NUMERICAL ANALYSIS OF THREE-DIMENSIONAL VISCOUS FLOWS AROUND AN ADVANCED COUNTERROTATING PROPELLER

Yuichi Matsuo*and Shigeru Saito†
National Aerospace Laboratory
7-44-1, Jindaijihigashi, Chofu, Tokyo 182, Japan

Abstract

Three-dimensional viscous flows around an advanced counterrotating propeller are investigated by solving unsteady Navier-Stokes equations. An efficient solution procedure has been developed and applied. The solver is originally based on an implicit finite difference method, and also uses unsteady algorithm, block structured grid system and sliding boundary technique in order to treat the relative motion between fore and aft rotors. Numerical results are presented for transonic flow of cruise flight and low-speed flow of take-off. Predicted aerodynamic loadings at take-off are compared with experimental data.

Introduction

An advanced turboprop propulsion system is currently of great interest since it offers higher propulsive efficiency than conventional turbofans even at high subsonic Mach numbers, leading to the potential savings of aircraft fuel consumption. In addition, counterrotating propeller (CRP) can afford to attain much higher propulsive efficiency than a conventional single-rotation propeller (SRP) by recovering swirl loss. However instead, the CRP flow field is more complex than SRP due to the mutual motion of fore and aft rotors. In order to understand such complex flow field and ultimately find optimized CRP configurations, a number of analytical and experimental studies have been conducted[1-3]. In the National Aerospace Laboratory also, basic low-speed wind tunnel test[4] is conducted for a preliminary designed scale model shown in Figure 1.

Meanwhile, with the advent of high speed computers, applications of computational fluid dynamics to CRP has been popular in recent years. It not only overcomes the limitations of classical propeller theories, but also provides a large amount of information not readily obtained by measurements. Roughly speaking, two types of approaches to solve the CRP flow field have emerged so far. One is a quasi-steady method which pursues a time-averaged solution. Celestina et. al[5] solved the averaged-passage equation system proposed by Adamczyk[6]. The other is purely unsteady approach which directly treats the relative motion of fore and aft rotors. So called zonal method using a sliding grid can often be seen in turbomachinery rotor-stator interaction problem, and Whitfield et al.[7] first applied that to CRP. Janus and Whitfield[8] proposed their own locally distorted grid approach.

Along these approaches, a number of computational efforts concerning CRP have been made[9-12], but most of them are focused upon inviscid flow or Euler analysis mainly due to the computer power restriction. Although such gives very useful information for design, its solution is rather sensitive to grid distribution, thus a poor distributed grid sometimes leads to divergence of the computation. Especially in a low-speed or take-off condition simulation, since the inflow has a relatively large angle of attack against each blade, Euler analysis sometimes predicts an unphysical suction peak. Further, important viscous effects caused by vortex-blade or wake-blade interaction are not predicted.

In order to predict the CRP flow field more accurately and evaluate such viscous effects, viscous flow analysis which solves Navier-Stokes equations in some way is required about CRP. The most formidable difficulty of Navier-stokes computation is how to deal with increased grid points necessary to resolve blade surface boundary layers. In turbomachinery application, Rai's zonal approach[13] using sliding grid with O-type grid is famous, and a direct extension of that method is one of the possible ways to compute the CRP flow. But the original method is too complicated to implement, and to make matters worse, thin airfoil profiles of the CRP blade prevent using the O-type grid topology around blade.

In this paper, an efficient approach to solve the viscous flow field around CRP is presented. The Reynolds-averaged Navier-Stokes equations are solved using finite difference method, unsteady algorithm, block structured grid with H-type topology and sliding grid technique. In the following sections, solution procedures to solve the CRP flow field are briefly described. Then, two representative results which correspond to different flow conditions, one is for transonic cruise and the other is low-speed take-off, are presented and discussed.

Solution Procedure

The actual flow through CRP is extremely complex, so it seems impossible to completely handle such flow field even by the current supercomputers. Therefore, suitable mathematical modelings or physical simplifications should be introduced to the computation. In this paper, it is assumed that the inflow is uniform and parallel to the rotating axis, the rotors' rotating speed is both constant and that the entire flow field is circumferentially periodic. Under these assumptions, the present code for CRP has been developed. It is based on the Navier-Stokes solver for SRP developed by Matsuo et al. Since the numerical implementation is

*Researcher, Member AIAA.
†Senior researcher, Member AIAA.

described in detail in Ref. 14, the outline and additional techniques are summarized hereafter.

Basic solver Basically the Reynolds-averaged Navier-Stokes equations written for a generalized curvilinear coordinate system fixed to a rotating system are solved by using finite difference method with Baldwin-Lomax[15] algebraic turbulence model. The time integration uses Euler implicit scheme and an approximate factorization technique[16]. The use of the implicit method removes the stiffness problem related to fine mesh and consequently reduces CPU time to obtain a solution. As a spacial discretization formula, an upwind biased TVD scheme[17] is applied in order to enhance numerical accuracy.

Unsteady algorithm Even in the simplest case treated here, the flow field is inherently unsteady. So a time-accurate unsteady algorithm is added to the above solver. In this paper, Newton subiterations are used at each time level. At the same time, a constant time step is used in the entire region instead of using a local time step. In addition, in order to treat the relative motion of fore and aft rotors, sliding grid technique is introduced in the same way as the turbomachinery application[13]. This algorithm is easily implemented without serious changes of the original code.

Block structured grid Once the unsteady algorithm is established, the next problem is how to reduce the computational time increased by many grid points. If the above unsteady algorithm is applied to the whole computational region, it takes a long time to obtain a solution because the subiterations are also used for entire region. However, considering that the flow field is almost steady except near rotors, it can be found that the unsteady algorithm is not always necessary for the far field from the rotors. Therefore, in this approach the whole grid is at first divided into four independent blocks. The first block is in front of the fore rotor and stationary, the second around the fore rotor and rotating, the third around the aft rotor and rotating, and the forth behind the aft rotor and stationary. Next, the Newton subiterations are applied only to the two blocks rotating with rotors. In all blocks H-type grid topology is utilized, and each is connected through sliding zonal boundaries. If doing so, grid points can be efficiently used even by using inefficient H-grid, and as a result the increase of the computational time is not so serious. Figure 2 shows a typical computational grid used here. On the assumption of flow periodicity in the circumferential direction, the computational domain contains only one blade passage. These H-type grids can easily be generated by using algebraic interpolation technique, and grid points are clustered in the vicinity of the walls so as to resolve viscous boundary layers.

Boundary conditions Final important issue in computing the CRP flow field is boundary conditions. Several types of boundaries are encountered in the CRP flow field. On the blade surfaces, non-slip and adiabatic conditions are imposed, whereas on the spinner surface slip conditions are imposed for simplicity. On upstream and downstream periodic boundaries, the values of two points which correspond to each other is set equal by using the averaged value of two interior points. For inflow and far field boundaries, free stream values are specified. At the exit boundary, only static pressure is fixed by solving radial equilibrium, and other flow variables are extrapolated from inside. On the sliding boundaries, special attention should be paid. First, the relative position of adjacent blocks is defined at each time level, and then linear interpolation is used to transfer information across the boundary. Although this treatment is nonconservative, it is used for computational simplicity and reduction of the CPU time. Instead, small area of adjacent blocks is overlapped as shown in Figure 2 in order to maintain the numerical accuracy on the sliding boundaries.

Results and Discussion

Two sets of viscous flow computations at different flow speeds are executed for the scale model shown in Figure 1. The configuration is an eight bladed, pusher type, counter-rotating propeller, and experimentally studied in the NAL. Parameters of the computations are listed on Table 1.

Transonic Flow Case The first results shown here is for transonic flow of cruise flight condition. The grid contains axially 174 points, radially 51 points and circumferentially 21, 41, 101 and 41 points with four blocks respectively. Note that the third grid is fine enough to resolve the interaction between fore rotor wakes and aft rotor blades, and that the overall distribution is determined after preliminary grid study investigating total number of points, minimum grid size and distance to the far field boundary.

Figure 3 shows time histories of aerodynamic loadings of both fore and aft rotors. In this case, time step is selected so that each rotor rotates by one passage (45 degrees) per 400 iterations, and in total 2000 iterations are carried out. As can be seen, the loading coefficients show a little irregular behaviour in their initial stages, but after about 1200 iterations they reach to sinusoidal mode. Therefore this iteration number is quite reasonable to get the unsteady solutions, and also the CPU time is not so long. This shows the present code is efficient enough to obtain unsteady viscous flow solutions of CRP.

Figure 4 shows time histories of pressure contours on a midspan section in a cyclic manner. In spite of the nonconservative treatment on the sliding boundaries, the shock system looks smoothly going through the boundaries. It can be seen that shock strength of the aft rotor changes periodically while that of the fore rotor don't change drastically. Unfortunately, since no data is available in this case, any comparison with experiment can't be made here.

Low-speed Flow Case To verify the present numerical method and compare with the experiment, the calculation at the take-off condition, namely at the low-speed condition (Mach No.=0.2), was carried out. In this case , the grid is the same as the previous case, but the time step is set half in case of divergence.

Figure 5 shows the computational result. In this figure, the total thrust coefficient is compared with the experimental data. The result carried out by the present method shows the reasonable agreement with the experiment.

Concluding Remarks

In this paper, an efficient solution procedure to analyze three-dimensional viscous and unsteady flows around CRP are described and its preliminary computational results are shown. Results show the present code gives plausible unsteady solutions with clear shock system. Note that the solutions are efficiently obtained within the reasonable computational time. However, in order to verify the accuracy and reliability of the code, parameter study by varying time step, grid size and flow conditions are required, further detailed comparison with experiment should be systematically performed. These points are concerned with future work.

Acknowledgements

The authors would like to acknowledge Mr. Y. Mizobuchi of University of Tokyo for his support throughout this work.

References

1. Smith, L.H., Jr., "Unducted Fan Aerodynamic Design," ASME Paper 87-GT-233, 1987.
2. Hughes, C.E. and Gazzaniga, J.A., "Summary of Low-Speed Wind Tunnel Results of Several High-Speed Counterrotation Propeller Configurations," AIAA Paper 88-3149, 1988.
3. Podboy, G.G. and Krupar, M.J., "Laser Velocimeter Measurements of the Flowfield Generated by an Advanced Counterrotating Propeller," AIAA Paper 89-0434, 1989.
4. Data in house, unpublished paper.
5. Celestina, M.L., Mulac, R.A. and Adamczyk, J.J., ""A Numerical Simulation of the Inviscid Flow Through a Counterrotating Propeller," ASME Paper 86-GT-138, 1986.
6. Adamczyk, J.J., "Model Equation for Simulating Flows in Multistage Turbomachinery," ASME Paper 85-GT-226, 1985.
7. Whitfield, D.L., Swaford, J.M., Janus, J.M. and Mulac, R.A., "Three-Dimensional Unsteady Euler Solutions for Propfans and Counter-Rotating Propfans in Transonic FLow," AIAA Paper 87-1197, 1987.
8. Janus, J.M. and Whitfield, D.L., "Counterrotating Prop-Fan Simulations which Feature a Relative-Motion Multiblock Grid Decomposition Enabling Arbitrary Time Steps," AIAA Paper 90-0687, 1990.
9. Nicoud, D., Brochet, J. and Goutines, M., "A Methodology Proposal to Design and Analyse Counterrotating High Speed Propellers," ASME Paper 89-GT-38, 1989.
10. Nallasamy, M. and Groeneweg, J.F., "Unsteady Euler Analysis of the Flow Field of a Propfan at an Angle of Attack," AIAA Paper 90-0339, 1990.
11. Miller, C.J. and Podboy, G.G., "Euler Analysis Comparison with LDV Data for an Advanced Counter-Rotation Propfan at Cruise," AIAA Paper 90-0438, 1990.
12. Srivastava, R. and Sankar, L.N., "An Efficient Hybrid Scheme for the Analysis of Counter Rotating Propellers," AIAA Paper 91-0703, 1991.
13. Rai, M.M., "Unsteady Three-Dimensional Navier-Stokes Simulations of Turbine Rotor-Stator Interaction," AIAA Paper 87-2058, 1987.
14. Matsuo, Y., Arakawa, C., Saito, S. and Kobayashi, H., "Navier-Stokes Simulations Around a Propfan Using Higher-Order Upwind Schemes," AIAA Paper 89-2699, 1989.
15. Baldwin, B.S. and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, 1978.
16. Coakley, T.J.,"Implicit Upwind Methods for the Compressible Navier-Stokes Equations," AIAA Paper 83-1958, 1983.
17. Chakravarthy, S.R. and Osher, S.,"A New Class of High Accuracy TVD Schemes for Hyperbolic Conservation Laws," AIAA Paper 85-0363, 1985.

Table 1. Parameters of the computations.

	CASE I	CASE II
Free stream Mach number	0.75	0.20
Advance ratio	3.2	1.0
Fore rotor blade angle	58°	36°
Aft rotor blade angle	56°	32°
Reynolds number	1×10^6	5×10^5

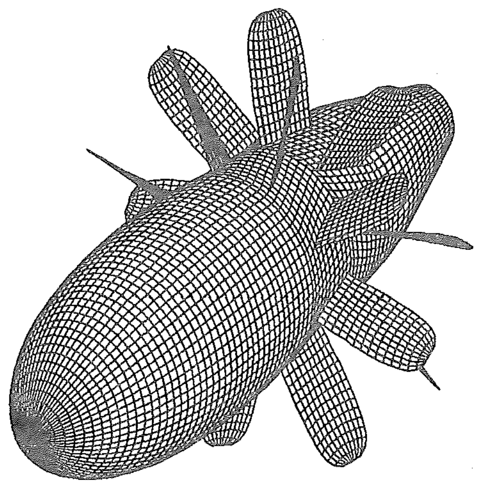
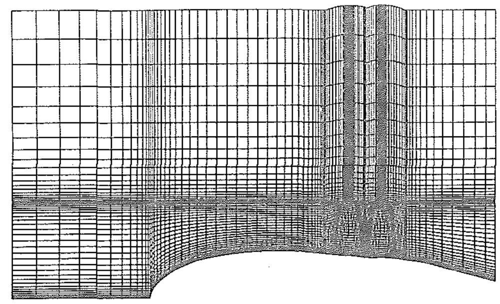
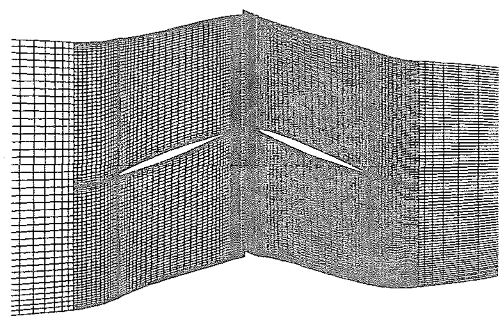


Fig. 1. Schematic of a counterrotating propeller.



(a) Side view



(b) Section view

Fig. 2. Computational grid.

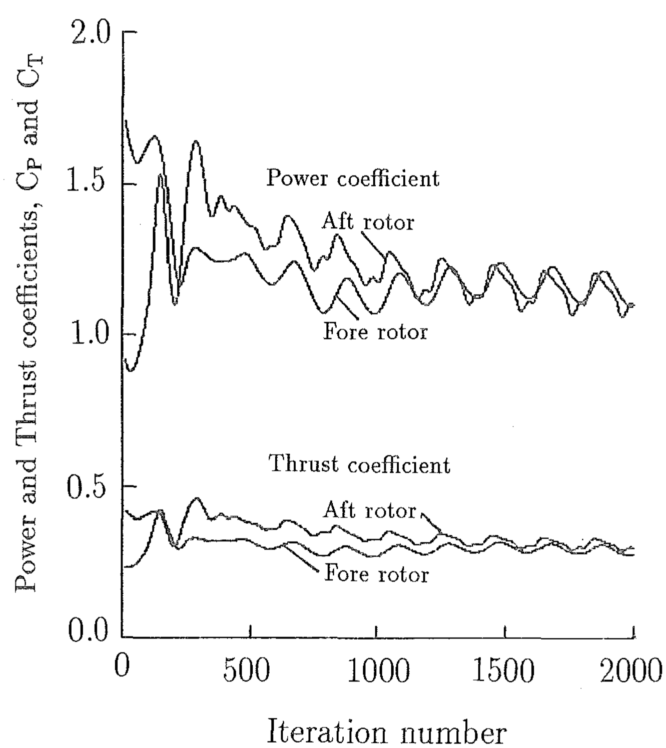


Fig. 3. Time histories of aerodynamic loadings of rotors in Case I.

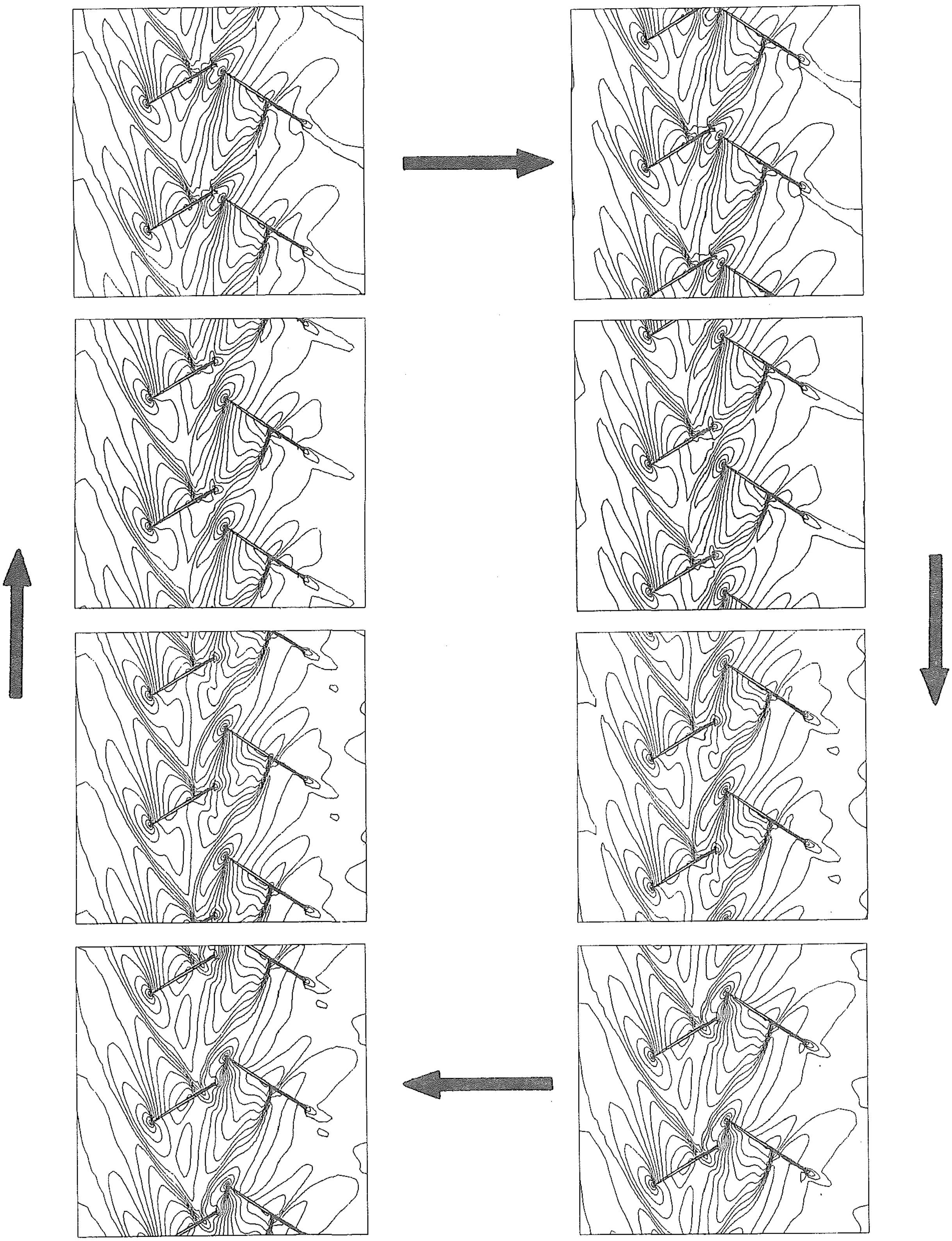


Fig. 4. Time histories of pressure contours on a midspan section in Case I.

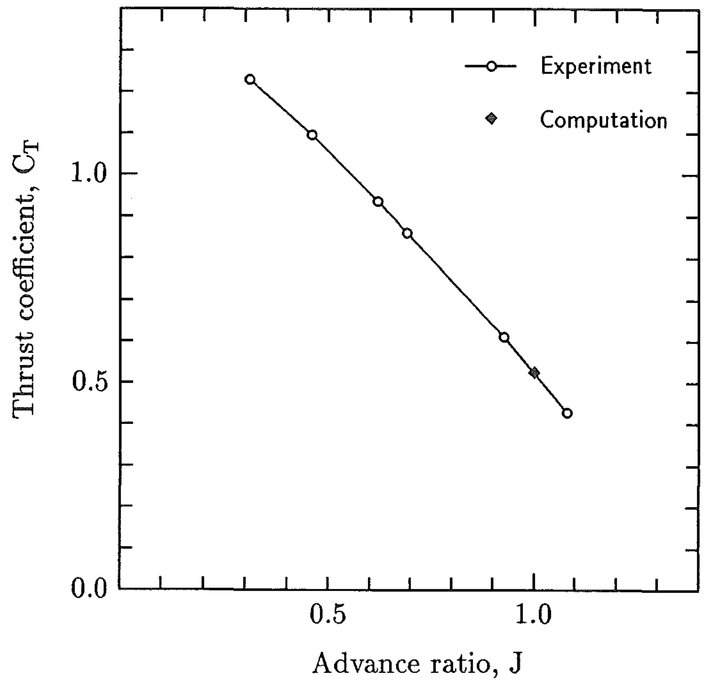


Fig. 5. Comparison of thrust coefficient with experiment in Case II.