Numerical Simulation and Analysis of Aerodynamic Interaction of Helicopter Rotor

Xiang TAO¹,* and Qi WANG¹

¹School of Aircraft Engineering, Nanchang Hangkong University, Nanchang 330063, China
*Corresponding author

Keywords: Rotor, Aerodynamic interference, CFD.

Abstract. Problems exist aerodynamic interference between the rotor blades, which directly affects the performance of the rotor. Approximation to paddle around two parallel movements for analysis using CFD method based on N-S equations, analyzed the mechanism of the aerodynamic interference between the rotor blade, simulated the different Angle of attack, Mach number and the influence of wings spacing to interfere with the flow field. Results show that: the behind blades influenced by the front blades changes with the angle of attack and Mach number showed a favorable or unfavorable influence, the smaller influencing distance is, the greater influencing intensity is. These conclusions will be of theoretical support for the design of the rotor.

Introduction

The numerical simulation method of rotor flow[1] generally have wake method and computational fluid dynamics method[2-7](CFD method). Using the traditional wake method in engineering practice was widely used, but due to the defects of the theory itself, details of the flow field, especially characteristics of transonic flow simulations in the presence of great difficulty, so difficult for new rotor air operated layout design. The principle of the CFD method is that it can capture the details of the transonic flow field. So frequently used in the rotor flow field numerical simulation.

As the rotation of the helicopter rotor lifting surface, its each slice blade in previous blade wake, at the same time its wake impact on the flow field of the next blade, namely each blade is "front wing", and "hind wings". In order to get the influence of different angles and different velocity, the rotor is simplified as two parallel blades, and analyzed the interference between the two-dimensional parallel blades. The influence of different conditions on aerodynamic parameters of blades is obtained, which provides data support for rotor design.

Numerical Simulation of Flow Field

Simulation Conditions

The airfoil is NACA0012. In the calculation of the before and after the two wings between the parallel interference phenomenon using unstructured grid. In order to reduce the computation time and ensure the accuracy, grid set encryption area, airfoil wall from the outer boundary distance is 10C (C for airfoil chord), maximum grid size is 0.25C; airfoil wall from the inner boundary distance is 3C, maximum grid size is 1/30C; In order to observe the airfoil nearly wall under surface pressure and velocity, on the surface of the airfoil set the boundary layer mesh, boundary layers is four. The total numbers of meshes are about 138000, and the grid is shown in Fig. 1:
The calculation conditions include: select the two-dimensional axial symmetry based on the pressure implicit steady-state solver, because the flow is compressible flow, so choice the energy equation. Select Spalart-Allmaras equation turbulence model, define the material as the ideal gas, Viscosity is Sutherland; the far field boundary condition is the pressure-far-field, far field pressure $p=101325\text{Pa}$; density $\rho=1.177\text{kg/m}^3$; far field temperature $T=300\text{K}$. The iterative method for solving the flow field is SIMPLE algorithm.

This computational method can use in the different angle of attack, Mach number and paddle distance.

**Flow Field Analysis**

The calculation conditions are as follows: angle of attack is $6^\circ$; Mach number is $0.6$; $4.0C$ distance, velocity contours, pressure contours and vector streamline diagram obtained in Fig. 2 to Fig. 4.

From Fig.2 and Fig. 3, we can see that velocity and pressure are different, the pressure of behind wing upper surface is greater than the front wing, the pressure of lower surface is less than the front.
wing, especially in front on the upper surface, wing area of low pressure is less than the front wing. From Fig. 4 can see that the flow velocity of behind wing is basically same with the front wing, so the drag coefficient is basically same, but part of the top surface of the front wing flow will flow under the behind wing lower surface, thus making the behind wing lower surface velocity decreased, reduce the pressure, lift coefficient decreased.

<table>
<thead>
<tr>
<th></th>
<th>Front wing</th>
<th>Behind wing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lift coefficient</td>
<td>0.7802</td>
<td>0.66927</td>
</tr>
<tr>
<td>Drag coefficient</td>
<td>0.02939</td>
<td>0.03451</td>
</tr>
<tr>
<td>Lift-drag ratio</td>
<td>26.546</td>
<td>19.393</td>
</tr>
</tbody>
</table>

From Table 1 can see that affected by the front wing, the lift drag ratio of the behind wing is changed, in 0.6Ma and angle of attack 6°. The lift to drag ratio of the behind wing is less than the front wing. The lift coefficient of the behind wing is 14% lower than that of the front wing, and the drag coefficient increases by 17%, lift-drag ratio decrease by 27%. It shows that they have a strong aerodynamic interference.

The Influence of Parameters on the Flow Field

Aerodynamic Interference in Different Angles of Attack

This section studied in 4 times of the airfoil spacing, flow velocity is 0.2Ma, angle of attack from 2° to 12°, then get the aerodynamic characteristics of the front wing and the behind wing, shown as Fig. 5-7.

From Fig. 5 to Fig. 7 can see that in 4 times of the airfoil spacing and flow velocity is 0.2Ma, the aerodynamic interference of the front wing to the behind wing is different, in the $\alpha$=2°~9° lift coefficient of the behind wing will be less than front wing, the drag coefficient is greater than the front wing, at high angle of attack 10°~12°, lift coefficient of the behind wing is greater than the front wing, drag coefficient is less than the front wing. Through the lift-drag ratio know that at the angle of attack for 2°~9° wing lift-drag ratio is less than the front wing lift-drag ratio, and in the 5°, 6° and 7° value of maximum disturbance is strong, in the angle of attack range two wing interference is unfavorable; at the angle of attack for 10°~12° wing lift-drag ratio is greater than the front wing, which is explained in the angle of attack range two wing interference is favorable. Compared the undisturbed blade and the disturbed blade, the latter is smaller than the change of the aerodynamic characteristic with the change of the angle of attack.
Aerodynamic Interference in Different Mach Number

In order to compare the aerodynamic interference characteristics of different mach number, select 4 times airfoil spacing, angle of attack is 6°, flow velocity from 0.2Ma to 0.8Ma, then get the aerodynamic characteristics of the front wing and the behind wing, shown as Fig. 8-10.

From Fig. 8 to Fig. 10 can see that in 4 times the airfoil spacing, angle of attack is 6°, the aerodynamic interference of the front wing to the behind wing is different, as the incoming flow velocity 0.2Ma~0.6Ma, lift coefficient of the behind wing will be less than the front wing, drag coefficient of the behind wing is greater than the front wing; at 0.7Ma and 0.8Ma, lift coefficient of the behind wing is larger than that of the front wing, drag coefficient of the behind wing is less than the front wing. Through the lift-drag ratio know that the front wing lift-drag ratio is greater than the rear wing in 0.2Ma~0.6Ma, which is explained under this mach number, aerodynamic interaction is unfavorable; the behind wing lift-drag ratio is greater than the front wing in 0.7Ma and 0.8Ma, which is explained under this mach number, aerodynamic interaction is favorable.

Compared the undisturbed blade and the disturbed blade, the latter is smaller than the change of the aerodynamic characteristic with the change of mach number.

Aerodynamic Interference in Different Airfoil Spacing

This section establishes four different models and the airfoil spacing are 10 times, 15 times and 20 times. Mach number is 0.2, angle of attack is 6° (by above know that the angle of interference is most), then get the aerodynamic characteristics of the front wing and the behind wing, shown as the following table:

<table>
<thead>
<tr>
<th>Spacing</th>
<th>Cl</th>
<th>Cd</th>
<th>Cl/Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>front wing</td>
<td>0.07346</td>
<td>0.0027</td>
<td>27.207</td>
</tr>
<tr>
<td>4 times</td>
<td>0.06349</td>
<td>0.00321</td>
<td>19.778</td>
</tr>
<tr>
<td>10 times</td>
<td>0.06785</td>
<td>0.00294</td>
<td>23.078</td>
</tr>
<tr>
<td>15 times</td>
<td>0.06954</td>
<td>0.00285</td>
<td>24.4</td>
</tr>
<tr>
<td>20 times</td>
<td>0.07203</td>
<td>0.00277</td>
<td>26.003</td>
</tr>
</tbody>
</table>

From the Table 2, with the increase of interference spacing, lift coefficient and drag coefficient of the behind wing is gradually close to the front wing, described the interference from the front wing gradually weakened.
Conclusions

In this paper, a numerical simulation method for the flow field of parallel interference is established, and analyzed the influence of the flow field characteristic, summarize the conclusion as follows:

Blade in continuous rotary motion, the forward blade wake sustained impact on the retreating blade, because the forward blade interference, the surface velocity and pressure of the retreating blade has changed, affected his lift coefficient and drag coefficient.

Under the same interference spacing, when angle of attack and mach number are different, aerodynamic interference is also different. In small angle of attack and small mach number, aerodynamic interference is unfavorable; in high angle of attack and high mach number, aerodynamic interference is favorable.

With the increase of the number of blades, the interference distance between two blade decreases, the forward blade has stronger interference to the retreating blade.

Literature References


