Aerodynamic Characteristics of a Ducted Fan System Based on Momentum Source Method

Hongming Cai¹*, Guoliang Ma², Zhongxin Li¹

¹School of Mechanical Engineering, Nanjing University of Science and Technology, Nanjing, China
²School of Energy and Power Engineering, Nanjing University of Science and Technology, Nanjing, China

*Corresponding author e-mail: chm@njust.edu.cn

Abstract. Ducted fan systems have received a considerable attention due to their potential for wide application in both civil and military missions. Compared with isolated fans, the presence of duct can substantially weaken the contraction of fan wake and gives the potential to fly efficiently with high security, compact structure and low noise. Based on the momentum source method (MSM) and the S-A turbulence model, a computational fluid dynamics (CFD) simulation model is established and validated as a reliable tool by comparing with calculation results based on sliding mesh technique. The effect of the revolution speed, flight velocity and angle of attack on the aerodynamic characteristics of the ducted fan system is evaluated in detail. The calculation results show that the thrust of the ducted fan system shows an approximate quadratic increase with the increasing revolution speed; As the vertical velocity increases, the thrust of the ducted fan system increases first and then decreases; As the forward velocity increases, the lift and drag of the ducted fan system increase; As the angle of attack increases, the drag of the ducted fan system increases and the lift first increases and then decreases.

1. Introduction

Nowadays, the military’s desire for improved awareness and information-gathering capability in combat situation has led to research into ducted fan VTOL UAVs which can provide a small group of soldiers with a bird’s eye view of the battlefield [1, 2, 3]. It has the ability to hover, take off and land vertically and fly forward with high security, compact structure and low noise. The ducted fan UAVs can be sent to a hostile territory or over the next hill to see what lies beyond and can be loaded with advanced camera and sensing device to report the accurate information back to the operator. However, the special layout of the ducted fan UAVs challenges the design and analysis methods on aircrafts with conventional layout.

The ducted fan systems behave extremely complex aerodynamic characteristics, especially in a crosswind or forward flight. Several reviews of aerodynamic characteristics of ducted fan systems found in literatures are given, for example, Zhao [4] used an effective but rough theoretical approach to model the aerodynamics of a ducted fan UAV in a preliminary design. Guerrero [5] employed AVID OAV, a multidisciplinary optimization code for designing and analyzing ducted fan VTOL UAVs, to predict the aerodynamics of the various components that make up a ducted fan UAV. Graf [6] adopted wind tunnel tests to study the aerodynamic characteristics of the ducted fan UAV. The propeller of the UAV was not
considered, which led to a big mistake. Ruzicka [7] employed the Navier-Stokes overflow mesh flow solver to analyze the flow field of the RAH-66 rotorcraft reversal system. Ryu [8] adopted a commercial computational fluid dynamics software to study the aerodynamics and duct lip separation of the ducted fan system in crosswind. Ohanian [9] explored the novel application of synthetic jet actuators for leading and trailing edge flow control on ducted fan UAVs. Thouault [10] studied the aerodynamics and flow field of a generic fan-in-wing configuration using CFD simulation, particle image velocimetry and wool-tuft visualization. The above works are of great significance for understanding the overall aerodynamic characteristics of ducted fan UAVs. However, there are few literature on aerodynamics of ducted fan systems with coaxial fans.

In this paper, a CFD simulation method based on MSM and S-A turbulence model is established and the aerodynamic characteristics of the ducted fan system are analyzed. This paper is organized as follows: In Section 2, the numerical method used for the ducted fan system is established, and validated in Section 3. Subsequently, section 4 dedicates to the aerodynamic characteristics of the ducted fan system and followed by conclusions in Section 5.

2. Numerical method

2.1. Governing Equations
The conservative unsteady RANS equations written in ALE form are adopted as the governing equations:

\[
\frac{\partial}{\partial t} \int_{\Omega(t)} W \, dV + \oint_{\partial \Omega(t)} (\mathbf{F}(W) - \mathbf{n} \cdot \mathbf{\hat{n}} W) \, dS = \Phi_{\Omega(t)} \mathbf{F}_v \, dS + \int_{\Omega(t)} \mathbf{S} \, dV
\]

(1)

Where \( \Omega \) is the flow control volume and \( \partial \Omega \) is the boundary of the control volume, \( W \) represents the conservative vector, \( \mathbf{F}(W) \) and \( \mathbf{F}_v \) represent the convective and viscous fluxes respectively, \( \mathbf{n} \) and \( \mathbf{\hat{n}} \) represent the velocity and normal vector of boundary respectively. \( \mathbf{S} \) represents the momentum source term.

2.2. Momentum Source Model
The rotational motion of the propellers with respect to the fuselage makes the flow field of the ducted fan system essentially unsteady and complex. The propellers should be simplified to predict the aerodynamic characteristics of the ducted fan system. MSM [11, 12, 13] mediates the complexity and accuracy between the body-fitted grid method and the theoretical method. In the momentum source model, the propeller is treated as a “black box”, where the energy is changed within the fluid by introducing a MSM in the cylindrical region enclosing the propeller. The momentum source term can be expressed as:

\[
\mathbf{S} = S(C_l, C_d, \alpha, \dot{\alpha}, \mathbf{V}_{abs}, \omega, x, y, z, t, C, \rho, \mu, R, M, B)
\]

(2)

where \( C_l \) and \( C_d \) represent the airfoil lift and drag coefficients respectively, \( \alpha \) and \( \dot{\alpha} \) represent the angle of attack and the rate of change of it, \( \mathbf{V}_{abs} \) denotes the absolute velocity, \( C \) represent the blade chord, \( \rho \) and \( \mu \) denote the air density and viscosity respectively, \( R \) and \( M \) represent the Reynolds number and Mach number respectively, \( B \) is the number of blades.

2.3. Turbulence Model
Compared with the other turbulence models, the S-A turbulence model has a small amount of calculation and good stability, and can better simulate the attached flow and the separated flow. S-A turbulence model is introduced to close the RANS equations:

\[
\frac{\partial \mathbf{\vec{v}}}{\partial t} + \frac{\partial}{\partial x_j} (\mathbf{u}_j \mathbf{\vec{v}}) = \mathbf{b}_i (1 - f_{j2}) \mathbf{S}_i \mathbf{\vec{v}} + \frac{1}{\sigma} \frac{\partial}{\partial x_j} \left[ (\mathbf{v}_j + \mathbf{\vec{v}}) \frac{\partial \mathbf{\vec{v}}}{\partial x_j} \right] + C_{12} \frac{\partial \mathbf{\vec{v}}}{\partial x_j} \frac{\partial \mathbf{\vec{v}}}{\partial x_j} - \left[ C_{41} f_{i2} - \frac{C_{42}}{\kappa} f_{j2} \right] \left[ \frac{\mathbf{\vec{v}}}{d} \right]^2
\]

(3)

where \( \mathbf{\vec{v}} \) is turbulence viscosity coefficient.
3. Method Validation

3.1. Vehicle Description
Fig.1 shows the ducted fan system studied in this paper. The ducted fan system consists of duct and upper/lower fans. Some parameters of the ducted fan system are presented in Table 1. The flow structure and aerodynamic characteristics of the ducted fan system is significantly different from isolated fans due to the presence of duct, which should be studied in detail.

![Ducted fan system](image1)

**Figure 1.** Ducted fan system.

| Parameter                  | Value  |
|----------------------------|--------|
| Diameter of duct           | 0.4m   |
| Chord of duct              | 0.24m  |
| Diameter of propeller      | 0.3m   |
| Maximal cruise speed       | 15m/s  |

**Table 1.** Some parameters of the ducted fan system.

3.2. Mesh Generation
The ducted fan system is meshed with refined unstructured meshes, as shown in Fig.2. Fig.2(a) and Fig.2(b) illustrate the calculation grids based on MSM and sliding mesh technology respectively. The wall Y plus values of the calculation grids are less than 1.0. The grow rates of the grid along the normal of wall are less than 1.2. In overall, the computational domains hold about 2800,000 and 4100,000 cells respectively.

![Calculation grids](image2)

**Figure 2.** Calculation grids of the ducted fan system.
3.3. Validation Results
Fig. 3 plots the thrust of the ducted fan system at revolution speed between 2000 and 3500. It can be seen that, all cases show an approximate quadratic increase with the increasing revolution speed. The calculated thrust based on MSM is slightly bigger than that based on sliding mesh technology at each revolution speed. In general, the numerical method established in this paper has been proven to be useful for numerical simulation of ducted fan systems.

![Figure 3. Thrust of the ducted fan system in hovering.](image)

4. Aerodynamics of the ducted fan system

4.1. Vertical Flight
Fig. 4 shows the pressure contour of the ducted fan system at different vertical velocities (-6m/s, -3m/s, 3m/s, 6m/s). It can be seen that the duct wall exhibit different pressure distribution characteristics at different vertical velocities. As the rising velocity of the ducted fan system increases, the airflow velocity on the wall of the duct lip increases, and the pressure on the duct lip decreases. It should also be noted that the pressure on the outer wall of the duct also decreases with the increase of the rising velocity.

![Pressure Contour](image)
Figure 4. Pressure contour of the ducted fan system at different vertical velocities.

Fig. 5 illustrates the thrust of the ducted fan system at different vertical velocities. It can be seen that the thrust of the ducted fan system does not monotonically increase or decrease as the vertical velocity increases. The ascending motion causes the effective angle of attack of the airfoil to decrease, and the descending motion causes the effective angle of attack of the airfoil to increase until stall. Therefore, the thrust of the ducted fan system increases first and then decreases with the increase of the vertical velocity.

Figure 5. Thrust of the ducted fan system at different vertical velocities.

4.2. Forward Flight
Fig. 6 shows the pressure contour of the ducted fan system in forward flight. It can be seen from Fig. 6(a) that a weak vortex is formed in the middle of the inner wall of the upwind side and a low pressure zone is formed in the lip. It can be seen from Fig. 6(b) that a large vortex is formed in the middle of the inner wall of the upwind side, a high pressure zone is formed in the middle part of the outer wall of the upwind side and the duct lip in the downwind side, and a large low pressure zone is formed in the duct lip of the upwind side. Therefore, the ducted fan system generates large lift, drag and positive moment in forward flight.
Figure 6. Pressure contour of the ducted fan system in forward flight.

Fig. 7 shows the lift and drag of the ducted fan system in forward flight. It can be seen that the lift of the ducted fan system increases first and then decreases as the angle of attack increases, and the maximum lift corresponds to an angle of attack of about 70 degree. The drag of the ducted fan system increases as the angle of attack increases. The main reason is that the thrust of the coaxial fans decreases and the drag of the duct increases. It can also be found that the lift and drag of the ducted fan system increase as the forward velocity increases.

Figure 7. Lift and drag of the ducted fan system in forward flight.

5. Conclusion
In this paper, a MSM-based Naiver-Stokes computational fluid dynamics model is established and validated by comparing with calculation results based on sliding mesh technology. The flow field of the vertical and forward flight of the ducted fan system is numerically simulated. The influence of various factors on its aerodynamic characteristics is analyzed. The following conclusions are drawn:

1). The CFD numerical method established in this paper is suitable for simulating the flow field of the ducted fan system.
2). The thrust of the ducted fan system shows an approximate quadratic increase with the increasing revolution speed.
3). The thrust of the ducted fan system increases first and then decreases with the increase of the vertical velocity.
4). The lift and drag of the ducted fan system increase as the forward velocity increases. The drag increases and the lift increases first and then decreases as the angle of attack increases.
Acknowledgments
This work was financially supported by the Natural Science Foundation of Jiangsu Province, China (Grant No.BK20160817) and the Fundamental Research Funds for the Central Universities (Grant No.30915118807, 30917011302).

References
[1] Modeling, control, and flight testing of a small ducted-fan aircraft. Journal of Guidance Control and Dynamics, 2006, 29(4): 769-779.
[2] Cai H M, Wu Z L, Deng S H and et al. Numerical prediction of unsteady aerodynamics for a ducted fan micro air vehicle. Proc IMechE Part G, 2015, 229(1), 87-95.
[3] Wang X.L., Xiang C.L., Najjaran H. and et al., Robust adaptive fault-tolerant control of a tandem coaxial ducted fan aircraft with actuator saturation, Chinese Journal of Aeronautics, 2018, 31(6), 1298-1310.
[4] Hui When Zhao and Cees Bil, P., Ducted fan VTOL UAV simulation in preliminary design. 9th AIAA Aviation Technology, Integration, and Operations Conference, 2009.
[5] Ignacio Guerrero, Kelly Londenberg, A powered lift aerodynamic analysis for the design of ducted fan UAVs. 2nd AIAA Unmanned Unlimited Systems, Technologies, and Operations, 2003.
[6] Will Graf, Jonathan Fleming and Wing Ng, P., Improving ducted fan UAV aerodynamics in forward flight. 46th AIAA Aerospace Sciences Meeting and Exhibit, 2008.
[7] Ruzicka, G. and Strawn, R., Meadowcroft, E., Discrete-blade, Navier-Stokes computational fluid dynamics analysis of ducted-fan flow. Journal of Aircraft, 42 (5), 1109-1117.
[8] Ronch, A., et al., Evaluation of dynamic derivatives using computational fluid dynamics. AIAA Journal, 2012, 50 (2), 470-484.
[9] Ohanian J. Ducted fan aerodynamics and modeling, with applications of steady and synthetic jet flow control, PhD Thesis, Virginia Polytechnic Institute and State University, USA, 2011.
[10] Thouault N, Breidamter C and Adams A., Numerical investigation of inlet distortion on a wing-embedded lift fan. Journal of Propulsion and Power, 2011, 27(1), 16-28.
[11] Cai H.M., Virtual flight simulation of a dual rotor micro air vehicle, International Journal of Computational Fluid Dynamics, 2015, 29(2), 192-198.
[12] Son C., Oh S. and Yee K., Ice accretion on helicopter fuselage considering rotor-wake effects, Journal of Aircraft, 2017, 54(2), 500-518.
[13] Mao X. and Sorensen J., Far-wake meandering induced by atmospheric eddies in flow past a wind turbine, Journal of Fluid Mechanics, 2018, 846, 190-209.