)
Min element quality	> 0.30

To ensure **mesh independence**, three mesh densities (coarse, medium, and fine) were tested. The final mesh was selected based on convergence behavior and result stability, with thrust deviations of less than 2% between medium and fine meshes—indicating reliable mesh-independent results.

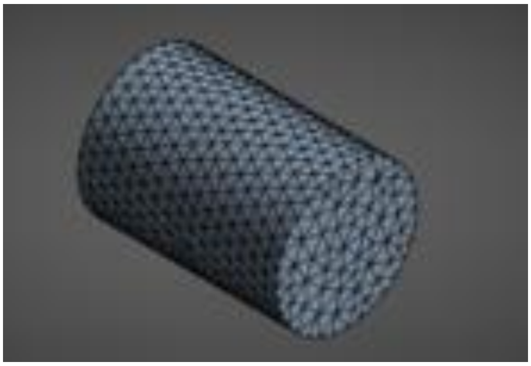

Figure 2.6. Meshing processed

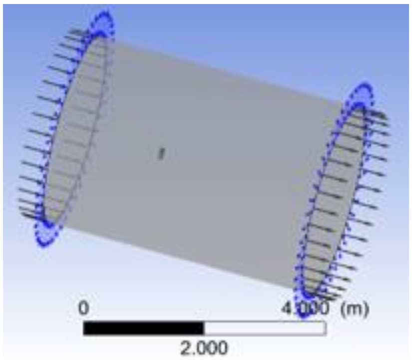

Figure 2.7. Setting of boundary condition

2.2.4 Boundary Conditions and Solver Setup

Two fluid domains were defined on **Figure 2.7.**

- **Rotating domain**: assigned as a rotating frame of reference with defined angular velocity.
- **Stationary domain**: with specified inlet velocity, outlet pressure, and openings.

Boundary conditions included:

- **Inlet**: velocity inlet,
- **Outlet**: pressure outlet,
- **Opening**: ambient flow interaction,
- **Propeller surface**: wall with rotational motion applied.

Printed ISSN: 1411-4348
Online ISSN: 2541-4577

The Multiple Reference Frame (MRF) model was employed to simulate the rotating region. Solver control was set to steady-state with high-resolution advection schemes. Once the simulation ran to convergence, results were analyzed for thrust, torque, pressure distribution, and flow streamlines.

2.3 Theoretical Background

In addition to the thrust and torque coefficients, the **power coefficient** and **propeller efficiency** are also important parameters for evaluating the aerodynamic performance of a propeller.

2.3.1 Power Coefficient

The performance of the propeller is evaluated using dimensionless coefficients, including thrust coefficient, torque coefficient, power coefficient, and propeller efficiency.

Thrust Coefficient:

$$C_T = \frac{T}{\rho n^2 D^4} \tag{2.2}$$

Power Coefficient:

$$C_P = \frac{P}{\rho n^3 D^5} \tag{2.3}$$

Advance Ratio:

$$J = \frac{V_a}{nD} \tag{2.4}$$

Propeller Efficiency:

$$\eta = \frac{J \cdot C_T}{2\pi \cdot C_Q} \tag{2.5}$$

Where:

- T = Thrust force (N)
- P = Shaft power input (W)
- Q = Torque (Nm)
- ρ = Air density (kg/m^3)
- D = Propeller diameter (m)
- n = Rotational speed (rev/s)
- Va = Advance speed (m/s)
- C_T, C_P, C_Q = thrust, power, and torque coefficients respectively
- η = propeller efficiency

These coefficients allow comparison between different propellers and operating conditions regardless of size or speed.

2.3.2 Propeller Efficiency

Propeller efficiency (η) is defined as the ratio of useful output (thrust power) to input shaft power. It can also be derived using dimensionless coefficients:

$$\eta = \frac{J \cdot K_T}{2\pi \cdot K_Q} \quad 2.4$$

This expression shows how efficiently the propeller converts rotational energy into thrust. A high efficiency indicates effective momentum transfer from the blades to the airflow.

2.3.3 Blade Element Theory (BET)

Blade Element Theory (BET) is a foundational analytical method used to estimate propeller performance. It divides the propeller blade into small elements along its span and analyzes the aerodynamic forces on each segment individually.

Each element experiences:

- **Lift force (L)** due to airflow velocity and blade angle,
- **Drag force (D)** due to resistance against motion.

The elemental thrust and torque contributions are calculated and then integrated along the blade radius:

$$dT = \frac{1}{2}\rho V^2 c\, C_L \cos(\phi)\, dr \tag{2.5}$$

$$dQ = \frac{1}{2}\rho V^2 c\, C_D \sin(\phi)\, r\, dr \tag{2.6}$$

Where:

- c = chord length of the blade element,
- C_L ,C_D = lift and drag coefficients,
- Φ = inflow angle,
- r = radial position along the blade,
- V = relative velocity at the blade section.

The integration of dT and dQ from root to tip gives total thrust and torque. BET can be further enhanced using Blade Element Momentum Theory (BEMT), which combines BET with momentum theory for higher accuracy.

3. RESULS AND DISCUSIONS

3.1 Performance of the Test Bench (with Efficiency Discussion)

The performance of a test bench refers to its ability to produce accurate and reliable data during static experiments. To evaluate its performance, a series of verifications, calibrations, and measurements were conducted. A well-performing test bench improves both the quality and credibility of experimental results.

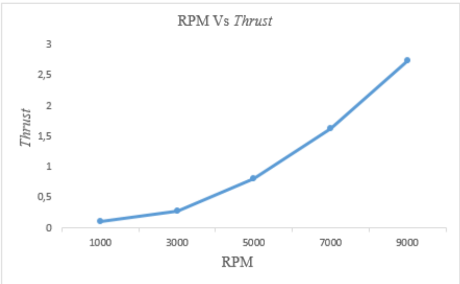

Figure 3.1. Correlation of rpm and Trust experimentally

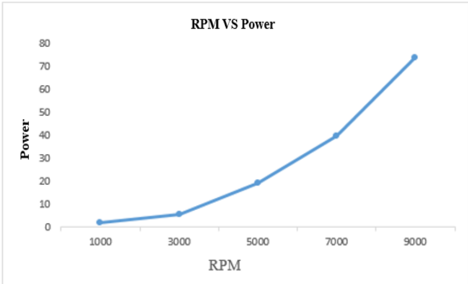

Figure 3.2. Correlation of rpm and Power experimentally

Figure 3.1 presents the static thrust performance of the 7×4 propeller as a function of rotational speed (RPM). The results show a clear increasing trend: as RPM increases, the generated thrust also increases proportionally. The maximum recorded thrust reached **2.7468 N at 9000 RPM**. This indicates a linear correlation between rotational speed and thrust under static conditions, demonstrating that the test bench can accurately capture the response of the propulsion system to speed variation.

Figure 3.2 shows the correlation between RPM and power consumption. At the highest test speed of **9000 RPM**, the measured power consumption reached **74.03 W**. Similar to the thrust trend, power consumption increases with RPM, reflecting the higher energy input required at higher motor speeds—this is typical behavior for brushless DC motors.

To evaluate how efficiently the propeller converts electrical power into thrust under static conditions, **propulsive efficiency** η is generally defined as:

$$\eta = \frac{T \cdot V_a}{P} \quad (3.1)$$

Where:

η = propeller efficiency

T = thrust (N)

Va = advance velocity (m/s)

P = input shaft power (W)

However, since this is a static test (Va=0), the actual propulsive efficiency is theoretically zero. To enable internal comparison, we define a pseudo-efficiency η^* as the ratio of thrust to power:

$$\eta^* = \frac{T}{P} \quad (3.2)$$

This ratio does not represent actual propulsion efficiency, but rather serves as an internal metric for comparing how much thrust is generated per unit of electrical power under static conditions.

At 9000 RPM, the pseudo-efficiency is:

$$\eta^* = \frac{2.7468}{74.03} \approx 0.0371\,\mathrm{N/W}$$

This indicates that the system produced approximately **0**.0371 N of thrust per Watt of power input. Although not a measure of true flight performance, this internal comparison is still valuable for assessing relative performance across configurations.

For dynamic performance prediction, efficiency analysis should incorporate forward velocity and dimensionless coefficients such as:

$$\eta = \frac{J \cdot K_T}{2\pi \cdot K_Q} \quad (3.3)$$

Where:

$J = V_o/nD$ = advance ratio

C_T, C_Q = thrust and torque coefficients

n = revolutions per second

D = propeller diameter (m)

This would allow full comparison with CFD and theoretical performance benchmarks under realistic flight conditions.

3.2 Comparison of Experimental and CFD Results

3.2.1 CFD Simulation Results

Following the meshing and setup process in ANSYS CFX, the simulation was executed for various rotational speeds (RPM). The output obtained was the thrust force (in Newtons) generated by the 7×4 propeller at each RPM setting. The maximum simulated thrust recorded was **3.63577 N at 9000 RPM**, as shown in **Figure 3.3**, which presents the thrust-RPM relationship from the CFD results.

3.2.2 Experimental vs. CFD Comparison

To enable comparison, experimental thrust data originally measured in grams were first converted into Newtons. The comparative thrust values at five different RPM levels are shown in **Figure 3.4**, where both CFD and experimental data are plotted on the same axes.

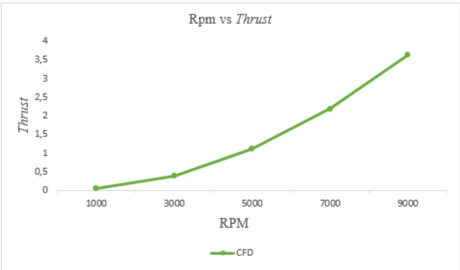

Figure 3.3. Correlation of rpm and Trust CFD

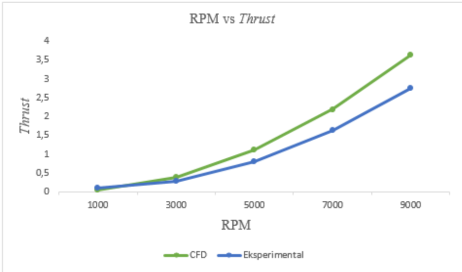

Figure 3.4. Correlation of rpm and Trust CFD and experiment

The comparison reveals a consistent trend: **CFD simulations generally predict higher thrust** than the experimental results at all RPM levels except for **1000 RPM**, where the experimental thrust is slightly higher.

Tabel 3.1 Data of experimental and CFD simulation operation

RPM	Experimental Thrust (N)	CFD Thrust (N)	Difference (N)
1000	0.1046	0.0410	**−0.0636**
3000	0.2874	0.3933	+0.1059
5000	0.5444	1.1092	+0.5648
7000	1.6252	2.1900	+0.5648
9000	2.7468	3.6358	+0.8890

The **largest deviation** occurred at **9000 RPM**, where CFD predicted a thrust of **3.63577 N**, while the experimental result was **2.7468 N**, resulting in a difference of **approximately 0.889 N**.

Analysis of Differences

While general differences between CFD and experimental results can be attributed to idealized simulation conditions and measurement uncertainties, further analysis provides deeper insights:

- At **low RPM (1000)**, the **experimental thrust was higher than CFD**, which could indicate that the **CFD model underpredicts low-speed performance**, possibly due to challenges in resolving low-Reynolds-number flow behavior or numerical dissipation in the near-stall regime. Additionally, **the load cell used in the experiment was calibrated with high sensitivity**, making it suitable for capturing small force values reliably.
- At **higher RPMs**, the **CFD thrust consistently exceeded experimental values**, which may be influenced by several factors:
  - **CFD assumes ideal flow** without mechanical friction, turbulence dissipation, or shaft losses, resulting in more optimistic predictions.
  - The **test bench may not fully capture 3D flow phenomena** such as **tip vortices**, which become more pronounced at higher RPMs and may affect thrust generation. These vortices may not be fully sensed by the load cell, leading to underestimation in experimental readings.
  - **Temperature rise and motor heating** at higher RPMs could reduce motor efficiency, affecting the real thrust produced—an effect not modeled in CFD.

Despite these quantitative differences, **both methods show a consistent upward trend in thrust with increasing RPM**, confirming the qualitative agreement between experimental and CFD approaches. This supports the conclusion that the CFD simulation provides a reliable estimation of performance trends, especially when validated with physical test data. Further improvements in the model—such as incorporating dynamic flow, heat effects, and turbulence refinement—are recommended to enhance accuracy.

4. CONCLUSION

This study successfully evaluated the performance of a 7×4 (7040) propeller using a combination of experimental static thrust testing and Computational Fluid Dynamics (CFD) simulation. A dedicated low-cost propeller test bench was developed and validated, and it proved capable of providing accurate and repeatable thrust and power measurement across various RPM levels. This setup demonstrated reliability in capturing propeller performance for small-scale UAV propulsion applications.

Both experimental and CFD results showed a consistent thrust increase with RPM under static conditions. However, CFD predicted higher thrust compared to experimental measurements at most RPM levels. The largest deviation occurred at 9000 RPM, where CFD thrust exceeded the experimental value by **0.889 N**, equal to a **31.4% difference**. This discrepancy arises from idealized CFD assumptions such as neglecting mechanical losses, shaft friction, air turbulence, and tip vortex dissipation—factors that occur in real experimental environments.

Despite quantitative differences, the **trend agreement** confirms that CFD can reasonably predict propeller behavior when properly validated. The experimental results verified the functionality of the test bench, while CFD provided detailed aerodynamic insights such as pressure distribution and flow phenomena, which are not measurable experimentally. **Therefore, using both methods in combination provides a more comprehensive and reliable evaluation strategy for small propeller performance assessment.**

Research Contributions

The main contributions of this study are:

1. **Development of a functional propeller test bench** using an Arduino-based measurement system capable of recording thrust, RPM, voltage, current, and power in real time.

2. **CFD validation of the 7040 propeller performance**, which has not been extensively analyzed in previous research despite its widespread use in micro UAVs.
3. **Establishment of a comparative framework** between CFD and experimental testing, demonstrating that CFD can be used as a reliable design and prediction tool when validated with physical data.

Future Work and Practical Impact
Future work will extend to:

- Dynamic testing using forward airflow (non-zero advance ratio) to analyze thrust coefficient, torque coefficient, and propeller efficiency more comprehensively.
- Mesh optimization and turbulence model refinement in CFD to minimize prediction errors.
- Integration of results into **UAV propeller selection and optimization**, improving flight endurance and energy efficiency for small-scale drone applications.

This combined experimental–CFD approach provides a practical reference for researchers and UAV developers working on propeller selection, performance prediction, and propulsion system optimization.

Acknowledgement

The Research was carried out at Fluid and Aerodynamics Research **(FAERO);** it is a research group within the Doctoral Programme in Mechanical Engineering at Universitas Muhammadiyah Surakarta. The authors would like to express sincere gratitude to our academic supervisor for their invaluable guidance throughout the completion of this research. We also deeply appreciate the support and encouragement from our colleagues, which contributed significantly to the successful execution of this project.

Special thanks are extended to Mr. Buyung Junaidin from ITDA for his assistance and insights during the development of the experimental setup. Furthermore, we gratefully acknowledge the support provided by the *Sub Directorate Research and Innovation (DRI-DRPPS)* and the *Directorate of Research and Community Service The ministry of higher education research and technology Indonesia*, whose contributions were instrumental in the realization of this study.

REFFERENCES

[1] B. Fahrureza and T. Priangkoso, “Perancangan Instrumentasi Alat Uji Test Bench Electronic Ducted Fan (EDF) Menggunakan Arduino Mega 2560,” *J. Ilm. Momentum*, vol. 19, no. 2, p. 150, 2023, doi: 10.36499/jim.v19i2.10083.

[2] H. S. Maulana, R. S. Dewanto, and E. H. Binugroho, “Rancang Bangun Alat Uji Gaya Dorong Propeller untuk Karakterisasi Motor Pada Sistem Propulsi Unmanned Aerial Vehicle,” *Rotasi*, vol. 26, no. 1, pp. 62–68, 2024, doi: 10.14710/rotasi.26.1.62-68.

[3] H. R. Mas Intan, Muhammad Luqman Bukhori, and Sabri Alimi, “Rancang Bangun Alat Uji Thrust Motor Brushless Dengan Variasi Propeller,” *Tek. STTKD J. Tek. Elektron. Engine*, vol. 9, no. 1, pp. 65–75, 2023, doi: 10.56521/teknika.v9i1.836.

[4] J. Unnikrishnan, S. Ambadi, and P. Ghughtyal, “Performance Analysis of a Small-Scale Master Airscrew Propeller using Experimental and Analytical Methods,” *Int. Res. J. Eng. Technol.*, no. June, 2020, [Online]. Available: www.irjet.net

[5] C. Sasmito, R. Pujiwat, D. H. Priatno, I. Iskendar, M. H. Khoirudin, and D. B. Eskayudha,

"Design of Airscrew Propeller as an Alternative Main Propulsion for Wing in Surface Effect (WiSE) A2C Using the Simplified Method Approach," *Mek. Maj. Ilm. Mek.*, vol. 23, no. 2, p. 124, 2024, doi: 10.20961/mekanika.v23i2.77937.

[6] S. W. . Mak, JH., "Analysis of Propeller Blade Through CFD Modellinge," *J. Des. Sustain. Environ.*, 2022.

[7] A. bin Aminuddin and W. S.-I. Wan Salim, "Aerodynamic Analysis of Propeller for Heavy Lifting Drone Applications Using Blade Element Momentum Analysis," *Prog. Aerosp. Aviat. Technol.*, vol. 2, no. 2, pp. 24–32, 2022, doi: 10.30880/paat.2022.02.02.004.

[8] L. C. Hang, N. A. N. Rosli, M. A. Wahid, N. Othman, S. Mat, and M. Z. M. Zain, "CFD Analysis on Propeller at Varying Propeller Disc Angle and Advance Ratio," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 96, no. 1, pp. 82–95, 2022, doi: 10.37934/arfmts.96.1.8295.

[9] H. A. Kutty and P. Rajendran, "3D CFD simulation and experimental validation of small APC slow flyer propeller blade," *Aerospace*, vol. 4, no. 1, 2017, doi: 10.3390/aerospace4010010.

[10] N. H. Nguyen and H. K. Ngo, "Experimental study on static thrust characteristics of Master Airscrew E9x6 propeller with duct," *Sci. Technol. Dev. J.*, vol. 20, no. K3, pp. 60–66, 2017, doi: 10.32508/stdj.v20ik3.1093.

[11] A. Kurniawan, "Pengukuran dan Analisis CFD Performa Propeller Master Airscrew G/F 3 9x6," *J. Ilm. Aviasi Langit Biru*, vol. 10, no. 1, pp. 73–87, 2017.