Flowfield analysis of helicopter rotor in hover and forward flight based on CFD

Zhao Qinghe, Li Xiaodong
School of Energy and Power Engineering, Beihang University, Beijing 100191, China

Email: zhaqinghe2001@163.com

Abstract. The helicopter rotor field is simulated in hover and forward flight based on Computational Fluid Dynamics(CFD). In hover case only one rotor is simulated with the periodic boundary condition in the rotational coordinate system and the grid is fixed. In the non-lift forward flight case, the total rotor is simulated in inertia coordinate system and the whole grid moves rigidly. The dual-time implicit scheme is applied to simulate the unsteady flowfield on the movement grids. The $k-\omega$ turbulence model is employed in order to capture the effects of turbulence. To verify the solver, the flowfield around the Caradonna-Tung rotor is computed. The comparison shows a good agreement between the numerical results and the experimental data.

1. Introduction
Helicopters have been used widely in the military, civil transportation, first aid domain and so on. To the present time the technology has already experienced great development. Modern helicopters request compact layout, high forward flight speed, manoeuvrability and low noise level, which lead to complex flowfield characteristics like following. Firstly, the flowfield include incompressible regions and transonic regions. Secondly, it is a typically unsteady flowfield, the airflow speed relative to the blades of the advancing side increase under the forward flight condition, some phenomenon of the transonic speed such as shock wave will appear. On the other side, the relative speed decrease with the flow angle is comparatively high usually lead to dynamic stall. Thirdly, there are the tip vortex which make a strong impact on the aerodynamic characteristic of flight vehicle.

After rapid development, computational fluid dynamics(CFD) has received practical application in aviation, as a result of the mentioned complexity above, its application to rotors is much less than fixed wings. [1-6] The grid system is an important issue in rotorcraft numerical simulation. The Chimera approach was used to solver the Reynolds-averaged Navier-Stokes equations on a multi-block mesh. The blade grids are embedded in a background mesh. Along the Chimera boundaries, a tri-linear interpolation method is used to couple the solution on the blade-fixed and background grids. Another approach is using sliding planes method. It couple a part of the mesh rotating with the rotor and a stationary lower part around an approximate helicopter fuselage. Besides, due to the convenience in the grid generation, the unstructured grid is also used wildly.

In the present work, multi-block structured meshes are used to model the non-lift forward flight with a rigid mesh motion. This approach was designed to overcome the complexity of the Chimera grid. The structure of the paper is as follows. Section 2 presents spatial discretization and temporal integration method in the rotational coordinate system and in inertia coordinate system. Section 3
gives the validation results for both hovering and forward flight rotor. Some conclusions are finally drawn in Section 4.

2. Numerical Solution Procedures

Assuming that the wake shed from the rotor is steady, the flow around a hovering rotor can be treated as a steady-state problem. For forward-flight simulations, the flow field is asymmetry and unsteady. So it introduces the simulation process separately.

2.1. Methodologies for steady flows

For hover cases, due to the symmetry of the flow in rotational coordinate system, the computational domain can be replaced by a domain around one blade. The periodicity boundary condition is used to take into account the influence of other blades. Furthermore, the periodicity boundary condition can be used to reduce the computational expense. The government equation is given below

\[ \frac{d}{dt} \int_{V(t)} \vec{v} dV + \int_{\partial V(t)} (\vec{F}(\vec{v}) - \vec{F}_v(\vec{v})) \vec{n} dS = \vec{S} \]  

(1)

Where \( \vec{v} \) is the conserved variables. \( \vec{F} \) and \( \vec{F}_v \) are the inviscid and viscous fluxes, respectively. There is a source terms in the non-inertial frame of reference. It means the influence of the centripetal and the Coriolis force. For an hovering rotor rotating around the y-axis with the angular velocity \( \vec{\omega} = [0, \Omega, 0] \), the source term \( \vec{S} \) can be expressed by equation 2.

\[
\vec{w} = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho E
\end{bmatrix}, \quad \vec{F} = \begin{bmatrix}
\rho(q - q_o) \\
\rho u(q - q_o) + p_i \\
\rho v(q - q_o) + p_j \\
\rho w(q - q_o) + p_k \\
\rho h(q - q_o) + p q_o
\end{bmatrix}, \quad \vec{S} = \begin{bmatrix}
0 \\
-\rho \Omega w \\
0 \\
\rho \Omega u \\
0
\end{bmatrix}
\]

(2)

Where \( q_o \) is the grid velocity, it produce a grid flux. The finite volume discretization with central differencing for the flux approximation leads to 2 order scheme. The implicit Lower-Upper Symmetric Gauss Seidel operator(LUSGS) is applied to obtain the update solution. Besides, the mesh around the rotor does not rotate about the rotational axis.

2.2. Methodologies for unsteady flows

For forward flight, the flowfield should be simulated in unsteady condition. The dual time stepping approach is introduced, and the system can be solved by an iteration through pseudo time to the steady state, as given by

\[
\frac{3V_{i,j,k}^n w_{i,j,k}^{n+1} - 4V_{i,j,k}^n w_{i,j,k}^n + V_{i,j,k}^{n-1} w_{i,j,k}^{n-1} + R_{i,j,k}(w_{i,j,k}^{n+1})}{2\Delta t} = 0
\]

(3)

\[
\frac{V_{i,j,k}^{n+1} w_{i,j,k}^{n+1,m+1} - w_{i,j,k}^{n+1,m}}{2\Delta t} + \frac{3V_{i,j,k}^{n+1} w_{i,j,k}^{n+1,m} - 4V_{i,j,k}^n w_{i,j,k}^n + V_{i,j,k}^{n-1} w_{i,j,k}^{n-1} + R_{i,j,k}(w_{i,j,k}^{n+1,m})}{2\Delta t} = 0
\]

(4)

\[
V_{i,j,k}^{n+1}(\frac{1}{\Delta \tau} + \frac{3}{2\Delta t} + A + B + C)\Delta w_{i,j,k} = -R_{i,j,k}(w_{i,j,k}^{n+1,m}) - \frac{3V_{i,j,k}^{n+1} w_{i,j,k}^{n+1,m} - 4V_{i,j,k}^n w_{i,j,k}^n + V_{i,j,k}^{n-1} w_{i,j,k}^{n-1}}{2\Delta t}
\]

(5)

Where \( \Delta t \) is the physical time step, \( \Delta \tau \) is the pseudo time step, \( \Delta w_{i,j,k} = w_{i,j,k}^{n+1,m} - w_{i,j,k}^{n+1,m} \), A, B, C is the Jacobi metrics of the inviscid flux. The accelerate convergence methods such as local time step can be used in the pseudo time cycles.

2.3. The grid topology and the boundary conditions
The grid generation procedures is an important issue for simulation, especially for complex flowfield and the multiple bodies in relative motion. Because the grid quality affects the accuracy and convergence rate of numerical solutions. Furthermore, in hover simulations, the periodicity boundary are required to form a perfect match at the boundary to avoid data interpolation errors. Figure 1 shows the topology, and the enlarged figure is given in Figure 2. The rotor centre of rotation is laid at the y-axis, the rotor blade is located on the x-axis and the quarter chord point is laid on at z-axis. The blade leading edge points at the negative x-axis direction.

The computational domain of the rotor is divided into 22 Multi-blocks. An O type blocking topology around the blade is used. The accurate viscous case computation can be performed in this type of mesh topology. There are 161 points along the airfoil surface with concentration near leading and trailing edges. The first cell size is $10^{-5}$ of the blade chord length normal to the surface and the cell ratio follows an exponential law. The rotor hub was modelled as a simple straight cylinder.

The computational domain is surrounded by 4 types boundary conditions: viscous wall, rotational periodicity, characteristic type boundary and the hub conditions. To maintain second-order accuracy at the boundaries, the domain is surrounded by two layers of ghost grids. Take the viscous wall boundary condition as an example

$$u_{\text{ghost}} = 2u_{\text{wall}} - u_i, \quad v_{\text{ghost}} = 2v_{\text{wall}} - v_i, \quad w_{\text{ghost}} = 2w_{\text{wall}} - w_i$$

where subscript ghost denotes the ghost grid, subscript wall denotes the value on the wall, it equal the wall movement velocity, in particular it equal 0 when the wall keep static. The subscript 1 denotes the velocity of the first level grid.

The hub conditions are applied when performing single-blade simulations, in hover and forward flight conditions. Characteristic type boundary conditions are applied at the far-field. They are based on free-stream values when the far field boundary is located relatively far from the rotor.

2.4. Mesh movement
The mesh movement should be used under the forward-flight condition. There is no change of the relative position of points in the same grids if the rotor moves rigidly. Assuming it rotate around the y-axis, the transformation matrices are introduced for rotation. And the grid coordinate and grid velocity can be calculated directly from formula 7. Where 0 denotes the original grid position and n denotes the updated grid position.
\[
\begin{bmatrix}
    x \\
y \\
z
\end{bmatrix}^n =
\begin{bmatrix}
    \cos \omega t & 0 & \sin \omega t \\
    0 & 1 & 0 \\
    -\sin \omega t & 0 & \cos \omega t
\end{bmatrix}
\begin{bmatrix}
    x \\
y \\
z
\end{bmatrix}^0
\begin{bmatrix}
    u \\
v \\
w
\end{bmatrix}^n = -\omega
\begin{bmatrix}
    \sin \omega t & 0 & -\cos \omega t \\
    0 & 0 & 0 \\
    \cos \omega t & 0 & \sin \omega t
\end{bmatrix}
\begin{bmatrix}
    x \\
y \\
z
\end{bmatrix}^0
\] (7)

There is no grid volume changing, so the geometric conservation law (GCL) is also kept.

3. Validation of the framework

Validation is now presented for the hover formulation and the forward flight method. A summary of cases is given in Table 1. The Mach number and the Reynolds number are based on rotor tip velocity. Where $\mu$ is the advance ratio, which means the ratio of the free flow velocity and the tip velocity. All the test case have been performed with the $k-\omega$ turbulence model.

**Table 1.** Simulation conditions of the helicopter rotors in hover and forward flight

|       | Ma  | Re              | $\mu$ |
|-------|-----|-----------------|-------|
| Hover | 23.56 | $2.27 \times 10^6$ | 0     |
| Forward flight | 27.9 | $3.58 \times 10^6$ | 0.2   |

3.1. In hover

An experimental and analytical study of a model helicopter in hover had been carried out by Caradonna and Tung [7]. The blade had a NACA0012 profile and were untwisted and untapered, which had a diameter of 2.286 meter, and a chord length of 0.191m, according to the aspect ratio was 6. In the Caradonna-Tung experiment, the surface pressure distribution was measured at five rotor blade sections ($r/R=0.50, 0.68, 0.80, 0.89, 0.96$). The $C_P$ is calculated by formula 8, where $\omega$ is the rotate velocity and $r$ is the span distance.

\[
C_P = \frac{p-p_0}{0.5 \rho (\omega r)^2}
\] (8)

The CFD results for the pressure coefficient distribution at four different sections of the blade were compared to the experimental data in Figure 3. The comparison shows a good agreement between the numerical results and the experimental data.
3.2. Non-lifting forward flight

The case is for a non-lifting rotor which has two blades and the airfoil is NACA0012. The rotor aspect ratio was 7. The experimental data [8] is available for the surface pressure on the advancing side. The grid moves rigidly with the blade surface. Therefore, the rotation is applied to the whole grid. The hover tip Mach number is 0.8 and advanced ratio is 0.2. There is a strong transonic effects on the advancing blade. The azimuth angle increment of 1° is used for the computation. The surface pressure distributions according to six azimuth angles \( \phi = 30°, 60°, 90°, 120°, 150°, 180° \) are shown in Figure 4. The wind flows from left to right, and the rotor rotate in anticlockwise direction. A strong shock wave is seen on the advancing side of the rotor blade.

Figure 4. Surface pressure distribution of forward flight rotor

Figure 5 shows the surface pressure distributions at 89% span for six azimuth angles. The \( Cp \) is calculated by formula 9, where \( V_\infty \) is the free stream velocity and \( \phi \) is the azimuth angle. There are some difference between the numerical results and the experimental data. The flow is highly unsteady and the shock is moving rapidly downstream.

\[
Cp = \frac{p - p_0}{0.5 \rho (V_r + V_\infty \sin \phi)^2}
\]  

(9)
Figure 5. Pressure distribution comparison on the forward flight
4. Conclusions

The helicopter rotor field is simulated in hover and forward flight. In hover case the single rotor field is simulated in the rotational system and the grid is fixed. In the non-lift forward flight case, the total rotor is simulated in inertia system, the whole grid move rigidly. The finite-volume solver adopts a dual-time implicit scheme on movement grids. The $k-\omega$ turbulence model is employed in order to capture the effects of turbulence. The unsteady flowfield of the Caradonna-Tung rotor has been investigated. The surface pressure distribution compared well with experimental data. Forward flight prediction capabilities have been validated through the Caradonna-Tung rotor. Some conclusions are drawn below:

a) In the rotational frame the hover flowfield can be transformed from the unsteady problem to a steady one. The accuracy can be kept and the simulation time decrease greatly.
b) In forward flight condition the rotor flowfield is unsteady. It can be simulated by dual-time marching with the moving mesh method.

References

[1] Hariharan N Egolf A and Sankar L 2014 Simulation of Rotor in Hover: Current state and challenges AIAA paper 2014-0041
[2] Srinivasan G R and Baeder J D 1992 Flowfield of lifting rotor in hover: A Navier-Stokes simulation AIAA Journal Vol 30 No 10
[3] Jacques S Klansdieter P and Michel C 2001 Numerical simulation of flows around helicopters at DLR and ONERA Aerosp Sci Technol 5:35-53
[4] Pomin H and Wagner S 2002 Navier-Stokes analysis of isolated rotor flow in helicopter hover flight Journal of Aircraft 39(5) 813-821
[5] Steijl R Barakos G and Badcock K 2006 A framework for CFD analysis of helicopter rotors in hover and forward flight International Journal for numerical methods in fluids 51:819-847
[6] Renzoli P D’Alascio A Kroll N Peshkin D and H.L.Hounjet M 2000 EROS-a common European Euler code for the analysis of the helicopter rotor flowfield Progress in Aerospace Sciences 36:437-485
[7] Caradonna F X and Tung C 1981 Experimental and analytical studies of a model helicopter rotor in hover Vertica 5(1):149-161
[8] Caradonna F X Laub G H and Tung C 1984 An experimental investigation of the parallel blade-vortex interaction NASA/TM 86005