JOURNAL OF AIRCRAFT
Vol. 44, No. 4, July–August 2007

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes should not exceed 2500 words (where a figure or table counts as 200 words). Following informal review by the Editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Hub Drag-Reduction Strategies of Canard Rotor/Wing Unmanned Aerial Vehicles

C. Kim* and J. Lee†
Korea Aerospace Research Institute, Daejon 305-333, Republic of Korea

DOI: 10.2514/1.24395

Introduction

CANARD rotor/wing (CRW) aircraft is a new conceptual vehicle having capabilities of vertical takeoff/landing and high-speed cruising. It takes off vertically with the thrust of the main rotor, then the rotor is fixed and generates lift for a high-speed cruise. The CRW aircraft concept was proposed in the 1990s by Rutherford et al. [1] and Bass , and several wind-tunnel tests [3–5] and analysis [6] were accomplished to develop key components of CRW technology in the last decade. In 1998, Boeing and the Defense Advanced Research Project Agency (DARPA) launched a joint project to develop a CRW-concept unmanned aerial vehicle (UAV) called Dragonfly. Demonstrator vehicles were manufactured and a flight test was performed.

Meanwhile, the Korea Aerospace Research Institute (KARI) reviewed the CRW concept for the new conceptual aircraft development program in 2002. Subsequently, they confirmed some favorable characteristics such as vertical takeoff/landing and high subsonic cruising. During the review process, computational fluid dynamics (CFD) analysis was intensively performed to understand the aerodynamic characteristics of the initial configuration. CFD analysis additionally identified the hub as the cause of significant drag increase (30% of the vehicle drag). In this paper, the mechanism whereby the hub increases drag significantly is first explained, then a couple of drag-reduction strategies are proposed: one is to install a fairing surrounding hub at cruise and the other is to modify the rotor shape around the hub.

Numerical Investigation of Hub Drag Increase

Grid Generation and Numerical Experiment

The geometry surface, saved in CAD format, was retrieved to generate the surface grid shown in Fig. 1. The initial CRW configuration is very similar to the Dragonfly developed by DARPA. It consisted of a canard, rotor/wing, and U-type tail. A hub connecting the rotor/wing and body is installed. The rotor section shape should be symmetric and ellipses are used. Its root and tip thickness ratios are 24 and 16 %, respectively. The vehicle's length, span, and height are 5.4, 3.6, and 1.3 m, respectively. The maximum takeoff weight is 600 kg and the payload is 90 kg.

The unstructured grid is prepared and the surface grid is the hybrid type. To save the grid number and cluster, more grid points along the main flow direction and most of the surfaces, especially the rotor and tail, are covered with quadrangles. Complex surfaces such as the hub and junction areas of the canard and body, etc., are covered with triangles. Figure 2 shows the hybrid surface grid around junctions of the body, canard, hub, and tail. The total number of the surface grid amounts to 140,000.

For accurate boundary-layer flow simulation, 15 wall prism layers were generated by projecting in the normal direction from the surface grid. The height of the first layer is 5.0×10^{-5} m and the growth rate is 1.2. The rest of the volume is filled with tetrahedrons and the total number of 3-D computational grids, including the boundary-layer grid, is 6×10^{6}. The far-field boundary is cylinder-shaped, and the distance from the vehicle to the front end is 5 times of the vehicle length and to the rear end is more than 10 times.

The compressible Navier–Stokes code Fluent [7] was implemented for numerical experiment. The far-field boundary condition was set with the freestream Mach number, static pressure, temperature, and turbulence intensity. The turbulence viscosity was computed with the Spalart–Allmaras one-equation model. The freestream velocity is 11 m/s (400 km/h), and the ambient pressure and temperature are 70,121 Pa and 268 K, respectively. The turbulence intensity and length are 2% and 0.01 m. After initializing the whole computational field with the far-field flow properties, numerical iteration was carried out with a first-order-accurate solver to accelerate the solution convergence. Then iteration was continued with a second-order-accurate solver until the flow solutions converged.

Numerical Results and Hub Drag Identification

Figure 3 shows the pressure coefficient Cp contour of the vehicle surface. The vehicle configuration appeared to be smooth, and any pressure jump or drop is not observed, except in the hub area. The hub connecting the rotor and body, even though it is a short cylinder (height is 70 mm and diameter is 240 mm), separates adjacent flow and generates a low-pressure region behind the hub. To explain the flow pattern around the hub, a low-speed contour plot is shown in Fig. 4, because a low-speed contour plot helps to identify the separated flow or thick boundary-layer region. The freestream

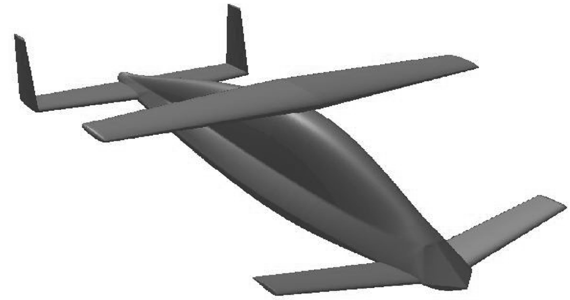

Fig. 1 Geometry of the canard rotor/wing UAV.

Received 3 April 2006; revision received 9 March 2007; accepted for publication 12 March 2007.

*Senior Researcher, Aerodynamics Department, 45 Eoeun-Dong, Yuseong-Gu. Member AIAA.
†Principal Researcher, Aerodynamics Department, 45 Eoeun-Dong, Yuseong-Gu.

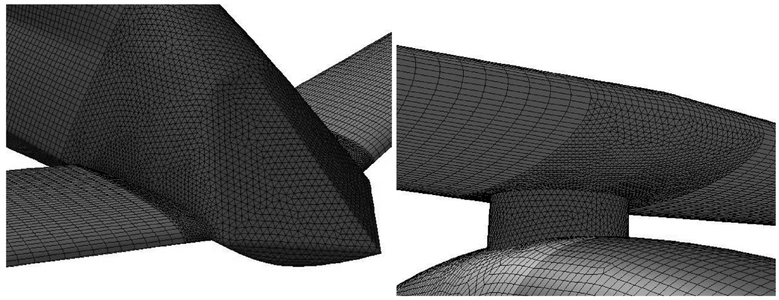

Fig. 2 Hybrid surface-grid demonstration: left, nose and canard junction and right, rotor and hub.

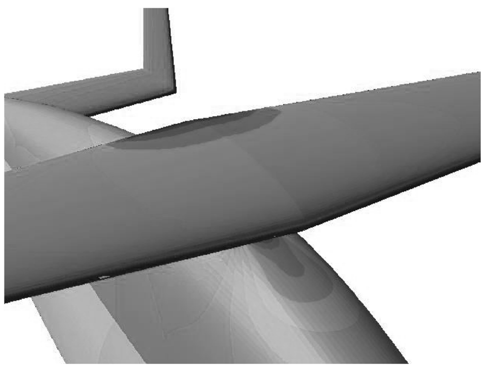

Fig. 3 Pressure coefficient *Cp* contour plots of the whole vehicle and the rotor center.

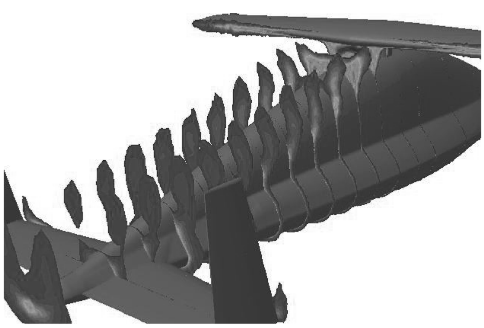

Fig. 4 Low-speed contour plot around the vehicle and hub area, $V_\infty = 111$ m/s and $V = 0$–100 m/s.

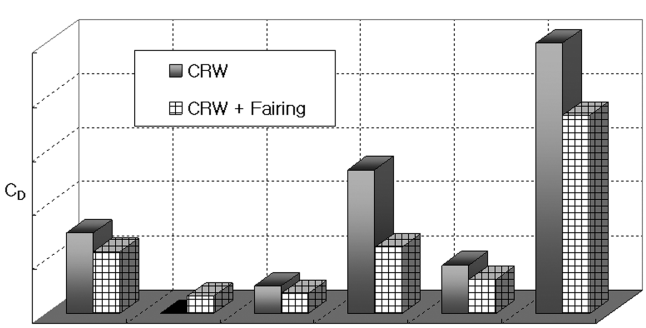

Fig. 5 Comparison of drag coefficients for the hub fairing effect.

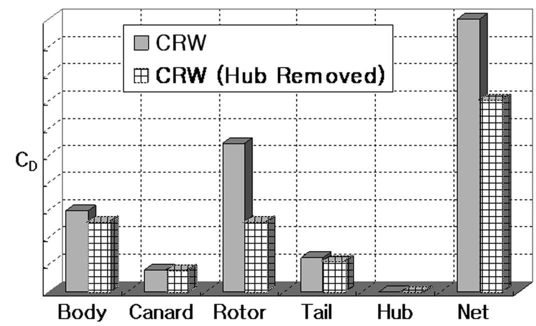

Fig. 7 Comparison of component drag for hub-effect identification.

velocity is 111 m/s and the low-speed contour ranges from 0 to 100 m/s. Figure 4 confirms that the flow behind the hub is fully separated.

The fully separated flow by the hub prevents the pressure from being recovered, generates a low-pressure region behind the hub, and accelerates the flow over the upper surface of the rotor. Therefore, the hub causes severe drag increase of the vehicle, especially the rotor. To estimate the drag increase caused by the hub, a CRW configuration without a hub was tested numerically. The comparison shown in Fig. 5 vividly explains the hub effect on the drag increase. The hub, even though its drag is minor, provides significant effect on the drag increase of the body, tail, and especially the rotor.

Hub Drag-Reduction Strategies

As confirmed in the previous section, drag increase of the rotor is mainly from the hub; thus, a couple of strategies are presented to reduce the hub drag, such as fairing installation and modification of the rotor shape around the hub.

Fairing Installation

A fairing is installed, as shown in Fig. 6, to prevent flow from contacting the hub at the airplane mode. The fairing makes the flow pass the hub region smoothly and does not induce any pressure drop around the hub. The numerical experiment for the fairing configuration revealed the amount of drag reduction and the drag increment by the hub. Figure 7 compares the component drag coefficients to demonstrate the effect of fairing. Fairing installation reduces drag of the body, canard, tail, and rotor, and the rotor drag reduction is highest. However the fairing should be opened for the rotor to rotate without any restriction at the hovering and conversion

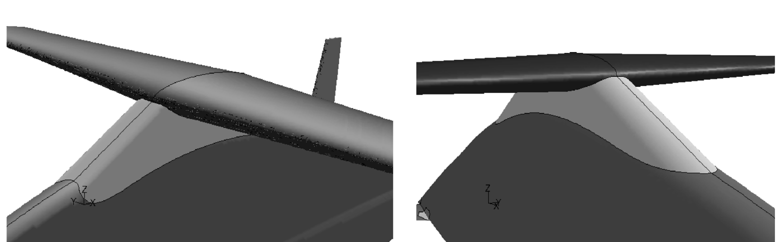

Fig. 6 Configuration of fairing: left, front view and right, rear view.

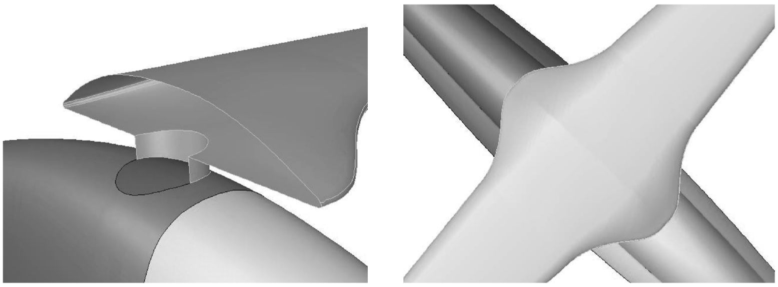

Fig. 8 Configuration of the modified rotor around the hub.

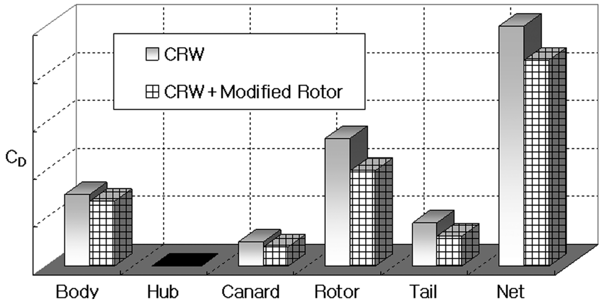

Fig. 9 Comparison of drag coefficients for the effect of rotor modification.

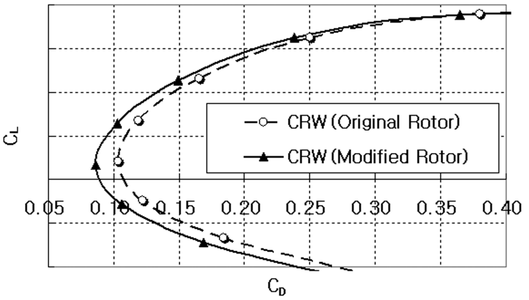

Fig. 10 Drag polar plot for the effect of rotor modification.

modes. Therefore, a mechanical device should be installed inside the fairing for operation.

Main Rotor Modification

Although the fairing installation demonstrates terrific rotor drag reduction, it requires an additional mechanical device, which may be burdensome on the vehicle weight management. As an alternative, main rotor modification, even though it does not reduce drag as much as fairing installation, was proposed.

Even though the hub separates the flow fully and generates lower pressure behind the hub, the lower surface of the rotor, being flat and not having any geometry contribution to the direction of the drag, does not increase drag. Therefore, the lower surface of the rotor was straightened and the chord was extended to maintain the same thickness. Figure 8 illustrates the configuration of the modified rotor around the hub. As explained, the chord was extended by double, but the thickness of the rotor center is the same as the original one, to occupy the space for ducts and mechanical devices.

Even though the drag reduction of the modified rotor is not as drastic as the fairing installation, drag reduction by modifying the shape of the rotor center is considerable (Fig. 9). The modification again decreases the drag force of the body, canard, rotor and tail. Figure 10 is a drag polar plot to demonstrate the drag-reduction extent of the modified rotor. The modified rotor decreases the drag of the vehicle over various angles of attack but does not deteriorate the lift of the vehicle. The lift of the vehicle is generally dependent on the azimuthal position of the rotor [6], and the circular disc shape of the rotor surface may produce lift that is less dependent on the azimuthal position.

Conclusions

A CRW UAV was simulated with CFD and the mechanism of drag increase by a hub was presented. The hub having a circular cylinder shape separates flow fully and prevents the pressure recovery behind the hub, and the significant pressure difference around the hub increases the drag of the vehicle, especially the rotor. Therefore, a couple of drag-reduction strategies are proposed and numerical experiments for each case were performed to show their effects on the reduction. Fairing installation reduces almost the same amount of drag as the increment by the hub, but rotor center modification reduces drag less than the fairing installation.

Acknowledgments

This work was supported by the Smart UAV Development program, one of the 21st Century Frontier research and development programs funded by the Ministry of Commerce, Industry and Energy of Korea.

References

[1] Rutherford, J. W., Bass, S. M., and Larsen, S. D., "Canard Rotor/Wing: A Revolutionary High-Speed Rotorcraft Concept," AIAA/AHS/ASEE Aerospace Design Conference, Irvine, CA, AIAA Paper 93-1175, Feb. 1993.

[2] Bass, S. M., "Canard Rotor/Wing: A Concept to Revolutionize the Rotary-Wing Industry," *International Powered Lift Conference*, Society of Automotive Engineers, Warrendale, PA, Sept. 1998.

[3] Bass, S. M., Thompson, T. L., and Rutherford, J. W., "Fixed-Wing Performance Predictions of the Canard Rotor/Wing Concept Based on Wind Tunnel Test Results," *A Collection of Technical Papers: 13th AIAA Applied Aerodynamics Conference*, AIAA, Washington, D.C., 1995, pp. 1003–1012; also AIAA Paper 1995-1887, June 1995.

[4] Bass, S. M., Thompson, T. L., Rutherford, J. W., and Swanson, S., "Low Speed Wind Tunnel Test Results of the Canard Rotor/Wing Concept," AIAA Applied Aerodynamics Conference, Monterey CA, AIAA Paper 93-3412, Aug. 1993.

[5] Thompson, T. L., Smith, R. L., Helwani, M., Shockey, G. A., and Schwimley, S. L., "Wind Tunnel Test Results for a Canard Rotor/Wing Aircraft Configuration," *AHS Annual Forum* [CD-ROM], Vol. 57, American Helicopter Society, Alexandria, VA, May 2001.

[6] Pandya, S. A., and Aftosmis, M. J., "Computation of External Aerodynamics for a Canard Rotor/Wing Aircraft," 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, AIAA Paper 2001-0997, Jan. 2001.

[7] FLUENT, Software Package, Ver. 6.2, Fluent Inc., Lebanon, NH, 2004.