# CFD MODELING EFFECTS ON UNSTEADY MULTISTAGE SIMULATION FOR A TRANSONIC AXIAL COMPRESSOR

Mai Yamagami, Hidekazu Kodama, Dai Kato, Naoki Tsuchiya **IHI** Corporation

Yasuo Horiguchi Advanced Science & Intelligence Research Institute Corp. Tokyo, Japan

Junichi Kazawa Japan Aerospace Exploration Agency Tokyo, Japan

## ABSTRACT

Unsteady three-dimensional multistage calculations are performed for a highly loaded, high-speed axial compressor to investigate the impact of real geometry modeling and different numerical approaches on the accuracy of the performance prediction. First, two features of the real geometries are separately compared with the calculation which consists of a pure flow path model except that rotor tip clearances are considered. One treats leakage generated by part gaps between variable stator vanes and the annulus lines. Another incorporates seal cavities to model leakage underneath the shrouded stators. Then, the influence of different numerical approach with different turbulence models is also investigated. Discussion on the impact of the CFD modeling on the performance prediction focuses on the prediction accuracies of stage operating points and spanwise mixing. It is suggested that a realistic simulation of turbulent-type flow unsteadiness in a multistage machine is important for an accurate prediction of spanwise mixing phenomena.

## NOMENCLATURE

- ADP: Aerodynamic Design Point HPC: High Pressure Compressor IGV: Inlet Guide Vane L/E: Leading Edge OGV: Outlet Guide Vane VSV: Variable Stator Vane T/E: Trailing Edge H: Blade Height P: Pressure R: Rotor S: Stator T: Temperature
- V: Velocity

V<sub>r</sub>: Radial Velocity e: Seal-teeth Clearances Height *m* : mass flow rate Dt: time span for one cycle t: time

Subscripts: r: Ratio t: Total (stagnation) conditions x: Axial direction ave: time averaged mid: at mid span of circumferentially mid pitch

## INTRODUCTION

The development of an advanced multistage axial compressor is very challenging, because the compressor design requirements are mutually conflicting, and the important physical phenomena occurring in a multistage axial compressor system are highly complex and nonlinearly interrelated. Therefore an advanced design tends to cause aerodynamic problems, and an additional program of the redesign and its verification test cycle will be required. This iterative cycle significantly impacts on the development time and cost. One vital key to reducing the time and cost of a development program is to identify and correct problems as early as possible in the program. Having better simulation that is capable of addressing the aerodynamic issues associated with the advanced design is essential to reduce the time and cost of developing an advanced multistage compressor.

The most important capability of simulation of multistage compressor is a prediction of stage matching. The term stage matching refers to the matching of the inlet flow requirements of a particular stage to the outlet flow of the stage upstream. In a high-speed multistage axial compressor, even a small error tends to force a stage to operate at the matching point far from its design point, because the operating range of mass flow from choke to stall is generally small at high rotating speeds. The mismatch extends to all stages, and this sometimes leads to high performance loss. Correcting a mismatch is a hard and time consuming task in the compressor development. It is demanded to identify aerodynamic design problems that would result in stage mismatching by using multistage simulations in the design phase of the development program.

Another important feature of a multistage compressor, for which an accurate prediction is required in the simulation, is spanwise mixing phenomena. This feature is significant in the rear stages where the spanwise distribution of total pressure and total temperature are flattened. Therefore incorrect prediction of spanwise mixing would calculate the incorrect spanwise distributions of inlet Mach number and inlet flow angle to the blades and the vanes, and these would result in incorrect predictions of aerodynamic performance. The mechanism of spanwise mixing is explained by two main reasons. One is the radial convective component of secondary flows suggested by Adkins and Smith [1], and another is the mixing due to turbulent type diffusion suggested by Gallimore and Cumpsty [2, 3]. The latter mechanism is purely an unsteady phenomenon, so only unsteady stage simulation is able to directly predict the turbulent type diffusion.

So far many numerical approaches have been proposed to perform the multistage simulation efficiently. The first approach [4] is based on a simple formulation of the Reynolds Averaged Navier Stokes (RANS) equations and employs 'Mixing-Planes' between stationary and rotating blade-rows. This approach is efficient and robust, but neglects unsteady flow physics. Therefore the mixing-plane approach is unable to predict the spanwise mixing phenomena. Adamczyk [5] proposed the approach based on the average-passage equation system which includes the stress and flux terms to account for the unsteady deterministic flow field. In this approach, a modeling is necessary to provide closure of correlation terms which appear in the averaging process of the governing equations. He and Ning [6] proposed an improved modeling to perform calculation efficiently and accurately. However, a lot of extensive validations are required to establish a reliable modeling in the use of average passage approach.

Recently, Yamagami, et al [7] performed an unsteady threedimensional stage calculation for a high-speed, six-stage advanced compressor to investigate the capability of predicting unsteady phenomena occurred in a multistage environment. The CFD code used in the calculation solves the RANS equations with a one-equation Spalart-Allmaras turbulence model. As for a single blade row, it has been verified by comparing with experimental data that this code accurately predicts the steady performance. However, in the comparison with the test results of the multistage compressor, they found that the unsteady RANS calculation had issues of accuracies in predicting stage operating points at the design operating condition as well as spanwise mixing phenomena. In spite of a more realistic simulation incorporating unsteadiness, the calculation resulted in only a small improvement compared with the results of the steady stage calculation using mixing plane model. A correct prediction of the stage matching and the spanwise mixing phenomena is essential for a simulation of multistage axial compressors. So it is demanded to identify root causes of the discrepancy between the CFD results and the test results, and solve the problems to improve the prediction.

One of the possible causes of the discrepancy is that the calculation used a pure flow path and neglected real geometry features, except that rotor tip clearances are considered. Some collateral flows like seal leakage flow, sink and source flows from cavity, leakage flow of variable stator clearance could change flow blockage and enhance spanwise mixing. The others might be related to matters of computational technique for an unsteady multistage analysis.

The present paper first investigates the impact of real geometry features on the accuracy of stage matching and spanwise mixing predictions. For the stage matching investigation, stage operating points at the design condition are compared to the rig test data. Two higher fidelity models are separately treated to clarify each impact of the modeling on the predictions. One is leakage generated by part gaps between the variable stator vanes and the annulus lines. Another is leakage underneath a shrouded stator. Then the different numerical code with different turbulence model is used to investigate the impact of a different numerical approach on the predictions.

## COMPRESSOR VALIDATION TEST CASE

The test case used in this study is a six-stage, highly loaded advanced axial compressor. This transonic compressor with pressure ratio over 12 is designed by IHI in Japanese "ECO engine project" [8, 9]. The compression system consists of an inlet duct, a front frame, an inlet guide vane (IGV) and six stages of compressor blade rows. For the sixth stage, a new concept, so called Diffuser Passage [10] design is introduced.

For the axial compressor, detailed flow measurements were made in the rig test. Mass flow rate was measured by venturi nozzles located upstream of the compressor. Overall compressor performance was measured by eight inlet pressure/temperature combo rakes which are located circumferentially mid pitch between each of the eight struts of the front frame and six exit rakes located at compressor exit. Total pressure and temperature were also measured at the inlet of Stators 1 through 5. Only total pressure was measured at Stator 6 inlet, due to spatial restrictions. For each of the pressure and temperature measurements at stator inlet, there were sets of four to five sensors (depending on the height of the vanes) mounted on the leading edge of the stator vanes, roughly equally spaced in spanwise direction between 10% span and 90% span from hub. The radial four to five sensors were separately mounted on two stator vanes to avoid interference between the sensors, and circumferential arrangement of the instrumented stator vanes was carefully chosen so that circumferential non-uniformity of the performance would be minimized. Wall static pressure measurements and tip clearance measurements were also performed for each stage. Mass flow rate measurement was accurate to within  $\pm 0.6\%$ . Accuracy of pressure measurements was  $\pm 0.1\%$  of each full scale, and accuracy of temperature measurement was  $\pm 0.4$ degC for data below 200degC and  $\pm 0.25\%$  for data above 200degC.

IGV, Stator 1, Stator 2 and Stator 3 were variable geometry vanes. At the design point in rig test, an inter-stage bleed of 3.5% of compressor inlet flow were extracted at Stator 4 exit.

## NUMERICAL PROCEDURE

#### CFD code

CFD code used in this study is UPACS [11, 12] which was developed by Japan Aerospace Exploration Agency. The code is an unsteady 3D flow solver for the Reynolds-Averaged Navier-Stokes equations based on a finite volume method using multi-block structured grids. In this study, the convection fluxes are discretized by Roe's flux difference splitting with 3<sup>rd</sup>-order MUSCL, and the viscous fluxes are discretized by 2<sup>nd</sup>-order central difference. Here the minmod limiter is applied to the MUSCL approach. The Spalart-Allmaras one-equation turbulence model [13] is selected, since the various validations have been conducted using experimental data. Time-integration is evaluated by 2<sup>nd</sup>-order Euler implicit method with Newton sub-iterations. The code is parallelized with MPI and its good parallel efficiency is shown in Takagi et al [12].

The rotor-stator interface between the rotating domain including rotor blades and the stationary domain including stator vanes are treated as a discontinuously sliding grid-block boundary. The flux on a grid surface across the sliding boundary is precisely calculated in a fully-conservative manner at each time step. The numerical flux on the boundary is evaluated by the same scheme as that of the inner region.

Numerical iteration is repeated until the residuals of conservative variables decrease below an acceptable value and the principal parameters such as mass flow rate, total pressure and total temperature at inflow and outflow boundaries converge in an acceptable range. After the convergence criteria are satisfied, the iteration is continued further until the convective variables at the inflow boundary reach to the outflow boundary. Then the calculation starts to store the time step results for a time average post processing.

## **Modeling Configurations**

Figure 1 shows a computational domain of the rig test model compressor, whose main flow path is the same as that of the baseline calculation previously performed by Yamagami, et al [7]. It is composed of Strut, IGV and following 6 stages of rotors and stators. An unsteady multistage calculation of a full annulus model would be extremely time consuming with restricted computer resources. To overcome this difficulty, computational domain is set to 1/10 sector of the whole annulus as shown in Figure 1. When it is done, rotor and stator airfoil geometries are scaled and the counts are changed to keep the same solidity as original one at each radial section. The chord and thickness of an airfoil are altered, but hub and casing radii are preserved. Although the chord is changed up to 6% for a couple of blade rows, the change in the chord is an average of 3%. In the previous paper [7], by using the steady stage calculations with mixing plane model, it had been confirmed that the influence of the chord change up to 6% on the performance prediction is negligible.



Figure 1 Computational domain of multistage compressor

Table 1 Matrix of CFD modeling

	Real Geometiry Modeling			
	Rotor Tip clearance	VSV clearance	Seal Cavity	CFD code
CaseA	0	-	-	UPACS with SA
CaseB	0	0	-	UPACS with SA
CaseC	0	-	0	UPACS with SA
CaseD	0	-	-	UCAS with BL

CFD modeling of the calculations performed in this paper are summarized in Table 1. Each of the three new calculation results (caseB, caseC, caseD) are compared with the baseline calculation results (caseA) so that each influence of the CFD modeling change is clarified separately.

In the actual compressor, there are some geometry features that generate interacting secondary flows with the main flow, like a bleed port, seal cavities, rotor tip clearances and gap clearances in variable stator vanes (VSV). Only rotor tip clearances were included in caseA. The effects of a bleed on the overall performance and the stage matching was investigated in the rig tests and found to be relatively small by the comparison between the test with a design bleed flow and the test with no bleed flow. Therefore in this study, the impact of the bleed port is not investigated.



Figure 2 VSV clearance model



Figure 3 Cross sectional view of model Compressor



Figure 4 Shrouded stator seal cavity model

 Table 2 Comparison of simulated seal leakage

 flow rates and estimated flow rates in the rig

	Cavity Leac		
	CFD	Analytical	e/H(%)
	CI D	model	
stator1	0.14	0.14	0.39
stator2	0.30	0.30	0.59
stator3	0.47	0.42	0.83
stator4	0.48	0.53	1.00
stator5	0.46	0.62	1.18

The current model compressor uses a variable stator system on the front 4 stators (from IGV to Stator 3). The part gap clearances between the VSV and the annulus lines are modeled for these stators as shown in Figure 2. The gap size for each VSV was measured when the model compressor was assembled. These measured gap sizes are used in the computational model of caseB.

In the actual rig, there are seal cavities (Figure 3) which provide clearance between the rotating rotor wheel and the stationary inner-band that stabilize the stator vanes. In caseC, these seal cavities are modeled as shown in Figure 4. For gridding the cavities with multi-block structured meshes, the actual cavity geometries of the rig are altered by a group of boxes in the meridional plane, and rotated around the engine centerline. The modeled cavity volume and the area of the rotating and stationary surfaces are maintained close to the actual figures. To pass seal leakage flow from upstream cavity (stator exit) to downstream cavity (stator inlet), cavity meshes are connected at the bottom of the stator shroud ring. The labyrinth seal-teeth are approximated by a single tooth of zero thickness. The seal tooth tip clearances are adjusted so as to simulate the leakage flow rate estimated by a semi-empirical analytical model of Kotomori and Miyake [14] with the actual clearance of the rig. Table 2 compares the simulated leakage flow rates and the estimated flow rates of the rig. The simulated flow rates are calculated by subtracting the mass flow rate upstream of the slit from the mass flow rate downstream of the slit. They are simulated fairly well. The gap clearances in the CFD turned out to be around 0.15mm compared to 0.2mm of the actual rig. The clearance in the CFD had to be tighter to simulate multiple-teeth Labyrinth seal characteristics with a single tooth model. The model simplification method mentioned above was established by the study using a CFD calculation which models seal cavity and an isolated stator. It was confirmed that there was almost no impact on the seal leakage loss and the interaction loss between cavity leakage flows and the main stream flows due to the simplifications.

### Numerical grid

O-H type structured grid is used for each blade passage. The O-H type grid in which O-type grid is surrounding the blade surface guarantees highly orthogonal grids on the blade surface. Outside of the O-type grid is filled in with an H-type grid. To the contrary, clearance gap between rotor tip and casing is filled in with an H-O type grid, which means that Htype grid located in the center is surrounded by O-type grid. At the rotor-stator boundary, tangential mesh widths on both upstrem and downstream grid blocks are kept as uniform as possible to avoid numerical diffusion across the interface.

Grid dependence study is conducted for each cascade separetely by steady stage calculation. The number of grid points in each direction is increased until there is almost no change in the converged solution. Consequently the total number of grid points becomes about 100 million for caseA, 130 million for caseB and 180 million for caseC. The first grid points off the solid walls are placed in the region of Y+ < 3 except around leading edge. Average spacing of the first grid points from the wall is Y+ = 1.4 over the whole solid walls, which is small enough to resolve the viscous sublayer with the Spalart-Allmaras model[13].

#### **Boundary Conditions**

Inlet boundary conditions are imposed at the upstream of strut where spanwise distributions of total temperature, total pressure and flow angles are specified. The distributions of these flow parameters at the strut inlet are obtained by using a separate calculation for an isolated strut model so that the calculated distributions of total temperature and total pressure at measurement combo rakes in the strut passage are matched to the measured distributions. Exit boundary conditions are imposed at the downstream of OGV (Stator 6) where the static pressure is specified. It is adjusted so that the overall pressure ratio matches to the measured value. Non-slip and adiabatic wall boundary conditions are applied to blade surface and hub/casing walls.

#### **RESULTS AND DISCUSSION**

The unsteady time-accurate simulations were performed with 30,000 time steps per cycle (i.e., one rotor revolution) and three inner sub iterations per time step. The simulations required approximately two cycles to converge. To calculate overall performance, time-averaged results are obtained by sampling data every 50 time steps over 1/10 cycle, or 3,000 time steps, and taking average of the sampled sixty data.

#### Effect of Real Geometry Modeling

Overall Performance. In the rig test, the rotor rotational speed was held constant at design speed, while a discharge valve in front of the exit scroll was throttled to obtain a constant-speed, overall mass flow vs. pressure rise characteristic. Steady performance measurements were made at several throttle settings. In Figure 5, overall performance near design point obtained by CFD of caseA, caseB and caseC are compared with the test data. Design point is also plotted. It should be noted that the test data point with the highest pressure ratio in the figure does not represent near stall point. Tested stall margin is well beyond this last plot. As shown in Figure 5, the computed corrected mass flow rates at IGV inlet of caseB and caseC are essentially unchanged. The predicted mass flow rates are still about 2.5 percent higher than the test data. Meanwhile, although not shown in the figure, the impacts of real geometries on the predicted overall efficiency are -0.8 points for CFD with VSV clearance (caseB), and -1.7points for CFD with seal cavities (caseC).



Igure 5 Mass Averaged overall Performance at Design Speed



Figure 6.1 Percent difference in stage total temperature ratio relative to design at design speed



Figure 6.2 Percent difference in stage total pressure ratio relative to design at design speed



(Stator1+Rotor2) Performance

(Stator2+Rotor3) Performance

Comparision of Stage Operating Points. Figure 6 represents percentage differences from the design intent in stage total temperature ratio (Figure 6.1) and stage total pressure ratio (Figure 6.2). The data obtained by CFD of caseA, caseB and caseC are compared with the test data. In this figure, the stage is defined between upstream stator leading edge to downstream stator leading edge for each rotor. For the CFD with cavity (caseC), the stator leading edge location is defined at downstream of the slit (B) as shown in Figure 4.

As shown in Figure 6.1, the stage-wise distribution of the deviations from the design intent in total temperature ratio predicted by the calculation with VSV clearance (caseB) is almost the same as that of the pure flow path calculation (caseA). This indicates that the leakage flows generated by the VSV clearances have little influence on the stage work. On the other hand, small differences in the total pressure ratio can be seen between caseA and caseB as shown in Figure 6.2. This is attributed to the leakage loss appeared in the calculation with VSV clearance (caseB). The lower pressure ratios of Stage 3 (Stator 2+Rotor 3) and Stage 4 (Stator 3+Rotor 4) in the caseB calculation are mainly due to higher endwall losses of the VSV's. In Stage 2 (Stator 1+Rotor 2) of caseB, where the total pressure ratio is higher than that of caseA, higher endwall losses are seen in Stator 1, but the Rotor 2 loss is reduced near endwalls. This is considered to be due to the fact that the Rotor 2 blade was designed to match to Stator 1 leakage flows, and it may result in the improvement in Stage 2 performance compared with caseA. The leakage losses of the VSV clearances seem to contribute to a growth of blockage. Although not indicated in the figures, in the calculation results of caseB, increase in the axial velocity relative to the caseA results can be seen in the main stream up to Stage 5. The resultant higher Mach number flows are thought to be the reason why the total pressure ratio of Stage 5 (Stator 4+ Rotor 5), where there is no VSV, is lower than that of caseA

In Figure 6.1 and Figure 6.2, the impacts of seal cavity modeling (caseC) are also compared with the baseline calculation (caseA). Although the effects of the modeling can be seen to some degree compared with those of the VSV clearance modeling (caseB), the overall trend of the stage-wise distributions of total temperature ratio and total pressure ratio is similar to that of the caseA results. The influences of leakage flows of the seal cavities on the aerodynamic performance are individually investigated and discussed in a companion paper by Kato, et al [15].

As seen in Figure 6, an improvement in the prediction of stage operating points at the design condition due to the introduction of higher fidelity geometry modeling is relatively small. This is mainly attributed to the Stage 1 matching where the deviations from the measurements are largest among the stages and almost no improvement can be seen due to the model changes. Figure 7.1 compares the calculated Stage 1 pressure rise – mass flow rate characteristics with the measurements. In the Figure, the calculation results of a 1.5 stage (IGV+Rotor 1+Stator 1) model with a pure flow path are also compared. It can be seen that all calculations (caseA, caseB and caseC) show almost the same Stage 1 matching point on the constant speed line predicted by the 1.5 stage calculation. The stage matching occurred in the calculations fixes Stage 1 at the point much lower than the measurement. It can be considered that this is caused by the fact that the CFD calculations predicted a stage characteristic with higher mass flow rate. The same trend is also seen in Stage 2 (Figure 7.2) and Stage 3 (Figure 7.3).

Figure 8 shows comparison of the mass flow rate predicted by the current UPACS code with the measured for in-house transonic fans and compressors. All six fans in the figure have no Inlet Guide Vane (IGV) and the 1.5 stage compressor is



Figure 8 Percent difference in predicted mass flow rate relative to test data



Figure 9 Spanwise distribution of Flow angle at IGV T/E

composed of IGV's, rotor blades and stator vanes, modeling front stage of HPC which is different from current computational target. It can be found that existence of IGV has a great impact on the accuracy of mass flow rate prediction.

In figure 9, the calculated spanwise distributions of IGV exit flow angle for the baseline calculation (caseA), the VSV clearance modeling (caseB) and the seal cavity modeling (caseC) are compared with the mean camber line angle at the IGV exit (metal angle). It can be seen that the effect of real geometry modeling on the IGV exit flow angle is very small and all the calculated exit flow angles almost agree with the metal angle except the endwalls. This indicates that the CFD prediction of higher mass flow rate is not attributed to the calculated IGV exit flow angles, because the IGV exit flow angles would be smaller than the metal angle with a large deviation angle, if the calculated IGV flow angles caused the higher mass flow rate prediction. It can be considered that an underestimation of flow blockage due to IGV might result



Figure 10.1 Comparison of Spanwise Mixing at HPC exit predicted by CFD with and without VSV clearance



Figure 10.2 Comparison of Spanwise Mixing at HPC exit predicted by CFD with and without cavity



Figure 10.3 Estimation of Spanwise Mixing coefficient by axisymmetric Through Flow

in the higher mass flow rate prediction. It can be considered that a highly probable cause remained might be underestimation of flow blockage at Rotor 1 inlet. However, further investigation is necessary to identify root cause of the inaccurate prediction of a mass flow rate.

**Spanwise Mixing.** The previous work performed by Yamagami, et al [7] suggested that the baseline calculation (caseA) underestimated the spanwise mixing phenomena in which the calculated mixing was a quarter level of the mixing in a real multistage compressor. In this study, effects of real geometry modeling on the spanwise mixing are investigated.

In Figure10.1, the predicted spanwise distributions of total temperature at OGV leading edge are compared with those at the location of compressor exit rakes for both the pure flow path calculation (caseA) and the calculation with VSV clearance (caseB). In the figure, the rig test data are also plotted. Here, the total temperatures are normalized by the time and mass averaged total temperature of the unsteady CFD results at OGV leading edge. The spanwise distributions are almost the same between caseA and caseB at both OGV leading edge and the location of compressor exit rakes. The comparison indicates that the leakage flows generated by the VSV clearances have almost no influence on the spanwise mixing.

In Figure10.2, the same comparison as in Figure10.1 is made between the pure flow path calculation (caseA) and the calculation with seal cavity (caseC). From this figure, it is difficult to evaluate the effect of the model change on the spanwise mixing, because the spanwise distribution of the caseC total temperature has already changed at OGV leading edge. Therefore the level of spanwise mixing occurred in the caseC calculations is investigated by conducting the same analysis as in the previous paper [7]. An axisymmetric through flow analysis, which can account for spanwise mixing by using a turbulent type diffusion model [3], is performed to determine the spanwise mixing coefficient that gives the mixing occurred in the caseC calculation from OGV leading edge to the location of compressor exit rakes. It is found that a similar radial profile is obtained when the mixing coefficient is reduced to a quarter of empirically correlated coefficient as shown in Figure 10.3. This level is the same as that of caseA. It suggests that modeling of seal cavity has little effect on the prediction of spanwise mixing.

## Effect of Code Difference

Before we developed the UPACS code, we had another multistage simulation code (UCAS) developed by JAXA. The UCAS code is also an unsteady 3D flow solver for the Reynolds-Averaged Navier-Stokes equations, but it uses a finite difference scheme and H type structured grid. The convection terms are discretized using the 3<sup>rd</sup>-order TVD scheme developed by Chakravarthy and Osher [16] and the viscous fluxes are discretized by 2<sup>nd</sup>-order central difference. Timeintegration is evaluated by 2<sup>nd</sup>-order Euler implicit method with Newton sub-iterations. So the numerical discretization accuracy is the same as that of the UPACS code. A remarkable difference is that the UCAS code uses a Baldwin-Lomax zero equation turbulence model (BL model) [17].

There was one example in which the results of unsteady multistage calculation by this code were compared with the test results of a 7-stage HPC. This comparison was not appropriate for discussion about stage matching prediction, because the test compressor incorporated casing treatments in the front stages, whereas the computation model neglected them. However the comparison brought an important finding that the UCAS code with BL model reasonably captured spanwise mixing phenomena. This fact however motivated us to perform an unsteady multistage calculation by using the UCAS code for the current 6-stage HPC.

In this study, a calculation by the UCAS code with BL model is performed for the pure flow path model (caseD). The grid density is maintained equal with that of CaseA except in the vicinity of solid walls. The first grid points need to be placed slightly further off the solid walls to aquire converged solution without numerical oscillation. The average distance is still kept within Y+<5. Total number of grid points consequently sums up to about 80 million for CaseD.

**Overall Performance.** In Figure 11, overall performance predicted by the UCAS code (caseD) is compared with the prediction by the UPACS code (caseA) and the test data. The corrected mass flow rate at IGV inlet predicted by the UCAS code is 0.75 % lower than that of caseA. This suggests that the calculation caseD predicted a stage 1 characteristic with lower mass flow rate than that of the calculation caseA.





**Comparison of Stage Operating Points.** Figure 12 represents percentage differences from the design intent in stage total temperature ratio (Figure 12.1) and stage total pressure ratio (Figure 12.2). In the figures, two predictions, by



Figure 12.1 Percent difference in stage total temperature ratio relative to design at design speed



Figure 12.2 Percent difference in stage total pressure ratio relative to design at design speed



Igure 13 Spanwise distribution of Flow angle IGV T/E

the UPACS code (caseA) and by the UCAS code (caseD), are compared with the rig test results. It can be seen that Stage 1 matching is improved in the prediction by the UCAS code (caseD). This is because the prediction of lower mass flow rate for Stage 1 resulted in Stage 1 matching point at higher pressure ratio than that of caseA.

In Figure 13, the spanwise distributions of IGV exit flow angle are compared between the calculation caseA and the calculation caseD. In the figure, the mean camber line angle at the IGV exit (metal angle) is also plotted. It can be seen that the impact of the code change on the prediction of IGV exit flow angle is very small. This suggests that the lower mass flow rate prediction for Stage 1 by UCAS code might be attributed to larger flow blockage due to IGV in the UCAS calculation. distribution Figure 14 compares the spanwise of circumferentially averaged axial velocity just behind the IGV between caseA and caseD. In the figure, the solid lines show mass averaged axial velocities and the dotted lines show area averaged axial velocities. All the averaged axial velocities are normalized by the overall mass averaged axial velocity over the plane just behind the IGV for the calculation caseA. Just behind the IGV T/E, mass flux is close to zero due to the wake. So mass averaging approximately represents axial velocity outside the IGV wake flows. On the other hand, area averaging accounts for the low axial velocity in the viscous regions, so the averaged value depends on the effective area of the main stream. Therefore the difference between the mass averaged value and the area averaged value roughly represents the flow blockage at the IGV exit. The comparison indicates that the mass averaged axial velocity, i.e. approximately the main stream axial velocity outside the IGV wakes, of caseD, in which the calculated total mass flow rate is 0.8% lower than that of caseA, is rather a little higher than that of caseA. And moreover, the difference between the mass averaged value and the area averaged value for caseD is about 1 point larger than that for caseA. These suggest that the lower mass flow rate prediction for Stage 1 by UCAS code might be attributed to larger flow blockage due to IGV in the UCAS calculation.



Figure 14 Comparison of Mass Averaged and Area Averaged Axial Velocity at IGV T/E (Normalized by overall mass averaged axial velocity at IGV T/E plane for CaseA)



Figure 15.1 Spanwise distribution of normalized Total Temperature at Stator L/E and HPC exit



Figure 15.2 Spanwise distribution of normalized Total Pressure at Stator L/E

**Inter-stage Performance.** Figure 15.1 and Figure 15.2 show spanwise distributions of normalized total pressure and total temperature, respectively, at the exit of rotor. In the figures, the calculation results by the UCAS code (caseD) are compared with those by the UPACS code (caseA) and the rig test results. The test data were obtained by instrumentations mounted on the leading edge of downstream stators. The simulation results are also compared at the leading edge of the downstream stators. Both total pressure and total temperature have been normalized by a mass average over the annulus plane at the leading edge of the upstream stators. In the test data analysis, data match through flow calculations were carried out to obtain the mass averaged total pressure and total temperature.

As seen in Figure 15.1, the use of the UCAS code significantly improves a prediction of spanwise distribution of total temperature, especially in the mid and the rear stages. This suggests that the difference between the UCAS code and the UPACS code has a great impact on the prediction of spanwise total temperature, especially in the mid and the rear stages.

**Spanwise Mixing Phenomena.** In Figure 16.1, the predicted spanwise distributions of total temperature at OGV leading edge are compared with those at the location of compressor exit rakes for both the prediction by the UPACS code (caseA) and the prediction by the UCAS code (caseD). It can be seen that the spanwise mixing predicted in caseD has been already accelerated at OGV leading edge compared with that in caseA.

In order to investigate the level of spanwise mixing occurred from the OGV leading edge to the location of compressor exit rakes in the prediction by the UCAS code (caseD), the spanwise mixing coefficient is determined by using the axisymmetric through flow analysis. It is found that the spanwise mixing coefficient is equivalent to a half of empirically correlated coefficient as shown in Figure 16.2. It is two times larger than that of the prediction by the UPACS code (caseA). This indicates that, in the calculation caseD, the level of turbulent type diffusion is larger than that in the calculation caseA.

To illustrate the level of unsteadiness, Figure 17 compares the time fluctuation of a radial component of the velocity between the prediction by the UPACS code (caseA) and the prediction by the UCAS code (caseD). The time fluctuations are compared at 3 locations on an axial plane of 10% chord downstream of the OGV, at mid span of the circumferentially mid pitch location (Figure 17.1), at 10% span from the tip of the circumferentially mid pitch location (Figure 17.2) and at mid span of the circumferentially OGV trailing edge location (Figure 17.3). Here the fluctuation velocities from the time average value are normalized by the time averaged axial velocity at the mid pitch and the mid span location. The level of the velocity fluctuation in the prediction by the UCAS code (caseD) is much larger than that in the prediction by the UPACS code (caseA) in the mid span region.



Figure 16.1 Comparison of Spanwise Total Temperature distribution at OGV L/E and at HPC exit



Figure 16.2 Estimation of Spanwise Mixing coefficient by axisymmetric Through Flow analysis



Figure 17.1 at mid span, circumferentially mid pitch location



circumferentially mid pitch location



Figure 17.3 at mid span, circumferentially OGV T/E location

Figure 17 Comparison of Time Fluctuation of a radial component of velocity between two codes on axial plane of 10% chord downstream of the OGV

The difference in the calculated unsteady behavior between the two predictions can be already seen in the front stage. Figure 18 compares the instantaneous entropy contours of the front stage at 50% span. The results of the calculation by the UPACS code (caseA) and the calculation by the UCAS code (caseD) are shown in Figure 18.1 and Figure 18.2, respectively. A remarkable difference between the two calculations can be seen in the wake behavior. The wakes predicted by the UCAS code are more vortical and unsteady than those in the calculation by the UPACS code.



Figure 18.1 Instantaneous Entropy contour in Full stage CFD with UPACS (caseA) at Mid-span : Strut to Stator 1



Figure 18.2 Instantaneous Entropy contour in Full stage CFD with UCAS (caseD) at Mid-span : Strut to Stator 1

One of the possible causes that make a difference in the unsteady features between the two predictions might be the level of turbulent eddy viscosity evaluated in the predictions. In Figure 19, contour plots of the calculated turbulent eddy viscosity by the UPACS code with SA model (Figure 19.1) are compared with those by the UCAS code with BL model (Figure 19.2) on an axial plane just behind Rotor 1. The SA model is able to take into account the transport and diffusion of turbulence, but the BL model can not simulate the history effects. It is considered that this might result in a higher level of turbulent eddy viscosity would suppress unsteady behavior of viscous flows. Further investigation on a turbulence model is necessary.



Figure19.1 Normalized Turbulent Eddy viscosity contour on an axial plane just behind Rotor 1 in caseA calculation



Figure19.2 Normalized Turbulent Eddy viscosity contour on an axial plane just behind Rotor 1 in caseD calculation

## CONCLUSIONS

Unsteady multistage calculations are performed to investigate the impact of real geometry modeling and different numerical approach on the accuracy of stage matching and spanwise mixing predictions.

1. Real geometry modeling has little effect on the predictions of stage matching at the design condition, and spanwise mixing in the present study. However, the UPACS code with Spalart-Allmaras turbulence model used in the study tends to suppress unsteady behavior of the viscous flows. So it is necessary to investigate the effects of the real geometry modeling by different numerical approach.

2. The major cause of the discrepancy in the prediction of overall stage matching is stage 1 matching, where a higher mass flow rate is calculated. Underestimation of flow blockage at Rotor 1 inlet might be highly probable cause of the discrepancy. However, further investigation is necessary to identify root cause of the inaccurate prediction of a mass flow rate.

3. It is found that a numerical approach in which turbulence eddy viscosity vanishes away from the solid walls adequately predicts the turbulent type diffusion in a multistage compressor. A realistic simulation of flow unsteadiness in a multistage machine is important for an accurate prediction of spanwise mixing phenomena. A future prediction method for the multistage analysis would be a LES type approach where larger turbulent structures are resolved by the grid instead of a turbulence model.

#### ACKNOWLEDGEMENT

This study is conducted under the contract with New Energy and Industrial Technology Development Organization (NEDO) as a part of "aircraft and space industry innovation program" and "energy innovation program" of Ministry of Economy, Trade and Industry (METI).

#### REFERENCES

[1] Adkins, G.G., Smith, L.H., "Spanwise Mixing in Axial-Flow Turbomachines", Journal of Engineering for Power, Vol. 104, pp. 97-110, 1982

[2] Gallimore, S.J., Cumpsty, N.A., "Spanwise Mixing in Multistage Axial Flow Compressors:Part1-Experimental Investigation", ASME Journal of Turbomachinery,Vol.108, pp.2-9

[3] Gallimore, S.J., "Spanwise Mixing in Multistage Axial Flow Compressors: Part2-Throughflow Calculations Including Mixing", ASME Journal of Turbomachinery, Vol.108, pp.10-16

[4] Denton, J.D.,"The Calculation of Three Dimensional Viscous Flow Through Multistage Turbomachines", ASME 90-GT-19,1990

[5] Adamczyk, J.J., "Model Equation for Simulating Flows in Multistage Turbomachinery", ASME paper 85-GT-226, 1985.

[6] He, L., Ning, W., "Efficient Approach for Analysis of Unsteady Viscous Flows in Turbomachines", AIAA Journal, Vol.36, No.11.

[7] Yamagami, M., Kodama, H., Kato, D., Tsuchiya, N., Horiguchi, Y., and Kazawa, J., "Unsteady Flow Effects in a High-Speed Multistage Axial Compressor," ASME GT2009-59583, 2009.

[8] Funatogawa, O., "Research and Technology Development in Japanese Environmentally Compatible Engine for Small Aircraft Project", ISABE-2005-1010, 2005.

[9] Kato, D., et al, "Development of Simple and Highperformance Technology for Compressors," IHI Engineering Review, Vol.41, No.1, 2008

[10] Kato, D., et al, "Development of Diffuser Passage Compressor Concept For Small Aircraft Engines", ISABE 2007-1166,2007 [11] Yamane, T., Yamamoto, K., Enomoto, S., Yamazaki, H., Takaki, R., and Iwamiya, T., "Development of A common CFD Platform-UPACS", Proc.Parallel CFD 2000 Conf.,Elsevier Science,pp.257-264,2001

[12] Takaki, R., et al, "Development of the UPACS CFD Environment", High Performance Computing, Proceedings of ISHPC 2003, Apring, pp307-319,2003

[13] Spalart, P. R., and Allmaras, S. R., "A One-Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 92-0439, Jan. 1992

[14] Komotori, K. and Miyake, K, 1977, "Leakage Characteristics of Labyrinth Seals with High Rotating Speed," Proc. Tokyo Joint Gas Turbine Congress, pp.378-380.

[15] Kato, D, Yamagami, M., Tsuchiya, N., Kodama, H., "The Influence of Shrouded Stator Cavity Flows on the Aerodynamic Performance of a High-Speed Multistage Axial-Flow Compressor," ASME GT2011-46300, 2011 (To be published).

[16] Chakravarthy, S.R., and Osher, S., "A New Class of High Accuracy TVD Schemes for Hyperbolic Conservation Laws," AIAA85-0363, 1985

[17] Baldwin, B.S., and Lomax, H., "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA78-257, 1978