Numerical Simulation of Rotor-aerodynamic Surface Interaction in Hover Using Moving Chimera Grid

LI Yibo, MA Dongli*
School of Aeronautic Science and Engineering, Beihang University, Beijing 100191, China
Received 19 April 2011; revised 27 May 2011; accepted 20 September 2011

Abstract

Three-dimensional unsteady Navier-Stokes equations are numerically solved to simulate the aerodynamic interaction of rotor, canard and horizontal tail in hover based on moving chimera grid. The variations of unsteady aerodynamic forces and moments of the canard and horizontal tail with respect to the rotor azimuth are analyzed with the deflection angle set at 0° and 50°, respectively. The pressure map of aerodynamic surfaces and velocity vector distribution of flow field are investigated to get better understanding of the unsteady aerodynamic interaction. The result shows that the canard and horizontal tail present different characteristics under the downwash of the rotor. The canard produces much vertical force loss with low amplitude fluctuation. Contrarily, the horizontal tail, which is within the flow field induced by the downwash of the rotor, produces only less vertical force loss, but the amplitudes of the lift and pitching moment are larger, implying that a potential deflection angle scheme in hover is 50° for the canard and 0° for the horizontal tail.

Keywords: unsteady flow; chimera grid; canard rotor/wing; aerodynamic interaction; numerical simulation

1. Introduction

Canard rotor/wing (CRW) aircraft is a new concept which combines the vertical take-off and landing capability of a helicopter with the high-subsonic cruise speed of a fixed-wing aircraft. The concept was promoted by McDonnel Douglas Helicopter Company (now belonging to the Boeing Company) in the 1990s, and was demonstrated under the X-50A “Dragonfly” program conducted by Boeing Company and Defense Advanced Research Projects Agency (DARPA) in 2003-2005 [1-3]. To explore its potential application, investigations have been made by China and Korea recently [4-5].

CRW concept is characterized by a stopped rotor and a three-surface configuration. It converts from rotary to fixed-wing flight by transferring the lift from the rotor to the canard and the horizontal tail until the conversion speed is reached when the rotor is completely unloaded and stops. It promises a wider flight envelope than the helicopter and tilt wing/tilt rotor aircraft. However, due to the complex unsteady interactions between rotor and aerodynamic-surfaces in hover and conversion flight, the CRW aircraft tends to be unstable and hard to control. Therefore, the unsteady aerodynamics simulation is a key point to its practical application [6-7].

The unsteady aerodynamics of the CRW aircraft hovering near the ground is simulated in Ref. [8] using a momentum source method. The momentum source method is an approximate approach to represent the flow field generated by the rotor. In this case, the moving chimera grid method could simulate the unsteady interaction more accurately [9-12].

In this paper, three-dimensional unsteady Navier-Stokes equations are solved to simulate the aerodynamic interactions between rotor, canard and horizontal tail of the CRW aircraft in hover, using a moving chimera grid method. The fluctuations of the aerodynamic
forces and moments are investigated with the deflection angle set at 0° and 50°, respectively. The pressure distributions on aerodynamic surfaces and the flow fields around the canard and the horizontal tail are analyzed to obtain a better understanding of the interaction mechanism.

2. Computational Scheme

2.1. Geometry and flow conditions

A CRW aircraft is shown in Fig. 1(a), and a simplified model without fuselage and vertical tail is shown in Fig. 1(b). Since the canard and the horizontal tail contribute most to the unsteady forces and moments in hover\cite{8}, it is feasible to remove the fuselage and vertical tail here.

The rotor collective pitch is set at 12°, and the rotary speed is 125.67 rad/s; the Mach number at the rotor tip in hover is 0.67. The inflection angle of the aerodynamic surface is set at 0° and 50°, respectively; the definition of deflection angle (where $\phi_c$ and $\phi_H$ are deflection angles of the canard and horizontal tail) is illustrated in Fig. 2.

2.2. Chimera grid method

The Chimera grid method has two main advantages when used to discretize the computation domain. On the one hand, calculations of moving body are allowed without remeshing; on the other hand, complex configuration meshing turns out to be more easily and quickly.

The moving Chimera grid method consists of three key parts: hole-cutting, inter-grid communication and six degrees of freedom integration\cite{13-14}.

The hole-cutting process requires several steps, as shown in Fig. 3. Firstly, the bounding boxes of all zones and wall boundary are determined. Secondly, an alternating digital tree(AD-Tree) is built if the bounding box of zone A overlaps with that of a wall boundary in zone B. The AD-Tree contains the coordinates of A’s cells located on the bounding boxes of the overlapped wall boundary face. Thirdly, the cut cells containing intersecting edges are found out. After that, the status of the two end points of each edge in cut cells is classified; the point inside the wall boundary is marked as IN, while the point outside the wall boundary is marked as OUT. Then, the status checking is propagated to other edges of the cut cells and then to the adjacent cells until all the grid points marked as IN are identified. Finally, the hole is generated by blanking out the cells containing IN points. Repeat step 2 to step 6 until all zones in the Chimera grid scheme are processed.

The Chimera boundary cells (including interpolation boundary cells and fringe boundary cells) represent the linkage between different zones of an overset grid. Every Chimera boundary cell must be associated with a donor cell, and the solution is interpolated from the donor cell to the respective Chimera boundary cell\cite{15-17}.

For a given interpolation boundary cell or fringe boundary cell, a donor cell can be found using the following algorithm. First, search all other grids and identify the cells whose bounding boxes overlap with that of the Chimera boundary; second, identify the donor cell containing the centroid of the Chimera boundary cell. If a donor cell could not be found, mark the Chimera boundary cell as an orphan. Then, interpolate...
solution from the donor cell to the respective Chimera boundary cell. If the latter is an orphan, interpolate a solution from its neighboring cells in the same zone.

In order to solve moving object problems, the general motion of bodies needs to be simulated. The equations of motion of a rigid body with constant mass of inertia are given by

\[ F = m \frac{dV}{dt} \]  
\[ M = \frac{\partial h}{\partial t} + \omega \times h \]  

where \( F \) and \( M \) are the resultant force vector and momentum vector of the body center of gravity, respectively; \( V, h \) and \( \omega \) are the linear velocity, momentum and angular velocity, respectively; \( m \) is mass weight.

The chimera grid used in this paper is illustrated in Fig. 4. The computational field is 40×45 times of the rotor radius, and the grid size is 4.2 million.

\[ \text{Fig. 4} \quad \text{Computational field and surface mesh.} \]

### 2.3. Numerical method

The Reynold average Navier-Stokes(RANS) equations are used in this paper. The governing equations can be derived by applying the mass and momentum balance relations to a control volume \( V_a \) with a boundary in a Cartesian coordinate system. The control volume moves and deforms depending on the volume surface velocity vector \( V_a \). The integral form of the unsteady compressible Navier-Stokes equations can be written as

\[ \frac{d}{dt} \left( \int_{V_a} \rho dV \right) + \int_{\partial V_a} \rho (V - V_a) \cdot n dA = 0 \]  
\[ \frac{d}{dt} \left( \int_{V_a} \rho V dV \right) + \int_{\partial V_a} \left( \rho V - \mathbf{G}_a - \mathbf{Q} \right) \cdot \mathbf{n} dA = \int_{V_a} \mathbf{S} dV \]  

where Eqs. (3)-(4) are the continuity, momentum and energy equation, respectively. \( \mathbf{n} \) is the cell-face normal; \( \mathbf{Q} \) is the conservative variables vector, \( \mathbf{Q} = [\rho u \, \rho v \, \rho w \, \rho E]^T \), \( \rho, E \) are the density and energy, and \( u, v \) and \( w \) are the components of velocity; \( \mathbf{G}_a \) is the convective flux and \( \mathbf{G}_b \) the diffusive flux; \( A \) is the area, \( V \) is the velocity, \( V_a \) is the volume surface velocity and \( S \) is the source term.

The governing equations (3) and (4) are spatially discretized using the finite volume method. The discretized form of the governing equations can be written as

\[ \frac{\Delta \mathbf{Q}}{\Delta t} + \sum_{f} F^f \frac{\partial \mathbf{Q}}{\partial n} \Delta A^f + \mathbf{S} \]  

\[ \begin{aligned} \Delta A^f & = A^f + \sum_{n} \mathbf{G}^n \Delta t \Delta A^f \\ \mathbf{G} & = \mathbf{G}_a - \mathbf{Q}_a \end{aligned} \]  

where \( \mathbf{G} = \mathbf{G}_a - \mathbf{Q}_a \) stands for the convective-moving flux; \( \partial F/\partial \mathbf{Q} \) and \( \partial S/\partial \mathbf{Q} \) are the flux Jacobian and source Jacobian, respectively.

The flux vector and the flux Jacobians are evaluated using Roe’s approximate Riemann solver, a flux difference scheme. High order spatial accuracy is achieved using the Osher-Chakravarthy limiter. The time marching algorithm applied is the Jacobi iterative implicit scheme, in which only the nearest neighbors to the center cell are taken into consideration.

A standard \( k-\varepsilon \) turbulence model is used to close the governing equations:

\[ \begin{aligned} \rho u_i k - \frac{\mu_k}{\sigma_k} k \frac{\partial k}{\partial x_i} & = \mathbf{G}_k - \rho \varepsilon \\ \rho u_i \varepsilon - \frac{\mu_\varepsilon}{\sigma_\varepsilon} \varepsilon \frac{\partial \varepsilon}{\partial x_i} & = \frac{\varepsilon}{k} \left( C_1 \mathbf{G}_k - C_2 \rho \varepsilon \right) \end{aligned} \]  

where

\[ \begin{aligned} \mu_k & = \mu + \mu_1 = \mu + \rho C_\mu \frac{k^2}{\varepsilon} \\ \mathbf{G}_k & = \mu \left[ (u_i + v_i)^2 + (v_i + w_i)^2 + (w_i + u_i)^2 + 2(u_i^2 + v_i^2 + w_i^2) \right] \end{aligned} \]  

where \( i = 1, 2, 3 \) represents the direction of \( x, y, z \), respectively; \( k \) and \( \varepsilon \) are the turbulent Kinetic energy and turbulent dissipation rate; \( \mu, \mu_1 \) and \( \mu_2 \) are the viscosity coefficient, turbulent viscosity and eddy viscosity, and \( C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3, C_1 = 1.44, C_2 = 1.92. \)
3. Results and Discussion

3.1. Fluctuations of force and moment

Affected by the flow field of the rotor, the aerodynamic forces and moments on the canard and the horizontal tail change periodically. The average value of the unsteady aerodynamic forces or moments in a cycle can be written as

\[
\bar{C} = \frac{1}{T} \int_0^T C_t \, dt
\]  

(10)

And the fluctuation amplitude is

\[
\Delta C = C_{\text{max}} - C_{\text{min}}
\]  

(11)

where \(C_t\) stands for the aerodynamic forces or moments at time \(t\); \(C_{\text{max}}\) and \(C_{\text{min}}\) are the maximum and minimum aerodynamic force or moment, respectively.

Fig. 5 and Fig. 6 show the time histories of the vertical force and pitching moment of canard and horizontal tail with the deflection angle set at 0° and 50°, respectively. The vertical force is positive up, and the pitching moment is defined as negative nose down. Fig. 5 shows that the canard produces a considerable vertical force loss at both the two deflections, and the loss reduces as the canard setting angle increases. Contrarily, the horizontal tail produces only exiguous vertical force loss, and the vertical force is positive when the deflection angle is set at 0°.

The general trend of the pitching moments is similar to that of the vertical force, as illustrated in Fig. 6. The canard with deflection angle of 0° causes strong nose down moment because of the negative vertical force, and the horizontal tail with deflection angle of 50° produces a positive pitching moment. The pitching moment approaches 0 when the deflection angle is set at 50° for the canard, and 0° for the horizontal tail.

![Fig. 5 Comparison of vertical force between canard and horizontal tail with deflection angle set at 0° and 50°, respectively.

![Fig. 6 Comparison of pitching moment between canard and horizontal tail with deflection angle set at 0° and 50°, respectively.

The average values and fluctuation amplitudes of the aerodynamic force and moment of the canard and horizontal tail are given in Table 1. The average force and moment of the canard are much larger than those of the horizontal tail, but their fluctuation of the horizontal tail is stronger.

| Table 1  Fluctuations of forces and moments |
|-----------------|-----------------|-----------------|-----------------|-----------------|
|                | Deflection angle(°) | Vertical force/N | Pitching moment/(N·m) |
| Canard         | Horizontal tail   | Canard          | Horizontal tail   |
| Average value  | 0                | −230            | 11               | −356            | −29             |
| Amplitude      | 50               | −103            | 29               | 2               | 135             |
|                | 50               | 22              | 56               | 12              | 240             |

From the above discussions, it can be concluded reasonably that the deflection angle of 50° for the ca-
nard and 0° for the horizontal tail could get the minimum vertical force loss and minimum pitching moment fluctuation in hover.

3.2. Pressure distributions

The fluctuations of aerodynamic force and moment of the canard and the horizontal tail are determined by the periodic rotor wakes. The pressure distributions at different rotor azimuthal angles ($\psi = 0^\circ$, $30^\circ$, $60^\circ$, $90^\circ$, $120^\circ$, $150^\circ$) in the $xOy$ plane at point $Z = -1.3$ m, where the canard and the horizontal tail are positioned, are presented in Fig. 7. As the rotor rotates, a high-pressure region followed by a relatively lower pressure region passes across the aerodynamic surfaces firstly, and then the pressure increases to a higher level gradually. These variational pressure zones pass through the canard and the horizontal tail periodically, causing fluctuant aerodynamic forces and moments.

![Fig. 7 Pressure distributions with different rotor azimuth angles, $xOy$ plane at point $Z = -1.3$ m, $\Phi = 50^\circ$.](image)

Figure 8 shows the variations in vertical force of the canard and the horizontal tail with the variation in the rotor azimuthal angle. It can be inferred that the frequency of the unsteady force is 2 times of the rotor rotational frequency; the canard reaches the minimum and maximum forces at the rotor azimuthal angle $\psi = 6^\circ$ (Point A) and $90^\circ$ (Point B), respectively. And the horizontal tail reaches its peaks at $\psi = 19^\circ$ (Point C) and $\psi = 165^\circ$ (Point D).

The pressure distributions on the upper surface of the horizontal tail at Point C and Point D are shown in Fig. 9. For Point C, the pressure on the upper surface of horizontal tail is much lower than that on the horizontal tail at Point D. The difference in pressure distribution of canard between Point A and Point B is similar to that between Point C and Point D, as presented in Fig. 10.

![Fig. 8 Fluctuations of vertical forces with rotor azimuth angle, $\Phi = 50^\circ$.](image)

![Fig. 9 Pressure distributions corresponding to the maximum and minimum forces of horizontal tail.](image)

![Fig. 10 Pressure distributions corresponding to the maximum and minimum forces of the canard.](image)
3.3. Velocity vectors

The flow fields around the canard and horizontal tail are illustrated in Fig. 11 and Fig. 12. The downward flow around the canard results in a negative vertical force. In Fig. 11(a), the velocity vectors are almost perpendicular to the canard surface at the deflection angle of 0°, producing a large magnitude of vertical force loss. As the deflection angle increases to 50°, a strong vortex has formed and the velocity vectors are more aligned with the chord line of the canard. As can be seen in Fig. 11(b), the vertical force loss is much smaller than that with the deflection angle of 0°.

\[ \text{(a) } \phi_c=0^\circ \]
\[ \text{(b) } \phi_c=50^\circ \]

Fig. 11  Velocity vectors around the canard.

The flow field around the horizontal tail is significantly different from that around the canard, as illustrated in Fig. 12. The forward velocity induced by the rotor downwash generates a positive angle of attack when the deflection angle of the horizontal tail is 0°, which subsequently produces a tiny positive vertical force, as mentioned earlier. The angle of attack turns to be negative when the deflection angle is changed to 50°, thus producing a vertical force loss.

\[ \text{(a) } \phi_h=0^\circ \]
\[ \text{(b) } \phi_h=50^\circ \]

Fig. 12  Velocity vectors around horizontal tail.

Due to the fact that the intensity of the flow field around the horizontal tail is much lower than that around the canard, the corresponding vertical force loss produced by the horizontal tail is much smaller, but the pitching moment of the horizontal tail at 50° is significant for its long force arm.

4. Conclusions

1) A moving chimera grid method and the three-dimensional unsteady Navier-Stokes equations have been used to simulate the unsteady aerodynamic interactions among rotor, canard and horizontal tail of a CRW aircraft in hover.

2) Affected by the rotor, the canard generally produces more vertical force loss and pitching moment compared with the horizontal tail. However, the horizontal tail has stronger force and moment fluctuations.

3) The deflection angle has different influences on the canard and the horizontal tail. The canard produces less vertical force loss and pitching moment at the higher angle of 50°. Contrarily, the horizontal tail has better performance at the lower angle of 0°, implying that a potential deflection angle scheme in hover is 50° for the canard and 0° for the horizontal tail.

4) The unsteady aerodynamic forces and moments of the canard and the horizontal tail are affected by the periodic rotor wakes consisting of high-low-high pressure regions. The wakes pass through the aerodynamic surfaces periodically, causing fluctuations of the forces and moments.

5) The flow field around the canard is quite distinct from that of the horizontal tail. The canard is under the downwash of the rotor, and the velocity vectors are almost perpendicular to the canard surface at the deflection angle of 0°. The horizontal tail is affected by the induced flow field of the downwash, and the forward velocity vectors generate a positive angle of at-
tack when the deflection angle of the horizontal tail is 0°.

References


Biographies:

LI Yibo received B.S. and M.S. degrees from Beihang University in 2006 and 2008 respectively, and is now pursuing his Ph.D. degree there. His main research interests are aircraft design and aerodynamics.
E-mail: liyibo1983@tom.com

MA Dongli is a professor of Beihang University. His main research interests are aircraft design and UAV technologies.
E-mail: madonglibuaa@126.com