

Numerical and Experimental Analysis of Electric Ducted Propulsion Fans across Operating Speed Ranges

Vu Thai Son*, Nguyen Binh Nguyen, Chu Hoang Quan, and Dinh Cong Truong

Abstract: Electric ducted propulsion fans (EDPFs) have become a key propulsion technology for next-generation electric aircraft, urban air mobility (UAM) vehicles, and unmanned aerial systems, offering high propulsive efficiency, a compact configuration, and reduced noise emissions. This study presents both numerical and experimental performance evaluations of an EDPF with a rotational speed range from 2,000 to 7,000 rpm. The aerodynamic thrust and torque performance of an EDPF were evaluated using 3D Reynolds–Averaged Navier–Stokes (RANS) simulations with the SST $k-\omega$ turbulence model and subsequently validated against experiment. The results showed that the maximum difference between the numerical and experimental thrust at 6,000 rpm was very small with only 0.86%. The peak efficiency of the EDPF reaches 93% at 7,000 rpm. These numerical and experimental results confirm the accuracy of the computational model with RANS and demonstrate its applicability in the performance analysis and optimization of EDPF. With this design, it could be used as a propulsion system for unmanned surface vehicles (USVs) if the rotating drive mechanism is driven by a drive shaft or the electric motor can operate in a water environment.

Keywords: Aerodynamic characteristics, electric ducted propulsion fans, experimental analysis, RANS simulations.

1. INTRODUCTION

In recent years, ducted fans have become known as important components in modern aerospace and mechanical engineering. Ducted fans have distinct advantages over traditional open rotors, offering benefits such as greater thrust and efficiency compared to open rotors of the same size, while noise is significantly reduced. This structural advantage makes ducted fans an excellent choice for new aerospace projects [1].

Compared to traditional open rotors, ducted fan designs make significant improvements in both safety and aerodynamic efficiency under the same operating conditions [2, 3]. However, the complex aerodynamics makes them difficult to study. To fully understand these systems, researchers combine computational simulation with experimental testing. First, researchers collect basic operating data by experimentally testing ducted fans. Then, they calculate, design, and simulate numerical

models to confirm those experimental findings. Abrego and Bulaga [4] studied wind tunnel experiments on ducted fans under both axial and lateral flow conditions. They found that the shape of the exhaust fan blades (specifically the deflection angle and the chord length) determined a great deal of the thrust. Martin and Tung [5] calculated the aerodynamic loads acting on a 10-inch scale VTOL/STOL UAV model at different flight phases. They measured velocity via hot-wire anemometry along the inner and outer duct walls and into the downstream wake. This data shows exactly how the clearance at the blade tip affects thrust performance. Fleming et al. [6] applied a similar technique by integrating computational fluid dynamics (CFD) with wind tunnel metrics for a 12-inch ducted fan. To address the requirements at the microscale, Urban et al. [7] evaluated 50 mm and 64 mm miniaturized EDPF. By monitoring voltage, torque, and thrust, they extracted highly realistic operational data for the development of small UAVs.

Manuscript received April 14, 2026; received in revised form July 20, 2026; accepted August 12, 2026; available online September 30, 2026.

Vu Thai Son is with the Department of Mechanical Engineering, Vietnam Maritime University, Haiphong, Vietnam (e-mail: sonvt@vimaru.edu.vn). Nguyen Binh Nguyen and Dinh Cong Truong are with the School of Mechanical Engineering, Hanoi University of Science Technology, Hanoi, Vietnam (e-mails: nguyen.nb181910@sis.hust.edu.vn and truong.dinhcong@hust.edu.vn). Chu Hoang Quan is with the Institute of Vehicle and Energy Engineering, Le Quy Don Technical University, Hanoi, Vietnam (email: chu.h-quan.fas@lqdtu.edu.vn).

* Corresponding author.

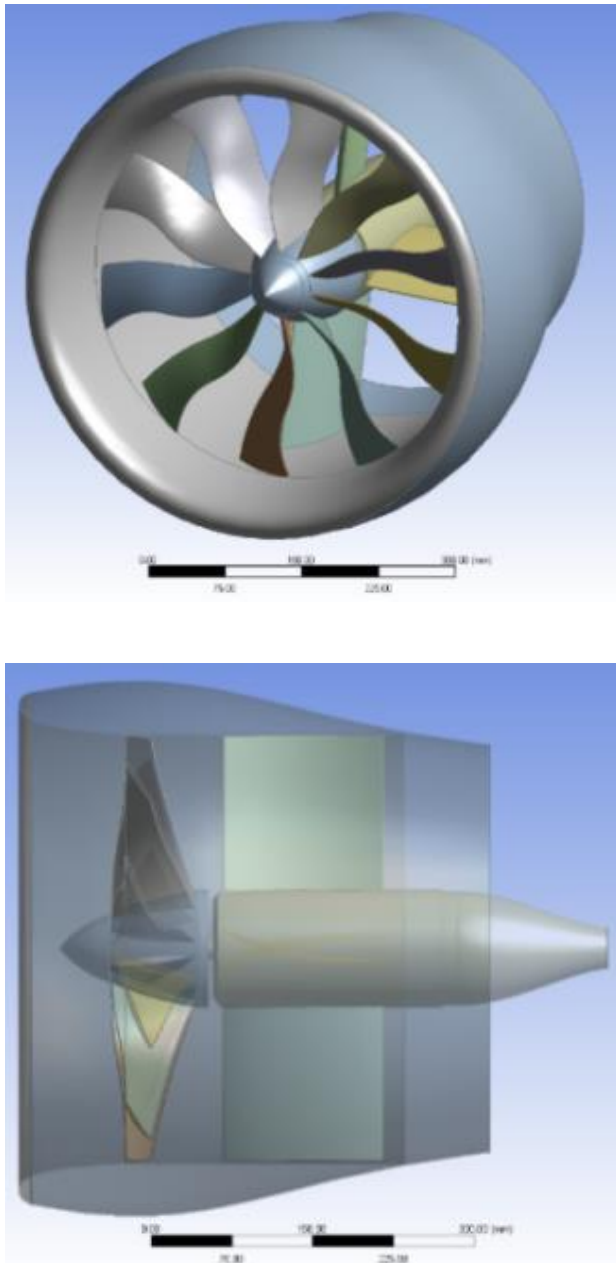


Fig. 1. 3D EDPF model.

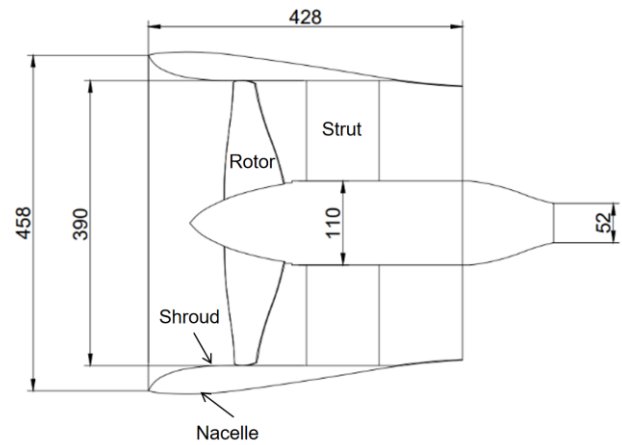


Fig. 2. Dimension of EDPF.

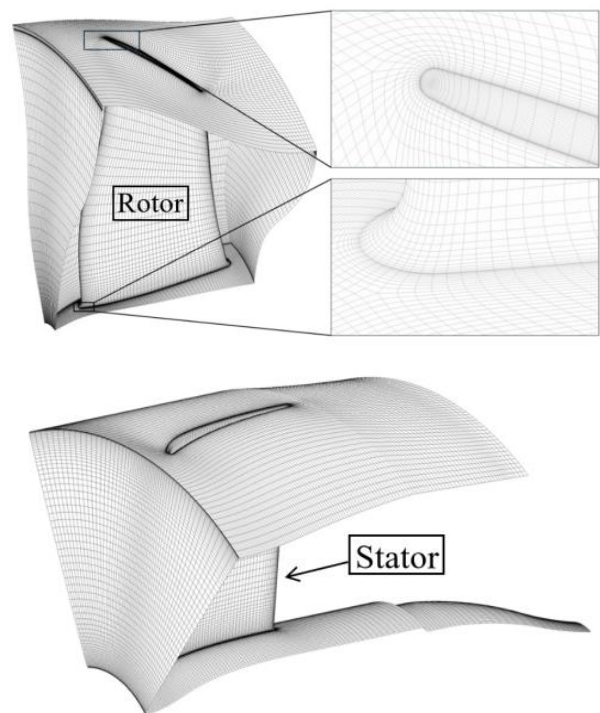


Fig. 3. 3D mesh of the EDPF.

Besides combined computational simulation and experimental ducted fan methods, some researchers use special methods for simulation studies of this system. Deng et al. [8] used sliding mesh techniques to simulate the unstable aerodynamic characteristics and high vortex flow at blade tips of ducted fans in hovering UAVs. To study complex aerodynamic conditions such as ground effects and flow splitting, recent studies are increasingly using Multi-Level Reference Frame (MRF) methods [9]. The result of these high-precision CFD models lies in their ability to detect small variations in thrust that simple physical sensors cannot capture. Inspired by these capabilities, Jiahao and Zhou [10–11] study a two-step strategy. They measured the thrust of a 150 mm ducted

fan at various. Their first publication detailed how to configure the rotor–stator layout to achieve the maximum thrust of a ducted fan, and their second paper presented a practical framework designed to balance power, thrust, and system performance in a ducted fan system. Dinh et al. [12] presented the ventilation hole configuration to reduce the temperature of the Rotor, End-Winding Forward, and End-Winding Rear of the EDPF with a decrease of 6.66 °C, 5.54 °C, and 6.8 °C, respectively.

However, all of the above studies have focused only on small diameter blades with low rotational speeds. Therefore, the system errors are difficult to control and achieve high accuracy. This study presents a numerical

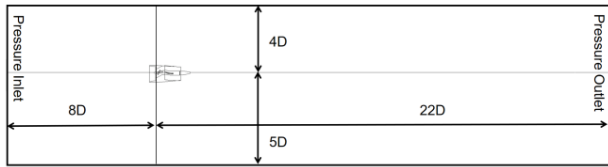


Fig. 4. Dimensions of the computational domain.

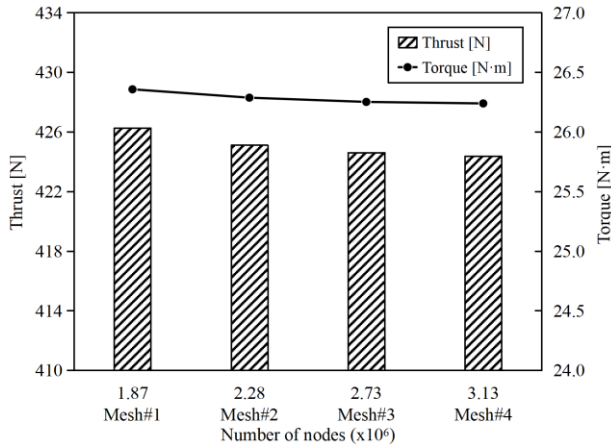


Fig. 5. Grid independence test at 6000 rpm.

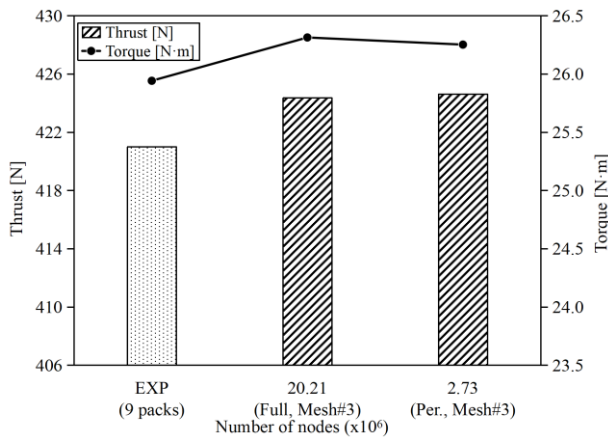


Fig. 6. Simulation and experiment thrust comparisons.

simulation analysis of a large EDPF with 390mm of diameter across a wide operating range from 2,000 to 7,000 rpm. The numerical results of thrust and torque of the EDPF are compared with the experimental results, which were measured by the Aerospace Propulsion Systems Laboratory, School of Mechanical Engineering, Hanoi University of Science and Technology.

2. NUMERICAL ANALYSIS

2.1. Geometry description

The EDPF used in this study has 10 rotor blades with an average blade tip clearance of 2 mm. An electric motor inside the central housing powers the fan and connects it to the duct with 6 strut blades as shown in Fig. 1. The overall dimensions of the EDPF are a shroud

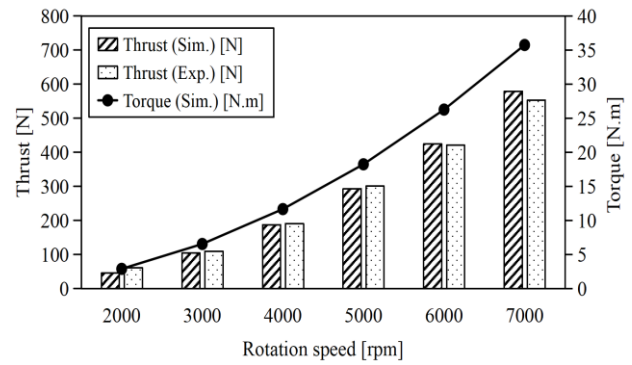


Fig. 7. Simulated and experimental thrust comparisons.

diameter of 390 mm, a length of 554 mm, and a nacelle diameter of 458 mm, operating at rotational speeds from 2000 rpm to 7000 rpm. The experimental EDPF used was of the exact same size as the simulation, fabricated from a carbon fiber composite material. The geometric dimensions of the EDPF are illustrated in Fig. 2.

2.2. Numerical method

To simulate the ducted fan, we used highly accurate 3D CFD simulations based on the steady state using $k - \omega$ SST (Shear Stress Transport) model, which provides high simulation accuracy for flows with adverse pressure gradients to capture the flow effects near the walls such as tip leakage flow, vortex behind the blades. This work utilizes ANSYS CFD 2025 R1 [13] to design, set up, simulate and export the results of the EDPF with ANSYS DesignModeler, CFX-Pre, CFX-Solver, and CFX-Post, respectively.

To reduce computational costs, a periodic simulation was executed utilizing a single-blade passage domain with periodic boundary conditions enforced on the rotational periodic interfaces. The circumferential symmetry of the steady axial flow field allows this periodic setup to capture the blade-to-blade flow physics identically to the full-domain setup.

2.3. Meshing

Cadence Fidelity AutoMesh package edition 2025.1 [14] was used for meshing both the fan zone (R), strut region (S), and other region (Fig. 3). The hexahedral mesh quality in this study was greater than 0.8.

2.4. Boundary conditions

In the CFD calculations of the ducted fan, only the fan is considered as the moving component, and it is necessary to define a region surrounding the fan. For sliding meshes, the rotating area must be structured in such a way that the local motion speed does not pass through the mesh boundaries as illustrated in Fig. 4.

2.5. Theoretical formulation

The total thrust (T_{total}) of an EDPF is the summation of the thrust generated by the core (T_{core}) (including the fan blade/hub, strut blade/hub, nose, and tailcone) and

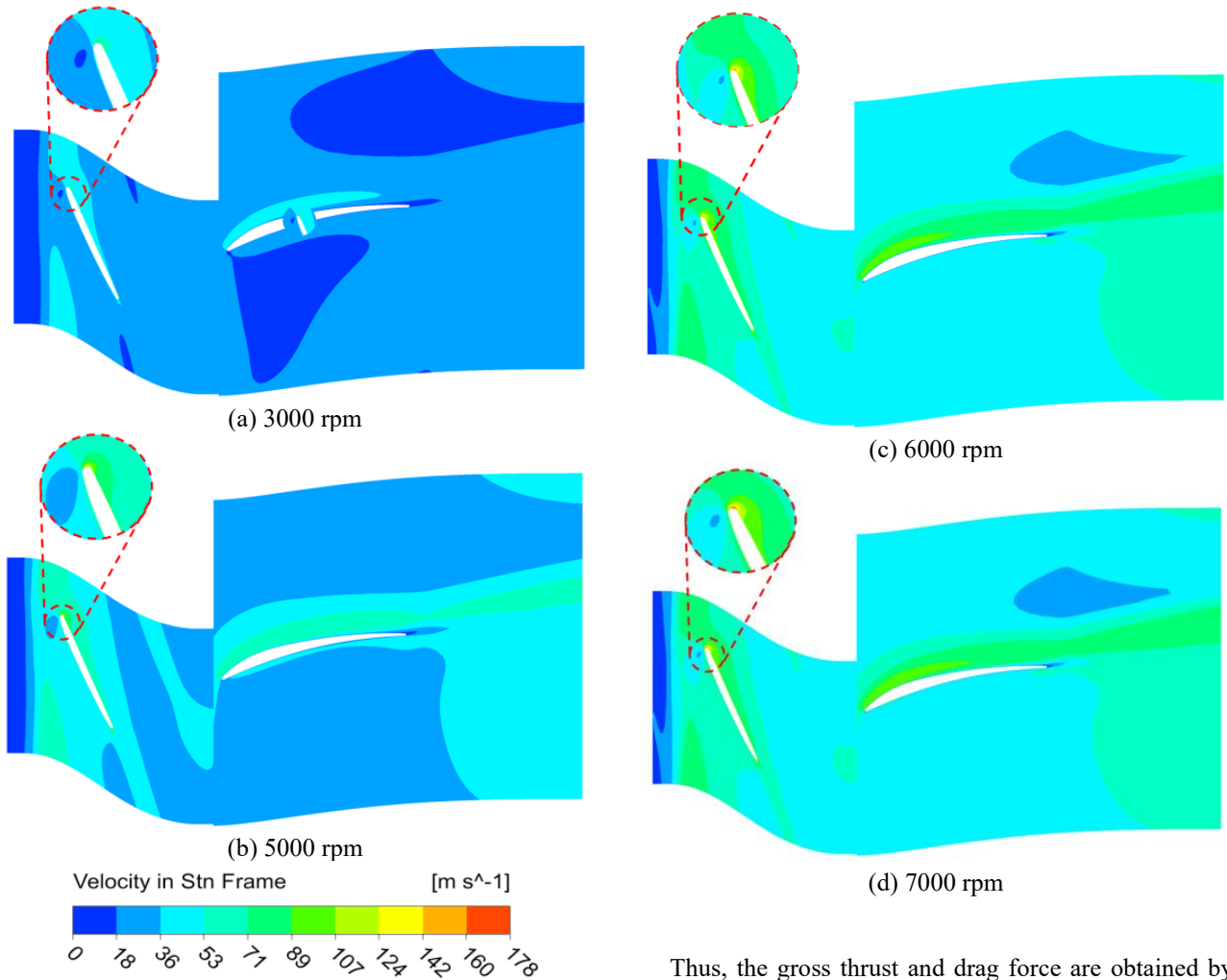


Fig. 8. Blade-to-blade velocity contours at the 95% span location.

the aerodynamic force generated by the duct (T_{duct})

$$T_{\text{total}} = T_{\text{core}} + T_{\text{duct}} \quad (1)$$

Based on the integral momentum theory [15, 16, 17], the net thrust can be expressed as:

$$T_{\text{total}} = \dot{m} (v_e - v_0) + (P_e - P_0)A_e \quad (2)$$

and

$$\tau = T_{\text{total}} l \quad (3)$$

where, T_{total} is the EDPF thrust (N), $\dot{m}$ denotes the mass flow rate (kg/s), and v_e, v_i are velocity (m/s) at the EDPF air exhaust and air intake, correspondingly, P_e and P_0 represent the exit static pressure and freestream static pressure and A_e is the exit area. τ denotes the fan torque (Nm), l symbolizes the lever arm (m).

Flow rate represents the quantity of fluid (air or water) passing through the fan per unit of time. The mass flow rate can be expressed as:

$$\dot{m} = \rho v A \quad (4)$$

where, ρ is fluid density with standard air density being approximately 1.225 kg/m^3 , v is average flow velocity across the cross-section (m/s), A is cross-sectional area of the duct/flow path (m^2).

Thus, the gross thrust and drag force are obtained by summing the surface force components acting opposite to and along the positive direction of the rotational z -axis, respectively (see Fig. 4). Accordingly, the gross thrust, drag, and net thrust are all calculated using the ANSYS CFD-Post module.

Electric power is the total electrical energy consumed by the system from the power source [18] as:

$$P_{\text{elec}} = VI \quad (5)$$

where, V is the input voltage and I is the current.

In a thruster system, mechanical power is the power at the motor shaft as:

$$P_{\text{mech}} = \tau \omega \quad (6)$$

where, τ is the motor torque (Nm) and ω is angular velocity of the shaft (rad/s). If measured in Revolutions Per Minute (RPM), the conversion is:

$$\omega = \frac{2\pi \text{RPM}}{60} \quad (7)$$

Overall system efficiency compares the final thrust power produced to the initial electrical power consumed:

$$\eta = (P_{\text{mech}}/P_{\text{elec}})100\% \quad (8)$$

3. NUMERICAL RESULTS AND DISCUSSIONS

3.1. Grid-dependency test and validation

The mesh independence study is carried out in two steps:

Step 1: Evaluating the mesh independence results

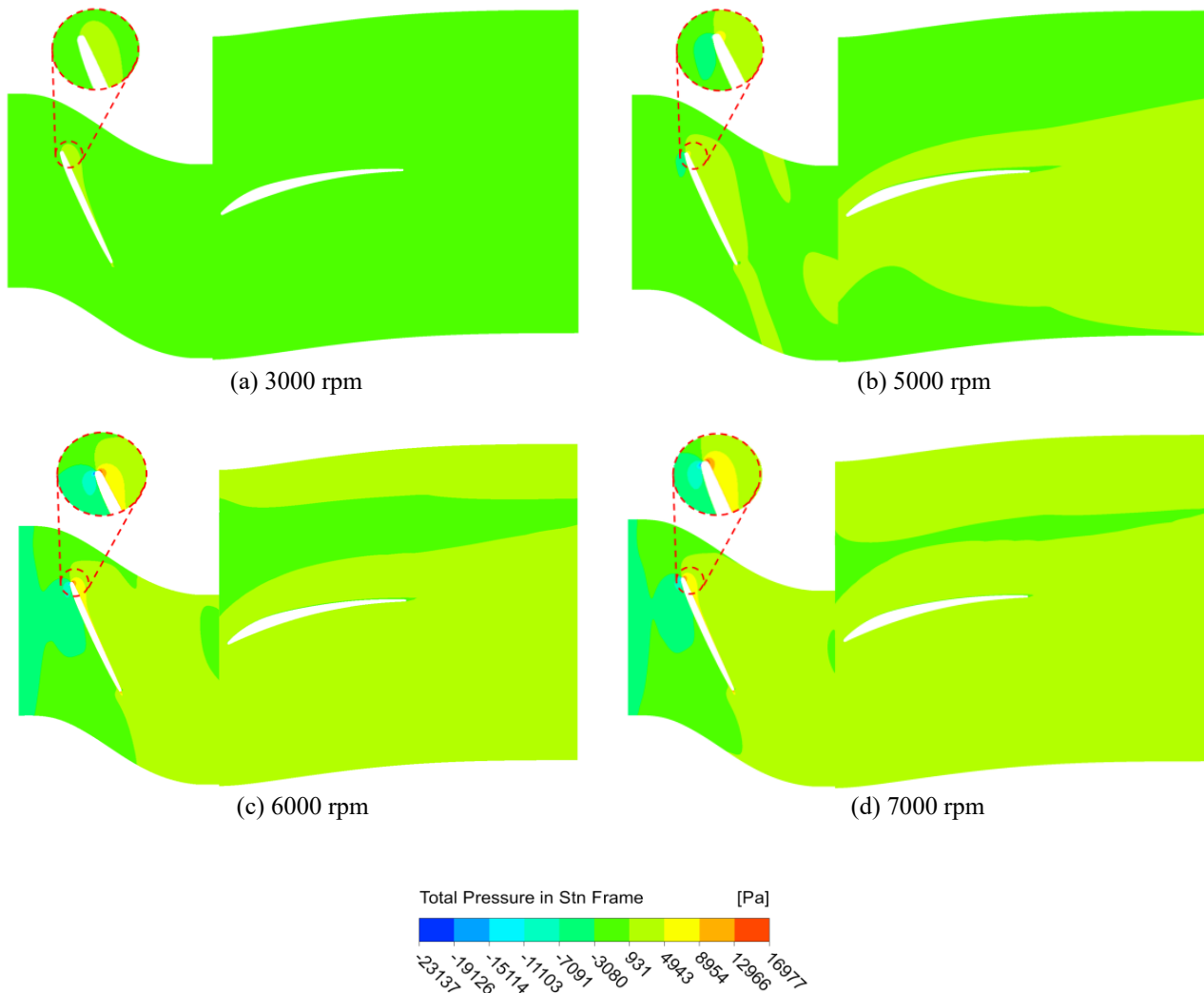


Fig 9. Blade-to-blade total pressure contours at the 95% span location.

using 4 periodic meshes. These meshes consist of varying element counts and are designated as Mesh#1 through Mesh#4.

Figure 5 shows the grid independence test results of EDPF at a rotation speed of 6000 rpm. The results show that the thrust of EDPF differs significantly from Mesh#1, Mesh#2, and Mesh#3; however, between Mesh#3 and Mesh#4, this difference is negligible and very difficult to notice (less than 0.2%). The same applies to the torque of EDPF. Therefore, Mesh#3 has the advantage of high accuracy while optimizing computation time with a significantly smaller number of nodes compared to Mesh#4.

Step 2: Analyzing the mesh independence results with experimental data: 9 battery packs, a full mesh based on Mesh#3 (Full, Mesh#3), and a periodic mesh based on Mesh#3 (Per., Mesh#3). The experimental data for the EDPF thruster have been previously published [19].

Figure 6 compares thrust and torque for three different configurations. Both numerical models predicted slightly higher values than the physical experiment testing. The experimental results recorded approximately 421 N for IJMST – Vol. 2, no. 3, pp. 46–57, 2026

thrust and 25.9 Nm for torque. In contrast, the simulations calculated approximately 424 N and 26.2–26.3 Nm. Simulations may have ignored mechanical losses or simplified flow physics. However, the deviation remains within acceptable engineering limits. The periodic Mesh#3 matched the results almost perfectly but required significantly fewer computational nodes when compared with full Mesh#3. Therefore, this study uses the periodic Mesh#3 for future simulation tasks. This independence test helps to reduce computational time and cost while maintaining robust predictive accuracy.

It is worth noting that while the global performance metrics (such as total thrust and torque) have been directly validated against experimental measurements, localized flow features—including turbulence kinetic energy (TKE) and entropy generation profiles—were evaluated purely through numerical prediction. Due to the high rotational velocities (up to 7,000 rpm) and the extremely tight blade tip clearance of 2 mm, embedding physical probes or employing non-intrusive optical diagnostics such as Particle Image Velocimetry (PIV) or hot-wire anemometry inside the duct was physically

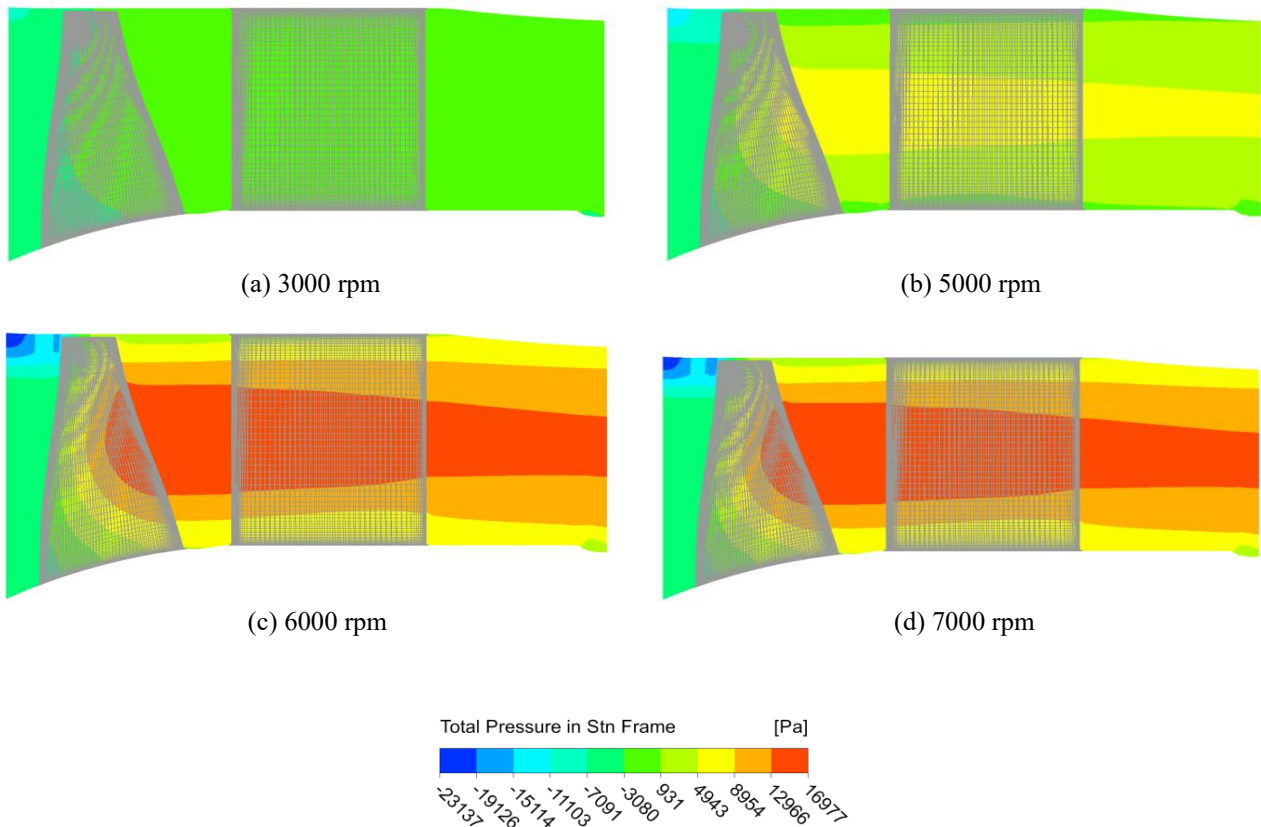


Fig. 10. EDPF meridional view of total pressure contours.

restricted without severely disrupting the internal flow dynamics.

To mitigate this experimental limitation and ensure the physical fidelity of the qualitative flow structures discussed in subsequent sections, the grid-dependency test (Mesh#1 to Mesh#4) was strictly extended to monitor localized variables. Crucially, spatial convergence was verified not only for macroscopic forces but also for local flow profiles across the blade span and tip regions. The localized distributions of TKE and static pressure exhibited strict grid independence and asymptotic behavior when progressing from Mesh#3 to Mesh#4. This rigorous numerical verification, combined with the marginal error (0.86%) in global thrust validation, provides strong secondary assurance regarding the reliability and qualitative fidelity of the internal flow phenomena analyzed herein

3.2. EDPF aerodynamic characteristics across operating speed ranges

(a) *Numerical and experimental comparison:* Figure 7 shows the comparison results between simulation and experiment regarding the thrust value of the EDPF. At different rotational speeds from 2000 rpm to 7000 rpm, the difference in thrust values is not significant. At a rotational speed of 6000 rpm, the difference is only 0.86% (424.6 N in simulation and 421 N in experiment). From

this comparison, it can be seen that the EDPF model in simulation has high accuracy and reliability and can be used for further in-depth simulations of the EDPF later.

(b) *Numerical qualitative result:* In this section, we analyze the static velocity, total pressure, turbulence kinetic energy, and static entropy distributions at the blade tip to understand the local aerodynamic behavior. This investigation focuses on three specific operating speeds: the low rotation speed at 3000 rpm, the stable rotation speed at 5000 rpm condition, and the highest rotation speed at 7000 rpm.

Figure 8 illustrates the static velocity contours at the 95% span location of EDPF at the rotor and stator regions. At low rotation speed, 3000 rpm, the flow field contains low kinetic energy, and the fluid speed is low within the low-velocity regions. The flow momentum increases noticeably at a stable rotation speed of 5000 rpm. Fluid accelerates over the rotor suction surface, and these moderate-velocity areas form a clear wake near the downstream stator. As the speed scales up to 6,000 rpm, this aerodynamic acceleration intensifies; the high-velocity zones expand considerably along the blade suction side, and the localized wake structures stretching toward the stator row become more elongated and turbulent. At the highest rotation speed, 7000 rpm, the velocity magnitudes are beyond 90 m/s. The suction side develops broad high-velocity zones alongside steep gradients. These features point to heavy aerodynamic

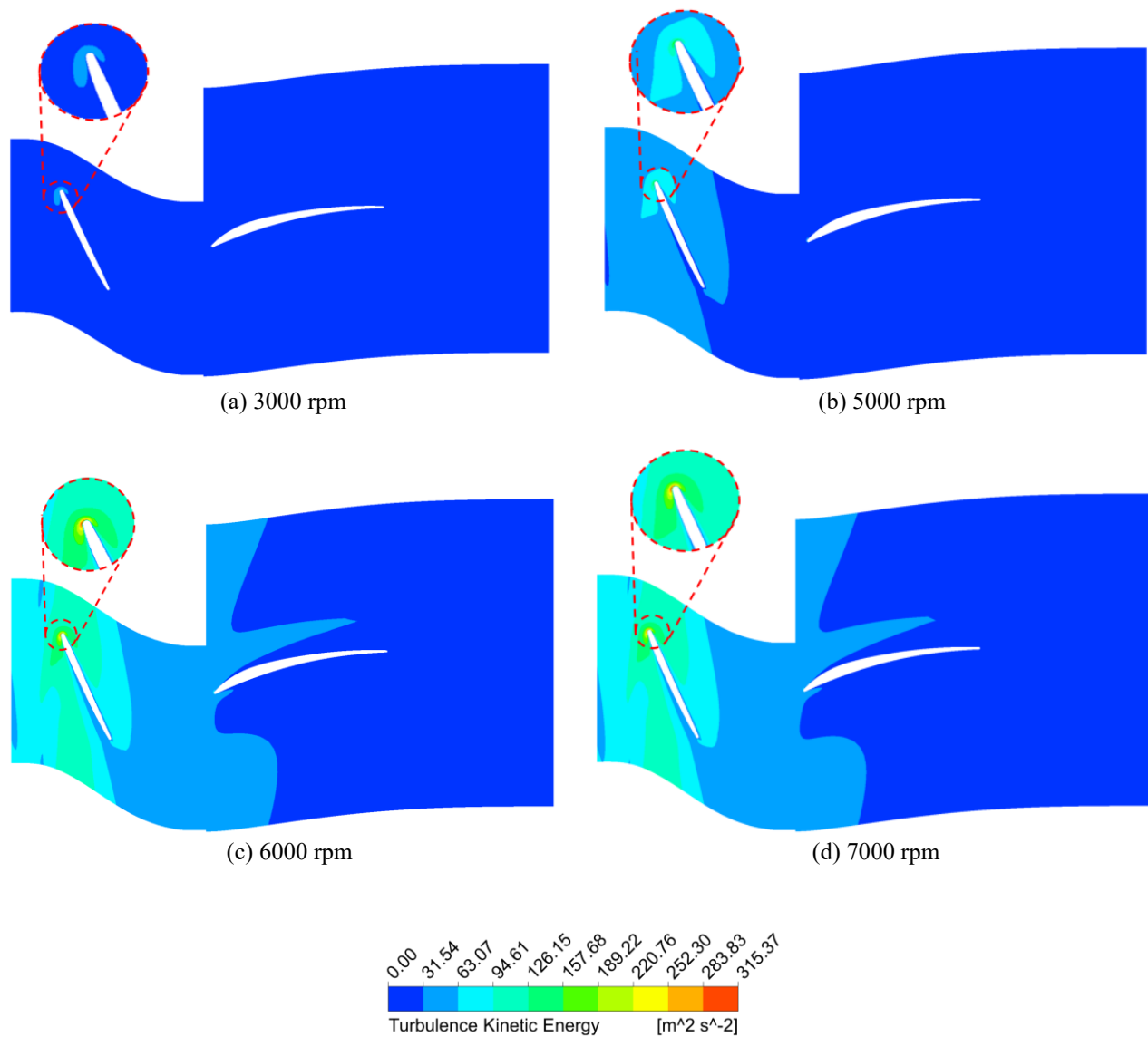


Fig. 11. Blade-to-blade turbulence kinetic energy at the 95% span location.

loading and strong tip-leakage flow. They also reveal an intense interaction between the rotor and stator. Ultimately, an increase in rotational speed changes the uniform low-energy flow into a highly complex aerodynamic environment.

Figure 9 illustrates the total pressure contours at the 95 % span location at the rotor and stator blades. At low rotation speed, 3000 rpm, the pressure field is almost uniform, and the impeller experiences very little pressure difference. At the stable rotation speed of 5000 rpm, the areas of higher pressure begin to appear around the rotating and stationary blades. The high-pressure area appears on the pressure side, while the low-pressure area appears on the suction side. As the rotational speed scales up to 6,000 rpm, this pressure field asymmetry intensifies noticeably; the localized high-pressure zones expand further along the pressure surface, while a more defined low-pressure core forms near the leading edge of

the suction surface. This distribution becomes even more noticeable at the highest speed of 7000 rpm. A strong low-pressure area appears with low-magnitude contours at the leading edge of the suction face. Meanwhile, the pressure-bearing face creates very high pressure represented by elevated pressure contours. This large pressure gap directly causes rapid growth of thrust and torque. Furthermore, the strong vortex pressure gradient at 7000 rpm highlights a much deeper interaction between the rotor and stator blades.

Figure 10 illustrates the total pressure contours at the meridional view at rotational speeds of 3,000 rpm, 5,000 rpm, 6,000 rpm, and 7,000 rpm. At a low speed of 3,000 rpm, the total pressure field displays nearly uniform distribution with minimal pressure values throughout the domains. At 5,000 rpm, regions of elevated total pressure begin to manifest prominently, as indicated by the widening contour bands of moderate pressure. When the

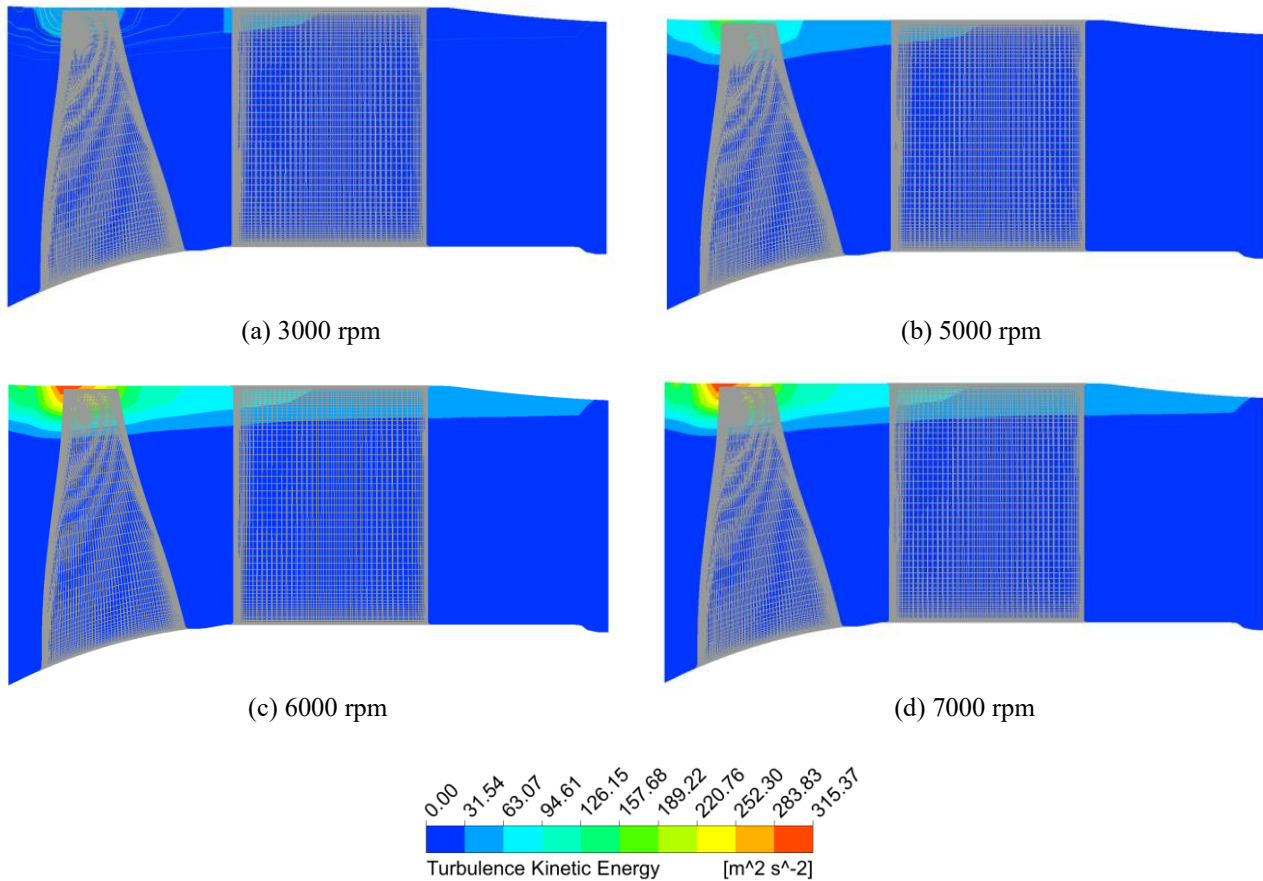


Fig. 12. EDPF meridional view of turbulence kinetic energy.

rotational speed increases to 6,000 rpm, the aerodynamic compression intensifies further, causing the higher pressure zones to expand extensively through the passage with increasing magnitudes. Ultimately, at the maximum speed of 7,000 rpm, regions of peak maximum pressure—represented by the peak-magnitude regions—dominate the core fluid domains. This progressive evolution demonstrates that a faster rotational speed substantially enhances the global static pressure capability and introduces a significantly higher aerodynamic loading across the EDPF stage.

Figure 11 illustrates the turbulence kinetic energy (TKE) contours at the blade-to-blade 95% span location across various rotational speeds. At a low speed of 3,000 rpm, the entire flow field exhibits minimal and negligible TKE values, indicating a highly stable flow environment. As the rotational speed increases to 5,000 rpm, localized regions with elevated TKE levels begin to emerge near the blade leading edge and the blade tip clearance. When the speed scales up further to 6,000 rpm, this turbulent fluctuation intensifies; the high-TKE zone at the leading edge expands noticeably (as indicated by the amplified contour levels of turbulent energy), and the wake deficit trailing down the suction side begins to broaden. This trend reaches its peak at the maximum rotational speed of 7,000 rpm, where the TKE values at the leading edge

region spike dramatically, showing a stark contrast to the surrounding flow fields. Ultimately, faster rotational speeds significantly augment turbulent dissipation in the near-tip region. An intense interaction between the mainstream flow and the tip leakage vortex actively drives this severe flow unsteadiness, magnifying the overall aerodynamic losses.

Figure 12 displays the meridional contours and reveals a direct link between rotational speed and turbulence kinetic energy (TKE), which occurs primarily near the shroud. At a low speed of 3,000 rpm, the flow field stays highly stable and contains minimal TKE values throughout the domain. The flow dynamics shift noticeably at 5,000 rpm, where a stronger tip clearance leakage vortex creates a moderate turbulence zone of localized high fluctuations that stretches axially along the casing. As the rotational speed scales up further to 6,000 rpm, this localized aerodynamic unsteadiness intensifies, causing the high-turbulence region to propagate deeper into the passage and expand toward the stator entrance, with progressively expanding high-magnitude bands. The TKE increases drastically at the maximum speed of 7,000 rpm, where an intense turbulence core forms right next to the shroud, reaching its highest aerodynamic dissipation levels. Overall, while the hub and mid-span sections remain relatively

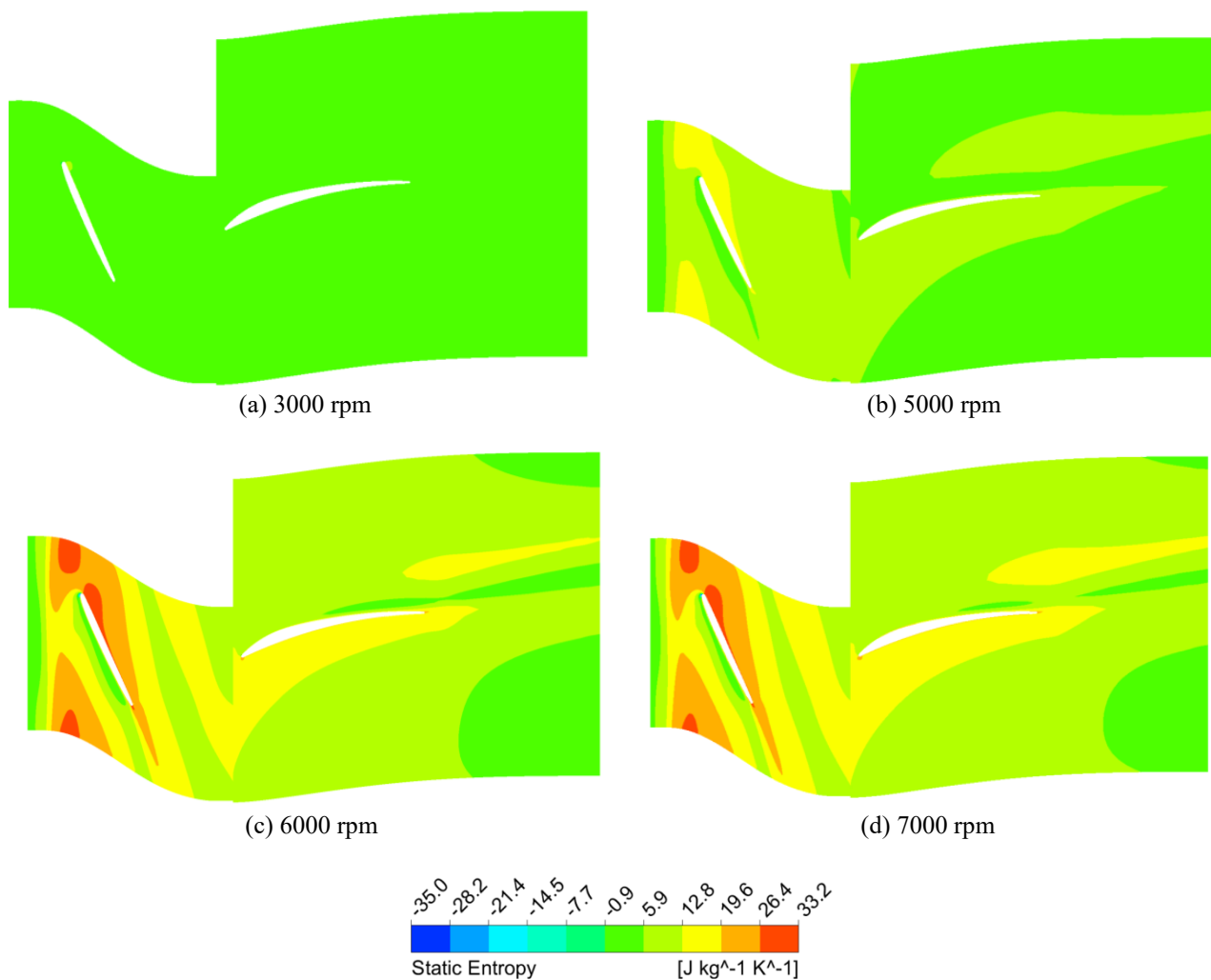


Fig. 13. Blade-to-blade static entropy at the 95% span.

undisturbed, the casing area suffers from severe energy dissipation. A faster rotor speed clearly worsens the aerodynamic unsteadiness, driven by fierce interactions between the mainstream flow and the tip leakage structures.

The blade-to-blade static entropy contours at the 95% span location across various rotational speeds are illustrated in the figures. Static entropy acts as a direct thermodynamic indicator of aerodynamic losses, highlighting regions of energy dissipation driven by tip leakage vortices, boundary layer separation, and wake mixing. At a low speed of 3,000 rpm, the entire flow domain displays a highly uniform, low-magnitude contour. This corresponds to minimal static entropy values fluctuating between -0.9 and $5.9 \text{ J kg}^{-1} \text{ K}^{-1}$, representing a highly stable flow environment with negligible dissipation. As the rotational speed increases to 5,000 rpm, moderate-loss regions—indicated by contours around $12.8 \text{ J kg}^{-1} \text{ K}^{-1}$ —begin to emerge near the rotor leading edge and extend along the suction side. At 6,000 rpm, the aerodynamic losses intensify dramatically. Severe high-entropy zones, represented by

high-magnitude contours peaking between 19.6 and $33.2 \text{ J kg}^{-1} \text{ K}^{-1}$, cluster densely at the rotor leading edge and cover a substantial portion of its suction surface. This loss pattern reaches its maximum expansion at 7,000 rpm, where the peak entropy core maintains its intense concentration at the rotor tip, while the elevated deficit wake propagates deeper downstream into the stator passage. Ultimately, this progressive trend confirms that at elevated speeds (especially at 6,000 rpm and above), tip-clearance leakage flows and the resulting rotor-stator aerodynamic interactions serve as the primary drivers of thermodynamic irreversibility and stage performance degradation.

These findings heavily influence the overall EDPF analysis. They establish a solid baseline for comparison with previous axial compressor studies [20–22]. This broad evaluation clearly details the unique traits of both systems.

(c) *Fan performance curves analysis:* This section shows the results of analyzing EDPF parameters by combining simulation, theoretical calculations, and experiments testing. Torque, mass flow, and pressure rise

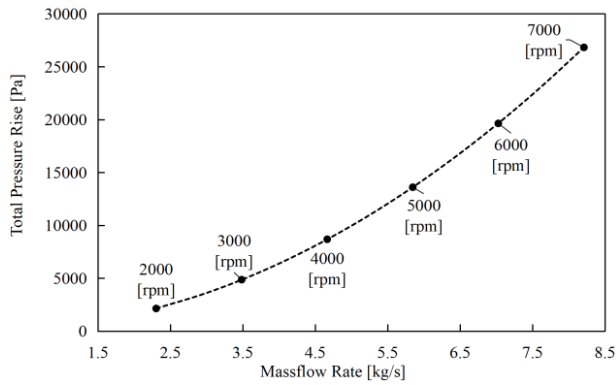


Fig. 14. Total pressure rise versus mass flow rate at various rotational speeds.

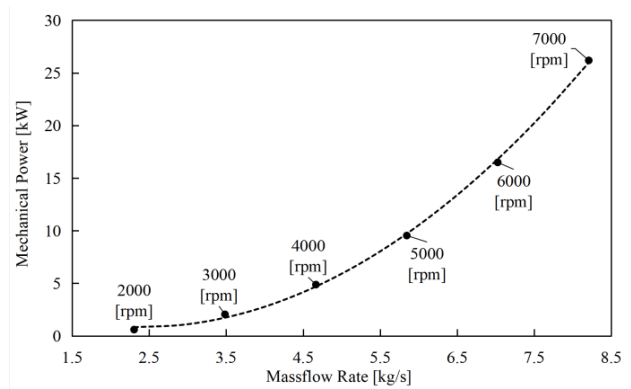


Fig. 16. Mechanical power versus mass flow rate at various rotational speeds.

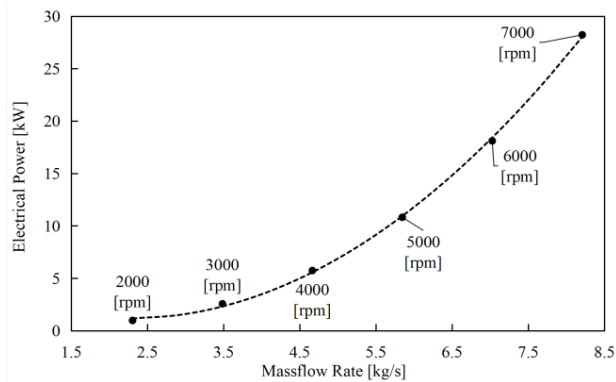


Fig. 15. Electrical Power versus massflow rate at various rotational speeds.

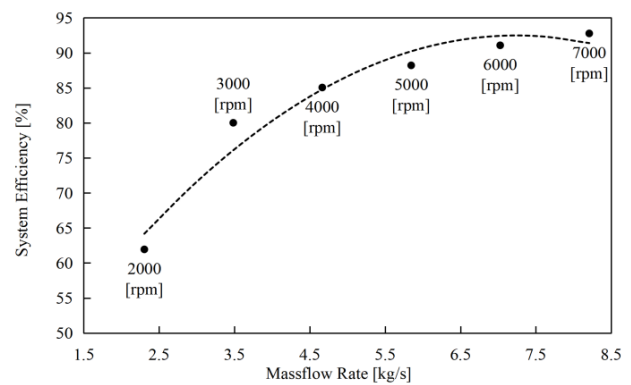


Fig. 17. Efficiency versus mass flow rate at various rotational speeds.

data were obtained from simulations. Parameters such as rotational speed, mechanical power, electrical energy, and system efficiency were derived from the calculation steps. Voltage, however, was measured directly through experiments. Details of all these data can be found in Table 1.

Figure 14 shows the total pressure rise versus mass flow rate at various rotational speeds. At the lowest rotational speed of the EDPF, it produces a mass flow rate of 2.3 kg/s with low pressure (2000 Pa). But at the high rotational speed of the EDPF (7000 rpm), the fan's power increases dramatically. It can push nearly four times the volume of air (over 8.0 kg/s) and generate thirteen times the thrust (27,000 Pa).

Figure 15 clearly highlights the trade-off associated with the increased power output: operating the fan at higher speeds results in a steep rise in electrical energy consumption. When running slowly at 2000 rpm, the fan is very economical, consuming only 1 kW of electricity. However, to reach maximum power at 7000 rpm, the electricity consumption not just increases slightly but jumps to 28 kW. In other words, push the EDPF to its fastest speed to get four times the airflow but pay 28 times the electric power. Figure 16 shows mechanical power versus mass flow rate. The results indicate that both parameters increase rotation speed. At 2000 rpm, EDPF produces a mass flow rate of 2.3 kg/s and consumes less

than 1 kW of mechanical power; at 7000 rpm, EDPF consumes up to 26 kW of power to produce a mass flow rate of 8.2 kg/s.

Figure 17 shows efficiency versus mass flow rate at various rotational speeds. As tabulated in Table 1, the overall system efficiency exhibits a continuous upward trend with the increase of rotational speed, successfully peaking at 93% at the maximum operating speed of 7,000 rpm. Although the qualitative flow diagnostics (such as TKE distributions in Fig. 12 and the static entropy contours in Fig. 13) indicate that localized aerodynamic losses and tip clearance leakage flows intensify at 7,000 rpm, the tremendous rise in total pressure delivery and effective aerodynamic power output at this high rotational speed successfully counteracts these localized irreversibilities. The robust design of the EDPF blade profile maintains highly favorable pressure-bearing characteristics across the span, ensuring that the electric-to-pneumatic energy conversion remains highly efficient even under high-load conditions. Consequently, 7,000 rpm represents the optimal operating point for maximum aerodynamic performance, delivering peak system efficiency of 93%.

4. CONCLUSION

This study evaluates EDPF at different speeds and

Table 1. Data from numerical simulations, theoretical calculation and experimental testing.

Parameters	Value					
Rotational velocity [rpm]	2000	3000	4000	5000	6000	7000
Total pressure rise [Pa] (Sim.)	2136	4863	8681	13601	19637	26824
Torque [N.m] (Sim.)	2.882	6.535	11.653	18.224	26.251	35.728
Thrust [N] (Sim.)	45.775	104.556	187.319	293.595	424.615	579.299
Thrust [N] (Exp.)	62	110	191	301	421	553
Mechanical power [kW] (Exp.)	0.604	2.053	4.881	9.542	16.494	26.190
Voltage [V] (Exp.)	117.90	117.20	115.70	113.63	110.35	106.20
Current [A] (Exp.)	8.27	21.89	49.60	95.20	164.12	265.82
Electric power [kW] (Exp.)	0.975	2.566	5.739	10.817	18.110	28.230
System efficiency [%] (Exp.)	62	80	85	88	91	93

achieves several important results by comparing simulation and experimental results:

(i) The established 3D RANS numerical framework utilizing the $k-\omega$ SST model demonstrates exceptional predictive accuracy, with the computed thrust deviating from experimental measurements by a mere 0.86% at the high-efficiency operating point (6000 rpm).

(ii) The overall system performance and energy conversion efficiency exhibit a continuous upward trend with the increase of rotational speed, successfully peaking at 93% at the maximum operating speed of 7000 rpm. Although qualitative flow diagnostics indicate that localized aerodynamic losses, flow separations, and tip clearance leakage flows (characterized by high static entropy and turbulence kinetic energy) intensify at 7,000 rpm, the dramatic rise in total pressure delivery and aerodynamic power output at this high rotational speed successfully counteracts these localized thermodynamic irreversibility.

(iii) While the absolute maximum aerodynamic performance is achieved at 7,000 rpm with 93% efficiency, a rotational speed of 5,000 rpm represents the optimal, most balanced operating point for practical maritime aerial missions, yielding a highly favorable trade-off between sufficient aerodynamic thrust, structural stability, and power economy.

The verified fidelity of this computational method provides a reliable simulation for future geometric optimization and acoustic minimization of electric ducted propulsion fans.

ACKNOWLEDGEMENT

This research is funded by Vietnam National Foundation for Science and Technology Development (NAFOSTED) under grant number NCUD.02–2023.20. The authors also gratefully acknowledge the collaboration and support of Viettel Aerospace Institute and Hanoi University of Science and Technology throughout the course of this research.

REFERENCES

- [1] T. Zhang and G. N. Barakos, "Review on ducted fans for compound rotorcraft," *Aeronautical Journal*, vol. 124, no. 1277, pp. 941–974, 2022.
- [2] D. Black and C. Rohrbach, "Shrouded propellers – A comprehensive performance study," *Proc. of 5th Annual Meeting and Technical Display*, Philadelphia, PA, USA, pp. 21–24, 1968.
- [3] H. Xu and Z. Ye, "Numerical simulation and comparison of aerodynamic characteristics between ducted and isolated propellers," *Journal of Aerospace Power*, vol. 26, pp. 2820–2825, 2011.
- [4] A. I. Abrego and R. W. Bulaga, "Performance study of a ducted fan system," *Proc. of AHS Aerodynamics, Aeroacoustic, Test and Evaluation Technical Specialist Meeting*, American Helicopter Soc., Alexandria, VA, 2002.
- [5] P. Martin and C. Tung, "Performance and flowfield measurements on a 10-inch ducted rotor VTOL UAV," *Proc. of 60th Annual Forum of the American Helicopter Society*, American Helicopter Soc., Alexandria, VA, 2004.
- [6] J. Fleming et al., "Improved control of ducted fan VTOL UAVs in crosswind turbulence," *Proc. of AHS 4th Decennial Specialist's Conference on Aeromechanics*, 2004.
- [7] D. Urban et al., "Effect of electric ducted fans structural arrangement on their performance characteristics," *Applied Sciences*, vol. 13, no. 5, 2023.
- [8] S. Deng, S. Wang, and Z. Zhang, "Numerical prediction of unsteady aerodynamics of a ducted fan unmanned aerial vehicle in hovering," *Aerospace*, vol. 9, no. 6, pp. 318, 2022.
- [9] Y. Jin et al., "Numerical investigation on unsteady characteristics of ducted fans in ground effect," *Aerospace Science and Technology*, 2023.
- [10] J. Guo and Z. Zhou, "An efficient blade design method of a ducted fan coupled with the CFD modification," *Aerospace*, vol. 9, no. 5, pp. 241, 2022.
- [11] J. Guo and Z. Zhou, "Multi-objective design of a distributed ducted fan system," *Aerospace*, vol. 9, no. 3, p. 165, 2022.
- [12] V. S. Dinh et al., "Flow analysis through center body of ducted fan to electric motor cooling," *Journal of Physics:*

- Conference Series, vol. 2707, Institute of Physics, 2024.
- [13] Ansys, Ansys CFX–solver theory guide, Release 2025 R1, 2025.
- [14] Cadence Design Systems, Inc., 2025, “Cadence fidelity CFD user guide,” Version 2025.1.
- [15] S. Farokhi, “Aircraft propulsion: Toward net–zero aviation,” 4th. Ed., Chapter 2, Wiley, 2025.
- [16] S. Gudmundsson, “General Aviation Aircraft Design: Applied Methods and Procedures,” Elsevier, Second Edition, 2014.
- [17] T. S. Vu et al., “Aerodynamic characteristics of ducted propulsion fan using secondary air intake,” *Eng.*, vol. 7, no. 6, pp. 295, 2026.
- [18] S. J. Chapman, *Electric Machinery fundamentals*, 5th ed., Mc Graw Hill, 2012.
- [19] H. Q. Chu et al., “Static aero–propulsion experiment of an electric ducted fan,” *Aerospace*, vol. 12, no. 6, pp. 509, 2025.
- [20] H. Q. Chu et al., “Aerodynamic and structural performances of a single–stage transonic axial compressor with blade fillet radius,” *International Journal of Intelligent Unmanned Systems*, vol. 11, no. 3, pp. 407–424, 2023.
- [21] A. T. Nguyen et al., “Aerodynamic performances of a single–stage transonic axial compressor using an inclined casing groove,” *Australian Journal of Mechanical Engineering*, vol. 22, no. 1, pp. 179–188, 2024.
- [22] A. V. C. Tan et al., “Effect of bleeding dual discharge port on aerodynamic performance of a single–stage transonic axial compressor,” *International Journal of Fluid Machinery and Systems*, vol. 17, no. 2, pp. 88–101, 2024.