Detailed Aerodynamic Analysis of a Shrouded Tail Rotor Using an Unstructured Mesh Flow Solver

By Hee Dong LEE and Oh Joon KWON

Division of Aerospace Engineering, Korea Advanced Institute of Science and Technology, Daejeon, Korea

(Received March 20th, 2003)

The detailed aerodynamics of a shrouded tail rotor in hover has been numerically studied using a parallel inviscid flow solver on unstructured meshes. The numerical method is based on a cell-centered finite-volume discretization and an implicit Gauss-Seidel time integration. The calculation was made for a single blade by imposing a periodic boundary condition between adjacent rotor blades. The grid periodicity was also imposed at the periodic boundary planes to avoid numerical inaccuracy resulting from solution interpolation. The results were compared with available experimental data and those from a disk vortex theory for validation. It was found that realistic three-dimensional modeling is important for the prediction of detailed aerodynamics of shrouded rotors including the tip clearance gap flow.

Key Words: Aerodynamic Performance, Shrouded Tail Rotor, Computational Aerodynamics, Unstructured Meshes

Nomenclature

$C_{T_{\text{tot}}}$: total thrust coefficient
 $\dot{m}$: mass flow rate
 M_{tip} : tip Mach number
 p : fluid static pressure
 R : rotor radius
 S : rotor disk area
 T_{rot} : rotor thrust
 T_{sh} : shroud thrust
 T_{tot} : total thrust
 U : inertial velocity component normal to control volume surface
 U_r : relative velocity component normal to control volume surface
 u, v, w : Cartesian velocity components in inertial frame
 V_w : velocity at outflow boundary
 V_{in} : normalized velocity at inflow boundary
 V_w : normalized velocity at outflow boundary
 v_i : induced velocity at rotor disk plane
 ρ : fluid density
 σ : area ratio of outflow boundary to rotor disk plane
 Ω : rotor angular velocity

1. Introduction

For conventional helicopters operating on a single main rotor, a tail rotor is required to compensate the torque reaction of the main rotor and to obtain yaw stability and directional control. Traditionally, an open-type tail rotor system has been widely used due to its simplicity for manufacturing and maintenance. Even though this tail rotor system is a very effective means to produce necessary control force for pilots, it is one of the major sources of helicopter noise, and also frequently causes concerns about the security and reli-

ability from failure. The NOTAR (No Tail Rotor) system utilizing the Coanda effect and ducted air flow as the anti-torque has also been developed to eliminate the difficulty associated with the tail rotor. However, this system requires a complex fan assembly and extra power for driving flow through the tail boom duct. Recently, some of the major helicopter companies have developed shrouded (or ducted) tail rotor systems as a replacement of the conventional open-type rotor.

Unlike the conventional open-type tail rotor, the shrouded tail rotor has been developed to reduce the risk of component damage and to enhance the operational safety and the security of the personnel involved in helicopter operation, particularly near the ground. In addition, significant improvement in aerodynamic performance has also been observed compared to that of the conventional tail rotor. The shrouded tail rotor experiences less three-dimensional effect, and additional thrust can be obtained from the shroud. Helicopters equipped with a shrouded tail rotor are known to have improved aerodynamic characteristics, smooth handling, and excellent yaw maneuverability. Also, noise attenuation with distance is normally stronger than that of a conventional tail rotor since the noise fundamental frequencies are higher approximately by an order of magnitude.

The first shrouded tail rotor, FenestronTM, was developed by French Aerospatiale and installed on the prototype GAZELLE Helicopter in 1968.¹⁾ The Russian helicopter, Ka-60, is equipped with the fan-in-fin anti-torque system.²⁾ In the USA, the RAH-66 Comanche helicopter developed by Boeing-Sikorsky is equipped with a ducted fan designated as FANTAILTM,^{3,4)} and Bell also developed a ducted tail rotor for an anti-torque system.⁵⁾ A similar shrouded tail rotor was also developed in Japan and installed on the XOH-1 helicopter.⁶⁾

Along with the development of the shrouded tail rotor systems, there have been several numerical and experimen-

tal researches to evaluate the aerodynamic performance of various types of shrouded tail rotors.²⁻⁹⁾ The aerodynamic performance of the Fenestron™ tail rotor has been studied by using two methods: a classical momentum/blade-element theory and a quasi three-dimensional method originally developed for compressor calculations.⁷⁾ Detailed aerodynamic analysis of the RAH-66 FANTAIL™ has been made by solving the incompressible laminar Navier-Stokes equations.³⁾ However, the analysis was made for an axis-symmetric flow by representing the rotor as an additional time-averaged momentum source term in the momentum equation. Performance calculation of the fan-in-fin configuration has been made by modeling the rotor using an ideal disk vortex theory with some empirical correction factors to account for the effect of realistic configuration such as shroud, number of blades, and tip clearance gap.²⁾

Even though these simplified analyses were quite successful for the prediction of the overall performance of shrouded rotors, detailed three-dimensional flow features such as blade tip vortex generation and tip leakage effect cannot be accurately predicted. Thus, first principle-based numerical simulation of the realistic flow is urgently required not only for the understanding of detailed aerodynamics but also for better design of the aerodynamic shape of shrouded tail rotors. However, only a limited number of realistic flow simulations have been previously made using steady and unsteady Euler solvers.^{8,9)}

In the present study, a detailed aerodynamic analysis of a shrouded helicopter tail rotor has been made using a three-dimensional unstructured mesh parallel Euler flow solver. The inviscid flux across each cell face is computed based on the Roe's flux-difference splitting. The governing equations are integrated in time by using an implicit time-marching method. Validation was made for a shrouded tail rotor configuration attached to Kamov Ka-60 helicopters by comparing the results with the experiment.²⁾

2. Numerical Method

2.1. Governing equations

The three-dimensional, compressible Euler equations can be written in an integral form on a rotational frame of reference using absolute flow variables:

$$\frac{\partial}{\partial t} \int_V Q dV + \int_{\partial V} F(Q, n) dS = \int_V S(Q) dV \quad (1)$$

where

$$Q = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho w \\ e_0 \end{pmatrix}, \quad F(Q, n) = \begin{pmatrix} \rho U_r \\ \rho u U_r + p n_x \\ \rho v U_r + p n_y \\ \rho w U_r + p n_z \\ e_0 U_r + p U \end{pmatrix},$$

$$S = \begin{pmatrix} 0 \\ \Omega \rho v \\ -\Omega \rho u \\ 0 \\ 0 \end{pmatrix}$$

In the equation, n_x , n_y , and n_z are the Cartesian components of the exterior surface unit normal vector on the boundary ∂V , and U_r is the normal component of the relative velocity on the rotational frame. All variables are normalized by freestream density, freestream speed of sound, and blade chord length.

Equation (1) is discretized by using a cell-centered finite-volume method. Inviscid flux across each cell face is computed using the Roe's flux-difference splitting formula. An implicit time integration algorithm based on the linearized Euler backward-differencing is used to advance the solution in time. At each iteration, the linear system of equations is solved by using the point Gauss-Seidel method.

The flow solver is parallelized by partitioning the computational domain into several subdomains using the MeTiS library. Load balancing is achieved by distributing an equal number of cells for each processor. Communication of data between processors is made using the Message Passing Interface library.

2.2. Boundary conditions

On the solid surface of the blade and shroud, the flow tangency condition for inviscid flows is applied by imposing no flux through the solid surface. The grid velocity due to rotation is also accounted for on the solid surface.

At the far-field boundary, a point sink boundary condition originally developed by Srinivasan¹⁰⁾ for conventional isolated rotors was modified for the present shrouded rotor. The modification was made on the one-dimensional momentum theory under the assumption that the slip stream from the rotor expands along the diffuser wall of the shroud. The momentum theory can be derived from mass conservation, momentum conservation and the Bernoulli equation under a basic assumption of the Froude theory^{7,11,12)}:

$$\begin{aligned} \dot{m} &= \rho v_i S = \rho V_w \sigma S \\ T_{\text{tot}} &= T_{\text{rot}} + T_{\text{sh}} = \dot{m} V_w \\ &= \rho \sigma S V_w^2 \end{aligned} \quad (2)$$

where σS , v_i and V_w represent the area of outflow boundary, the induced velocity at rotor disk plane, and the flow velocity at the outflow boundary, respectively. Then the velocity at the outflow boundary, V_w , can be expressed in terms of the total thrust as follows:

$$V_w = \sqrt{\frac{T_{\text{tot}}}{\rho \sigma S}} \quad (3)$$

After normalization, the velocity can be written as

$$\bar{V}_w = M_{\text{tip}} \sqrt{\frac{C_{T_{\text{tot}}}}{\sigma}}$$

The outflow velocity, $\bar{V}_w$, obtained from the momentum

Fig. 1. Far-field boundary condition for shrouded rotor.

theory is not applied directly at the outflow boundary, but used to evaluate the far-field inflow velocity by introducing a three-dimensional point sink concept which satisfies the conservation of mass property.^{10,13)} When observed from a distance, the rotor can be considered as a three-dimensional point sink whose strength is equivalent to the mass flow rate out through the far-field boundary obtained from the momentum theory. Then the inflow velocity induced by the point sink at an arbitrary distance r can be written as follows:

$$\bar{V}_{in} = -\frac{1}{4}\sigma\left(\frac{R}{r}\right)\bar{V}_w \quad (4)$$

Using this inflow velocity, the characteristic boundary condition is applied at the far-field boundary for incoming flow. At the outflow far-field boundary, the static pressure is obtained from the isentropic relation using the outflow velocity, and other flow variables are extrapolated from the interior. Figure 1 shows a schematic of the point sink far-field boundary condition applied to the present shrouded tail rotor. Since this boundary condition closely simulates the actual flow physics, the far-field boundary can be located closer to the rotor such that the size of the required computational domain can be reduced. This also speeds up the convergence of the solution to a steady state.

Due to the periodic nature of the flow for hovering rotors, calculation was made for a single blade of the rotor by imposing a periodic boundary condition between the adjacent blades. The grid periodicity is also enforced at the grid generation process, and the cells adjacent to this boundary are treated as interior cells.¹⁴⁾

3. Results and Discussion

3.1. Geometric configuration and mesh generation

In order to validate the present method, calculation was made for a model of a shrouded tail rotor tested at TsAGI,²⁾ which has been installed on the Ka-60 helicopter developed by Kamov. Figure 2 shows a full configuration of the Ka-60 helicopter and the section across the shrouded tail rotor. The test model of the shrouded tail rotor is shown in Fig. 3, and its geometric dimensions are presented in Table 1 as report-

Fig. 2. Ka-60 helicopter and the cross section of the shrouded tail rotor.²⁾ Fig. 3. TsAGI test model of the Ka-60 shrouded tail rotor.²⁾Table 1. Geometric dimensions of the shrouded rotor.²⁾

Rotor	Radius	0.297 m
	No. of blades	11
	Solidity	0.4951
	Twist angle	-12 deg
	Tip speed	74.6 m/s ($M_{tip} = 0.22$)
	Airfoil	NACA23012 ($r/R = 0.35 \sim 1.0$)
Shroud	Tip clearance gap	0.01R
	Inlet radius	0.2R
	Diffuser length	0.7R
	Diffuser angle	4 degrees

ed in Ref. 2). The rotor has 11 equally-spaced blades with a rectangular planform shape and a linear twist of -12 degrees from root to tip. The size of the blade tip clearance gap is one percent of the rotor radius.

Since some of the exact geometric information of the test model is not provided, particularly about the vertical fin and azimuthally-varying shroud radial thickness, the present calculation was made for a simplified configuration by neglecting the vertical fin and by assuming an axially symmetric shroud having uniform radial thickness in the circumferential direction. The simplified configuration of the shrouded rotor used for the calculation is presented in Fig. 4.

Fig. 4. Simplified configuration of the shrouded rotor used for calculation.

Fig. 5. Surface triangulation for the shrouded rotor.

Fig. 6. Surface triangulation at the periodic boundaries.

An unstructured mesh was generated around the simplified TsAGI test model, and the surface triangulation for the half configuration is presented in Fig. 5. The total number of tetrahedral cells and node points is 884,698 and 167,609, respectively. A grid sensitivity study was also made by performing the calculation on a coarse mesh having 283,273 tetrahedral cells and 56,013 node points, which showed less than one percent difference in the predicted total rotor thrust. The surface triangulation on the artificial periodic boundary between the blades is shown in Fig. 6, which confirms the grid periodicity. The bold lines represent the subdomain boundary used for parallel computation. It indicates that small cells are mostly distributed along the blade and inside the diffuser of the shroud. The far-field

Fig. 7. Spanwise distribution of the blade sectional thrust.

boundary is located at five radii away from the rotor. Test calculations have been made to check the effect of the far-field boundary location, which showed no significant difference in aerodynamic performance between $3R$ to $10R$.

Calculations were made on a LINUX-operating PC cluster having Pentium IV 2.4 GHz CPUs, which took approximately three hours to obtain a converged solution using 29 processors.

3.2. Aerodynamics of the shrouded rotor

An initial validation was made for an isolated rotor without shroud at a tip mach number of 0.22 and a collective pitch angle of 28 degrees at the blade root. At this operating condition, it is known that the rotor produces thrust equivalent to that of the shrouded rotor at a collective pitch angle of 45 degrees.²⁾ The predicted thrust from the present calculation was 95 N, which is approximately 8% over-prediction to that of the experiment. This may be attributed to the inviscid flow nature of the present solver.

Next, a numerical analysis of the shrouded tail rotor was made at a tip mach number of 0.22. In order to compensate the simplification involved in the removal of the vertical fin and making the shroud radial thickness uniform, calculation was made for a collective pitch angle at which the predicted rotor thrust compares with that of the experiment in a similar degree obtained for the isolated rotor. This correction method is normally used for the aerodynamic performance prediction of helicopter rotors to eliminate the effect of uncertainties involved in experiment and to guarantee correct flow acceleration through the rotor disk. The resultant collective pitch angle at blade root was 40 degrees while the measurement was made at 45 degrees.

In Fig. 7, the predicted sectional thrust distribution along the blade is compared with the result based on the disk vortex theory,²⁾ in which the center body was not modeled. However, the effect of the center body on the blade sectional thrust is very small. At the inboard sections of the blade, the present calculation predicts slightly higher loading than the disk vortex theory. When approaching toward the outboard sections, the result from the present method shows slightly lower thrust peak and a flatter load distribution, representing less three-dimensionality due to shroud. The present result also shows a slight increase in thrust at the tip, which is due to the formation of the blade tip vortex rolling up

Fig. 8. Spanwise distribution of the induced downwash.

Fig. 9. Spanwise distribution of the circumferential induced velocity.

Fig. 10. Spanwise distribution of the induced angle-of-attack.

through the tip clearance gap.

In Figs. 8, 9 and 10, spanwise distributions of the induced downwash, the circumferential induced velocity and the induced angle-of-attack as defined in Fig. 11 are compared with the experiment and the results from the disk vortex theory. These flow properties are extracted at a half chord length below the rotor disk plane and averaged in the circumferential direction. It is shown that the induced downwash and the induced angle-of-attack are generally in good agreement with the experimental results. The present results also compare well with those of the disk vortex theory. However, when approaching toward the tip, the present results indicate better comparison than those of the disk vortex theory, which demonstrates the importance of the realistic three-dimensional tip modeling. The disk vortex theory results show a tendency of going in the direction opposite to the experiment. The spanwise distributions of the circumfer-

Fig. 11. Definition of velocity components at a section of the blade.

a) Present calculation

b) Result of Fenestron™ tail rotor calculation⁷⁾

Fig. 12. Static pressure distribution around the section of the shroud.

ential induced velocity from both predictions are slightly off from the experiment at inboard sections, which may be attributed to the inviscid nature of the prediction methods. However, the present method indicates a better behavior when approaching toward the tip, which is physically correct as shown from the experiment. The predicted value obtained from the disk vortex theory increases very high at the tip, which is due to the lack of the physical blade bound vortex in the theory.

Figure 12 a) shows the calculated static pressure distribution around the sectional contour of the shroud, which is normalized by the blade tip speed and is azimuthally averaged. It is shown that the low pressure peak is obtained on the shroud lip (B-C) as the flow accelerates entering the rotor disk, which is the main source of the thrust obtainable from the shroud. At the blade zone (C-D), the pressure rapidly increases, and this trend continues at further down-

Fig. 13. Numerical particle traces at the periodic boundary.

Fig. 14. Vorticity contours at the periodic boundary.

stream along the diffuser wall as the flow expands and decelerates. The pressure is maintained almost uniformly at the bottom surface and around the outer boundary of the shroud. The overall tendency of the shroud pressure distribution is in good agreement with that of other predictions⁷⁾ as shown in Fig. 12 b).

In Fig. 13, the numerical particle traces are presented at the periodic boundary between the adjacent blades. It is indicated that the flow enters the computational domain except at the wake boundary, which reflects the point sink method applied at the far-field boundary. The figure also shows the flow entrainment through the rotor disk and the acceleration into the wake with higher kinetic energy provided by the rotor.

The non-dimensionalized vorticity contours at the periodic boundary in Fig. 14 show that the tip vortex from primary blade and the layers of the discrete tip vortices from preceding blades are well captured using the present method. Four layers of inboard vortex sheet are also well captured. The result illustrates that the outer edge of the inboard vortex sheet travels downstream approximately twice the velocity of the corresponding tip vortex, which is similar to that of conventional open-type rotors. However, the tip vortices do not show any contraction, but are convected downstream along the diffuser wall. The discrete tip vortices eventually merge into a well-developed vortex sheet downstream of the bottom edge of the shroud. A strong vortical region is also observed in the wake of the center body, which results from the continuous addition of vorticity from the inboard vortex

Fig. 15. Iso-vorticity surface for the full configuration (vorticity level of 0.025).

sheets from upstream as expanding around the corner of the center body. Development of the vorticity is also observed at the lip region of the shroud and the center body as the flow accelerates.

A three-dimensional perspective view of this vorticity distribution is shown as an iso-vorticity surface in Fig. 15 for the full rotor configuration. The figure confirms the existence of blades and the layers of tip vortex forming a helical pattern as convected downstream. The inboard vortex sheets generated from the blade trailing edge are also clearly seen in the figure.

4. Conclusion

In the present study, a three-dimensional inviscid flow analysis has been made for the detailed study of shrouded tail rotor aerodynamics. A point sink boundary condition and the one-dimensional momentum theory were modified to satisfy the far-field boundary condition for the rotor with shroud. Calculations were made for an isolated rotor without shroud and a shrouded tail rotor for validation. The results were compared with experiments and those obtained using an ideal disk vortex theory. It was demonstrated that realistic three-dimensional aerodynamic analyses are required for the simulation of detailed flow fields of shrouded tail rotor involving formation of tip vortex, inboard vortex sheets, and flow acceleration and expansion through the diffuser.

References

- 1) Mouille, R.: The Fenestron, Shrouded Tail Rotor of the SA. 341 Gazelle, *J. Am. Helicopter Soc.*, **15** (1970), pp. 31–37.
- 2) Bourtsev, B. N. and Selemenev, S. V.: Fan-in-Fin Performance at Hover Computational Method, Proc. 26th European Rotorcraft Forum, 2000.
- 3) Rajagopalan, R. G. and Keys, C. N.: Detailed Aerodynamic Analysis of the RAH-66 FANTAIL™ Using CFD, *J. Am. Helicopter Soc.*, **42** (1997), pp. 310–320.
- 4) Keys, C., Sheffler, M., Weiner, S. and Heminway, R.: LH Wind Tunnel Testing: Key to Advanced Aerodynamic Design, Proc. 47th American Helicopter Society Annual Forum, 1991.
- 5) Andrews, J. R., Richard, G. R., Jr. and Rahnke, C.: Design and Testing of a Ducted Tail Rotor Concept Demonstrator for a Model 222U Helicopter, Proc. 22nd European Rotorcraft Forum, 1996.
- 6) Bandoh, S., Fudamoto, M. and Akiyama, T.: The Ducted Tail Rotor

- System of the New Observation Helicopter (XOH-1), Proc. AHS International Meeting on Advanced Rotorcraft Technology and Disaster Relief, Gifu, Japan, 1998.
- 7) Vuillet, A. and Morelli, F.: New Aerodynamic Design of the Fenestron for Improved Performance, Proc. 12th European Rotorcraft Forum, 1986.
 - 8) Kawamoto, H., Yoshida, H., Murashige, A. and Yamakawa, E.: CFD Simulations of Ducted Tail Rotor, Proc. 37th Aircraft Symposium, Japan, 2C15, 1999.
 - 9) Kawamoto, H., Yoshida, H., Murashige, A. and Yamakawa, E.: Unsteady Aerodynamic Simulations of Ducted Tail Rotor, Proc. 38th Aircraft Symposium, Japan, 1D11, 2000.
 - 10) Srinivasan, G. R., Raghava, V. and Duque, E. P. N.: Flowfield Analysis of Modern Helicopter Rotors in Hover by Navier-Stokes Method, *J. Am. Helicopter Soc.*, **38** (1993), pp. 3–13.
 - 11) Leishman, J. G.: *Principles of Helicopter Aerodynamics*, Cambridge University Press, New York, 2000.
 - 12) Xu, G. H. and Wang, S. C.: Effects of the Shroud on Aerodynamic Performance in Helicopter Shrouded Tail Rotor System, *J. Aircraft Eng. Aerospace Technol.*, **73** (2001), pp. 568–572.
 - 13) Strawn, R. C. and Djomehri, M. J.: Computational Modeling of Hovering Rotor and Wake Aerodynamics, Proc. 57th American Helicopter Society Annual Forum, May 9–11, 2001.
 - 14) Kang, H. J. and Kwon, O. J.: Effect of Wake Adaptation on Rotor Hover Simulations Using Unstructured Meshes, *J. Aircraft*, **38** (2001), pp. 868–877.