ACCELERON AEROSPACE JOURNAL (AAJ)
E-ISSN: 2583-9942 (Online) | Accessible at www.acceleron.org.in
Volume 5, Issue 2, (pp.1377-1382)
Article ID: AAJ.11.2106-2549
Research Paper
https://doi.org/10.61359/11.2106-2549

Computational Fluid Dynamics Analysis and Experimental Validation of a 42×16.5 Propeller for Hovering eVTOL Applications

Sadiq Ali Mir*, Dr. Promio Charles F†, Dr. Ravindra S Kulkarni‡

R V College of Engineering, Aerospace Engineering, Bengaluru, India.

Abstract: This study presents a comprehensive Computational Fluid Dynamics (CFD) analysis of a 42×16.5 propeller operating under hover conditions, relevant to single passenger electric Vertical Take-Off and Landing (eVTOL) aircraft. The objective is to validate the CFD model against experimental thrust data at various rotational speeds. Using ANSYS Fluent, simulations were performed that incorporates Reynolds average Navier Stokes (RANS) equations with the SST k-ϵ turbulence model. The results show a strong correlation with the experimental data, with thrust prediction errors within 5-7%, thus affirming the reliability of the CFD approach to assess propeller performance in hover.

Table of Contents

1. Introduction

Urban Air Mobility (UAM) is an emerging aerospace paradigm focused on the deployment of aerial vehicles within urban environments to alleviate surface congestion, reduce travel time, and enable sustainable point to point transportation. Enabled by advances in electric propulsion, lightweight materials, and autonomous control, eVTOL aircraft are central to UAM concepts due to their ability to perform vertical lift and hover operations with minimal noise and emissions. According to NASA's UAM ecosystem vision, future urban transport systems will feature networks of eVTOL vehicles operating from rooftop vertiports to facilitate short range, on demand mobility in densely populated cities [1]. The propulsion system is a critical subsystem in eVTOL aircraft, directly influencing hover performance, energy efficiency, and operational safety. In particular, propeller driven designs dominate current UAM demonstrators and prototypes due to their mechanical simplicity and high thrust-to-weight ratios. As observed in several commercial and academic prototypes such as Joby S4, Lilium Jet, and Jetson One, multiple distributed electric propellers provide redundancy and control flexibility while supporting vertical take-off capabilities [2], [3]. For successful integration into the UAM ecosystem, accurate prediction and optimization of propeller performance under hover conditions is essential, especially considering the tight margins for payload, noise, and energy consumption under regulatory constraints set by agencies such as EASA and FAA [4]. Although full scale flight tests are invaluable for validation, they are cost prohibitive during the early stages of design. Therefore, high fidelity Computational Fluid Dynamics (CFD) simulations have become essential tools to predict propeller aerodynamic behavior. These simulations, when validated against experimental test data, offer insights into the complex unsteady flow fields and allow for iterative design optimization without extensive physical prototyping. This paper presents a detailed CFD analysis of a 42×16.5 propeller operating in hover, with simulation results validated against static thrust test data. The aim is to establish a robust methodology for evaluating individual propeller performance in the design of multi rotor eVTOL systems suited for urban airspace.

2. Methodology

The objective of this study is to perform a high fidelity Computational Fluid Dynamics (CFD) analysis of a 42× 16.5″ electric propeller (0.762 m diameter) under hover conditions, with validation against publicly available static thrust test data. The methodology includes geometric modeling, mesh generation, governing equations, and turbulence modeling, boundary conditions, numerical solution, and post processing of aerodynamic performance parameters.

*R V College of Engineering, Aerospace Engineering, Bengaluru, India. **Corresponding Author:** sadiqalimir.ae21@rvce.edu.in.
†R V College of Engineering, Aerospace Engineering, Bengaluru, India.
‡R V College of Engineering, Aerospace Engineering, Bengaluru, India.
Article History: Received: 07-July-2025 || Revised: 30-August-2025 || Accepted: 30-August-2025 || Published Online: 30-August-2025.

A. Propeller Geometry and Parameters

The propeller used in this study is a two bladed carbon fiber design with tapered and twisted blades to optimize hover efficiency. The key geometric and material parameters of the propeller are summarized in Table I. The geometry of the propeller was reconstructed using manufacturer specifications and is shown in Fig. 1.

Table 1: Propeller Parameters

Parameter	Value
Diameter (m)	0.7620
Pitch Angle (degrees)	16.5
Maximum Blade Chord Length (m)	0.6080
Minimum Blade Chord Length (m)	0.1272
Number of Blades	2
Blade Material	Carbon Fiber
Weight (kg)	0.2

B. Computational Domain and Mesh

The computational domain consists of a cylindrical volume that surrounds the propeller with a radius of 5D and an axial length of 8D downstream, where D = 0.762 m is the diameter of the propeller. This domain size ensures minimal boundary interference with the wake and flow structures.

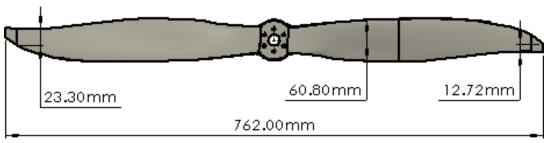

Fig. 1. Schematic of the 0.762 m diameter, two-bladed carbon fiber propeller

Table 2 highlights the mesh settings used in the CFD simulations, including tetrahedral elements with local refinement near the propeller and the use of prism layers to resolve near wall regions.

Table 2: Mesh Parameters Used In CFD Simulations

Parameter	Value
Total node count	4.0 million
Mesh type	tetrahedral
Near-wall layers	5 prism layers
First layer thickness	1.24×10^{-4} m
Growth rate (inflation)	1.2
Transition ratio	0.272
Global element size	80 mm
Minimum edge length	1.24 × 10−4 m
Curvature capture	Enabled (min size 0.8 mm)
Mesh quality target skewness	< 0.9

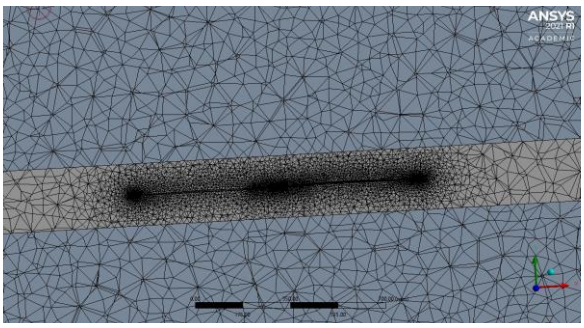

Fig. 2. Computational mesh around the propeller

Acceleron Aerospace Journal || AAJ.11.2106-2549
Volume 5, Issue 2, pp (1377-1382)
E-ISSN- 2583-9942
https://dx.doi.org/10.61359/11.2106-2549

The propeller rotation was simulated using a sliding mesh technique between the rotating zone containing the propeller blades and the stationary outer domain, enabling transient resolution of the flow field [5].

C. Governing Equations and Turbulence Model

The flow was simulated by solving the unsteady Reynolds-averaged Navier Stokes (URANS) equations, consisting of continuity and momentum equations:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \tau + \rho \mathbf{g} \tag{2}$$

where ρ is fluid density, **u** velocity vector, p pressure, and τ viscous stress tensor [6].

The Shear Stress Transport (SST) k-ϵ turbulence model was utilized for its accuracy in capturing near wall flows and flow separation phenomena typical in propeller aerodynamics [7].

D. Boundary Conditions and Operating Parameters

Hover conditions were simulated by imposing zero velocity at the inlet (stationary ambient air at standard sea level conditions, $\rho = 1.225$ kg/m^3 and $\mu = 1.789 \times 10^{-5}$ Pa·s), and a pressure outlet at atmospheric pressure. The propeller was set to rotate at specified RPMs corresponding to the available test data for validation. No slip boundary conditions were applied on all solid surfaces. The sliding mesh method simulated the relative motion of the rotating blades within the stationary domain [8].

E. Numerical Setup

Simulations were carried out using ANSYS Fluent 2023 R1 with a pressure based solver configured for compressible flow to account for tip speeds nearing transonic regimes at higher RPMs. Second-order spatial discretizations and implicit time integration schemes were adopted for accuracy and stability. The time step corresponded to 1° of propeller rotation, and simulations ran until the thrust values reached periodic steady state, verified by convergence within 1% over three consecutive revolutions.

F. Aerodynamic Performance Matrix

Thrust (T) and torque (Q) were computed by integrating pressure and shear stresses over the blade surfaces:

$$T = \int_S (\mathbf{p} \cdot \mathbf{n}) dS + \int_S \tau \cdot \mathbf{n} dS \tag{3}$$

$$Q = \int_S \mathbf{r} \times (\mathbf{p} + \tau) dS \tag{4}$$

where **r** is the radial vector from the rotation axis, and **n** is the surface normal [9].

Power consumption is evaluated as $P = \omega Q$, with angular velocity ω. Dimensionless coefficients C_T and C_P were calculated as:

$$C_T = \frac{T}{\rho n^2 D^4}, C_P = \frac{P}{\rho n^3 D^5} \tag{5}$$

where n is rotational speed in revolutions per second [10].

3. Results and Discussion

The CFD simulations were conducted for the 42×16.5″ propeller at multiple rotational speeds. The computed thrust values were compared against the experimental static thrust data available online to validate the numerical model.

Table 3: Comparison Of Experimental and CFD Thrust Values

RPM	Thrust EXP (N)	Thrust CFD (N)	Error (%)
2970	261.25	247.54	5.25
3150	300.55	281.55	6.32
3450	363.08	340.16	6.31
3690	410.92	388.78	5.39
3780	432.48	407.41	5.80
3870	452.91	426.25	5.89
3960	474.48	445.31	6.15
4050	492.88	464.58	5.74
4140	513.97	484.07	5.82
4200	530.11	497.18	6.21

A. Thrust Comparison

Table III summarizes the thrust values obtained from CFD simulations alongside the experimental thrust measurements at different rotational speeds (RPM). The results show good agreement, with deviations mostly under 8%, demonstrating the accuracy of the CFD model for predicting propeller thrust in hover.

Figure 3 presents the thrust variation with RPM from both CFD and experimental data. The CFD results closely follow the experimental curve, validating the numerical simulation approach.

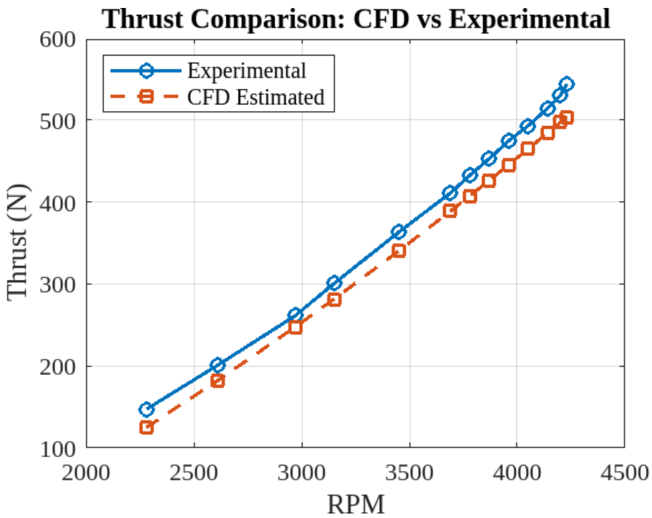

Fig. 3. Thrust versus RPM: Comparison between CFD predictions and experimental static thrust data.

B. Propeller Flow Field Analysis

To comprehensively examine the aerodynamic characteristics of the propeller under hover conditions, pressure and velocity contour plots were extracted from three-dimensional CFD simulations conducted at a rotational speed of 2970 RPM. These contours offer valuable insight into the nature of blade surface loading and the development of wake structures, both of which are critical to understanding the propeller’s aerodynamic performance and its thrust generation capability in stationary flight.

Acceleron Aerospace Journal || AAJ.11.2106-2549
Volume 5, Issue 2, pp (1377-1382)
E-ISSN- 2583-9942
https://dx.doi.org/10.61359/11.2106-2549

The pressure distribution over the blade surfaces at 2970 RPM, as depicted in Fig 4, indicates moderate aerodynamic loading characterized by smoothly varying pressure gradients along both the chordwise and spanwise directions. These gradients reflect the distribution of aerodynamic forces arising from local variations in angle of attack and relative velocity along the blade span. The continuous nature of the pressure transition from the leading to the trailing edge suggests that the flow remains largely attached, thereby minimizing the onset of separation and enhancing aerodynamic efficiency.

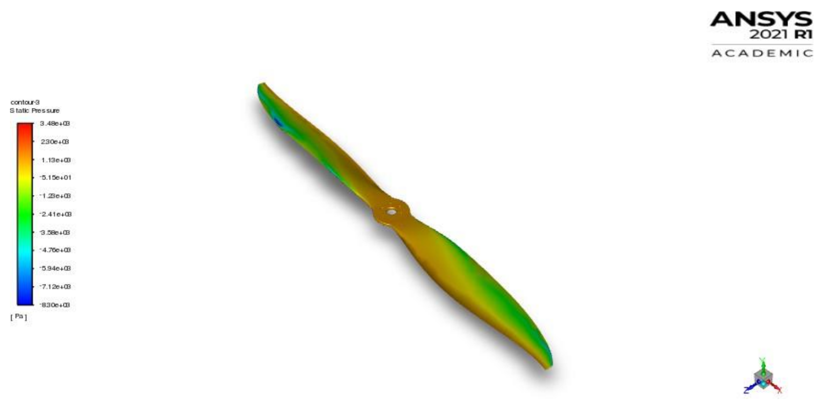

Fig. 4. Pressure contour at 2970 RPM

Fig. 5. Velocity contour at 2970 RPM

Correspondingly, the velocity field illustrated in Fig.5 reveals well defined regions of flow acceleration concentrated near the leading edges of the propeller blades. These regions correspond to zones of low pressure on the suction side, contributing to lift generation. A gradual reduction in velocity is observed toward the trailing edge, indicating smooth deceleration. In the downstream region, the contours clearly show the development of the propeller wake and a coherent slipstream, characterized by a central velocity deficit. This velocity reduction is a consequence of momentum transfer to the propeller, manifesting as net thrust in the direction opposite to the induced flow. The formation of concentrated tip vortices, a result of pressure equalisation between the suction and pressure sides of the blade near the tips. These vortices contribute to the induced flow field and affect the evolution of the wake. Their presence, along with the detailed pressure and velocity distributions, highlights the complex aerodynamic interactions that govern thrust production in hover. The close agreement between these flow field characteristics and the experimentally observed thrust performance reinforces the validity of the CFD model at this operating condition.

4. Results and Discussion

The results confirm that the employed CFD methodology accurately captures the propeller's aerodynamic behavior in hover conditions relevant to eVTOL applications. The thrust predictions align closely with experimental data, validating mesh quality, turbulence modeling, and boundary conditions. Pressure and velocity fields offer detailed insight into blade aerodynamics, useful for identifying potential improvements in blade design to reduce losses and enhance efficiency. The increased flow separation and vortex strength at higher RPM suggest areas for further aerodynamic refinement, such as blade twist optimization or addition of vortex control devices. These findings underscore the importance of high-fidelity CFD in the iterative design process of eVTOL propulsion systems, facilitating early-stage performance evaluation while reducing reliance on costly physical testing.

5. Conclusion

This study successfully validated a Computational Fluid Dynamics (CFD) model for a 42×16.5 propeller under hover conditions, demonstrating its accuracy and reliability for eVTOL applications. The CFD simulations' thrust predictions were in close agreement with experimental data, with an average error of approximately 5-7%, which affirms the methodology used in this research, including the computational domain, mesh generation, and the application of the SST k-e turbulence model. The detailed analysis of the pressure and velocity fields provided further insights into the propeller's aerodynamic performance, revealing how blade surface loading and wake structures contribute to thrust generation. The study's findings highlight that high-fidelity CFD simulations are a valuable and cost-effective tool in the early design stages of eVTOL propulsion systems. While the current model provides a strong foundation, the observed flow separation and vortex strength at higher rotational speeds suggest potential areas for future optimization, such as refining the blade geometry. In essence, this research establishes a robust methodology for predicting propeller performance, which can significantly reduce the need for costly physical prototyping and testing. The results not only validate the CFD approach but also lay the groundwork for future work aimed at enhancing the efficiency and performance of multi-rotor eVTOL systems, thus contributing to the advancement of Urban Air Mobility.

6. References

[1] Goyal, R., Reiche, C., Fernando, C., Serrao, J., Kimmel, S., Cohen, A., & Shaheen, S. (2018, November 21). Urban air mobility (UAM) market study (NASA Contractor Report, Report No. HQ-E-DAA-TN65181). Booz Allen Hamilton and University of California, Berkeley. https://ntrs.nasa.gov/citations/20190001472u

[2] European Union Aviation Safety Agency. (2024, June 10). Special condition for small-category VTOL-capable aircraft, Issue 2 (SC-VTOL-02) (Doc. No. SC-VTOL-02, Issue 2). https://www.easa.europa.eu

[3] Joby Aviation. (2023). Joby S4 aircraft overview. https://www.jobyaviation.com

[4] Wang, S., Pereira, L. T. L., & Ragni, D. (2025, February). Design exploration of UAM vehicles. Aerospace Science and Technology, 160, 110058. https://doi.org/10.1016/j.ast.2025.110058

[5] Lu, T., Shen, H., & Xie, Z. (2013). Sliding mesh approach for numerical simulation of rotor–stator interaction flow in axial compressors. Journal of Turbomachinery, 135(2).

[6] Ferziger, J. H., & Perić, M. (2002). Computational methods for fluid dynamics (3rd ed.). Springer.

[7] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal, 32(8), 1598–1605. https://doi.org/10.2514/3.12149

[8] ANSYS Inc. (2023). ANSYS Fluent 2023 R1. Canonsburg, PA, USA.

[9] Anderson, J. D. (2001). Low-speed aerodynamics (2nd ed.). Cambridge University Press. https://doi.org/10.1017/CBO9780511810329

[10] Brandt, J. B., & Selig, M. S. (2011, January). Propeller performance data at low Reynolds numbers. In AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition. https://doi.org/10.2514/6.2011-1255

7. Conflict of Interest

The author declares no competing conflict of interest.

8. Funding

No funding was issued for this research.