Design and analysis of component for non-rotor unmanned aerial vehicle

Y R Shiau¹, J Fang² and Y L Chen³*

¹Department of Industrial Engineering and Systems Management, Feng Chia University, Taichung 407, Taiwan
²Department of Aerospace and Systems Engineering, Feng Chia University, Taichung 407, Taiwan
³Ph.D. Program of Mechanical and Aeronautical Engineering, Feng Chia University, Taichung 407, Taiwan
*Email: p0460369@mail.fcu.edu.tw

Abstract. Unmanned aerial vehicles (UAVs) are widely applied in various industries and fields. Because they use a variety of rotor types, from single rotors to multi-rotors, UAVs offer a wide array of functions. However, continuously rotating rotors can be dangerous if they accidentally encounter foreign objects or bare hands. Therefore, non-rotor UAVs are the focus of discussion and modification in the present study. Non-rotor UAVs do not contain visible rotor mechanical components. Compared with the development of popularized multirotor UAVs, that of non-rotor UAVs is challenging in terms of structure and flight control. To address this challenge, wind tunnel structure models were developed in this study for different levels of aerodynamic force, and computer-aided engineering was used to conduct structural and flow field analyses to determine the key elements that reinforce non-rotor UAV structures. The research results revealed that the thickness and fillet design at the joint of the non-rotor UAV aerodynamic wind tunnel system were crucial factors influencing the system’s performance. Motor rotors can be embedded inside support structures to reduce airflow turbulence in the wind tunnel.

1. Introduction

Unmanned aerial vehicles (UAVs) are widely applied in various industries and fields. UAVs employ various rotors designs, including single-rotor to multirotor designs, enabling them to achieve a wide variety of functions. However, if rotors under continuous rotation accidentally encounter foreign objects or bare hands, then they can pose a serious hazard. Therefore, the present study focused on non-rotor UAVs and their potential modifications.

Non-rotor UAVs do not contain visible rotor mechanical components. Compared with popularized multirotor UAVs, non-rotor UAVs have certain disadvantages in terms of structure and flight control. To address these shortcomings, wind tunnel structure models based on aerodynamic force were developed in this study, and computer-aided engineering (CAE) was employed to conduct structural and flow field analyses.

In recent years, small turbine engines, particularly microturbine engines, have attracted considerable attention for their applications in UAVs or radio-controlled aircraft because of their extremely high thrust-to-weight ratio [1]. Fan-wing UAVs employ cross-flow turbines, which are based on a new aerodynamic concept. The fan wings can concurrently produce lift and thrust [2]. Flying structures with different thicknesses form different lift and drag [3]. The use of simple-structure devices and wind tunnel systems is conducive to stabilizing the flow field and enhancing the vertical movement and thrust effectiveness [4].
When the fluid velocity increases, the relative instantaneous pressure decreases according to Bernoulli’s principle, which is widely applied in aircraft engineering. Aircraft push air downward to generate lift. When the lift is greater than the weight of the aircraft, the aircraft moves upward. However, well-developed structures are required to maximize the benefits of fluid velocity. Thus, this study endeavored to determine the key elements for improving the aerodynamic wind tunnel structure and performance of non-rotor UAV structures.

2. Literature review

2.1. Impeller

Impellers are the key component for generating driving force and are widely applied in the aeronautical industry. Impeller models are subject to strict requirements in terms of precision and strength [5]. For enhanced fluid dynamics performance, an impeller’s geometric shape can be altered to reduce its weight, and its materials can be carefully selected according to their density to ensure the maintenance of adequate performance under high-speed stress and centrifugal force; this optimizes the impeller’s topology to enhance its strength efficiency and minimizes displacement [6]. The smaller the inlet angle of the impeller, the vortex formed by it will expand from the low pressure to outside of the impeller, which is conducive to fluid recirculation [7].

The structure of the impeller affects the performance of aerodynamics, and its vortex motion flows into the cavity through the impeller to form a vortex [8]. The air multiplier manufactured by Dyson, United Kingdom uses a turbine impeller to drive air flow and eject a jet through a narrow annular cavity (Figure 1). The resulting convergence and suction effectiveness are 10–15 times those of turbines. Over time, the flow parameter distribution in the flow field of an annular slot jet outlet reveals a higher mean speed at the annular upper part of the jet. The vertical pressure field exhibits axial asymmetry, whereas the horizontal pressure field exhibits axial symmetry. The near-field pressure is lower than the far-field pressure [9].

![Figure 1. Sectional view of an air multiplier [9].](image)

This study explored the relationships between air flow at the outlet and that generated by an impeller under the same wing length conditions but with different geometric parameters, including out-wind frame cross-section height, outlet angle of air flow relative to the fan axis, outlet slot thickness, fluid diameter, and fluid aspect ratio. To evaluate aerodynamic performance, unsteady conservation of mass and momentum equations were solved [10].

2.2. CAE analysis

CAE refers to the computer-aided design, manufacturing, and assembly of manufacturing systems used in production and manufacturing industries. CAE is widely applied in the fields of aeronautical and mechanical engineering to solve problems regarding the simulation and analysis of structural strength, buckling load stability, dynamic response, fluid properties, and mechanical properties (e.g., elasticity),
as well as to optimize structural performance. Consequently, these systems can be ensured to possess adequate physical properties and efficacy for use in research and development [11].

Optimization design in mechanical engineering involves the determination of the maximum or minimum value of target functions based on geometric variables from design parameters, such as the required length, thickness, radius, and number of nodes, within given constraints. When the action force and other dynamic principles are also considered, the optimal distribution of stress or deformation can be determined [12-13]. In the fields of aerospace and aeronautical, automobile, marine industries etc., light weight, strong resistance to thermal and mechanical loads, corrosion and wear resistance materials are commonly used to strengthen the structure [14]. Relevant factors should be considered in the optimization process, such as mass, strain failure, twist, displacement [15]. Ansys can be used to analyze the stress of an applied force and weight as well as structural performance characteristics such as fatigue to simulate movements. The total deformation volume, stress ratio, and strain can be used to analyze the fatigue limit to achieve design optimization [11].

To meet both aerodynamic and structural requirements, finite element analysis and parameter design were used to estimate the optimal features for achieving high aerodynamic performance and stability with consideration of a UAV’s dimensions, material, and structural characteristics. Accordingly, the weight of the UAV structure was reduced, and its aerodynamic performance was optimized [16]. Computational fluid dynamics (CFD), which combines theoretical and experimental fluid dynamics, was employed to analyze fluids and determine the different factors affecting engine jets [17].

3. Research methods

This study involved four steps. Step 1 was the aircraft design stage, in which Solidworks was used to construct geometric models, including for the main body, impeller structure, and numerical analysis of materials. Step 2 was the structural analysis stage, which involved establishing meshes, materials, loads, and constraints to analyze stress and strain relevance. Step 3 was the flow field analysis stage, which involved establishing the air domain, constructing meshes, setting initial conditions, preprocessing boundary conditions, and performing Ansys simulations. Step 4 was the Fluid-Structure Interaction analysis based on impeller. This study mainly analyzed wind tunnel structures in aerodynamics; thus, device components were used as the main analysis targets. The research methods and steps are displayed in Figure 2.

![Figure 2. Research methods and steps.](image-url)
4. Research design and analysis

4.1. Aircraft design

This study used a non-rotor UAV aerodynamic model to construct three-dimensional models. The streamlined and abrupt narrow design of the wind tunnel included four sets of components, each including an impeller, impeller box, wind tunnel, non-rotor outlet, and motor rotor. The components were assembled with an upper plate and a lower plate to form a UAV. The material selected was aluminum alloy, which is broadly applied in the transport vehicle and aeronautical engineering industries. Table 1 lists the analytical data, and Figure 3–5 present illustrations of the model.

Table 1. Material analysis (Aluminum alloy).

| Component  | UAV Model (mm) |
|------------|----------------|
| Length X   | 182.8 mm       |
| Length Y   | 58.912 mm      |
| Length Z   | 77 mm          |
| Volume     | 52569 mm³      |
| Mass       | 0.14562 kg     |

Figure 3. Impeller model.

Figure 4. Component model.

Figure 5. Non-rotor UAV model.
4.2. Structural analysis

4.2.1. Setting the structure simulation conditions. The material used in this study was aluminum alloy, which is the default material in the Ansys material library. Its characteristics are listed in Table 2.

| Item                  | Value       |
|-----------------------|-------------|
| Density               | 2770 kg/m³  |
| Young’s Modulus       | 71000 MPa   |
| Poisson’s Ratio       | 0.33        |
| Tensile Yield Strength| 280 MPa     |
| Tensile Ultimate Strength | 310 MPa |

Table 2. Material analysis (Aluminum alloy).

To facilitate simulation and analysis, the mesh allocation algorithm was set to “Mechanical” for the physical preference, “Program Controlled” for the element order, “High” for the smoothness level, “Yes” for options to activate mesh defeaturing and capture curvature, and “No” for the option to capture proximity (Figure 6). The analysis results revealed that the number of nodes in the structural domain was 646,360 and the number of elements was 352,971.

Constraints involved the use of the outlet bottom end as a fixed support to restrict the degree of freedom of Ux, Uy, and Uz. With reference to the coordinates, the rotation speed was set at 4,000 rpm to satisfy the load demand for clockwise rotation of the impeller, as shown in Figure 7.

4.2.2 Static analysis. Static analysis was performed with the aforementioned parameters to analyze the deformation of the static structure under the application of stress and obtain the static and fatigue simulation results. Static analysis helped to determine the position most likely to generate material fatigue. The airflow outlet load applied to the impeller with a rotation speed of 4,000 rpm was 0.14562 kg. The total deformation under this load is illustrated. The structural displacement resulting from the application of the maximum stress load to the impeller area was 3.173.5 mm. The maximum equivalent elastic strain, which was 0.006194 mm/mm, and the maximum equivalent stress, which was 368.34 MPa, were both observed at the junction of the wind tunnel and non-rotor. Fatigue analysis was conducted to analyze the safety factor of the aluminum alloy, the minimum value of which was determined to be 0.76016 (Figure 8).
4.3. Flow field analysis

4.3.1. Setting the flow field simulation condition. To evaluate changes in the fluids during the flow process, the air domain of the impeller was first established (Figure 9). The mesh model was developed, and the initial conditions, boundary conditions, and simulation solution parameters were set to help perform the simulation and evaluate the convergence results. Regarding the settings for the basic conditions of the air domain mesh, the physical preference was set to CFD, CFX was selected for the correlation, the smoothness level was set to “Medium,” and “Yes” selected for the options of mesh defeaturing, capture curvature, and capture proximity. The simulation analysis results are shown in Figure 10. The number of nodes was 1,406,308 in the structural domain and 1,017,231 in the flow field domain. The numbers of elements in the structural and flow field domains were 7,689,570 and 5,216,474, respectively.

Regarding the settings for the initial conditions, the operating fluid setting was “Air” at a temperature of 25°C. The reference pressure was set to 1 atm, the buoyancy model was set to “Non-Buoyant,” the model motion setting was “Stationary,” the turbulence model was set to “Shear Stress Transport,” the initial velocity of the inlet and outlet was set to 0 m/s, the relative pressure with respect to the initial pressure was set to 0 atm, and the actual pressure was set to 1 atm. Next, the boundary conditions were set, with the inlet and outlet belonging to the “Opening” type. The relative pressure and actual pressure were set to 0 and 1 atm, respectively (Figure 11). The rotational domain fan wall type was selected as “Wall,” with a wall velocity of 4,000 rpm. The domain contact surfaces were the interface and frozen rotor (Figure 12).
4.3.2 Simulation results. For the simulation, the calculation of laminar flow was set to “High Resolution,” and the turbulence level was set to “First Order”; the maximum and minimum convergence were controlled at 1,000 and 1 time steps, respectively. The convergence residual value was set at 1e-4 with the type as MAX. The simulation residual convergence value after 200 steps approximated 1e-4. Thus, the result was determined to be convergent, as shown in Figure 13.

When the airflow entered the tail section, the motor rotor occupied an excessive proportion of the space in the tunnel and prevented airflow from smoothly entering the tail section, thereby generating turbulence, as shown in Figure 14.

The airflow from the impeller is severely disturbed by the support structure, thereby reducing the velocity of the airflow entering the tail section and causing reflux (Figure 15).

In summary, when the impeller was activated, the inlet velocity was 1.55431 m/s, with a mass flow rate of 0.000762615 kg/s; the outlet velocity was 2.33623 m/s, with a mass flow rate of 0.000762276 kg/s.
4.4. Fluid-structure interaction analysis

4.4.1. Setting CFX mesh. Perform grid encryption of the target impeller (Figure 16). As a result of the analysis, the Nodes of the structure domain is 2876600, and the Elements is 1824147.

4.4.2. Pressure distribution. The results of the flow field analysis (inlet and outlet velocity value) are imported into the structure setting and analyze it to obtain the pressure distribution of impeller structure (Figure 17) and flow field (Figure 18).

5. Results and conclusions

The analysis results indicated the following findings:

- The maximum strain (0.006194 mm/mm) and stress (368.34 MPa) were both observed at the joint of the wind tunnel and non-rotor. We recommend increasing the thickness of the joint or incorporating a fillet design at the bonding area to limit the concentration of stress.
- The minimum safety factor was 0.76016, which is not ideal for UAVs. We recommend modifying the structure or using materials with higher strength.
- The velocity streamline chart reveals that the motor rotor occupied most of the space in the tunnel, thereby causing turbulence when airflow entered the tail section. We suggest that the motor rotor size be reduced or the tunnel be enlarged.
- The airflow generated by Impeller is disturbed by the support structure, thereby reducing the airflow velocity and causing reflux. The support structure should thus be simplified.
- When the pressure in the flow field merges into the structure, the blades are evenly distributed to bear the pressure, and the impact is minimal. The yield strength (280 Mpa) and maximum pressure of the comparative material is (4.482e-12 Mpa), which hardly affects the structure.

In summary, the thickness and fillet design at the joint for the non-rotor UAV aerodynamic wind tunnel system were crucial factors influencing its performance. Embedding motor rotors inside support structures can reduce airflow turbulence in the wind tunnel.
Acknowledgments
This work was supported by the Ministry of Science and Technology of the Republic of China (MOST108-2221-E-035-021)

References
[1] Tang W, Wang L, Gu J and Gu Y 2020 Single Neural Adaptive PID Control for Small UAV Micro-Turbojet Engine. Sensors, 20(2), 345
[2] Siliang D and Zhengfei T 2018 The aerodynamic behavioral study of tandem fan wing configuration. Int. J. Aerosp. Eng. 2018, Article ID 1594570, 1-14
[3] Balakrishnan S, Basri A A, Basri E I & Yazik M 2021 Effects of Different Thickness of UAV Airfoil on Aerodynamics Performance Using CFD Techniques. J. Aeronaut. Astronaut. Aviat. 53(2), 275-282
[4] Tsai C 2014 Development of an Innovative Unmanned Aerial Vehicle with Cross-Flow Fan Thrust System. Master’s thesis of Department of Energy and Refrigerating Air-Conditioning Engineering of National Taipei University of Technology, Taipei City
[5] Zhang C and Kan C 2016 The Reverse Reconstruction and Finite Element Analysis of the UAV Impeller. Int. J. Sci. Res., 5(3), 595-597
[6] Meli E, Furfieri R, Rind A, Ridolfi A, Volpe Y and Buonamici F 2020 A General Framework for Designing 3D Impellers Using Topology Optimization and Additive Manufacturing. IEEE Access, 8, 60259-60269
[7] Ciocănea A, Marin A & Burețea D L 2019 The influence of housing on the internal flow of a cross flow fan impeller. In 2019 CIEM 464-468 IEEE
[8] Li C, Ji C, Shi M & Sun Q 2019 Influence of rotor–stator component partition structure on the aerodynamic performance of centrifugal compressors. Eng. Appl. Comput. Fluid Mech., 13(1), 1080-1094
[9] Li H, Deng H S and Lai YB 2016 Numerical and experimental research on the outlet flow field for the air multiplier. Appl. Therm. Eng., 93, 652-659
[10] Jafari M, Afshin H, Farhanieh B, and Sojoudi A 2016 Numerical investigation of geometric parameter effects on the aerodynamic performance of a Bladeless fan. ALEX ENG J, 55(1), 223-233
[11] Chen Y L, Fang J, Shiau Y R and Hung W C 2017 A study on static structural of non-rotor unmanned aerial vehicle. MATEC Web of Conferences, 119, 01053. EDP Sciences
[12] Rokosz K, Hryniewicz T, Malorny W, Valiček J, and Harničárová M 2015. ANSYS analysis of stress and strain after cones plastic deformation. Tehnički vjesnik, 22 (2), 503-508
[13] Panagiotou P, Giannakis E, Savaidis G and Yakinthos K 2018 Aerodynamic and structural design for the development of a MALE UAV. Aircr. Eng. Aerosp. Technol
[14] Abderrahmane S, Merabbi F, Lousdad A & Megueni A 2020 Optimization design based approach for the determination and minimization of the displacement under tensile load in hybrid composite joint. Frat. ed Integrità Strutt., 14(52), 281-29
[15] Dillinger J K, Meddaikar Y M, Lübker J, Push M & Kier T 2020 Design and optimization of an aeroservoelastic wind tunnel model. Fluids, 5(1), 35
[16] Lan Z S and Lai W H 2015 The Design and Application of UAV for Disaster Prevention. Journal of the Chinese Institute of Civil & Hydraulic Engineering, 27(3), 191-199
[17] Chen Z, Cai W, Cai L, Cen G and Wang H 2020 Simulation and Experimental Analysis of the Temperature Field of Jet Flow of Aircraft Based on CFD Theory. IEEE Access, 8, 55881-55892