A Study on the Far-Field Boundary Condition Effects of CFD/Time-Marching-Free-Wake Coupled Method

Seong-Yong Wie*, Dong-Kyun Im*, Jang-Hyuk Kwon*, Duck-Joo Lee*
And
Ki-Hoon Chung**, Seung-Bum Kim**
* Department of Aerospace Engineering,
Korea Advanced Institute of Science and Technology, Daejeon, Korea
(Tel : +82-042-869-3756; E-mail: wsy278@kaist.ac.kr)
** Korea Aerospace Research Institute, Daejeon, Korea

Abstract: To simulate rotor aerodynamic characteristics, a wake generated during flights should be considered very carefully, because the wake induces velocity fields around the rotor to change the effective angle of attack. Rotor aerodynamics in hovering is particularly governed by wake geometry and strength. In this study, a helicopter rotor is simulated by tightly coupled CFD/Free-wake method to describe wake characteristics. Rotor blade and flow field aerodynamics are calculated by CFD, and wake motions are simulated by Time-Marching-Free-Wake(TMFW) method. This tightly coupled CFD/Free-wake method can describe wake characteristics as well as rotor aerodynamic properties. Through coupling the two methods, the far field boundary condition in the CFD are provided from the induced flow field obtained by TMFW method at each time step. The wake strength is determined from sectional lift calculated in CFD at both the subsonic and transonic speeds. To show advantage of the method, it is compared with other boundary conditions such as source-sink model located at the far wake boundary in the CFD domain. The accuracy of the present method depends on computational domain size because the induced velocity at the boundary affects rotor aerodynamic properties. To investigate grid effects, computation results in the large and small size grids are compared with experimental data.

Keywords: Tightly coupled method, CFD, Time Marchimg Free Wake, Far-Field boundary condition

1. INTRODUCTION

Aerodynamic characteristics of helicopter rotor are very unsteady, because motions of strong rotor wake is complicated..

For that reason, numerical simulation has a difficulty to predict accurate rotor aerodynamics. Particularly, wake characteristics are not clearly described due to numerical dissipation in CFD. This numerical problem also causes diminishing flow vorticity and circulation around the vorticity. Even though there is physical dissipation due to kinematic viscosity in the wake, the circulation around the vorticity is conserved. The numerically diminished vorticity cannot make sufficient induced velocity and inflow describing rotor aerodynamic performance. To overcome this problem, various mathematical model and numerical methods have been developed previously.

Srinivasan et al.[1] proposed a source-sink model that can provide inflow and outflow conditions at the far-field boundary to compensate the reduced inflow around the airfoil due to the diminished vorticity. It is very useful model, but this model cannot be used in describing the detailed wake motions owing to the forced source-sink effects even in hovering case. In forwarding flights as well as hovering case, wake model or numerical treatment should be applied to consider the wake effects more realistically.

Time Marching Free-wake method is an effective method to describe the wake, because it can capture wake properties without numerical dissipation. For that reason, many researchers have been studied hybrid method coupled with free-wake method. Berkman[2], Yang[3] and Zhao[4] used free-wake model to consider induced velocity effects in CFD domain. They used solver for Navier-Stokes equation in the near field, and used potential solver in the far field. Effects of velocity induced by rotor tip vortex were described free-wake model.

In this paper, rotor aerodynamics is also analyzed by using Time-Marching-Free-Wake method. The TMFW can describe inboard vortices as well as tip vortex. Therefore, detailed geometry of the wake can be prediced. Without potential solver, Euler solver in the near field is tightly coupled with the free-wake method. At each time, the free-wake method provides inflow and outflow conditions to,the Euler solver and the wake strength required in the free-wake method can be obtained by using the calculated velocity around the rotor from the Euler solver.

This tightly coupled method is applied to simulate rotor aerodynamics, and the results are compared with ones obtained from conventional rotor CFD and experimental data.

2. METHODLOGY

Code for analyzing rotor aerodynamics is based on structured grid, and is unsteady compressible solver. It can be possible to perform multi block and multi processing [6].

2.1 Governing equation

3dimensional Euler equation is below.

$$\frac{\partial q}{\partial t}+\frac{\partial f_i}{\partial x_i}=\alpha^F \vec{H} \qquad (1)$$

From (1), q is conservative variable, f_i means inviscid flux term. And $\vec{H}$ indicate noninertial force terms.

$$q = \begin{pmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho E \end{pmatrix}, \qquad f_i = \begin{pmatrix} \rho \vec{U} \\ \rho u_1 \vec{U} + \delta_{i1} p \\ \rho u_2 \vec{U} + \delta_{i2} p \\ \rho u_3 \vec{U} + \delta_{i3} p \\ (\rho e + p)\vec{U} + p\vec{U}_g \end{pmatrix} \tag{2}$$

$$\vec{H} = \begin{pmatrix} 0 \\ \vec{h} \bullet \hat{i} \\ \vec{h} \bullet \hat{j} \\ \vec{h} \bullet \hat{k} \\ \vec{h} \bullet \vec{u} \end{pmatrix}, \qquad \vec{h} = -\rho \vec{\Omega} \times \vec{u} \tag{3}$$

$\vec{U}$ is a flow velocity that is relative to grid velocity. This is shown below.

$$\vec{U} = n_x u + n_y v + n_z w - \vec{U}_g \tag{4}$$

From eq.(1), α^F is zero in the inertial frame, and is 1 in the non-inertial frame. In the inertial frame, $\vec{U}_g$ of eq.(4) is grid velocity, then $\vec{U}$ is relative velocity to the grid. In steady flow of non-inertial frame, $\vec{U}_g$ is a frame velocity, and $\vec{U}$ is relative velocity to the rotational frame.[7]

2.2 Numerical method

For spatial discretization, differential governing equation can be transformed to the semi-discretized equation using Finite Volume Method (FVM), and this equation is integrated by time marching method. In this FVM, cell centered method is used.

Roe's FDS(Flux Difference Splitting) and van Leer's MUSCL(Monotone Upstream Scheme for Conservation) is used for inviscid flow calculation, and van Alada's limiter is used for stability. For time marching, DADI(Diagonalized Alternating Direction Implicit) is used.[9,10] To improve time accuracy, dual time stepping is applied. And multigrid method and local time stepping are used to accelerate convergence.

2.3 Time Marching Free-Wake model

Wake is described as vortex filaments. Induced velocity of vortex filament can be obtained from Biot-Savart law. Biot-Savart law is shown below.

$$d\vec{V} = \frac{\Gamma}{4\pi} \frac{d\vec{l} \times \vec{r}}{|\vec{r}|^3} \tag{5}$$

$d\vec{l}$ is vortex filament length, $\vec{r}$ is distance between velocity position and vortex filament, and Γ is strength of vortex.

From eq.(5), singularity is occurred at r=0. To prevent this singularity, vortex model is used. In this study, Vatistas's vortex model is applied.

Vatistas's vortex model:

$$v_\theta = \frac{\Gamma}{2\pi r_c} \frac{r / r_c}{\sqrt{(1 + (r/r_c)^4)}} \tag{6}$$

r_c : radius of vortex core

Present free-wake model uses whole trailed vortex filaments as well as tip vortex, and this vortex filaments are described as rotor wake. Time marching scheme for wake motion is 4th order Adams-Bashforth-Moultion method. This method is more stable than conventional explicit method.

2.4 Tightly coupled method

This coupled method is that rotor CFD is tightly coupled with full trailed free-wake method. At each time step, CFD makes lifting line for free-wake, and this lifting line can make trailed vortex filament from rotor blades. Using lifting line theory and Kutta-Joukowsky theory, bound vortex strength in spanwise direction can be obtained, and difference between sectional bound vortices provide trailed vortex strength at each time. Trailed vortex strength is used in wake roll-up process of free-wake. Wake characteristics are investigated in this free-wake procedure. Fig.1 indicates bound vortices position and trailed vortices motion, and Fig.2 shows wake in CFD domain using tightly coupled method.

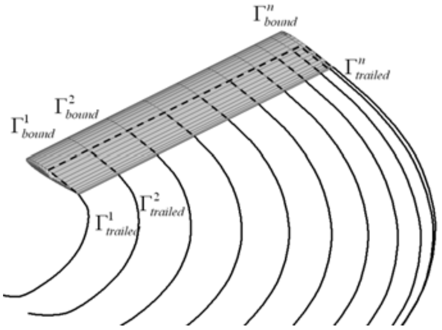


Fig. 1 Schematic of trailed vortices and bound vortices

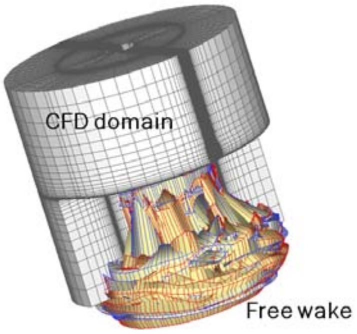


Fig. 2 Freewake in the CFD domain

Wake generation and CFD boundary condition correction are performed at each time. Procedure of tightly coupled method is shown Fig 3.

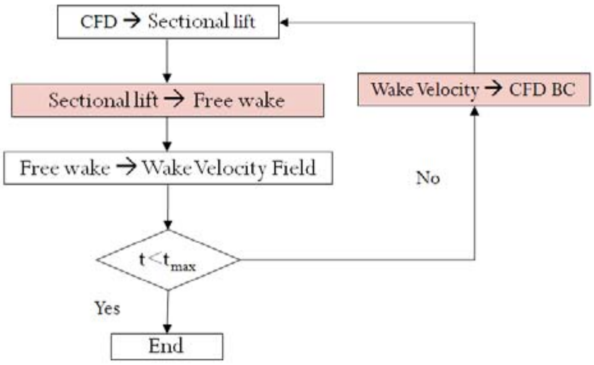


Fig. 3 Procedure of tightly coupled method

3. NUMERICAL RESULTS

3.1 Rotor aerodynamics in hovering

Caradona-Tung's rotor experiment [11] is analyzed. This rotor radius is 3.75ft (1.143m), and aspect ratio is 6. This blade is no tapered and no twisted. Sectional airfoil is NACA0012.

This rotor is simulated at pitch angle 8degree in subsonic (Tip Mach No.=0.439) and transonic (Tip Mach No.=0.877) region. Numerical results of the coupled method are compared with results of the conventional rotor CFD using source-sink model, unsteady panel method and experiment. To investigate boundary condition effects of grid size, rotor is simulated in 5 × R, 1.8 × R and 1.8 × R-disk grid system. At each grid system, .pressure coefficients are shown at r/R= (a) 0.68, (b) 0.89, and (c) 0.96

3.1.1 Analysis in Grid size 5 × R

This cylindrical grid is composed of 4 blocks, and radius of grid is 5 times rotor's radius. The number of grid points is 850212.

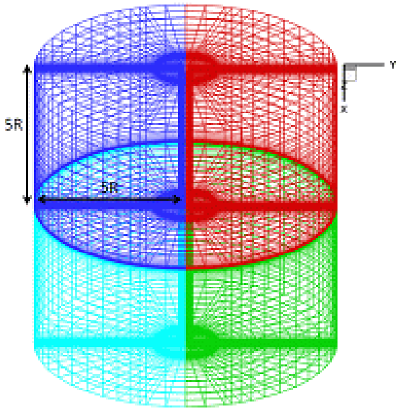
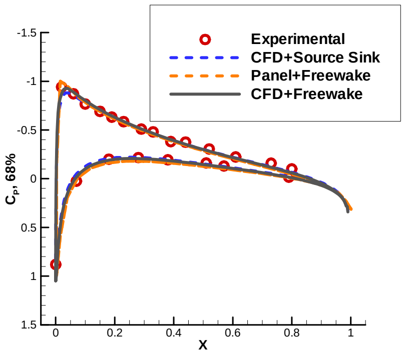


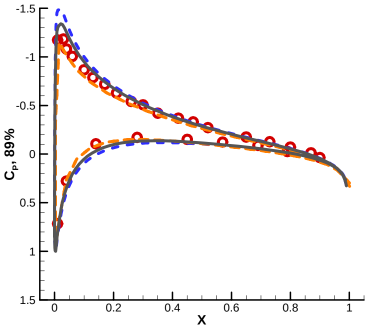
Fig. 4 5×R Grid system

Pressure coefficient at blade surface is shown in Figs. 5~6. Fig. 5 indicate tip Mach number 0.439, and Fig. 6 indicate tip Mach number 0.877. These coefficients are obtained at r/R= (a) 0.68, (b) 0.89, and (c) 0.96

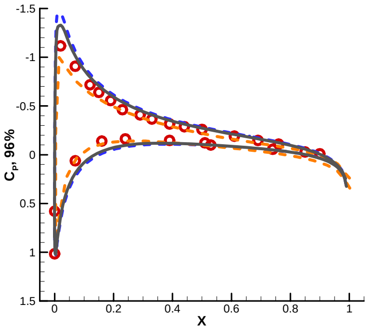
These numerical results of coupled method are compared with source-sink model, unsteady panel and experiment's result. We can confirm that tightly coupled method, conventional method with source-sink model and unsteady panel method results correspond with experimental data in general with this relatively large grid size.



(a)

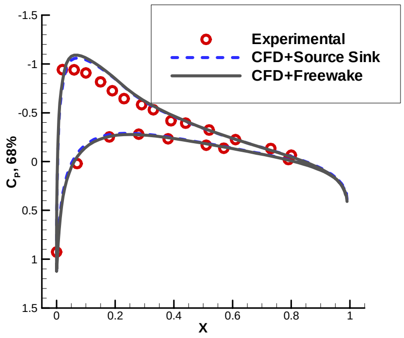


(b)

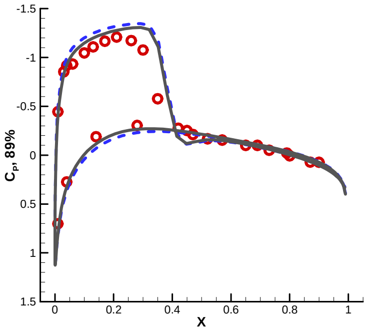


(c)

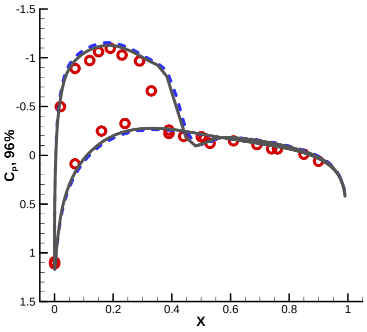
Fig. 5 Pressure coefficients (5×R grid, M_{tip}=0.439)



(a)



(b)



(c)

Fig. 6 Pressure coefficients (5×R grid, M_{tip}=0.877)

3.1.2 Analysis in Grid size 1.8×R

To investigate effects of boundary condition correction using free-wake, domain size is reduced. Radius of cylindrical grid is 1.8 times rotor's radius. The number of grid points is 553700.

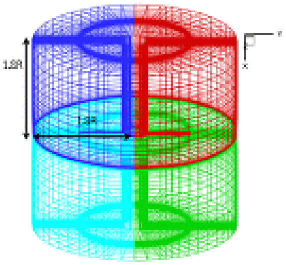
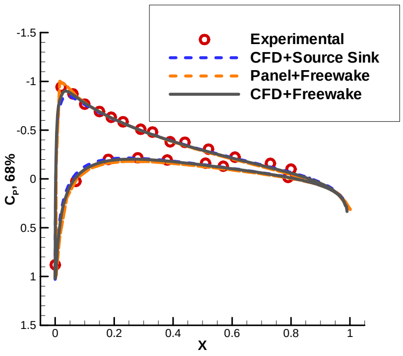
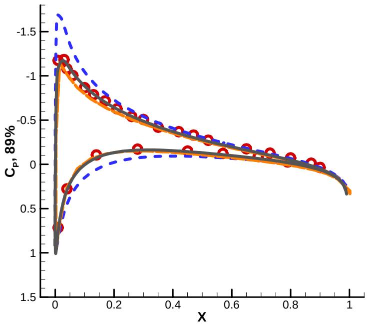


Fig. 7 1.8×R Grid system

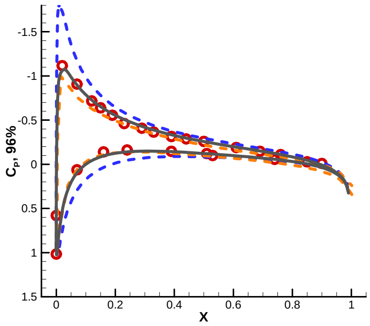
Figs. 8~9 indicate pressure coefficients at tip Mach number 0.439, 0.877. The tightly coupled method's results are more accurate than source-sink model's with this relatively reduced grid size especially at the blade tip.



(a)

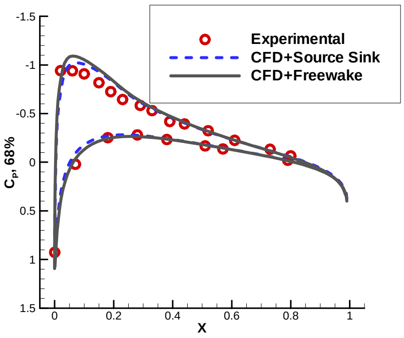


(b)

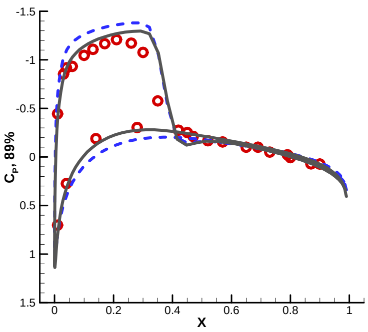


(c)

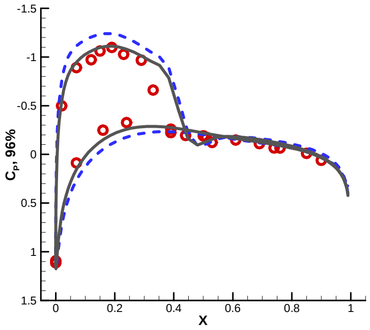
Fig. 8 Pressure coefficients (1.8×R grid, M_{tip}=0.439)



(a)



(b)



(c)

Fig. 9 Pressure coefficients (1.8×R grid, M_{tip}=0.877)

3.1.3 Analysis in Grid size 1.8×R-disk

We can see that coupled method is effective in small grid as well as large grid. To maximize this boundary condition correction effects due to grid reduction, height of 1.8×R grid is reduced. Reduced grid height is 0.6×R, and the number of grid points is 376516.

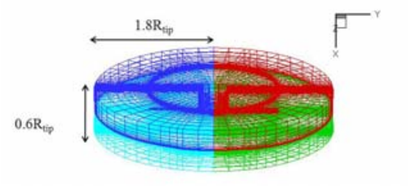
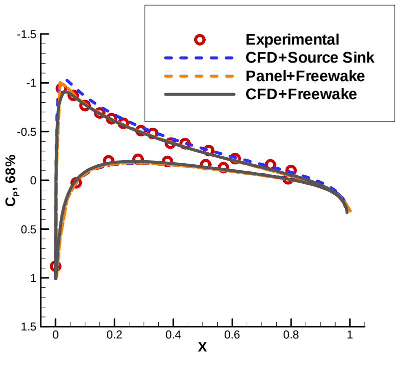
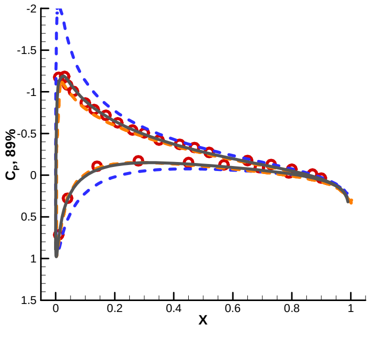


Fig. 10 1.8×R-disk Grid system

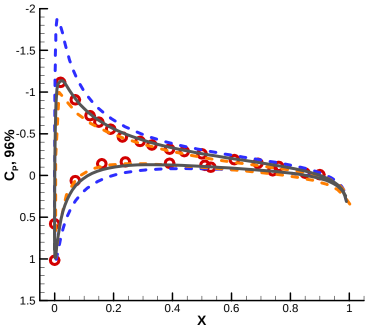
From Figs. 11~12, boundary condition effect using free-wake model is remarkable. Particularly, source-sink model's result is poor in transonic region. In small grid, conventional rotor CFD with source-sink model cannot predict rotor aerodynamics. On the other hand, coupled method predicts rotor aerodynamics in hovering very well.



(a)

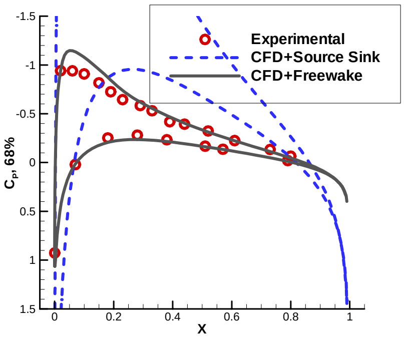


(b)

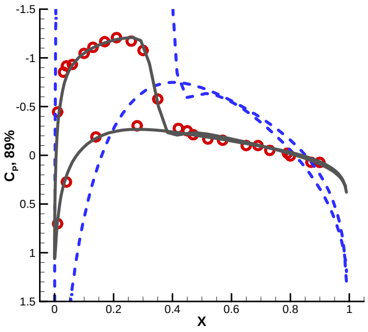


(c)

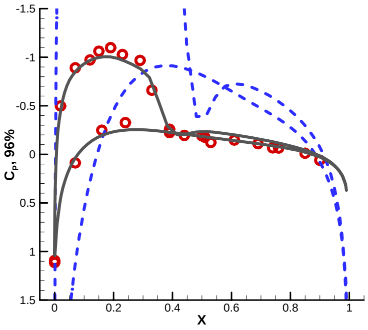
Fig. 11 Pressure coefficients (1.8×R-disk grid, M_{tip}=0.439)



(a)



(b)



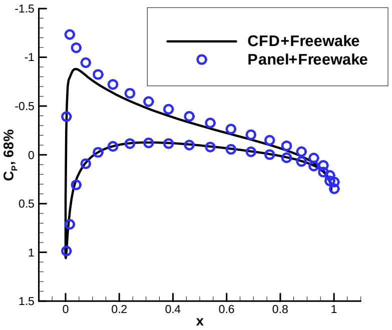
(c)

Fig. 12 Pressure coefficients (1.8×R-disk grid, M_{tip}=0.877)

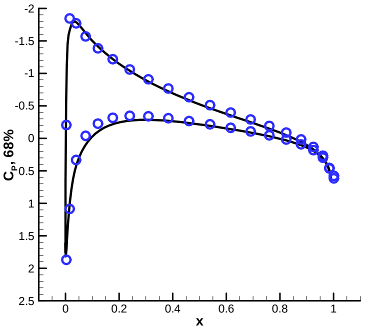
3.2 Rotor aerodynamics in forwarding

This coupled method is applied to the forwarding flight. Rotor model is Caradona-Tung's rotor, and it is used without feathering and flapping in forwarding. This rotor is simulated at pitch angle 8degree, Tip Mach No. is 0.439, advance ratio is 0.2, and climb velocity is 5.96m/s. Grid used in forwarding is 1.8×R-disk grid. Figs. 13~14 show pressure coefficient at each azimuth angle (ψ). Fig. 13 indicate value of r/R=0.68, and Fig. 14 indicate value of r/R=0.96. These are compared with unsteady panel method combined free-wake. This unsteady panel method is incompressible solver, but it is also coupled with free-wake.

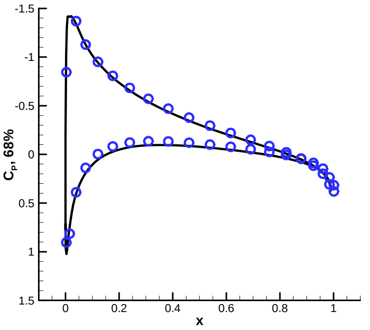
From Figs. 13~14, we can confirm that CFD/Free-wake coupled method results accord with unsteady panel results in forwarding.



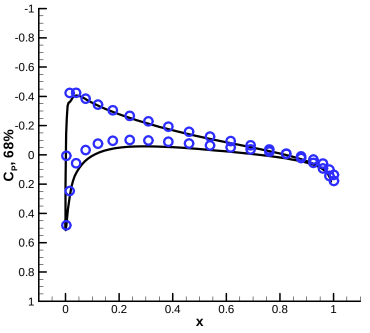
(a) ψ = 0 degree



(b) ψ = 90 degree

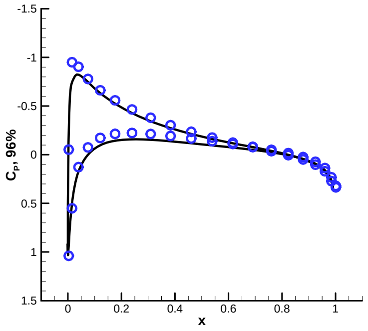


(c) ψ = 180 degree

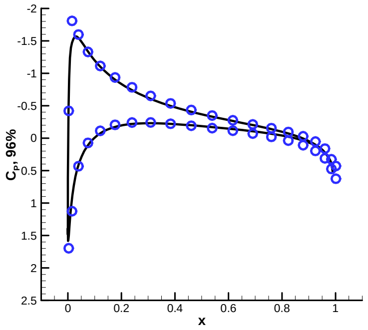


(d) ψ = 270 degree

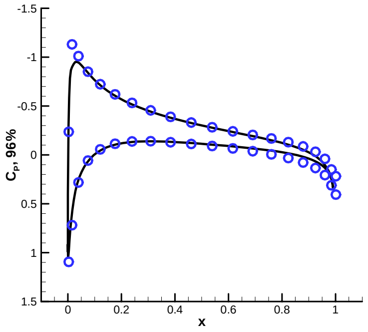
Fig. 13 Pressure coefficients with azimuth (1.8×R-disk grid, r/R=0.68)



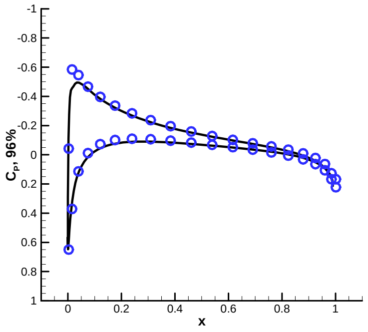
(a) ψ = 0 degree



(b) ψ = 90 degree



(c) ψ = 180 degree



(d) ψ = 270 degree

Fig. 14 Pressure coefficients with azimuth (1.8×R-disk grid, r/R=0.96)

3.3 Computational efficiency

To investigate efficiency, computational time by CFD/Freewake coupled method is compared with time required conventional rotor CFD with source-sink model. Whole procedure use 4 CPU(Pentium4-2.8GHz), and parallel computation technique. Computational time of 10 revolutions is measured and compared.

Table 1 Computational time

	5×R	1.8×R	1.8×R-disk
Source-sink	26.6 hour	18.6 hour	13.8 hour
Freewake	29.5 hour	20.6 hour	15.3 hour

From Table 1, because of induce velocity calculation and wake convection in free-wake routine, method coupled with free-wake requires a little bit more computational time in the same grid. But conventional method with source-sink model cannot be used in small grid (1.8×R, 1.8×R-disk), since this method cannot predict rotor aerodynamics. Actually conventional method is only allowed in 5×R grid. For that reason, the coupled method is more efficient than conventional method. Unsteady flow calculation for noise prediction should be required very small Δt to capture acoustic wave more exactly. Therefore computation efficiency problem is important if we want to use rotor aerodynamics for rotor noise.

4. CONCLUSIONS

In this study, rotor CFD is tightly coupled with free-wake to describe wake effects. And rotor is simulated by this coupled method. We can confirm that method coupled with free-wake is applicable to the rotor aerodynamic simulation.

Effects of boundary condition correction using free-wake depend on the distance between rotor and far-field of CFD domain. Therefore, numerical results in small grid are better than results in large grid. In small grid, inflow and outflow calculated by free-wake are well provided. It means that CFD/Free-wake coupled method can describe wake well.

This tightly coupled method has a good agreement with experimental data. And this method is more accurate and efficient than conventional method. It is also applicable in transonic region as well as subsonic region. All of this procedure can be applied to the forwarding flight.

.

ACKNOWLEDGMENTS

This study has been supported by the KARI under KHP Dual-Use Component Development Program funded by the MOCIE.

REFERENCES

[1] G. R. Srinivasan, V. Raghavan and E. P. N. Duque, "Flowfield Analysis of Modern Helicopter Rotors in Hover by Navier-Stokes Method," *Journal of the American Helicopter Society*, Vol. 38, No. 3, pp.3~13, 1993.

[2] M. E. Berkman, L. N. Sankar, "Navier-Stokes/Full Potential/Free-Wake Method for Rotor Flows," *Journal of Aircraft*, Vol. 34, No. 5, pp.635~640, 1997.

[3] G. Yang, L. Zhuang, "Numerical Simulation of Rotor Flow in Hover," *Journal of Aircraft*, Vol. 37, No. 2, , pp.221~226, 2000.

[4] Q. J. Zhao, G. H. Xu and J. G. Zhao, "New Hybrid Method for Predicting the Flowfields of Helicopter Rotors," *Journal of Aircraft*, Vol. 43, No. 2, pp.372~380, 2006.

[5] K. H. Chung, J. W. Kim, K. W. Ryu, K. T. Lee and D. J. Lee, "Sound Generation and Radiation from Rotor Tip-Vortex Pairing Phenomenon," *AIAA Journal*, Vol. 44, No. 6, pp.1181~1187, 2006.

[6] Y. S. Kim, S. H. Park and J. H. Kwon, "Drag Prediction Workshop Results Using the Parallel Multigrid Solver KFLOW3D," *Proc. of 2nd AIAA CFD Drag Prediction Workshop* , 2003.

[7] S. H. Park, "Prediction Methods of Dynamic Stability Derivation Using the Naver-Stokes Equations," *KAIST Ph. D. Thesis*, 2003.

[8] S. H. Park, Y. S. Kim and J. H. Kwon, "Prediction of Damping Coefficients Using the Unsteady Euler Equations," *Journal of Spacecraft and Rockets*, Vol. 40, No. 3, pp.356~362, 2003.

[9] T. H. Pulliam and D. S. Chaussee, "A Diagonalized Form of an Implicit Approximate-Factorization Algorithm," *Journal of Computational Physics*, Vol. 39, pp.347~363, 1981.

[10] C. H. Sung, S. H. Park, J. H. Kwon, "Multigrid Diagonalized-ADI Method for Compressible Flows," *Proc. of 15th AIAA Computational Fluid Dynamics Conference*, AIAA Paper 2001-2556.

[11] F. X. Caradonna and C. Tung, "Experimental and Analytical Studies of a Model Helicopter Rotor in Hover," NASA TM-81232, 1982.

[12] D. J. Lee and S. U. Na, "Numerical simulations of wake structure generated by rotating blades using a time marching, free vortex blob method," Eur. J. Mech. B/Fluids, Vol. 18, pp.147~159, 1999.

[13] Z. Yang, L. N. Sankar, M. J. Smith and O. Bauchau, "Recent Improvements to a Hybrid method for Rotors in Forward Flight," *Journal of Aircraft*, Vol. 39, No. 5, pp. 804~812. 2002.