Paper No. 071

A NEW HYBRID METHOD OF OVERLAPPING STRUCTURED GRIDS COMBINED WITH UNSTRUCTURED FUSELAGE GRIDS FOR ROTORCRAFT ANALYSIS

Yasutada Tanabe, Shigeru Saito
Aviation Program Group, Japan Aerospace Exploration Agency (JAXA)
7-44-1 Jindaiji Higashi-machi, Chofu, Tokyo 182-8522, JAPAN
Tel.(+81) 422-40-3230, Fax. (+81) 422-40-3235
e-mail: tan@chofu.jaxa.jp

Oki Takayama, Daisuke Sasaki, Kazuhiro Nakahashi
Department of Aerospace Engineering, Tohoku University
6-6-01 Aoba-yama, Sendai, 980-8579, JAPAN

ABSTRACT

The objective of this study is to develop a new integrated analysis platform for rotorcraft, *JANUS,* through combining structured and unstructured grid methods to capture blade tip vortices and handle the complicated flowfield around a realistic geometry of a helicopter fuselage. The *rFlow3D*, a moving overlapped structured grid CFD/CSD solver developed at JAXA specifically for rotorcraft analysis, is used to solve the flowfields around rotating blades and in the Cartesian background grids. The *TAS-code*, an unstructured grid CFD solver developed at Tohoku University, is used for the fuselage. Information exchange between these two codes is carried out via interpolations with the background grid. A common global time integration step is prescribed but a divided time step is determined for each type of grids to satisfy each CFL limit associated with the solver. Validation with the JMRTS experimental database is conducted and a good agreement is attained. The ability of the present solving platform is demonstrated through a test case combining a set of rotor system with a realistic helicopter fuselage geometry.

INTRODUCTION

There is considerable difficulty in solving flow equations around a helicopter by a CFD method for the following reasons; firstly, the flowfield around a helicopter has a wide range of speeds from subsonic to transonic. On the advancing side, the blade tip Mach number reaches as high as 0.9 when the cruise Mach number is only 0.3, while the flow speed may become even zero on the retreating side; secondly, the geometry of a helicopter fuselage may be highly complex due to a large amount of equipment attached on the surface including the landing gear in addition to the horizontal and vertical aerodynamic stabilizer surfaces; finally, tip vortices generated by rotating blades need to be captured accurately to evaluate the effects of interaction between tip vortices and rotating blades.

To tackle these problems, a variety of CFD techniques, such as sliding mesh algorithm [1], vorticity based computational model [2] and multi-block structured grid method [3], have so far been applied to numerical computation of the flow around a helicopter, yet none has conclusively been proven to be universally applicable, in part because of the intricate behavior of flows caused by the main rotor. The highly complex geometry of the fuselage poses a challenging problem for grid generation when using structured grid methods. Generally, it is highly time consuming and requires considirable user expertise.

It is arguably a natural thought to apply the structured and unstructured grid methods to different parts around a helicopter. The unstructured grid method is much more flexible and efficient to handle complicated geometries but the spatial accuracy are generally limited to the 2nd-order, which is not sufficient to capture blade tip vortices. On the other hand, the structured grid method can be easily exended to a higher order of spatial accuracy and is efficient to solve flows around simple geometries. Blade tip vortices can be captured and preserved much better when using a fourth-order scheme than a second-order scheme. With the approach of overlapped grids, the background grid can be chosen as a Cartesian structured grid with a near-body unstructured grid around the complicated fuselage. The shapes of the rotaing blades are generally not so complicated and can thus be treated using structured grids. The wide range of flow speeds in the flowfield around rotorcraft can be treated using an all-speed numerical flux scheme [4] or with a preconditioned method [3].

The Helios (Helicopter Overset Simulations) Computational Platform is under development in the US which uses an unstructured near-body grid together with Cartesian meshes in the off-body region and is currently applied to rotorcraft flowfields [5]. Several well-established solvers are utilized to build a multi-solver paradigm for CFD using overset unstructured and structured adaptive Cartesian grids [6] . In [7], NSU3D code is used for the near-body solver and SAMARC code for the off-body solver to solve the complex flow around a helicopter fuselage. With a software integration framework, many software modules that handle the Comprehensive Analysis, Fluid-Structure Interface, Domain Connectivity, Mesh Deformation and so on are implemented in the Helios Computational Platform. Several test cases for validation of Helios is described in [8] and reasonable agreement with the experiments and computational results with other well-known codes are found. Although the result around a realistic full configuration helicopter has not been published yet, the test results till now indicate that it is a promising approach to carry out the multidisciplinary analysis around rotorcraft.

JAXA has been working on CFD/CSD analysis based on a moving overlapped structured grid method and the *rFlow3D* code was developed especially for rotorcraft applications [4]. A fourth order spatial resolution offers a desirable capturing capability of tip vortices from blades. Although it only has limited capability to handle complex fuselage shapes, good results are obtained for test cases with relatively simple fuselage shapes using the built-in SLAU all-speed scheme [9]. This code has an integrated module to carry out the CSD analysis for elastic deformations of rotor blades [10]. A trim module for a full-configuration helicopter is also under development. To handle the realistic helicopter fuselage and release the burden of grid generation around the complex shapes, a proposal was made by JAXA to Tohoku University to use the unstructured grid solver TAS-code [11] for the fuselage and develop an information exchange interface between these two codes. This newly developed hybrid code is named JANUS (Japan's Advanced Numerical platform based on Unstructured and Structured grids), after the two-faced Greek God of gateways.

The TAS-code (stands for Tohoku University Aerodynamic Solver code) [11] was developed by the research group at Tohoku University led by Prof. K. Nakahashi, one of the co-authors of this paper. It is a successful code based on unstructured grid method in Japan and has been selected as a main aerodynamic analysis tool for the MRJ (Mitsubishi Regional Jet). To handle the wide range of speeds around a helicopoter fuselage, SLAU scheme was implemented into this unstructured grid CFD solver. Its validation was conducted with d'Alembert's paradox and it was confirmed that the SLAU scheme can diminish numerical errors at low Mach numbers, which are often typical of flows around a helicopter [12].

Validation of the hybrid code was first carried out using the ROBIN test cases as described in [12]. In this paper, validation with test cases selected from original experimental JMRTS database [13] is reported. In addition, a computation with a rotor and a realistic fuselage is carried out to demonstrate the high capability of the JANUS code.

NUMERICAL METHODS

Overset of Structured and Unstructured Grids

The base of the present coupling flow solver is the structured grid CFD solver (rFlow3D), which covers almost the entire computational domain. The unstructured grid CFD solver (TAS-code) is adopted only for the helicopter fuselage.

Figure 1 shows the computational domains for rotor blades and a fuselage. As shown in the figure, the grid is composed of the following four grids: the outer background grid, the inner background grid, the blade grids and the fuselage grid. The outer background grid covers the entire computational domain. The inner background grid is created to cover both blade and fuselage grids, and is a fine structured grid to capture the tip vortices precisely. The blade grids are structured grids around blades and can rotate and deform in simulation. Only the near field of fuselage is meshed by an unstructured grid method for treating complicated helicopter geometries. Each grid solves its own computational domain independently and exchanges flow variables via background grids at the same time step.

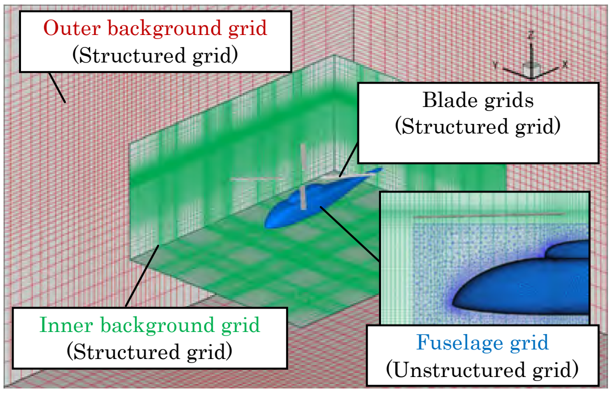


Fig. 1 Computational domains of four layers of grids.

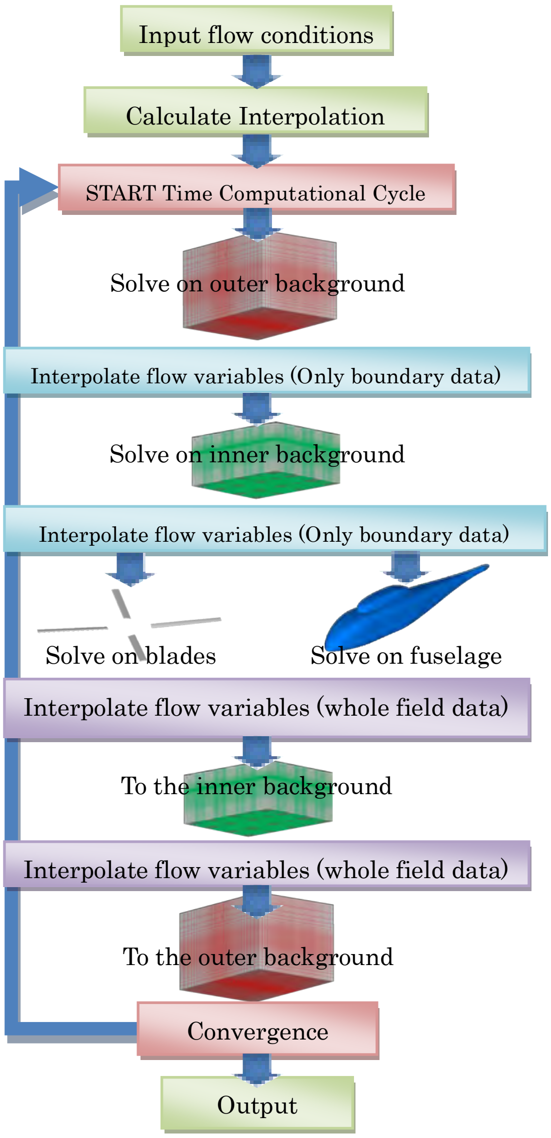


Fig. 2 The procedure of the present hybrid method

Procedure for the Hybrid Method

Figure 2 shows the computational procedure of the hybrid method. The base CFD solver is the *rFlow3D*, and the *TAS-code* is built into the *rFlow3D* as one of the subroutines that solve flows around a fuselage in an unstructured grid.

The detailed procedure for the hybrid method is as follows: pre-processing is required to calculate the interpolation factors, which are calculated from each grid before starting the calculation. All interpolation factors are saved in the memory, thus it is not necessary to calculate them again during the computational cycles. In the computational cycle, firstly, the outer background grid solves the own domain with the imposed free-stream condition. The physical quantities are interpolated to the outer boundary of the inner background via the trilinear interpolation. Secondly, making use of interpolated boundary data, the flowfield in the inner background is solved. Similarly, the inner background data is interpolated to the outer boundaries of the blades and the fuselage grids. The flows around the blades and fuselage are then solved separately. After the computation of the blades and fuselage grids, all the computed quantities in the blades and the fuselage grids are interpolated to the whole inner background grid points. Finally, the inner background results are interpolated to the outer background. This computational cycle is repeated at each time step. In the present CFD solver, the main computational domain is the inner background. The flow variables of the fuselage and the blade grids exchange via the inner background only at each time step. Using this procedure, the flow-field around the complicated geometry such as a helicopter body can be solved.

Interpolation Method for Flow Variables between Structured and Unstructured Grids

In the present solver, the flow variables are exchanged between grids at each time step. Tri-linear interpolation is used to interpolate flow variables from the background grids (structured) to the outer boundaries of other grids (structured/unstructured). For instance, physical quantities at "node*X*" of a tetrahedral element *A* need to be interpolated from the hexahedral element *B* in the background grids as shown in Fig. 3. The following tri-linear interpolation is used:

$$f(nodeX) = [\{f_1(1-s)+f_2 s\}(1-t)+\{f_3(1-s)+f_4 s\}t](1-u) + [\{f_5(1-s)+f_6 s\}(1-t)+\{f_7(1-s)+f_8 s\}t]u \quad (1)$$

where "f(node*X*), f1, f2, …" are the physical quantities or conservative quantities of a node such as density, velocities, pressure, momentum and energy. The indices 1 to 8 represent nodes of a hexahedral element *B* of the structured grid. "s, t, u" are the interpolation factors that are defined by the distance for each coordinate between target node (nodeX) and surrounding nodes. For example, if the "nodeX" is located at the center of the cell, the interpolation factors "s, t, u" are 0.5. When the coordinates and the physical quantities of eight nodes of the cell and the coordinates of the "node*X*" are specified, the flow variables are automatically interpolated. This tri-linear interpolation is also used to update a node of the structured grid.

When the whole data are interpolated from an unstructured grid to a structured grid, the linear approximation equation is needed to be solved by calculating an inverse matrix. When the "node*X*" of a hexahedral element *B* (structured grid) is considered to be interpolated from the tetrahedral element *A* (unstructured grid) as shown in Fig. 4, the formula of the linear approximation equation is as follows:

$$f(nodeX) = f_0 + f_x x + f_y y + f_z z \quad (2)$$

$$\begin{pmatrix} 1 & x_1 & y_1 & z_1 \\ 1 & x_2 & y_2 & z_2 \\ 1 & x_3 & y_3 & z_3 \\ 1 & x_4 & y_4 & z_4 \end{pmatrix}\begin{pmatrix} f_0 \\ f_x \\ f_y \\ f_z \end{pmatrix} = \begin{pmatrix} f_1 \\ f_2 \\ f_3 \\ f_4 \end{pmatrix} \quad (3)$$

where "f(node*X*), f1, …, f4" are the physical quantities or conservative quantities of a node. "f0, fx, fy, fz" are the interpolation factors that are obtained by calculating the inverse matrix (3). "x, y, z" are the coordinates of each node. The indexes 1 to 4 represent nodes of a tetrahedral element A of an unstructured grid.

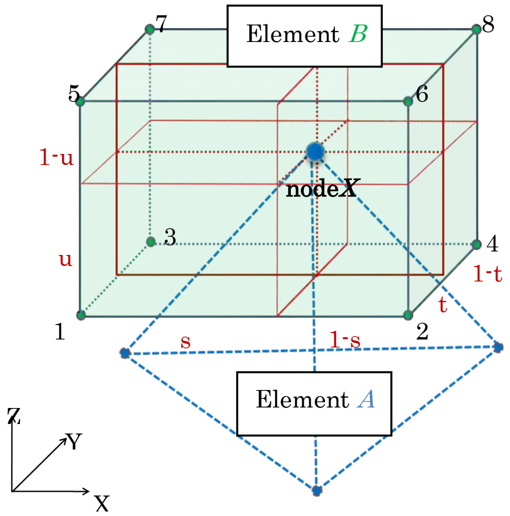


Fig. 3 The trilinear interpolation from structured grid to unstructured grid.

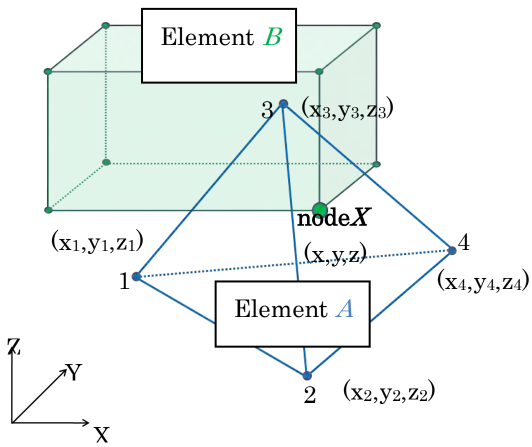


Fig. 4 The linear interpolation from unstructured grid to structured grid

Flow Solvers

In this study, the structured grid CFD solver (rFlow3D) and the unstructured grid CFD solver (TAS-code) are combined to solve flows around the helicopter blades and the fuselage. Details about the *rFlow3D* code can be found in [4] and details about the *TAS-code* can be found in [11]. Brief summaries for each flow solver are as follows:

rFlow3D (Structured grid CFD solver):
- Governing Equation: Compressible Euler equations
- Space discretization: Cell-centered finite volume method
- Inviscid flux: SLAU (Simple-Low-dissipative AUSM)
- Time Integration: Four stages Jameson integration method (for Cartesian background grids) & LUSGS/DP-LUR implicit method extended by Dual Time-stepping method (for blade grids, also for structured fuselage grids)
- Reconstruction: 4th order Compact MUSCL TVD interpolation method (FCMT)

TAS-code (Unstructured grid CFD solver, for unstructured fuselage grid):
- Governing Equation: Compressible Euler equations
- Space discretization: Cell-vertex finite volume method
- Inviscid flux: SLAU (Simple-Low-dissipative AUSM)
- Time Integration: LUSGS implicit method
- Reconstruction: Volume average method (2nd order)

A common global time integration step is prescribed by the user but a divided time step is determined for each type of grids to satisfy each CFL limit associated with the solver. In this study, a global time step corresponding to main rotor azimuth angle of 0.1 deg is prescibed but generally, 3 or 4 time divisions are required in the solver of the blade grid and 1 or 2 steps in the fuselage grid.

RESULTS AND DISCUSSIONS

Test cases based on the JMRTS experimental database [13] are selected for the validation of the newly constructed JANUS code. The rFlow3D code is validated based on the JMRTS in [14]. The results obtained using JANUS code are compared with the experimental results and also with the results obtained using rFlow3D code, where a structured grid around the JMRTS fuselage model is generated and a fourth order solver is used.

The unstructured grid on the surface of the JMRTS model is shown in Fig. 5. A near-body grid around the JMRTS model is formed and overlapped with the Cartesian background grid as shown in Fig. 6. There are in total 92,862 nodes in the unstructured fuselage grid.

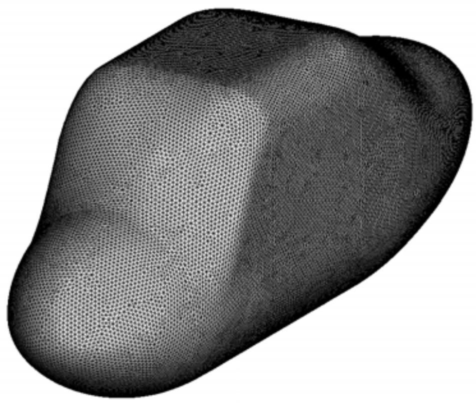


Fig. 5 Unstructured grids on the surface of JMRTS fuselage model

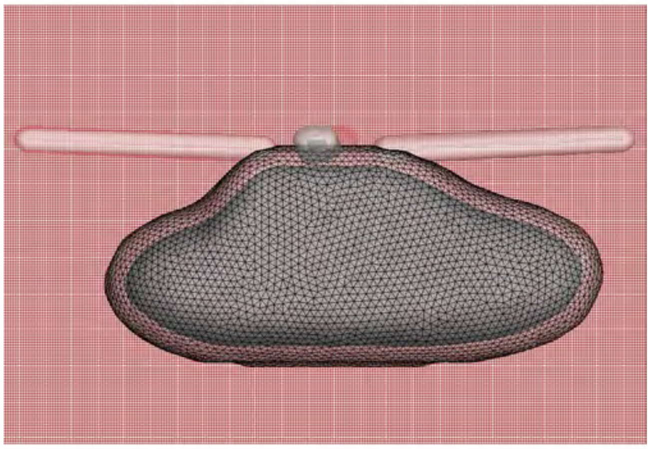
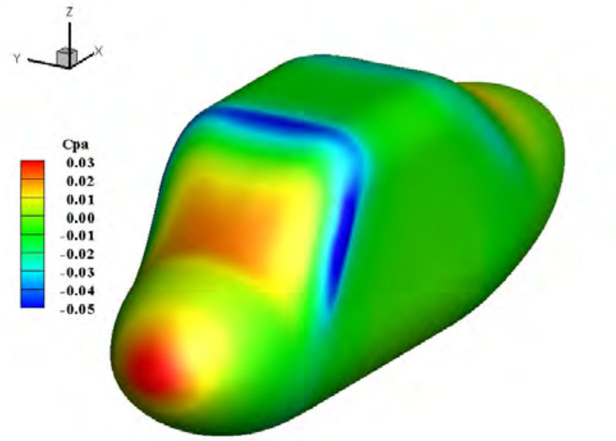
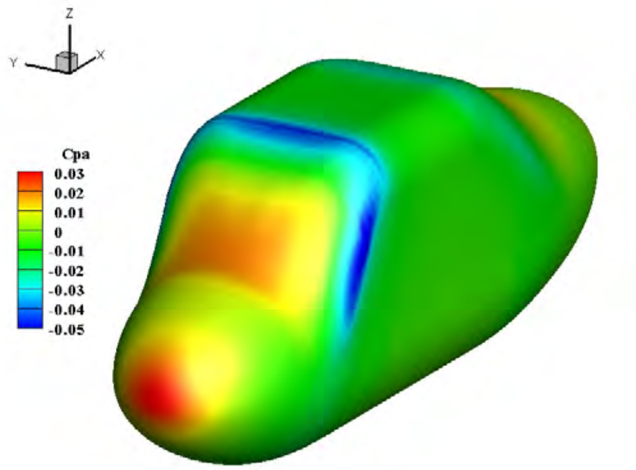


Fig. 6 Overlapping grids for JMRTS model

At first, a test case around the isolated JMRTS fuselage is simulated. The freestream Mach number $M_\infty = 0.175$ and the fuselage is tilted nose down as $\alpha_s = -2$deg. The calculated pressure distributions are compared in Fig. 7. Nearly same distributions are obtained.



(a) rFlow3D result



(b) JANUS result

Fig. 7 Comparison of surface pressure distributions on the isolated JMRTS fuselage model

The pressure distributions along the center line on the isolated JMRTS fuselage model are shown in Fig. 8 with comparison to the experimental results. Good agreement between the calculations and experimental measurement is observed in the forebody part of the fuselage. On the aft body of the fuselage, good agreement remains between the two calculations but significant discrepancies with the experimental measurement. There may be several possible reasons: a) the calculations are based on the Euler equations where the viscous terms are omitted. Possible flow separations in the aft body can not be predicted; b) in the experiment, only the blades are removed. The drive shaft and hub are not removed and rotates during the test. The flowfield downward of the drive shaft should be largely influenced by the wake of the rotating shaft and hub; c) also the opening around the drive shaft is not modeled in the calculations. The pressure distribution along the crossline is shown in Fig. 9. Discrepancies between the predictions and the measurement are observed in the upper surface of the fuselage where the rotating hub and drive shaft are considered to be the cause. However, for the isolated fuselage case, there are few differences between predictions using the *rFlow3D* and *JANUS* codes.

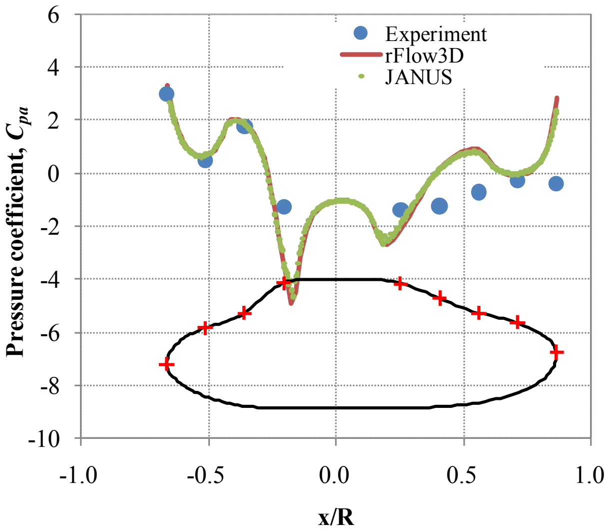


Fig. 8 Comparison of pressure along centerline on isolated JMRTS fuselage model

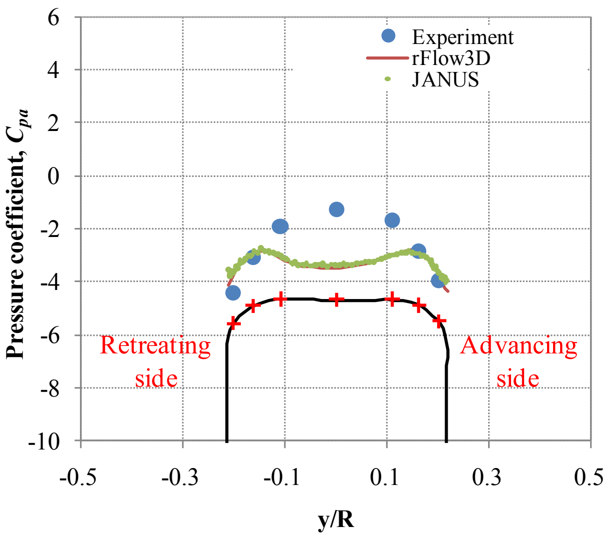


Fig. 9 Comparison of pressure along crossline (X/R=-0.21) on isolated JMRTS fuselage model

For the simulation of rotor/fuselage interaction, a test case with advance ratio of 0.16 from Table 1 is selected. The control inputs to the rotor are fixed based on the experiment. Thrusts predicted with fixed control are shown in Fig. 10. The *rFlow3D* result covers the whole test range of advance ratios, and underprediction in thrust is observed with increasing advance ratios. At advance ratio of 0.16, about 10% underpredictions compared with measurment exist both for the *rFlow3D* and *JANUS* results, where the *JANUS* is slightly lower than the *rFlow3D*. Although the causes for the underpredictions in thrust need further studies, the influences of the wind-tunnel walls and the support strut below the fuselage are arguably in question. The good agreements between the *rFlow3D* and *JANUS* results indicate that the unstructured grid solver *TAS-code* for the fuselage part is successfully integrated into the overlapped grid system.

Table 1: JMRTS forward flight test conditons

Test condition						
M_∞	0	0.029	0.059	0.088	0.131	0.161
M_{tip}	0.562	0.561	0.561	0.561	0.560	0.559
α_S	0	-2	-2	-2	-2	-2
μ	0	0.05	0.1	0.16	0.23	0.29
C_T	4.74E-03	4.73E-03	4.72E-03	4.77E-03	4.76E-03	4.78E-03

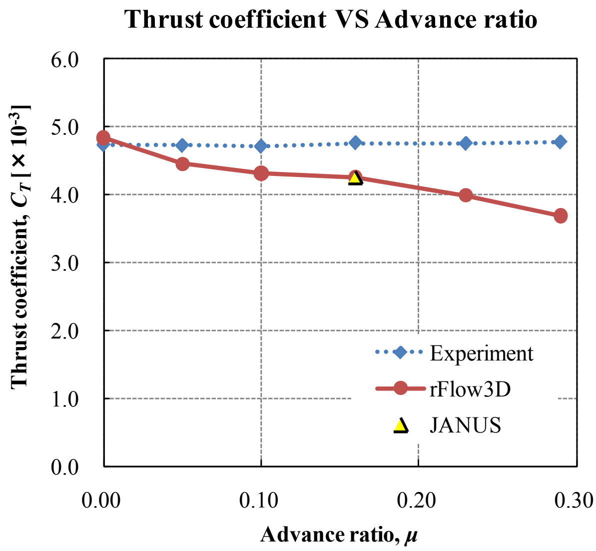


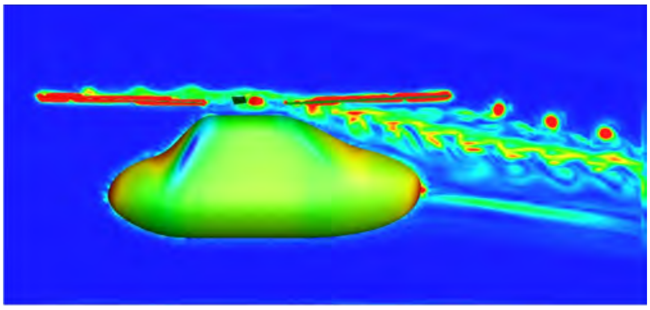
Fig. 10 Comparison of rotor thrust

For the case of advance ratio μ=0.16, the flowfield is visualized with the iso-surface plots of Q-criterion as shown in Fig. 11. The tip vortices of the rotor are captured fairly well in both cases. Some differences are observed in the fuselage wake, where the *JANUS* result gives a more diffused wake. To study this difference more closely, slice plots of this flowfield are made as shown in Fig. 12. The vortices from the inner root of the blade can be clearly seen in the *rFlow3D* result as shown in Fig. 12(a). In the *JANUS* result (Fig. 12 (b)), however, the wake near the fuselage is diffused and vortex cores can not be identified. This is due to a relatively coarse mesh used in the unstructured grids around the fuselage in this case and the spatial accuracy of the unstructured grid solver is 2nd-order, compared with the 4th-order solver used in the *rFlow3D*. To obtain a better solution around the fuselage, finer unstructured grids around the fuselage should be used with the *JANUS* code.

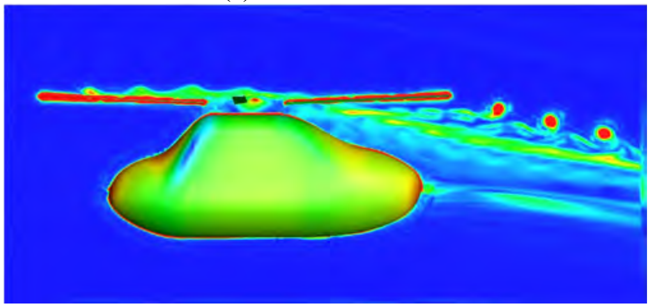
The comparison of surface pressure for μ=0.16 case at Ψ=0 deg is shown in Fig. 13. No significant difference can be seen in these two results.

The variations in the surface pressure compared with the experiments are shown in Fig. 14. Good agreement is obtained with regard to the peak phases. The amplitudes of the variations are also well predicted except in the area near the hub. However, the wave forms predicted by the *JANUS* is much more oscillatory than that by the *rFlow3D* code especially in the area near the hub. Further studies are required to improve the accuracy of unsteady flow predictions with the unstructured solver.

Variations in pressure on the blade surface are also compared as shown in Fig. 15. Good agreement is obtained with the measurements. No differences can be observed in the results between the *rFlow3D* and *JANUS*. The solvers for the blade grids are identical and it is considered that the influences from the fuselage are not remarkable here.

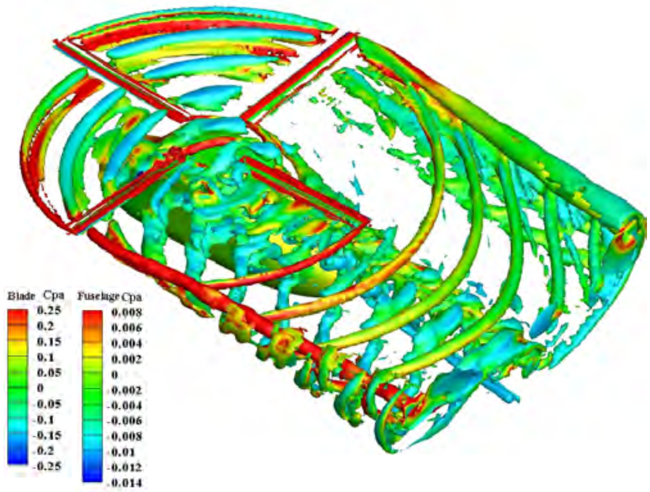


(a) rFlow3D result

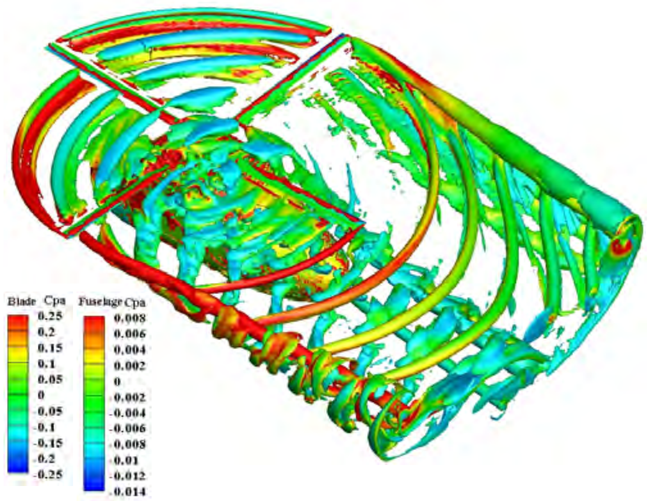


(b) JANUS result

Fig. 12 Comparison of wakes on center slice for advance ratio μ=0.16

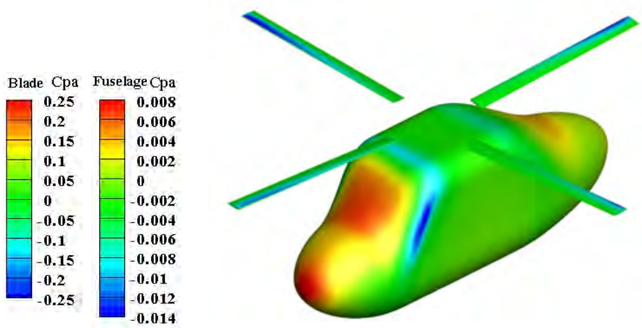


(a) rFlow3D result

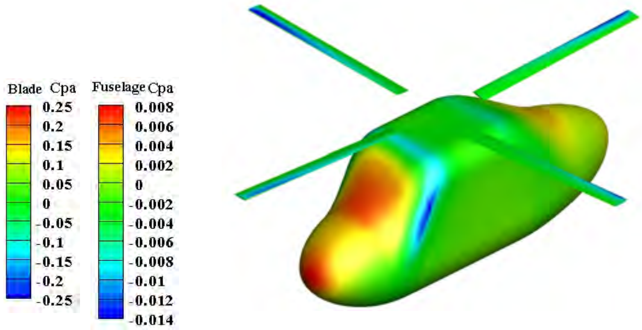


(b) JANUS result

Fig. 11 Comparison of iso-surface of Q-criterion (Q=0.008) for advance ratio μ=0.16



(a) rFlow3D result



(b) JANUS result

Fig. 13 Comparison of pressue distributions on fuselage and blade for advance ratio μ=0.16 at Ψ=0 deg

Finally, the new CFD solver was applied to a flow around a fuselage model which was designed after an existing helicopter model as a test case of the realistic helicopter geometry as shown in Figure 16. In this computation, the new solver correctly predicted tip vortices and differences between the starboard and port in the surface pressure on the horizontal and vertical stabilizers. It is concluded that the new CFD solver is remarkably effective in the numerical computation of flows around the realistic geometry of a helicopter.

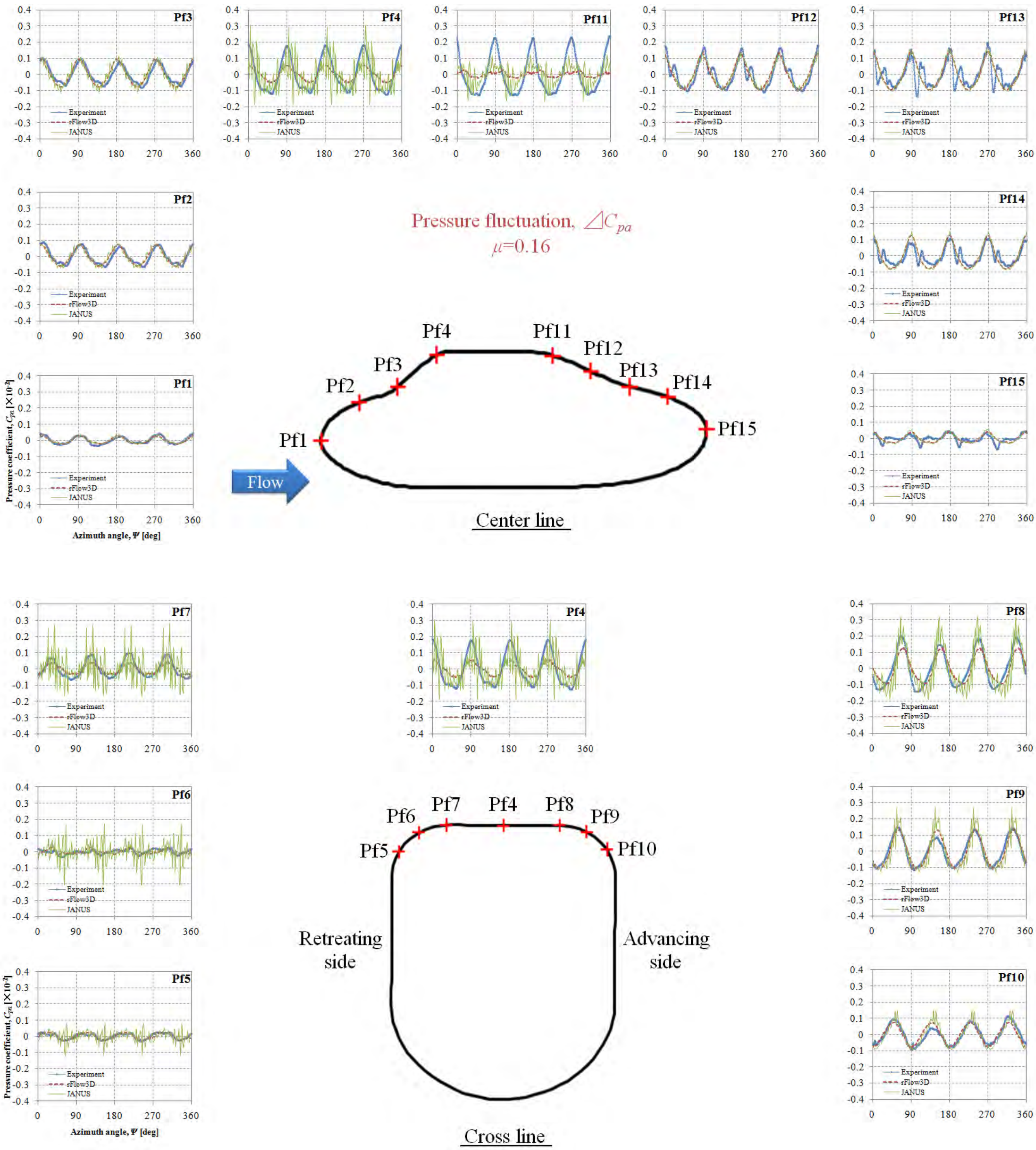


Fig. 14 Comparison of pressure fluctuations on fuselage surface for advance ratio μ=0.16

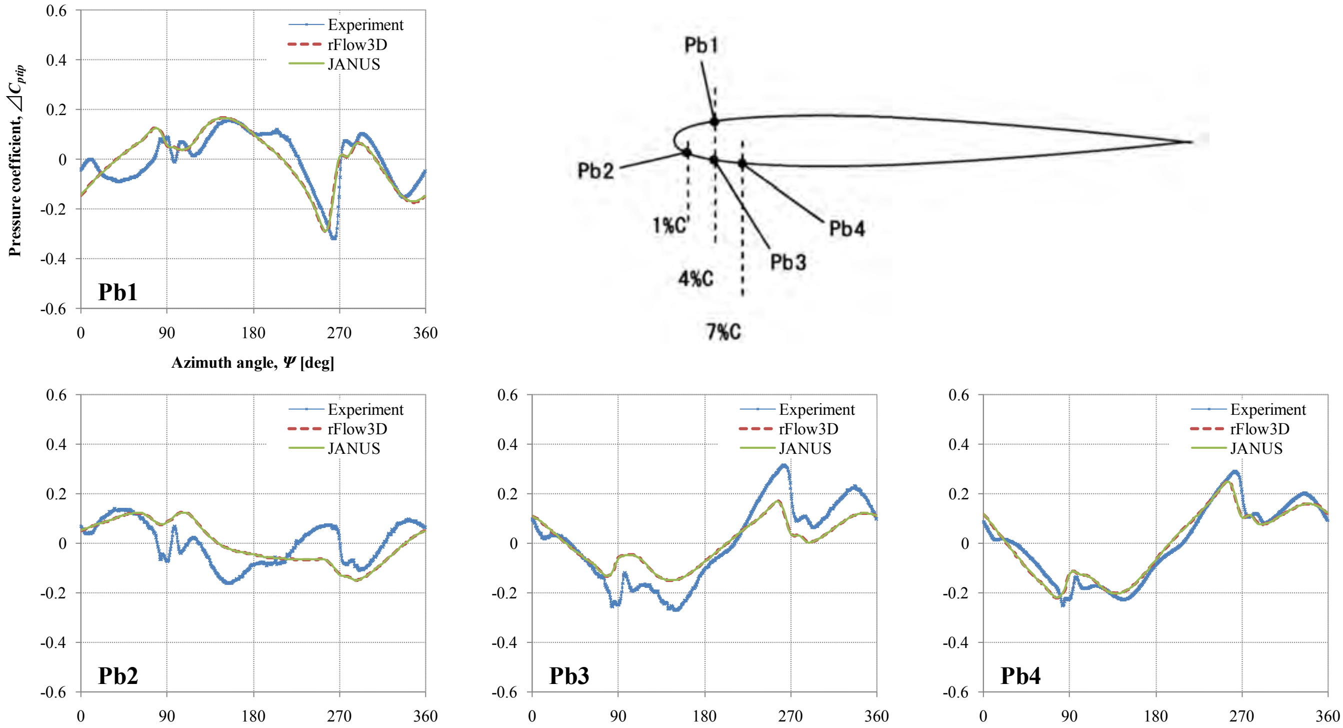


Fig. 15 Comparison of pressure fluctuations on blade surface for advance ratio μ=0.16

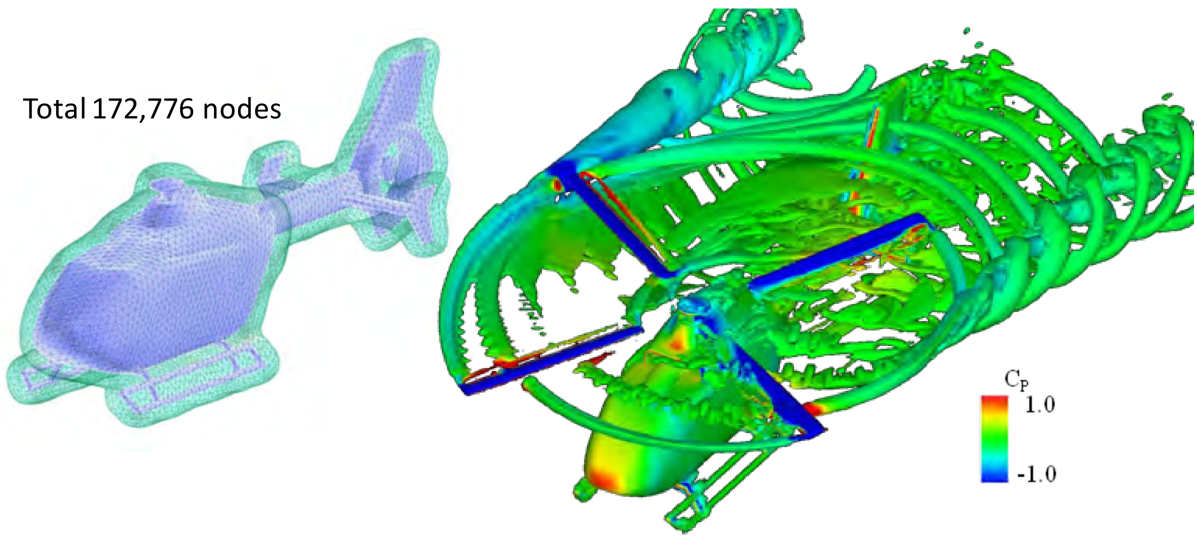


Figure 16: Test calculation for a realistic helicopter configuration

CONCLUDING REMARKS

A new integrated analysis platform for helicopter, *JANUS*, is developed through combining structured and unstructured grid methods to capture tip vortices from blades and handle the complicated flowfield around a realistic geometry of a helicopter fuselage.

A moving overlapped structured grid CFD/CSD solver *rFlow3D*, developed in JAXA specifically for rotorcraft analysis, is used to solving the flowfields around the rotating blades and for the Cartesian background grids. An unstructured grid CFD solver, *TAS-code*, developed in Tohoku University, is used for the complex fuselage. Information exchange between these two codes are carried out via interpolations with the background grid.

A common global time integration step is prescribed by the user but a divided time step is determined for each type of grids to satisfy each CFL limit associated with the solver.

Validations with the JMRTS experimental database are performed and a good agreement is attained. However, the pressure variations on the surface of the fuselage predicted by the *JANUS* code is oscillatory compared with the *rFlow3D* and the experimental results. Further improvements are required to inprove the accuracy of unsteady flows with the unstructured solver for the fuselage.

The ability of JANUS solving platform is demonstrated through a test case combining a set of rotor with a realistic helicopter fuselage geometry.

ACKNOWLEDGMENT

The authors greatly appreciate the assitance of Mr. H. Sugawara, Ryoyu Systems, Co. Ltd in grid generation and data processing. The wind tunnel testing to generate the JMRTS database was supported by Mitsubishi Heavy Industry.

REFERENCES

[1] Steijl, R., Barakos, G. and Badcock, K., "A Framework for CFD Analysis of Helicopter Rotors in Hover and Forward Flight," *International Journal for Numerical Methods in Fluids,* Vol.51, Issue 8, pp.819-847, 2006.

[2] Kenyon, A.R. and Brown, R.E., "Wake Dynamicsand Rotor-Fuselage Aerodynamic Interactions," *Journal of the American Helicopter Society*, Vol.54, No.1, January, 2009.

[3] Potsdam, M., Sankaran, V. and Pandya, S., "Unsteady Low Mach Preconditioning with Application to Rotorcraft Flows," *18th AIAA CFD Conference,* 25-28 June 2007, Miami, FL, *AIAA 2007-4473*, 2007.

[4] Tanabe, Y. and Saito, S., "Significance of All-Speed Scheme in Application to Rotorcraft CFD Simulations," *The 3rd International Basic Research Conference on Rotorcraft Technology,* Nanjing, China. October, 2009.

[5] Sankaran, V., Sitaraman, J., Wissink, A., Datta, A., Jayaraman, B., Potsdam, M., Mavriplis, D., Yang, Z., O'Brian, D., Saberi, H., Cheng, R., Hariharan, N., and Strawn, R., "Application of the Helios Computational Platform to Rotorcraft Flowfields." *AIAA-2010-1230*. 2010.

[6] Sitaraman, J., Katz, A., Jayaraman, B., Wissink, A., and Sankaran, V., "Evaluation of a Multi-Solver Paradigm for CFD using Overset Unstructured and Structured Adaptive Cartesian Grids." *AIAA-2008-660*. 2008.

[7] Wissink, A., Sitaraman, J., Sankaran, V., Mavriplis, D., and Pulliam, T., "A Multi-Code Python-Based Infrastructure for Overset CFD with Adaptive Cartesian Grids." *AIAA-2008-927*. 2008.

[8] Sitaraman, J., Floros, M., Wissink, A. and Potsdam, M., "Parallel Unsteady Overset Mesh Methodology for a Multi-Solver Paradigm with Adaptive Cartesian Grids," *AIAA-2008-7177*, 2008.

[9] Shima, E. & Kitamura, K., "On New Simple Low-Dissipation Scheme of AUSM-Family for All Speeds," *47th AIAA Aerospace Sciences Meeting, Orlando, FA, January 5-8 2009, AIAA Paper* 2009-136.

[10] Tanabe, Y. and Saito, S., "A Simpleified CFD/CSD Loose Coupling Approach for Rotor Blade Deformation," *JAXA-RR-08-008E*, 2009

[11] Nakahashi, K., Ito, Y. and Togashi, F., "Some challenges of realistic flow simulations by unstructured grid CFD", *Int. J. for Numerical Methods in Fluids*, Vol.43, pp.769-783, 2003.

[12] Takayama, O., Sasaki, D., Nakahashi, K., Tanabe, Y. and Saito, S., "Coupling Method of Unstructured and Structured Grids for Flow Around ROBIN Configuration," *2nd International Forum on Rotorcraft Multidisciplinary Technology*, Seoul, Korea, October 19-20, 2009.

[13] Tanabe, Y., Saito, S., Kobiki, N., Murota, K., Sugawara, H., Hayashi K., and Hiraoka, K., "An Experimental Study of Rotor/Fuselage Interaction," *35th European Rotorcraft Forum*, Hamburg, Germany, September 22-25, 2009.

[14] Tanabe, Y. and Saito, S., "Validation of a New Rotor Flow Code Using JMRTS Experimental Data," *The American Helicopter Society 66th Annual Forum*, Phoenix, AZ, May 11-13, 2010.