Predicting Aerodynamic Rotor-Fuselage Interactions by Using Unstructured Meshes

By Jong-Kook LEE and Oh Joon KWON

1) Agency of Defense Development, Taejon, Korea
2) Department of Aerospace Engineering, Korea Advanced Institute of Science and Technology, Taejon, Korea

(Received March 8th, 2001)

A numerical method has been developed for the analysis of interactional aerodynamics between helicopter rotor and fuselage in forward flight. A 3-D steady compressible Euler solver is used to compute time-averaged interactional effects between the rotor and the fuselage. The rotor is modeled as an actuator disk with zero thickness carrying pressure jump across it. Collective and cyclic pitch angles are calculated to satisfy the rotor trim condition. Unstructured meshes are used to model complex rotor-fuselage configurations. Calculations are performed for two generic helicopter fuselage geometries. The results are compared with available experimental data for validation.

Key Words: Helicopter Aerodynamics, Rotor-Fuselage Interaction, Unstructured Meshes, Actuator Disk Modeling

1. Introduction

The flow around helicopter rotors in forward flight is very complex because of the unsteady, asymmetric loading characteristics over the rotor disk, even for steady forward flight conditions. Interaction between the rotor and the helicopter fuselage further amplifies the complexity of the flow as shown in Fig. 1. A large portion of the helicopter fuselage is usually submerged in the wake of the rotor, which can change the overall aerodynamic characteristics of performance. The effect of rotor wake on the fuselage can also lead to problems such as fuselage vibration and aeroacoustic noise. Thus the mutual effect of rotor and fuselage must be evaluated accurately for the proper assessment of helicopter performance at the design stage of helicopters. Even though the performance of isolated rotor and isolated fuselage is analyzed separately, the interactional aerodynamics around the rotor-fuselage combination cannot be obtained by a simple linear superposition of each individual result because of the inherent nonlinear behavior of the flow.

Traditionally, the effect of rotor wake on the fuselage has been studied by using singularity methods. Freeman modeled the fuselage as a discrete distribution of source panels and represented the rotor as concentric vortex tubes. Marvis et al. studied the rotor-fuselage interaction by using coupled source and doublet distributions for the fuselage and a lifting-line/free-wake model for the rotor. Lorber and Egolf used a lifting-line/prescribed wake model to represent the rotor and source panels for the fuselage to study the unsteady interactional effect. Quackenbush et al. used an analytical-numerical matching technique to predict the fuselage surface pressure under the influence of the rotor.

The recent development of high-speed computers enables the use of Euler and Navier-Stokes methods for the prediction of helicopter rotor-fuselage interactional problems. Whitfield and Jameson used the Euler equations for analyzing the performance of propellers by introducing a source term in the equation. Rajagopalan and Mathur replaced the rotor by a momentum source to solve the rotor flow field by using the incompressible laminar Navier-Stokes equations. Zori and Rajagopalan introduced an overlapping grid methodology for solving the rotor-fuselage interference problems. Hariharan and Sankar introduced an overlapping grid methodology for solving the rotor-fuselage interference problems. Chaffin and Berry solved the rotor-fuselage interactional aerodynamic problem by modeling the rotor as an actuator disk and solving an incompressible Navier-Stokes code on structured Chimera grids.

Meanwhile, several experimental studies have been performed not only for the understanding of the complicated rotor-fuselage interactional aerodynamic phenomena, but also for providing useful data for the validation of the previously described numerical approaches. Unsteady and time-averaged flow behavior and fuselage surface pressure measurements were conducted at Georgia Tech for a generic cylindrical airframe under a two-bladed rotor system in forward flight. Experimental results have also been reported for a generic fuselage configuration in the form of fuselage surface pressure distribution and inflow at the rotor disk plane.

In the present study, an attempt is made for the prediction of rotor-fuselage interactional aerodynamic problems, using a 3-D Euler solver based on unstructured meshes. Unlike multiple overlapping structured grid topology, the flow field around complex rotor-fuselage configurations can be modeled as a simple single block grid with unstructured meshes. The time-averaged mean flow of the unsteady rotor in forward flight is calculated by modeling the rotor as an actuator
disk carrying pressure jump. The effect of rotor wake on the fuselage is predicted in the form of time-averaged surface pressure on the fuselage and induced inflow distribution at the rotor disk plane. The results are compared with available experimental and computational data for validation.

2. Computational Method

The present Euler solver uses a cell-centered finite-volume scheme to discretize the inviscid flux based on Roe’s flux-difference splitting on unstructured meshes. To achieve second-order accuracy, the estimation of the flow variables at each cell face is calculated by interpolating the solution with a Taylor series expansion in the neighborhood of each cell centroid. The cell-averaged solution gradient required at the cell centroid for the above expansion is computed by using Gauss’s theorem, evaluating the surface integral for the closed surface of each tetrahedron. This process can be simplified by using some geometrical invariant features of the tetrahedra. The expansion also requires nodal values of the solution, which can be computed from the surrounding cell centroid data by using a second-order accurate pseudo-Laplacian averaging procedure as suggested by Holmes and Connell.

For steady-state computations, the governing equations are linearized and advanced in time by using the first-order Euler backward difference. The Jacobian matrix that arises from the linearization process is a very large sparse matrix, since the stencil of high-order flux discretization at each cell face is based on its nodes and neighboring cell centroids. To reduce the size of the stencil and to enhance the diagonal dominance for numerical stability, only a first-order upwind approximation of the inviscid flux is used in the present implementation. A direct solution of the system of simultaneous equations for entire cells requires the inversion of a large spare matrix, which is computationally very expensive. Instead, a Gauss-Seidel relaxation approach is used to iteratively solve the system of equations. A cell-coloring scheme is implemented by regrouping cells so that no two in each group share a common face. The cells belonging to the first group are solved by the use of a Jacobi-type iteration. The cells in the subsequent groups are solved by the use of the Gauss-Seidel iteration. To accelerate the convergence of the solution to a steady state, local time stepping is also used.

At the solid wall, the slip boundary condition is used for the present inviscid flows. The flow tangency condition is implemented by imposing no flux through the wall. Density and pressure are extrapolated from the interior. At the far-field boundary, a characteristic boundary condition is applied by using the fixed and extrapolated Riemann invariants corresponding to the incoming and outgoing waves. The boundary condition applied to the rotor disk is described in the following section.

3. Rotor Disk Model

In the present study, the rotor is modeled as an actuator disk with zero thickness, which represents an imaginary disk carrying pressure jump between the upper and lower surfaces while allowing the flow to pass through. This approach reduces the computational resource requirement and eliminates the computational complexity of modeling each blade and performing time-accurate calculations. In the actual computation, the disk plane is represented by a finite number of triangles inside the computational domain, which exactly match the triangular surfaces of the tetrahedral cells for the present unstructured mesh methodology.

The pressure difference between the upper and lower disk surfaces is calculated by using the blade element theory so that the summation of the pressure difference for the entire disk matches the total thrust of the rotor. The local flow velocity, $\mathbf{V}$, is continuous through the rotor disk plane, and its magnitude and direction are known as a part of the computation. Then the induced angle of attack, $\alpha_i$, at the center of each triangular element on the rotor disk can be calculated as

$$\alpha_i = \tan^{-1} \frac{\mathbf{V} \cdot \hat{n}}{\mathbf{V} \cdot \hat{r}} \quad (1)$$

where $\mathbf{V} \cdot \hat{n}$ is the velocity component in the direction normal to the rotor disk plane, as in Fig. 2. $\mathbf{V} \cdot \hat{r}$ is the velocity component parallel to the rotor disk plane, which includes contributions from the local flow velocity and the rotor rotational speed:

$$\mathbf{V} \cdot \hat{r} = \bar{\mathbf{V}} \cdot \hat{r} + (\bar{\mathbf{f}} \cdot \hat{\mathbf{e}}) \times \hat{p} \quad (2)$$

where $\bar{\mathbf{r}}$ is the distance from the rotor axis of rotation to a point on the blade. $\hat{p}$ represents the unit vector parallel to the blade path. The effective angle of attack, $\alpha_e$, on the blade is then

$$\alpha_e = \Theta - \alpha_i \quad (3)$$
The pitch angle of the blade can be represented in terms of the blade azimuth angle, $\Psi$:

$$\Theta = A_0 - A_1 \cos \Psi - B_1 \sin \Psi - \frac{r}{R} \alpha_i \tag{4}$$

where $A_0$ is the blade collective pitch setting; $A_1$ and $B_1$ are the coefficients of lateral and longitudinal cyclic pitch angles; and $\alpha_i$ is the blade twist angle at the tip.

Since the velocity relative to the blade is known, elemental lift and drag acting at each section of the rotor blade can be calculated as

$$\Delta L = \frac{1}{2} \rho V^2 C_l \Delta A, \quad \Delta D = \frac{1}{2} \rho V^2 C_d \Delta A \tag{5}$$

The elemental area, $\Delta A$, represents each triangle on the rotor disk. Thus the elemental lift and drag calculated above are the aerodynamic forces obtainable from each triangular element during one revolution of the rotor. The lift and drag coefficients are determined by using the 2-D airfoil theory. For $N$ blades of the rotor, the elemental thrust, $\Delta T$, for each triangle is scaled by a time factor, $N(d\Psi/2\pi)$, to obtain time-averaged contribution while the rotor sweeps the azimuth angle of $d\Psi$.

$$\Delta T = N \frac{d\Psi}{2\pi} (\Delta L \cos \alpha_i - \Delta D \sin \alpha_i) \tag{6}$$

For the present unstructured triangular surface mesh, the thrust expression is modified as

$$\Delta T = \frac{N \Delta A}{4\pi r} \cos \rho V^2 (C_l \cos \alpha_i - C_d \sin \alpha_i) \tag{7}$$

The difference in pressure between the upper and lower surfaces of the rotor disk plane can be represented as

$$\Delta P = \frac{\Delta T}{\Delta A} \tag{8}$$

The upper surface of the rotor disk is considered to be an exit of the flow, and the pressure is calculated by extracting the pressure difference from the pressure of the lower surface. The velocity on the upper disk is set equal to the velocity on the lower surface to maintain continuity. The lower surface of the rotor disk is considered to be an inlet of the flow, and the pressure is extrapolated from the interior of the computational domain.

4. Computational Details

The present numerical method has been applied to two experimental geometries for validation. The first case, which has a two-bladed rotor and a generic cylindrical fuselage configuration, was tested at Georgia Tech.$^{10,11}$ The second validation is made for the configuration tested at the NASA Langley Research Center with a more realistic configuration known as ROBIN (ROTor Body INteraction) fuselage.$^{12-14}$ Calculations are made for both fuselage-alone and rotor-fuselage configurations. The computational domain is stretched 11 times the rotor radius upstream and downstream, 8 times sideward, and 9 times in a vertical direction for both. Comparisons are made for the time-averaged pressure around the fuselage and the inflow distribution at the rotor disk plane.

After each time iteration, the local induced angle of attack is calculated at the centroid of each triangle on the rotor disk as in equation (1), using the flow velocity, the rotor rotational speed, and the rotor geometric information. The local effective angle of attack is then obtained from the known blade pitch angle and the given rotor trim information. Thus the elemental lift and drag on each triangle are calculated, which results in the time-averaged contribution of thrust and the pressure jump as in equations (7) and (8). This pressure jump is used to update the rotor disk boundary condition. Other boundary conditions are also updated as necessary, and the calculation is advanced in time to the next time step. This procedure is continued until satisfactory results are obtained for the flow convergence and the total rotor thrust.

If necessary, the rotor trim is calculated to update the blade collective pitch setting and the lateral and longitudinal cyclic pitch angles so that the moment summation at the hub cancels out. This calculation requires an iterative procedure by solving the coupled trim equations, which are well-known formulas expressed in terms of the rotor advance ratio, inflow ratio, blade coning angle, and others.$^{18}$

5. Results and Discussion

5.1. Georgia Tech configuration

The rotor-fuselage configuration tested at Georgia Tech has a two-bladed rotor and a cylindrical fuselage. The blade is made of an NACA0015 airfoil section. The rotor operates at an advancing ratio of 0.1 with the freestream Mach number of 0.03. The blade collective pitch angle is set at 10 degrees, and the cyclic variation is not enforced. The rotor thrust coefficient is approximately 0.0091. The rotor shaft has a front tilt angle of 6 degrees.

At first, the fuselage-alone case was calculated for the validation of the present solver. Figure 3 shows the surface triangulation on the cylindrical fuselage. The mesh contains 115,161 cells and 21,082 nodes, out of which 3,126 node points are distributed on the fuselage surface. The result is compared with experiment$^{11}$ and the computational result based on a singularity method$^{20}$ as shown in Fig. 4. A
good comparison is made with other results showing a stagnant flow at the nose and the subsequent acceleration. At the midsection of the fuselage, the experimental result shows a dip in pressure, which is due to the interference with a rotor hub closely situated above the fuselage for blade attachment. The present result shows a flat pressure distribution without modeling the hub. The prediction based on the singularity method without the hub shows an almost identical result with the present calculation.

Figure 5 shows the surface triangulation of the rotor-fuselage configuration. The mesh contains 139,376 cells and 25,068 node points. It is shown that the rotor disk is represented by several triangles, as explained earlier. To validate the time-averaged rotor disk modeling of the present method, the inflow distribution beneath the rotor is compared with the experiment\(^{10}\) at four azimuthal positions of the rotor in Fig. 6. Fair agreement is obtained at all azimuthal positions between the present result and the experiment, even though a slight difference in magnitude is observed near the tip of the rotor because of the lack of the strong concentrated tip vortex modeling for the present method.

Figure 7 compares the time-averaged velocity component distributions along the top of the fuselage. Again, good agreement is obtained between the present prediction and the experiment except at the nose section of the fuselage because of the underprediction of the tip vortex contraction of the present method. The result based on a singularity method\(^{2}\)
shows a slightly better comparison than the present method by modeling the realistic unsteady tip vortex at the expense of computational complexity.

In Fig. 8, the time-averaged pressure distribution along the top of the fuselage is presented at various advancing ratios. Two high-pressure peaks at relatively low advancing ratios are due to the blade passage effect and the impingement of the front and rear tip vortices on the fuselage. Even though the present method shows slight differences in the magnitude and location of the peaks, the overall agreement between the present result and the experiment is considered to be good for the simplified rotor model incorporated in the present method. As the advancing ratio increases, the magnitude of pressure variation diminishes, representing less rotor-airframe interaction.

5.2. ROBIN configuration

The ROBIN configuration is designed for the experimental study of rotor-fuselage interactive aerodynamics. The fuselage has a more realistic configuration, including nacelle, than the Georgia Tech geometry, but can still be described with a simple formula. The rotor has four blades made of a NASA RC-10-(B) M002 airfoil section. The as-

Fig. 7. Time-averaged velocity distribution along the top of the fuselage.

Fig. 8. Time-averaged pressure distribution along the top of the fuselage at various advancing ratios.
pect ratio of the blade is approximately 14.6. The rotor shaft has a front tilt angle of 2 degrees.

Figure 9 shows the surface triangulation on the ROBIN fuselage-alone configuration. The total number of cells consists of 239,782 tetrahedra and 42,806 node points, of which 3,753 are distributed on the body surface. Small cells are distributed near the nose and around the sharp corners of the fuselage for accurate capturing of the complicated flow behavior.

The time-averaged surface pressure distribution on the fuselage-alone configuration is shown in Fig. 10 at four streamwise stations around the fuselage at the freestream Mach number of 0.125. It is shown that the predicted pressure distributions from the present flow solver compare well with the experimental result, demonstrating the capability of the present solver for predicting flow around complex configurations. The result is also compared with other calculations based on the structured grid method and the singularity method. Almost identical results are obtained for all three methods, confirming the accuracy of the present approach.

Surface triangulation on the combined rotor-fuselage con-
configuration is shown in Fig. 11. For this configuration, 255,071 tetrahedral cells and 45,260 node points are used, of which 3,446 are distributed to represent the rotor and fuselage surfaces. The rotor disk plane is again modeled with several triangular elements. The rotor has a 20% root-cutout, which is excluded from surface modeling, as shown in the figure.

In Fig. 12, the time-averaged surface pressure distributions at four streamwise stations of the fuselage are shown for the combined rotor-fuselage configuration on the advancing side of the fuselage at the rotor-advancing ratio of 0.2 and the freestream Mach number of 0.116. A good comparison is obtained between the present calculation and the experiment not only near the nose of the fuselage, but also further downstream where the rotor wake effect becomes more significant. The overall pressure shows a slightly higher value than the fuselage-alone configuration because of the impingement of wake carrying higher energy added by the rotor.

In Fig. 13, the time-averaged downwash inflow velocity is normalized by the rotor tip rotational speed and compared with the experimental data at four azimuthal positions for the rotor-fuselage configuration at the rotor-advancing ratio of 0.22 and the freestream Mach number of 0.126. The downwash inflow is defined as the velocity component normal to the rotor disk after the freestream contribution has been subtracted. The results compare well with the experimental data at all azimuthal positions, which demonstrates the validity of modeling the rotor as an actuator disk. The effect of flow acceleration, energy addition, and the time-averaged distortion of the flow is well modeled on the present unstructured mesh methodology.

Figure 14 represents the typical side and rear views of the numerical flow visualization around the rotor and the fuselage in the form of velocity vectors and particle traces. It is clearly shown that the fuselage is almost completely submerged in the wake of the rotor demonstrating the strong influence of the rotor on the fuselage. It is also shown from the rear view of the flow that the present approach approximately simulates the vortical flow structure at the tip of the rotor. The acceleration and turning of the flow through the rotor disk are also well captured in the side view.

The rotor trim analysis is performed for the present configuration at the freestream Mach number of 0.196 to match the experimental rotor thrust coefficient of 0.0064. The collective and cyclic pitch settings are compared with the experiment and other analytical results in Table 1. These collective and cyclic pitch angles are calculated by iteratively adjusting the values as the calculation proceeds. The total of 650 time iterations is performed, including 4 cycles of inner trim

<table>
<thead>
<tr>
<th>Table 1. Comparison of trim calculations.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
</tr>
<tr>
<td>$A_0$</td>
</tr>
<tr>
<td>$A_1$</td>
</tr>
<tr>
<td>$B_1$</td>
</tr>
</tbody>
</table>
iterations to obtain the converged solution. The result shows a reasonable agreement with the experiment within the accuracy of the assumptions used for the present calculation. The result obtained by the structured overset grid based on a Navier-Stokes simulation shows slightly higher values than the present method.

6. Conclusion

A time-averaged rotor disk model compatible to unstructured meshes is proposed to simulate interactions between the helicopter rotor and the fuselage. An actuator disk model of the rotor sustains a nonuniform pressure jump across the rotor upper and lower surfaces while allowing the flow to pass through. Calculations are made for flows around generic fuselage configurations with and without the effect of the rotor. The results compare well with the experiment for both pressure distribution on the fuselage surface and inflow distribution at the rotor disk plane. It is demonstrated that the present method is a useful tool to predict the complicated rotor-fuselage interactional aerodynamic phenomena in an efficient time-averaged manner.
References