CFD Simulation Methods for Rotor Hovering Based on N-S Equation

Zhou Cheng-Long*, Chen Ming, Xu An-An and Wang Fang

School of Aeronautic Science and Engineering, Beihang University, Xueyuan Rd, 37th Haidian Dist, Beijing 100191, Peoples R China.

*E-mail: 278427938@qq.com

Abstract. CFD simulation of the hovering condition of a single-rotor helicopter was carried out by three methods: moving reference frame (MRF), sliding mesh and overset mesh, respectively. The parameters such as velocity field, pressure field and pressure coefficient of the helicopter rotor and its surrounding flow field are obtained, and the simulation results are compared with the experimental data of wind tunnel test. The three methods are compared from the differences between the model and the grid, the accuracy of the calculation results, and the calculation convergence time. The characteristics, application and other meaningful results of each method are obtained.

1. Introduction

The main rotor of the helicopter generates lift and propulsion, so it is the most important part of the helicopter. Its aerodynamic performance not only has a great impact on the flight performance of the helicopter, but also has an important impact on the characteristics of vibration and noise. The design of the main rotor is largely influenced by the requirements of its hovering state, so it is of great significance for the aerodynamic performance of the helicopter hovering state[1].

At present, the research methods of rotor aerodynamic performance mainly include experimental methods and numerical simulation methods. The numerical simulation method effectively overcomes the shortcomings of high cost and long period of the experimental method. Therefore, this method has received more attention. The numerical simulation method of helicopter rotor is mainly divided into vortex method[2, 3] and computational fluid dynamics method[4]. The vortex method is relatively mature, and has good effects in the simulation and prediction of the aerodynamic characteristics of the conventional blade, and the calculation efficiency is high. However, since the vortex method is an approximation based on the potential flow hypothesis, it is difficult to accurately simulate the aerodynamic characteristics and flow field of modern helicopter rotors, especially the detailed characteristics of the flow field near the tip of the blade. The CFD method based on the N-S equation can capture the induction of the wake including the interaction of the tip vortex and the continuous blade as part of the entire flow field without specifying any wake model, and the calculation results can meet the accuracy requirements of the new blade flow field simulation, so the CFD method has become a new research hotspot. In particular, this method has unique advantages in the analysis of new blades or blades with strange shapes[5].

In the hovering condition of a single-rotor helicopter, there is no asymmetry of the pulling force and the moment caused by the asymmetry of the airflow on the rotating plane compared to the forward flight, so there is no flapping motion, shimmy motion and periodic variable pitch motion of the blade.
Therefore, a variety of methods can be used for this condition, including momentum source method, MRF method, sliding mesh method, and overset mesh method. The momentum source method is based on the disc assumption, and there is no precise three-dimensional model of the blade, so it is difficult to consider the transport of the vorticity and the viscosity of the gas. Compared with the momentum source method, the MRF, sliding mesh and overset mesh are more detailed and more close to the actual situation. Therefore, they have been widely used in rotor flow field simulation in recent years[6].

This paper mainly discusses the problems such as calculation accuracy and computational efficiency of MRF, sliding mesh and overset mesh for helicopter under hovering conditions. The calculation results of different methods are compared with the wind tunnel test data to obtain the characteristics and application scope of each method.

2. Grid system

2.1. Mesh generation of MRF and sliding mesh

The MRF method is mainly used for the steady state calculation of CFD. In this paper, the rotating reference frame system is mainly used. The solution provided by this method represents the time-averaged behavior of the fluid, not the time-accurate behavior. When using the MRF technique, the flow area is generally divided into two or more calculation areas, and adjacent calculation domains are connected by one or more interfaces. In the calculation, different regions are assigned different coordinate systems, such as assigning a rotating coordinate system to the region containing the main rotor and assigning a static coordinate system to other flow field regions. When using this method, a constant grid flux determined by the appropriate conservation equation will be generated. The mesh flux is calculated based on the properties of the reference frame, rather than from the local motion of the mesh cell nodes. The MRF method does not change the position of the grid cell node.

It should be noted that the MRF can provide results with practical physical meaning only when the fluid in the surrounding volume is axisymmetric. The MRF cannot be used when any large component of the external flow rate is perpendicular to the axis of rotation. The main reason is that when any component of the surrounding fluid is perpendicular to the axis of rotation, a non-physical result proportional to the vertical component will result.

The sliding mesh method is used for the unsteady state calculation (transient analysis) of CFD, which implements time-accurate behavior by moving the region mesh nodes. When using the sliding mesh technique, as with the MRF method, the flow area is generally divided into two or more calculation areas, and adjacent calculation domains are connected by one or more interfaces. Unlike the MRF method, the motion domain slips along the interface during the calculation. At each time step, the position of the grid is recalculated and the relative position of the grid points on the interface is determined to allow flux to be delivered at the interface.

From the above analysis, it is found that the single-rotor helicopter CFD simulation using the MRF method or the sliding mesh method creates a separate area for the rotating paddle, which is the interface with the air domain that wraps it. It should be noted that the grids on both sides of the interface do not need to be aligned, but the grid scales on both sides cannot be too different, otherwise it will be difficult to ensure the accuracy of the interface flux transfer. In fact, in the process of simulation, the MRF method and the sliding mesh method can all use the same set of grids. Since the vertical flight field of the rotor has rotational symmetry, it is only necessary to take one blade and its corresponding calculation domain for the simulation. The influence of the other blades can be achieved by using periodic boundary conditions on the circumferential boundary. In this paper, the MRF method and the sliding mesh method adopt a three-dimensional body-fitted structure grid, in which the sliding region (motion reference frame region) grid is shown in Figure 1, and the static region and the sliding region assembly grid diagram are shown in Figure 2.
2.2. Overset mesh method
The overset mesh method can only be used for unsteady computing of CFD, for discrete computing domains with multiple different meshes that overlap each other in any way. Research involving overlapping meshes must have a background region that encloses the entire solution domain, as well as one or more smaller regions that contain the bodies in the domain.

Figure 3 provides an example of how the airfoil curve mesh is located in the background Cartesian mesh. Airfoil grids capture boundary layer, tip vortices and shock waves. The background mesh surrounds the airfoil mesh and takes the solution to the far field. Some background grid points are located inside the airfoil entity area and must be removed from the solution. After deletion, the hole area will remain inside the larger background mesh and create a set of boundary points called hole fringe points. The airfoil mesh inserts the data into the background mesh through the hole fringe points of the background, and the background mesh inserts the data into the airfoil mesh at the airfoil outer boundary points[7].

Figure 3. Overset mesh of a two-dimensional airfoil
The advantage of overset mesh is that it allows us to divide several subdomains according to the specific research object, generate meshes for each subdomain separately, and allow the subdomain grids to overlap each other. Therefore, the overset mesh method can simulate the blade boundary layer
flow and the rotor wake vortex with high precision, and has satisfactory calculation efficiency. It is generally considered to be a grid method suitable for rotor CFD calculation.

The overset mesh system of this paper is mainly composed of two parts; one is the nearly orthogonal O-H type body structure grid generated around the blade surface, as shown in Figure 4. The second is the background structure grid surrounding the body structure grid. The assembly diagram of the background grid and the body structure grid is shown in Figure 5. The disadvantage of the O-H mesh compared to the O-O mesh is that the paddle tip mesh transition is not smooth enough. The reason why the H-type grid is used in this paper is that the H-type grid has more applications in capturing the tip shock wave. The background grid is static, used to simulate the change of the entire flow field, and the information is transmitted on the hole boundary with the blade body structure grid, thereby completing the information update of the entire flow field. In order to facilitate the overlap of the two grids and to accurately capture the tip vortex, local encryption of the nearby area swept by the tip is required in the background grid. The size of the grid is as large as possible to the outer boundary grid size of the blade body structure grid. At the same time, it is necessary to moderately expand the far-field boundary of the background grid to reduce the influence of far-field boundaries.

Regardless of the grid format, it should be noted that accurately capturing the rotor wake requires multiple grid points in the vortex core to prevent the tip vortex from spreading and maintaining its strength. Observing Figures 1 and 4, it is not difficult to find that the area mesh is encrypted in the vicinity of the vortex core.

3. Governing equations
Considering the continuity equation, the momentum equation and the energy equation, we can obtain the Reynolds average Navier-Stokes equation of the three-dimensional, viscous, unsteady and compressible flow in the inertial coordinate system:

$$\frac{\partial}{\partial t} \iiint_V \vec{W} dV + \iint_S (\vec{F}_c - \vec{F}_v) \cdot \vec{n} dS = 0 \quad (1)$$

Where \( \vec{W} = [\rho, \rho u, \rho v, \rho w, \rho e]^T \) is a conserved variable, \( \vec{F}_c = (f, g, h) \) is the convective flux, and \( \vec{F}_v = (a, b, c) \) is the viscous flux. The specific expression is:
In viscous flux, the expression of stress is:

\[ \tau_{xx} = \frac{2}{3} \mu \left( 2 \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} - \frac{\partial w}{\partial z} \right), \]  
\[ \tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial v}{\partial y} + \frac{\partial u}{\partial x} \right), q_x = -K \frac{\partial T}{\partial x} \]  
\[ \tau_{yx} = \tau_{xy} \]  
\[ \tau_{yy} = \frac{2}{3} \mu \left( 2 \frac{\partial v}{\partial y} - \frac{\partial u}{\partial x} - \frac{\partial w}{\partial z} \right), \]  
\[ \tau_{yy} = \mu \left( \frac{\partial w}{\partial z} + \frac{\partial u}{\partial x} \right), q_y = -K \frac{\partial T}{\partial y} \]  
\[ \tau_{zz} = \frac{2}{3} \mu \left( 2 \frac{\partial w}{\partial z} - \frac{\partial u}{\partial x} - \frac{\partial v}{\partial y} \right), \]  
\[ \tau_{zz} = \mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right), q_z = -K \frac{\partial T}{\partial z} \]  

In order to obtain a closed system of equations, the relationship between pressure and temperature is introduced as follows:

\[ p = \rho (\gamma - 1) \left[ e - \frac{1}{2} (u^2 + v^2 + w^2) \right] \]  
\[ p = \rho RT \]
4. Numerical methods and boundary conditions

4.1. Numerical methods

4.1.1. Spatial discretization method. The grid elements of the rotor flow field are spatially discrete based on the finite volume method, and different interpolation formats are formed according to the different discrete methods of the non-viscous flux on the interface of the grid elements. The central format is difficult to simulate the unsteady characteristics of the rotor flow field because of the second-order accuracy in space and the addition of artificial dissipation. Therefore, this paper adopts the format of Roe in upwind style[11]. This format calculates the numerical flux by the approximate Riemann solution method and has strong shock capture capability.

Approximate Riemann solution in Roe format may lead to non-physical understanding. Therefore, the entropy correction method proposed by Harten is used to correct the eigenvalues in Roe format[12].

4.1.2. Time stepping procedures. The slip grid and overlap grid method solves the unsteady state of the rotor. For the solution of the unsteady flow field of the rotor, the time dispersion of the flow control equation adopts the dual time stepping method. This method introduces the concept of physical time and pseudo-time, and pseudo-time advancement in each physical time step is equivalent to solving a steady-state problem. Therefore, measures such as local time step and implicit residual smoothing in the steady solution can be applied to the calculation of the flow field, thus improving the efficiency of the calculation. Jameson[13] explained the method in detail.

4.2. Boundary condition

In order to accurately predict the effects of viscous convection on the flow field and rotor blade parameters, the flow of the turbulent boundary layer in laminar, semi-laminar and fully turbulent zones needs to be considered. The solution is to set a number of grid points in the viscous boundary layer so that the values near the wall satisfy $y^+ \approx 1$[14]. The innermost layer height of the mesh generated in this paper is less than $1.0 \times 10^{-5} c$, where $c$ is the chord length.

In the hovering state, if the far-field boundary condition is determined based on the static flow outside the calculation box, the flow rate into and out of the box is zero, and the blade is in a closed box atmosphere. When the rotor rotates, the airflow in the box will cycle. Through multiple calculations, it is found that it is difficult to obtain trustworthy calculation results by using such boundary conditions. The reason is that the box size must be very large under such boundary conditions, even beyond the memory limitations of today’s supercomputers. Therefore, another far-field boundary condition based on momentum theory is used: when there is a small computational domain around the rotor, a non-zero flow field boundary condition should be specified that allows the airflow to enter and exit the box without violating the conservation law. Strawn and Barth explain in detail the method of determining the boundary conditions[15].

The outflow boundary is in the circle with the radius $R/\sqrt{2}$ directly below the paddle, and the outflow velocity is as follows:

$$V_{out} = M_{tip}\sqrt{C_T}$$

(7)

The remaining boundaries are inflow, and the directions are all pointing to the center of the hub. The inflow velocity is as follows:

$$V_{in} = \frac{M_{tip}}{8}\sqrt{C_T(R/r_F)^2}$$

(8)

Where $M_{tip}$ is the Mach number of rotor tip, $C_T$ is the tension coefficient of the rotor, $R$ is the blade radius, and $r_F$ is the distance from the far field boundary point to the center of the hub.

Srinivasan et al.[16] used this boundary condition to obtain satisfactory results.
5. Examples and discussions
The Caradonna-Tung[17] single-rotor experimental model with experimental data was used as a verification example. The experimental model is a double-leaf, rigid rotor with no distortion. The blade is stretched from the NACA0012 airfoil section with an aspect ratio of 6.0. In this calculation, the pressure coefficient distributions of the upper and lower surfaces of the blade profile at 0.96R in four different hovering states were selected from the experimental data: 1) Mach number at the tip of the blade $M_{tip} = 0.815$, Pitch $\theta = 5^\circ$; 2) Mach number at the tip of the blade $M_{tip} = 0.439$, Pitch $\theta = 8^\circ$; 3) Mach number at the tip of the blade $M_{tip} = 0.877$, Pitch $\theta = 8^\circ$; 4) Mach number at the tip of the blade $M_{tip} = 0.794$, Pitch $\theta = 12^\circ$. The specific parameters of the rotor model are shown in Table 1:

| Parameter                      | Value         |
|-------------------------------|---------------|
| Number of blades (N)          | 2             |
| Rotor radius                  | 1.143m        |
| Plane shape of blade          | Rectangle     |
| Airfoil                       | NACA0012      |
| Chord length (c)              | 0.1905m       |
| Undercut ($R_{cut}$)          | 0.243R        |
| Twist                         | 0°            |

Figure 6 is the pressure contour of working condition 3 calculated by MRF method, and Figure 7 is the velocity contour of working condition 3 calculated by MRF method at 0.96R profile of blade. It can be seen that there is obvious shock wave near the tip under this working condition.

Figure 8 shows the comparison of the surface pressure coefficients obtained by MRF, sliding mesh and overset mesh methods with the experimental data, as follows:
It can be seen from the figure 8 that under the four working conditions, all the three methods can obtain reasonable calculation results. Among them, the accuracy of MRF method is worse than that of the other two methods, especially in the acquisition of shocks at propeller tip. However, the time required for convergence of MRF method is much shorter than that of sliding mesh and overset mesh. Sliding mesh method and overset mesh method can obtain more accurate results than MRF. The convergence time of the two methods is almost the same when the mesh number difference between the two grids is very small. Sliding mesh method needs to re-model and draw meshes in the rotating region when changing the total pitch angle. Therefore, the workload is large and heavy. The overset mesh method can change the pitch angle by simply rotating the mesh in the moving area, and the workload is relatively small.

When calculating the rotor forward flight conditions, due to the complex motion forms such as periodic pitch variation, neither MRF nor sliding mesh method can accurately calculate the rotor forward flight conditions. The overset mesh method can be used to precisely define the forward flight state of the rotor by specifying the angular velocity of the mesh around the variable pitch axis in the rotating region and the initial pitch angle of each blade. Therefore, the overset mesh method has a wider scope of application.

In conclusion, the MRF method is suitable for cases where only the aerodynamic performance of the rotor in hover state is concerned and the accuracy requirement is not high. For example, in the initial stage of engineering design, the reasonable calculation results can be obtained quickly. The sliding mesh method is suitable for further improving the accuracy of calculation on the basis of MRF method. It can directly take the results of MRF method as the initial solution and use the grids of MRF method to calculate. The overset mesh method is applicable to the aerodynamic performance of the
rotor in hover, forward flight and other conditions. Compared with MRF method, the calculation amount of overset mesh method is larger, and the calculation results in hover condition are more accurate than that of MRF method.

In addition to the influence of different mesh methods, the aerodynamic simulation of the rotor is also affected by factors such as mesh quality and turbulence model. For unsteady calculations, the size of the time step will also affect the result. In this paper, the analysis of the three methods makes the other influencing factors as consistent as possible, but the rigor is still inevitable.

Acknowledgments

This research was financially supported by the National key research and development project of China (No.2017YFD0701004). Thanks to Professor Chen Ming from the Beihang University, School of Aeronautic Science and Engineering for his valuable comments.

References

[1] Costes M, Renaud T and Rodriguez B 2012 Rotorcraft simulations: a challenge for CFD J. International Journal of Computational Fluid Dynamics 26 383-405
[2] Bagai and Ashish 1995 Rotor free-wake modeling using a pseudo-implicit technique - including comparisons with experimental data J. Journal of the American Helicopter Society 40 29
[3] Bagai A, Leishman J G and Park J 1999 Aerodynamic analysis of a helicopter in steady manoeuvring flight using a free-vortex rotor wake model J. Journal of the American Helicopter Society 44 109
[4] Cao Y H, Wu Z L and Huang J S 2014 Numerical simulation of aerodynamic interactions among helicopter rotor, fuselage, engine and body of revolution J. Science China Technological Sciences 57 1206-18
[5] Srinivasan G R, Raghavan V and Duque E P N 1993 Flowfield analysis of modern helicopter rotors in hover by navier-stokes method J. Journal of the American Helicopter Society 38 3
[6] Kang H J and Kwon O J 2002 Unstructured mesh navier-stokes calculations of the flow field of a helicopter rotor in hover J. Journal of the American Helicopter Society 47 90
[7] Ahmad J and Duque E P 1996 Helicopter rotor blade computation in unsteady flows using moving overset grids J. Journal of Aircraft 33 54-60
[8] Baeder J D, Gallman J M and Yu Y H 1997 A computational study of the aeroacoustics of rotors in hover J. Journal of the American Helicopter Society 42 39
[9] Wesseling P 2001 Principles of computational fluid dynamics J. Springer 22 3-9
[10] Spalart P R and Allmaras S R 1994 A one-equation turbulence model for aerodynamic flows J. Recherche Aerospatiale 1 5-21
[11] Roe P L 1997 Approximate Riemann solvers, parameter vectors, and difference schemes J. Journal of Computational Physics 135 250-258
[12] Harten A and Hyman J M 1983 Self adjusting grid methods for one-dimensional hyperbolic conservation laws J. Journal of Computational Physics 50 235-269
[13] Jameson A 1991 Time dependent calculations using multigrid, with applications to unsteady flows past airfoils and wings 10th Computational Fluid Dynamics Conference
[14] Kang H J and Kwon O J 2002 Unstructured mesh navier-stokes calculations of the flow field of a helicopter rotor in hover J. Journal of the American Helicopter Society 47 90
[15] Strawn R C and Barth T J 1993 A finite-volume Euler solver for computing rotary-wing aerodynamics on unstructured meshes J. Journal of the American Helicopter Society 38 61-67(7)
[16] Srinivasan G R, Raghavan V, Duque E P N and Mcroskey W J 1993 Flowfield analysis of modern helicopter rotors in hover by navier-stokes method J. Journal of the American Helicopter Society 38 3
[17] Caradonna F X and Tung C 1981 Experimental and analytical studies of a model helicopter rotor in hover J. Vertica 5 149-161