Unsteady Flow Analysis of Fan Rotor with Inlet Distortion

K.Hirai and H.Kodama (Ishikawajima-Harima Heavy Industries)

O.Nozaki, K.Kikuchi, T.Nishizawa, A. Tamura and Y. Matsuo (National Aerospace Laboratory)

Abstract

A three-dimensional time-accurate Reynolds-averaged Navier-Stokes code has been extended for use in a parallel supercomputer called Numerical Wind Tunnel(NWT) developed by the Japanese National Aerospace Laboratory. The power of parallel supercomputer enables us to perform a more accurate computation of the unsteady flowfield within a multipassage configuration. The numerical simulations was aimed at investigating the unsteady flow behavior for a transonic fan rotor blade with inlet total pressure distortion, which is inclined to appear in a SST propulsion system.

Introduction

In an intake of the SST propulsion system, shock and boundary layer interaction effects generate total pressure distortions that could cause an unacceptable loss of performance of the front fan of the propulsion engine. It is necessary to understand the flow phenomena occuring in the fan rotors. A computational approach has a great advantage in terms of time and cost. However, such flow is unsteady phenomena extending the whole circumferential fan rotor passages, so the simulation based on the three-dimensional unsteady Reynolds-averaged Navier-Stokes equations is well beyond the capabilities of a single supercomputer.

The Japanese Aerospace Laboratory has developed a parallel supercomputer called Numerical Wind Tunnel (NWT). The NWT is composed of 162 processing elements, a control processor, and linking networks. Each processing element itself is a supercomputer with a peak performance of 1.7 GFLOPS and main memory of 256 MB. By parallel processing a 275 GFLOPS peak speed with total main memory of 41 GB can be realized. The power of parallel supercomputer enables us to perform a more accurate computation of the unsteady flowfield within a multipassage configuration.

The purpose of the present investigation is to advance our understanding of the flow phenomena inside fan rotors subject to a circumferencial total pressure distortions by using parallel supercomputer system.

Numerical Procedure

Computational scheme

The governing equations are the three-dimensional unsteady Reynolds-averaged Navier-Stokes equations. The turbulent viscosity is determined by the two-layer Baldwin-Lomax (1978) algebraic turbulence model.

An implicit finite difference scheme which is capable of using large CFL numbers is used. The convection terms are discretized using the TVD scheme developed by Chakravarthy and Osher (1985) and central differencing is used for the diffusion terms. Further details on the implicit scheme can be found in Matsuo (1991). For the implicit time integration approach, a Newton sub-iteration is performed at each time step to increase stability and reduce linearization errors. For all cases investigated in this study, four Newton sub-iteration were performed at each time step.

A multipassage computational domain is divided into sub-domains, in which one blade passage is allocated to one sub-domain. The algorithm in the present code is parallelized so that each sub-domain is executed on a different processing element of NWT and exchanges boundary information with neighboring sub-domains.

Boundary condition

At the inlet boundary, the distribution of total temperature, total pressure, flow angles are fixed. Static pressure is fixed at the exit boundary. Non-slip and adiabatic wall boundary conditions are applied to blade surface and hub/casing surfaces.

Results and Discussion

The design details and experimental results of a single stage fan are reported by Monsarrat,N et al (1969) and Sulam,D.H. et al (1970) respectively. The fan stage with the rotor of 1600ft/sec tip speed was tested with circumferentially distorted inlet flow. The inlet distortion patterns were generated by screens of varied porosity. A circumferential distortion parameter of 0.2 covering a 90-degree arc was achieved at the rotor inlet with the throttle wide open at design speed using a 120-degree full span screen.

The fan consists of 30 rotor airfoils and 44 stator airfoils. For the numerical simulations of unsteady interaction with inlet distortion, only rotor airfoils were treated. Hence a multipassage computation including 30 rotor blade passages was conducted (see Figure 1) imposing circumferentially non-uniform total pressure at the inlet boundary.

The grid system for the fan rotors is shown in Figure2. An H-type structured grid is used for each blade passage. The computational domain was extended in both upstream and downstream directions until the circumferential static pressure perturbations generated by the rotor sufficiently attenuated. Each grid for one passage consists of 119 nodes in the axial direction, 51 nodes in the circumferential direction, 61 nodes in the spanwise direction. Five nodes in the spanwise direction are used to describe the tip clearance region. The total number of grid nodes for 30 rotor blade passages reached 11.1 million points. The time steps of 9000 per one rotation were needed to proceed calculation successfully.

In Figure 3, total temperature contours in the absolute frame of reference at the downstream of the rotor are compared between the measurement and the calculation at 100% rotor design speed. The measured contour was obtained from the traverse data for 9 radial and 12 equally spaced circumferential locations, while the predicted contour was constructed using calculated temperatures for 61 radial and 280 circumferential locations. The overall features of the non-uniform temperature rise due to inlet distortion is successfully predicted by the calculation.

Figures 4(a), 4(b) show the circumferential total temperature distributions at 50 percent and 85 percent span from the hub respectively. The measured total temperatures were obtained at just downstream of the stator, and the calculated total temperatures at the location of stator leading edge are compared in the figures. Good agreement exists between the calculated results and the experimental data except around the circumferentail position of 300 degrees at 50 percent span, where the calculated variations are greater than those observed experimentally.

In Figure 5, instantaneous relative Mach number contours across the distorted inlet flow region are shown, and Figure 6 shows the static pressures on the blade surfaces (the passages are numbered in Figure 5). When the fan rotor enters the low total pressure region (No.15 in Figure 5), a passage shock near the leading edge of the fan rotor begins to move upstream. Mach number upstream of the shock is increased during the fan rotor passes through the distorted flow. When the fan rotor is in the middle of the distorted region (No.21 in Figure 5), flow separation begins to occur due to strong interaction between the shock and the blade boundary layer. Then the separated region extends to the downstream of the shock and becomes large. It may cause the high total pressure loss (No.27 in Figure 5). After the fan rotor goes back to the uniform inlet flow region, the shock begins to return to the original strength, and get to the steady condition after traveling almost the same circumferential length as the distorted region.

Conclusions

A three-dimensional Navier-Stokes code based on an implicit algorithm using Newton sub-iteration has been successfully extended for use in a parallel supercomputer.

The simulation of the flow fields in fan rotors with inlet total pressure distortion explained the mechanism of pressure loss production associated with the passage shock motion.

References

Baldwin, B.S., and Lomax, H., 1978, "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows", AIAA-78-257, 1978

Chakravathy, S.R. and Osher, S., "A New Class of High Accuracy TVD Schemes for Hyperbolic Conservation Laws," AIAA 85-0363, 1985

Matsuo, Y., "Computations of Three-Dimensional Viscous Flows in Turbomachinery Cascades", AIAA-91-2237,1991

Monsarrat, N., Keenan, M.J., and Tramm, J.A. "Design Report, Single-Stage Evaluation of Highly-Loaded, High-Mach-Number Compressor Stages", NASA CR-77562, PWA-3546, 1969.

Sulam, D.H., Keenan, M.J., Flynn, J.T. "Single-Stage Evaluation of Highly-Loaded, High-Mach-Number Compressor Stages II, Data and Performance Multiple-Circular-Arc Rotor", NASA CR-72694, PWA-3772, 1970.

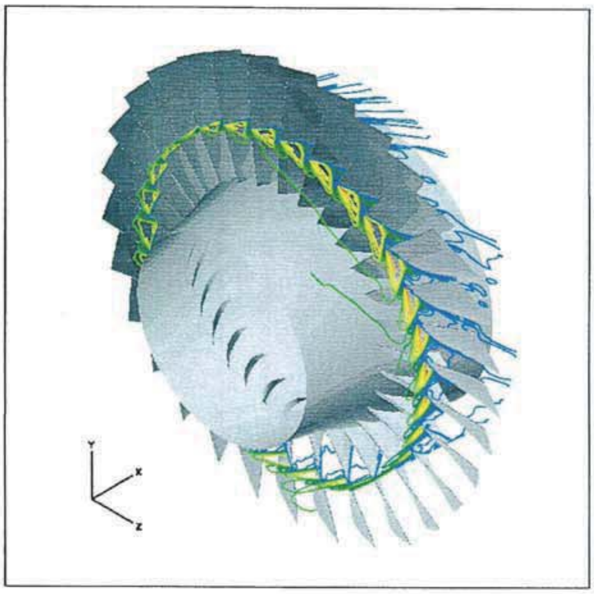


Fig.1 Configuration

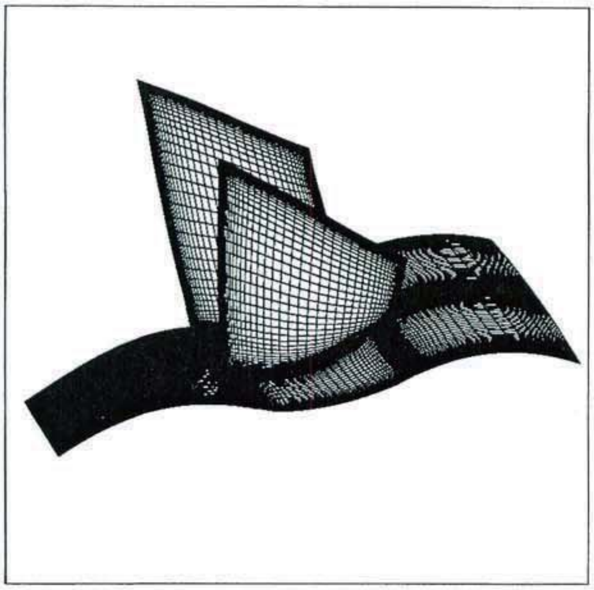
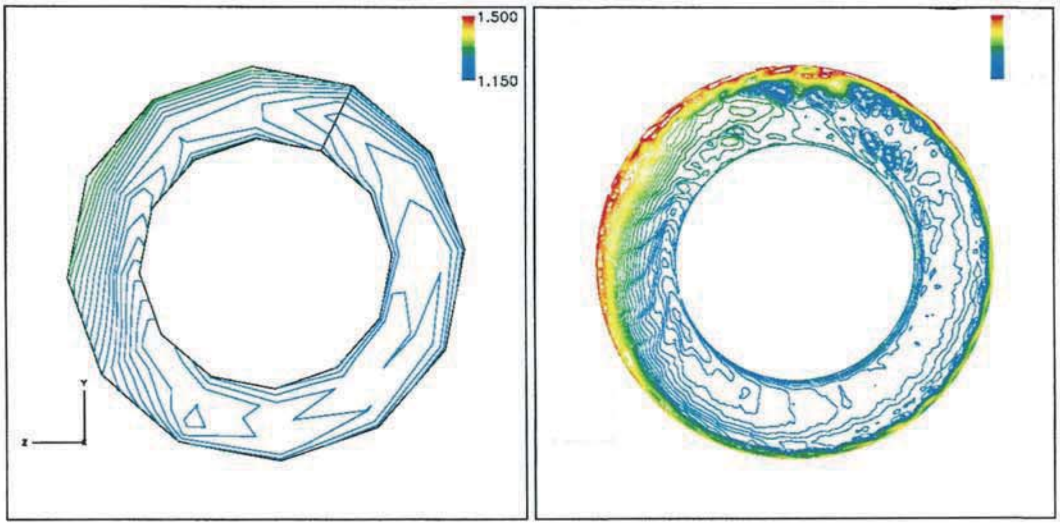
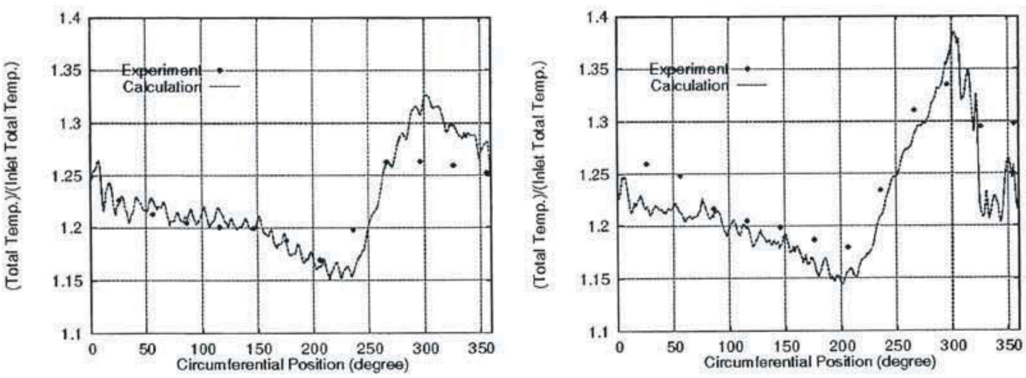


Fig.2 Grid System



(a) Experiment (b) Calculation

Fig.3 Absolute total temperature contours at the downstream of fan rotors



(a) 50 percent span (b) 80 percent span

Fig.4 Circumferential absolute total temperature distribution

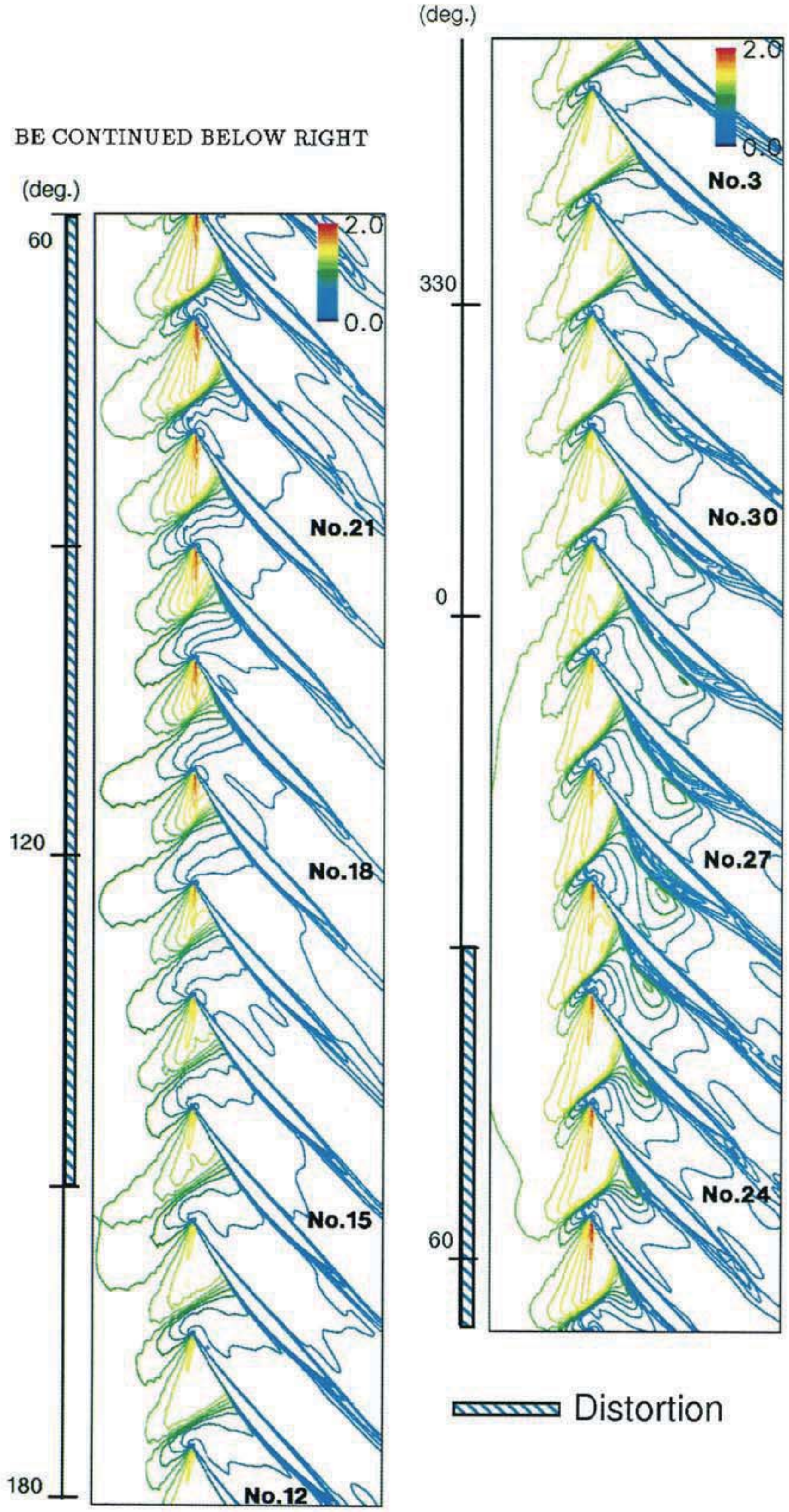


Fig.5 Relative Mach Number Contours

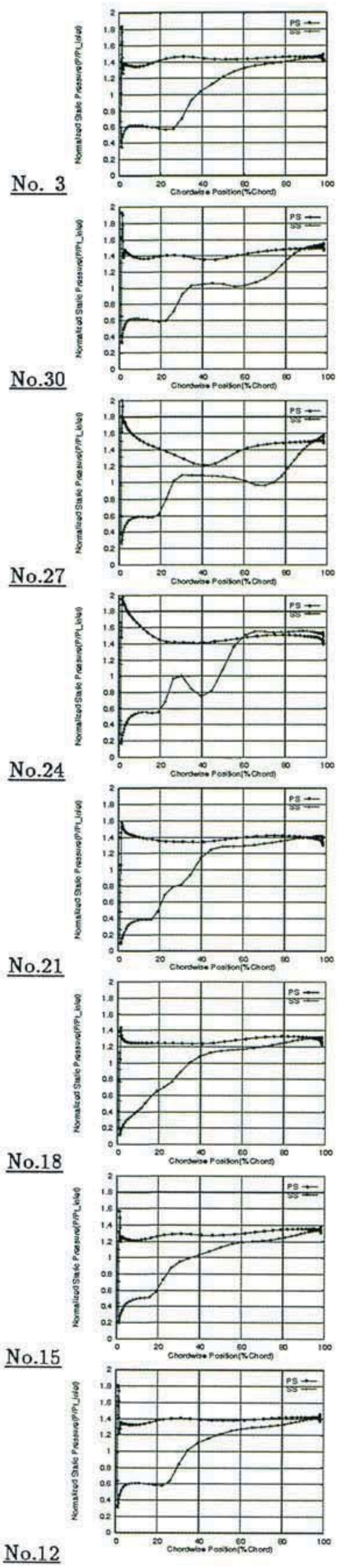


Fig.6 Static Pressure on Blade Surface