Design and Optimization of a High-altitude Long Endurance UAV Propeller

K You, X Zhao*, S-Z Zhao and M Faisal
School of Aeronautics, Northwestern Polytechnical University, Xi’an, Shaanxi, 710072, China

*Corresponding author: xuzhao@nwpu.edu.cn

Abstract. This paper presents a propeller designed for a solar unmanned aerial vehicle (UAV) at 22km altitude. The design objective for the 3-blade propeller is to achieve 72% efficiency and 7 N thrust, with 0.5588m diameter propeller rotating at 5500rpm under freestream velocity 50m/s. For such High-Altitude Long Endurance (HALE) flight vehicle, firstly, three low Reynolds number airfoils were assessed by airfoil aerodynamic performance analysis software Profili. As a result, FX 63-137 was selected for the best lift to drag ratio at given altitude. Secondly, a MATLAB program based on Joukowski propeller vortex theory was employed to determine three dimensional configuration of the blades. Then, CFD simulation was carried out on the preliminary design. Furthermore, optimal design method was employed to improve propeller efficiency of the preliminary design using software ISIGHT. Ten variables including chord lengths and blade pitch angles distribution at five spanwise sections were employed. Finally, the optimized propeller with power 467W was obtained, performance satisfying the requirement.

1. Introduction
Over the last 20 years, there are growing interest in the field of unmanned high-altitude aerial vehicles. Especially the high-altitude long endurance aircraft become popular. The propeller drive still remains the best propulsion system for airplanes powered by electricity or by fuel[1,2]. High altitude planes powered by propellers are able to fly over the wide range of altitude from 0 to 25000 meters. However, lots of difficulties have been found in propeller design in research of high-altitude aircraft propulsion system. Such wide range of altitude is connected with variability of propeller work conditions (for example the density of the air is changing 10 times) where the air density is much lower (low Reynolds number below $10^5$) and the advance velocity of aircraft is extremely slow (about 10–30m/s). Therefore, the airfoils generate smaller lift and higher drag. This is connected with laminar flow separation phenomenon[3], which makes it a big challenge to design efficient propeller propulsions for stratospheric aircrafts[4]. Hence, the focus on improving the propulsive efficiency is a top priority.

There are two main propeller geometrical adjustments that could be made to improve the propeller performance[5]. Firstly, different airfoil cross-sectional shapes will affect the lift coefficients at different altitudes, affecting the propeller efficiencies under different conditions[6,7]. So, need to select some good functioning airfoil at high altitude. Secondly, the variation of blade pitch angles and chord lengths will also alter the propulsive efficiency[8]. Hence a theory must be chosen for accurate measurement.

This paper has focus on only designing the propeller efficiency for cruising condition at high altitude (Low Reynolds number) of UAV, ignoring the takeoff and landing conditions which work at low altitude (high Reynolds number). Therefore, particularly important is to find the airfoils, which are able to
achieve good performance at low altitude. Airfoils for such aircraft typically operate in the Reynolds number range $2 \times 10^5 \sim 5 \times 10^5$. For computational tests, three of airfoils were selected. All of them represent airfoils dedicated for low Reynolds number flow regime: S1223, FX 63-137 and LOCKHEED L-188[9]. The Wortmann FX 63-137 is one of the most desirable airfoils for high-lift low Re models. The high-lift capability and mild-stall characteristics are among its key attributes[9]. the design philosophy employed in the design of the S1223 airfoil involves combining the favorable effects of both a concave pressure recovery and aft loading to achieve maximum lift at a design Reynolds number of 200k[9].

In this study, it is significant to evaluate the aerodynamic characteristics of the full-scale propeller for the purpose of validation. There are several methods, such as the momentum theory, blade element theory (BET), and vortex method. This papers is based on the vortex theory. Joukowski propeller vortex theory employed to determine the blade section pitch distribution and angle of flow.

To complete the proposal, the algorithm for the propeller optimizer is described, followed by the analysis of the aerodynamic characteristics of the best airfoil. Then parameters were applied established the 3-blade CATIA model. Afterwards, the validation of the design were carried by CFD method. Finally, the analysis of the optimum efficiency and design parameters of propellers are presented.

2. Primary design of the propeller

2.1. Problem description

The design objective for the 3-blade propeller is to achieve 72% efficiency and 7 N thrust, for the lowest power, with 0.5588m diameter rotating at 5500rpm under freestream velocity 50m/s.

At 22km altitude, air temperature is 218.6K, the density is 0.0645kg/m$^3$, the speed of sound is 296.4m/s. The static pressure is 4048Pa, viscosity is $1.4 \times 10^{-5}$kg/(m·s). At cruise status, the relative velocity at 0.7R of the propeller is 127m/s. The local Ma number is 0.43 and Reynolds number is around $1.47 \times 10^4$. The relative blades chord length c/D and blade thickness distribution were selected, shown in figure 1. More details can be found in table 2.

2.2. Low Reynolds airfoil selection

For high-altitude long endurance UAVs, the chord length of the propeller is small, and the air density is quite low, therefore resulted in low Reynolds number flow across the propeller. Generally, low Reynolds number airfoils has a poor performance of lift to drag ratio, which makes the high performance propeller design to be difficult.

Low Reynolds number airfoils were selected, such as S1223, FX 63-137 and LOCKHEED L-188. The aerodynamic performance under Re=$2 \times 10^4$ and Ma=0.43 were compared according to the same
thickness design. Considering a higher lift to drag ratio and a relative steady pitching moment coefficient, airfoil FX 63-137 (thickness 13.7%) and its series thickness airfoils were selected. According to figure 1, the thickness distribution of the airfoil is related to radial location. At each section, the typical relative velocity of the blade element has typical Reynolds number. Therefore, corresponding lift to drag ratio of each airfoils were estimated via airfoil aerodynamic performance analysis software Profili. The best angles of attack were obtained under the highest Cl/Cd by of those airfoils, shown in Table 1.

| r/R  | Airfoil   | Thickness % | Re   | Max Cl/Cd | Cl   | Angle of attack (°) |
|------|-----------|-------------|------|-----------|------|--------------------|
| 0.2  | FX 63-239 | 23.93       | 14000| 2.65      | 0.5  | 13                 |
| 0.4  | FX 63-150 | 15.04       | 19000| 6.36      | 0.69 | 7                  |
| 0.6  | FX 63-101 | 10.12       | 25000| 16.34     | 1.4  | 11                 |
| 0.8–0.95 | FX 63-081 | 8.07       | 37000| 35.53     | 1.38 | 7.5                |

Table 1. Airfoil performance data

3. Primary propeller designed by Joukowski propeller vortex theory
To obtain the chord length and blade section pitch distribution, Joukowski propeller vortex theory was employed and developed into a MATLAB program, where data in table 1 is involved. The thickness of the airfoils were modified a little bit. The preliminary design results is in table 2. The geometry of the propeller is shown in figure 2.

| r/R  | Relative chord length c/D(%) | Chord length c(mm) | Blade section pitch (°) | Relative thickness(t/c) % | Airfoil   |
|------|------------------------------|--------------------|------------------------|--------------------------|-----------|
| 0.036| 2.21                         | 12.27              | 72.4                   | 30                       | Ellipse   |
| 0.1  | 3.20                         | 17.78              | 79.2                   | 24.1                     | FX 63-241|
| 0.2  | 3.78                         | 21.00              | 75                     | 24.1                     | FX 63-241|
| 0.4  | 6.18                         | 34.33              | 52.5                   | 15.2                     | FX 63-152|
| 0.6  | 6.71                         | 37.30              | 41                     | 10.3                     | FX 63-103|
| 0.8  | 4.81                         | 26.72              | 30.5                   | 8.2                      | FX 63-082|
| 0.95 | 1.52                         | 8.43               | 25.9                   | 8.2                      | FX 63-082|

4. CFD simulation of the primary propeller
To evaluate the aerodynamic performance of the propeller, CFD method was carried out based on a steady flow simulation. Software NUMECA is employed to create structural grid and execute the solution.

In the numerical model, propeller is fixed in radial direction, with a radius R. Free stream flow is arranged in z direction. The computational domain covers a domain [0,5R] × [0,2 π / 3] × [−10R,25R] in radial, azimuth and axial directions, respectively where R is the rotor radius, see figure 3. The propeller is placed at z=0. The boundary conditions include: (1) far field boundary for front, rear and circumferential faces of the cylinder, where the free flow is 50m/s in right direction; (2) periodic boundaries for the longitudinal sections; (3) Navier-Stokes wall for blade surface; (4) Euler wall for hub. The whole domain is rotation at 5500RPM around negative z direction, and the discrete precision is second order.

Structured mesh were created, the division is 129×73×215 in radial, azimuth and axial directions, the total grid is 2 million in 33 blocks. Near wall boundary, the thickness for the first layer is 1×10-5c (c is the cord length of the blade) and around 15 layers in boundary layer zone. Steady flow Shear Stress
Transport (SST) turbulent flow model and Turbulent Navier-Stokes mathematical model were built on the model using Fine Turbo. The RANS equation is applied to simulate the steady flow across the propeller. Central difference method and multigrid approach are applied. The flow is considered to be converged under the condition that mass residual is less than $10^{-3}$, thrust and torque of the rotor are constant.

Figure 3. CFD model of propeller

The thrust and torque of the propeller were shown in table 3. And efficiency is calculated according to equation (1). In equation 1, T represents thrust, V is the cruise velocity and P is the power of the propeller, it is the product of torque with rotating speed (M·Ω).

$$\eta = \frac{TV}{P}$$

(1)

Compared to the design objectives, the thrust of the primary propeller is a little smaller. Therefore, the chord lengths of each sections were increased 15%, and optimal design were applied to adjust the chord length of each sections.

Table 3. CFD results of the propeller

|                | Thrust (N) | Torque (N·m) | Efficiency % |
|----------------|------------|--------------|--------------|
| Design objectives | 7          | 0.844        | 72           |
| Primary design   | 6.1        | 0.759        | 69.76        |
| Optimal design   | 7.47       | 0.92         | 70.49        |

5. Geometry optimization of the primary propeller

To improve the performance of preliminary design, the optimization problem was proposed to maximize the propeller efficiency at the design point with a constraint on thrust. The controlled variables are chord length at different sections. Altogether, there are 5 variables for sections 0.2R to 0.95R. The purpose of the approximate model is to produce a rapid method to evaluate propeller aerodynamic performance. This method requires the existence of a database containing several blade geometries and their associated aerodynamic performances. For better results, the NLPQL method is used. The NLPQL algorithm expands the objective function with a second-order Taylor series, and linearizes the constraints to obtain the next design point by solving the quadratic programming. Then perform a linear search based on two alternative optimization functions, where the Hessian matrix is updated by the BFGS formula, the algorithm is stable.

*search chord length $x = (x_1, x_2, \cdots, x_5)$*  
*make the efficiency largest*  
*Design variables*  
*Objective function*
thrust $F \leq -7N$

$0.8x_i \leq x_i^* \leq 1.2x_i \quad i = 1, 2, \ldots, 5$

Inequality constraint

Boundary constraint

The geometrical parameters and CFD results of this sample are stored into the database and the process goes on. As the number of samples in the database grows after each design cycle, the approximate model will become more and more accurate and better blade shapes will be found rapidly. Insight software were applied in optimal approach. The chord length distribution were given in figure 4. CFD results were shown in table 3. The power of the optimal result is 530W. It meets the thrust request but a little smaller than the efficiency request.

The CFD results show that the static pressure distribution on the blade, there is no flow separation on the blade wall, see figure 5. The streamline of the flow across the propeller is shown in figure 6.

Figure 4. Chord length comparison of the primary and optimal design

Figure 5. Static pressure distribution on the propeller

Figure 6. Streamline across the propeller

6. Conclusions
The low Reynold number airfoils FX 63-137 was selected and transformed for 4 thickness series. The primary design a HALE UAV propeller was completed by Joukowski propeller vortex theory and validated by CFD simulation. Results shows that Joukowski theory underestimates the chord length.
Finally, an optimal design approach was applied using software INSIGHT to obtain the best blade section pitch distribution. CFD result of the optimal design shows that the thrust meet the design requirement, the efficiency is 70.49%, which is a little bit smaller than the requirement 72%.

References
[1] Ilieva G, Páscoa J, Dumas A and Trancossi M. A critical review of propulsion concepts for modern airships. Open Engineering, Vol. 2, No. 2, pp 189-200, 2012.
[2] Liu P, Duan Z and Ma L. Aerodynamics properties and design method of high efficiency-light propeller of stratospheric airships. International Conference on Remote Sensing, Environment and Transportation Engineering (RSETE), pp 8041-8044, 24-26 June 2011.
[3] Hu H and Yang Z. An experimental study of the laminar flow separation on a low- Reynolds-number airfoil, Journal of Fluids Engineering, Vol. 130 / 051101-1, 2008.
[4] Tang Z, Liu P, Chen Y, and Guo H. Experimental study of counter-rotating propellers for high-altitude airships. J Propulsion and Power, 31, (5), pp 1491-1496, 2015.
[5] Nie Y. The design and experiments study on high efficient propeller used for stratospheric airship. MS thesis, Center for Space Science and Applied Research, Graduate University of Chinese Academy of Sciences, Beijing, China, 2008(in Chinese).
[6] Koch LD. Design and performance calculations of a propeller for very high altitude flight. MS thesis, Case Western Reserve University, Cleveland, Ohio, US, 1998.
[7] Ma R and Liu P. Numerical simulation of low-Reynolds-number and high-lift airfoil S1223. Proceedings of the World Congress on Engineering, Vol. 2. 2009.
[8] D’Angelo S, Berardi F and Minisci E. Aerodynamic performances of propellers with parametric considerations on the optimal design. Calculus of Variations, Vol. 4, No. 5, p 6, 2002.
[9] Selig M S, Guglielmo J J, Broeren A P and Giguere P. Summary of low-speed airfoil data, Volume 1. 1995.