Numerical Simulations for the Tip-Jet Drive Rotor Based Upon Momentum-Source Method

CUI Fei¹, ZHUANG Jian¹ and WANG Bo²
¹Innovation & Research Institute of HIWING Technology Academy, Beijing 100074, China
²Science and Technology on Rotorcraft Aeromechanics Laboratory Nanjing University of Aeronautics and Astronautics, Nanjing 210016, China
E-mail: 381343120@qq.com

Abstract. Based upon a momentum-source method, a numerical simulation method has been developed to calculate the tip-jet drive rotor/fuselage flowfield. The momentum-source term is used to describe the effect of rotor on its flowfield. The mass-source term is used to describe the tip-jet. According to the characteristics of rotor/fuselage flowfield and requirements of momentum-source and mass-source method, a hybrid mesh generation approach is developed. In the present method, on the basis of the known movement, geometric and aerodynamic characteristics on blade section, a scheme containing momentum-source and mass-source term is adopted for the solution of N-S equations. By the present method, the downwash flowfields of rotor and rotor/fuselage are simulated, and some meaningful conclusions are obtained.

1. Introduction
The jet-driven rotor generates a reaction force to drive the rotor to rotate through the high-speed jet of the wing tip. Compared with the shaft-driven rotor, the heavy gear transmission system and the anti-torque system are eliminated, which effectively reduces the complexity of the system, and is simple and easy to use[1-2]. With the increasing demand for vertical take-off and landing and high-speed cruise aircraft, a new concept of rotorcraft with stagnable rotor, such as X-50A in the United States[3], has emerged. The rotor of this type of aircraft has two major characteristics. First, the rotor is driven by jet, which can not only meet the power demand of vertical take-off and landing, but also meet the thrust demand of high-speed leveling in a single power situation. Second, the rotor can rotate at high speed to provide tension for vertical take-off and landing, and can be locked into a fixed wing to provide lift for high subsonic speed cruise.

In order to reduce the computational cost and improve the efficiency of simulated rotor downwash flow field, some researchers have made many attempts. At the end of the last century, Rajagopalan[4] and Chaffin[5] calculated the rotor flow field by using a paddle instead of a rotor, and made significant progress.

Based on the principle of momentum source[6-8], the aerodynamic characteristics of CRW(Canard Rotor/Wing) are numerically simulated by CFD method.

2. Principle and geometric characteristics of tip-jet drive rotor
A typical tip-jet rotor system consists of an air intake, an engine, a main nozzle, a main shaft duct, jet pipes, tip-jet nozzles, and a rotor/wing, as shown in Figure 1. The jet enters the main shaft pipe from...
the tail nozzle of the engine, and is divided into two blades that enter the rotation through the “Y”-shaped pipe at the hub. The main nozzle prevents the back pressure from being too high and causes the engine to surge.

The jet pipe is located inside the blade, as shown in Figure 2. At the tip of the blade, the cross-sectional shape of the pipe changes with the shape of the airfoil, as shown in Figure 3.

![Figure 1. Typical tip-jet rotor system.](image1)

![Figure 2. Blade cross section at hub.](image2)

![Figure 3. Blade cross section at tip.](image3)

3. Momentum-source method
By projecting the lift and the drag at \((r, \phi, n)\) in the blade profile coordinate system to the \(n, \phi\) direction, respectively, the tensile force \(f_n\) perpendicular to the rotational direction and the rotational drag \(f_\phi\) in the rotational direction can be obtained.

\[
\begin{align*}
    f_n &= dY \cos \beta - dX \sin \beta \\
    f_\phi &= -dY \sin \beta - dX \cos \beta
\end{align*}
\]  

(1)

(2)

The resultant force \(\vec{F}\) of the tensile force \(f_n\) and the resistance \(f_\phi\) and the radial force (which can be ignored here) is the force of the surrounding airflow on the blade micro-segment.

4. Mass-source method
Under the premise of not considering the generation of internal gas, transportation mode and whether it has the property of high temperature and high pressure, the driving mode of the tip jet can be expressed by the cross section of the propeller disk that continuously emits a certain mass of gas in synchronization with the blade:

Jet function = mass source item + mass source item corresponding to the momentum source item

Equivalent algorithm for mass flow \(\dot{m}\): The mass of gas ejected per unit time of the nozzle is evenly distributed to the volume of space swept by the nozzle per unit time.

5. Characteristics of flow field in the hovering state

5.1. Computational grid and conditions

![Figure 4. Selection of flow field calculation domain for CRW.](image4)

![Figure 5. A cross-sectional view of the flow field grid in the hovering state of CRW.](image5)
The selection of computational region should adapt to the characteristics of flow field, taking the hovering state as an example, select cuboid of $9R \times 9R \times 14R$ (R is the radius of rotor) as the calculation domain, as shown in Figure 4. Figure 5 is a cross-sectional view of the flow field grid in the hovering state of the CRW aircraft.

5.2. The flow field in hovering state

Figure 6 shows the flowfield of the fuselage in hovering state. The black arrow represents the flow line inside the surface. The far-field airflow above the fuselage has a downward acceleration trend when approaching the paddle. The downwash airflow shrinks into the paddle plate and flows downward around both sides of the fuselage when encountering the fuselage. After bypassing the fuselage, the airflow continues to accelerate downward. The light yellow part on both sides of the fuselage in the figure is the tip vortex.

Figure 7 shows the velocity vector of the airflow in the longitudinal direction of the fuselage. It can be seen that in the longitudinal direction of the fuselage, with the center of the hub as the boundary, the downwash flow in front of the fuselage hits the back of the fuselage and flows up and down along the front fuselage surface respectively. The downward airflow leaves the fuselage at the nose and flows downwards, while the upward airflow flows along the wall to the hub, and then flows vertically upward along the hub wall to form the upward airflow. Similarly, due to the influence of the tip vortex, the airflow behind the fuselage is sucked by the paddle, which is also divided into two parts along the wall surface upward and downward. The upward airflow to the hub forms the upward airflow along the wall of the hub, and the downward airflow flows along the fuselage to the tail, which is gradually upstream under the influence of the tail tip vortex.

6. Analysis with/without mass jet

Aiming at the forward flight state of CRW aircraft, the flow field with/without mass jet is compared and calculated. Among them, the forward flying speed of the CRW aircraft is 42m/s, the rotor speed is 1000r/min, the angle of attack is 0°, the collective pitch is 10°, the wing tip jet velocity is 400m/s, the wing tip jet flow rate is 2.5kg/s, and the flow temperature is 723K.

The following data are derived from the calculation results of two groups of flow fields: a) only added momentum source term (not simulated tip jet); b) added both momentum source term and mass source term (simulated tip jet). Figure 8 and Figure 9 shows the comparison of the density distributions with and without mass jet. It is assumed that the distance below and above the flow field space is 20H, and the central position in the y direction is at the point where $h/H = 0$. 

![Figure 6](image1.png)  
**Figure 6.** Velocity distribution of downwash flow field of CRW aircraft in hovering state. 

![Figure 7](image2.png)  
**Figure 7.** The velocity vector of the fuselage surface. 

![Figure 8](image3.png)  
![Figure 9](image4.png)
By comparing the results of group a) and group b), it can be seen that the density contour is significantly different. First, in group a), the density of the rotor area is significantly higher than that of the non-rotor area. And due to the influence of the CRW fuselage, the density of the outer ring portion of the paddle is higher than other area of the paddle. But the phenomenon gradually fades with the increasing vertical distance, the density on the entire paddle gradually becomes uniform. On the contrary, group b) with the mass source added has a reasonable density distribution. A ring is formed around the fuselage in the area with enhanced density corresponding to the area with jet tip injection. With the increasing of vertical distance, the influence of fuselage gradually weakens, and a ring is formed in the area with increased density under the influence of jet flow. Second, the upper limit of the density of group b) is much larger than that of group a). It can be analyzed that because of the introduction of the jet the density of the rotor nozzle area will increase sharply, and the increasing trend will continue to spread around the rotor, making the surrounding area also increases. The density of the so-called "ring" area actually reaches the corresponding density of the group a) value.

In addition, due to the influence of fuselage (vertical tail, canard wing), under the action of jet flow, the distribution of density cloud shows the phenomenon of left- right asymmetry.

Figure 8. Comparison of flow field density distribution with/ without tip-jet mass source.

(a) h/H=0.1 (momentum-source). (b) h/H=0.1 (mass-source term). (c) h/H=0.5 (momentum-source). (d) h/H=0.5 (mass-source term). (e) h/H=0.9 (momentum-source). (f) h/H=0.9 (mass-source term).
Figure 9. Comparison of velocity component and flow direction distribution of nozzle source with/without tip-jet mass source. (h/H=1.1)

In Figure 10, the pressure distribution cloud diagram of fuselage surface in three different states (rotation only, jet only and rotation plus jet) are listed respectively. The four maps in each column represent the pressure distribution at different viewing angles of the fuselage in the same state. Comparing (a) rotation only and (b) jet only state, we can see the effect of the induction speed caused by the rotation. In (a), the red area indicates the high pressure area, and the blue area indicates the low pressure area. The high pressure area is mainly concentrated near the canard wing and near fuselage, the front part of the fuselage and the middle part of the fuselage slant back before the tail, and the distribution of left and right sides are asymmetric. Group (b) just simulates the effect of the tip jet. The induction speed generated in the direction perpendicular to the paddle is too small, causing no significant fluctuation of the pressure. The entire surface is in the same color block area set by the current cloud image, and the pressure change can only be seen when the numerical range of the cloud image is reduced. Group (c) simulates the combination of rotation and jet, the area of high pressure expand, and the value of the pressure increases.

In addition, it is not difficult to see that the place where the high pressure region appears is corresponding to the tip end region where the rotor down washing speed is the largest. Due to the maximum speed of the tip of the blade, the induced velocity is also correspondingly the largest, which results in the formation of a high-pressure region on the surface of the fuselage. It can also be seen from the figure that the disturbance of rotor down-wash flow on the pressure of CRW tail is very small.

Figure 10. Comparison of pressure distribution on fuselage surface in different states.
In Figure 11 the pressure distribution of the flow field section in three different states (rotation only, jet only and rotation plus jet) are listed respectively. The cross sections shown are in the same spatial position and are located on the symmetrical plane of the fuselage. It can be seen from the figure that the disturbance of rotor downwash flow on the tail fin is indeed relatively small, which is consistent with the previous conclusion. The low pressure area near the paddle is mainly concentrated in the area near the tip of the propeller, and there are few low pressure areas in the central area of the paddle. Similarly, the high pressure area under the paddle plate is mainly concentrated at the tip, and the distribution law of the rotor-induced speed is reflected from the side, which is consistent with the actual situation.

(a) Rotation only. (b) Jet only. (c) The combined of rotation and jet.

Figure 11. Comparison of pressure distribution on the symmetrical plan

7. Conclusion

Based on the momentum source principle, the aerodynamic characteristics of CRW are simulated by CFD method. Firstly, the CFD numerical simulation method is described, and the flow field simulation flow charts are given. Then, the flow field characteristics in the hovering state are given. It can be seen that the distribution trend generated by momentum source is basically reasonable. Finally, the density distribution, velocity distribution and streamlines comparison of the fuselage under the action of momentum source only and the interaction of momentum source and mass source are given.

References

[1] Clark A. Mitchell, Barbara J. Vogel. The Canard Rotor Wing(CRW) Aircraft-A New Way to Fly[R]. AIAA 2003-2517, 2003.
[2] J.W. Lee, K.S. Jeon, M.Kim, et al. Rotor Deisign for the Performance Optimization of Canard Rotor/Wing Aircraft[C], The 2nd International Basic Research Conference on Rotorcraft Technology, Nanjing, China, 2005:322-335.
[3] J.W. Rutherford, S.M. Bass, S.D. Larsen. Canard Rotor/Wing: A Revolutionary High-speed Rotorcraft Concept[R]. AIAA 93-1175, 1993.
[4] Rajagopalan R. G.. A procedure for rotor performance, flowfield and interference - A perspective[R]. AIAA-2000-116. 2000.
[5] M. S. Chaffin, J. D. Berry, S. D. Larsen. Navier-Stokes Simulation of a Rotor Using a Distributed Pressure Disk Method, 51st American Helicopter Society International Annual Forum 1995. 112-136.
[6] Rajagopalan R. G, Mathur S. R. Three dimensional analysis of a rotor in forward flight. AIAA 89-1815, 1989.
[7] Rajagopalan, R. G. A procedure for rotor performance, flowfield and interference- A perspective[R] AIAA-2000-116, 2000.
[8] Leishman J. G. Principles of helicopter aerodynamics. Cambrideg Aerospace Series, Cambridge University Press, 2000.