COMPUTATION OF SUBSONIC AND TRANSONIC COMPRESSOR ROTOR FLOW TAKING ACCOUNT OF REYNOLDS STRESS ANISOTROPY

Toshiyuki Arima
HONDA R&D Co., Ltd.,
Wako Research Center,
Saltama, JAPAN

Toyotaka Sonoda
HONDA R&D Co., Ltd.,
Wako Research Center,
Saltama, JAPAN

Masatoshi Shirotori
HONDA R&D Co., Ltd.,
Wako Research Center,
Saltama, JAPAN

Yoshihiro Yamaguchi
HONDA R&D Co., Ltd.,
Wako Research Center,
Saltama, JAPAN

ABSTRACT

A two-layer k-ε/algebraic Reynolds stress model (ARSM) has been adopted to the three-dimensional, Reynolds-averaged Navier-Stokes code to include explicitly the Reynolds stress anisotropy. The code has been used to study the complex flow fields of a transonic axial compressor rotor (i.e., NASA Rotor 37) and a subsonic centrifugal compressor impeller (i.e., the backswept impeller of Krain, first reported in 1988). The computed results have been compared with those from a Baldwin-Lomax model, a low-Reynolds number k-ε turbulence model and actual experimental data. Calculated results for the axial compressor are compared with data reported by Suder in 1994. The suitability of the turbulence model to predict accurately the overall performance of the rotor, spanwise distributions of aerodynamic characteristics, and the wake flow profiles is assessed. Calculations for the centrifugal compressor impeller are compared with the experimental data reported by Hah and Krain in 1989. The usefulness of the turbulence models to predict accurately the overall performance of the impeller, the impeller-exit-velocity profile, and the meridional velocity and flow angle profiles at the cross-channel planes (via L2F measurements) has also been investigated. Calculations for modeling the turbulence of both the rotor and the impeller, reasonably good predictions have been obtained with the ARSM and the low-Reynolds number k-ε models, but have not been attainable using the Baldwin-Lomax model. The solutions obtained with the ARSM show better agreement with experimental data than those obtained with the other models. However, in some cases, the predicted differences between the ARSM and the low-Reynolds number k-ε models are not significant. The computed secondary flow and the relative helicity have also been used to investigate the effect of wall curvature and frame rotation on the flow field inside the centrifugal impeller for three operating conditions (i.e., design point, choke, and near surge) and the results are discussed.

NOMENCLATURE

\( C_1, C_2 = \) ARSM modeling parameter
\( G = \) Production rate of turbulent kinetic energy
\( F_{ij} = \frac{\partial u_i}{\partial x_j} \)
\( J = \) Jacobian of transformation = \( \partial(x_1, x_2, x_3)/\partial(x_1, x_2, x_3) \)
\( k = \) Turbulent energy
\( P_{ij} = \) ARSM production by mean shear stress
\( Re = \) Reynolds number
\( R_{ij} = \) ARSM production by system rotation
\( T_{ij} = \) ARSM modeling tensor
\( U_{yi} = \) component of relative velocity
\( U_2 = \) Impeller tip speed
\( V_m = \) Meridional velocity
\( V_\theta = \) Tangential velocity
\( x_i = \) Cartesian coordinate
\( y = \) Normal distance from wall
\( y^+ = \) Wall coordinate
\( \alpha = \) Absolute flow angle, \( \tan^{-1}(V_m/V_\theta) \)
\( \delta_{ij} = \) Kronecker's delta
\[\epsilon_{ij} = \text{Alternating tensor}\]
\[\epsilon = \text{Isotropic turbulent kinetic energy dissipation rate}\]
\[\mu = \text{Molecular viscosity}\]
\[\mu_t = \text{Turbulent viscosity}\]
\[\rho = \text{Density}\]
\[-\rho u_i \epsilon_{ij} = \text{Reynolds stress tensor}\]
\[\tau_{ij} = \text{Viscous stress tensor}\]
\[\Omega = \text{Angular velocity of rotation around the x_1 axis}\]

**INTRODUCTION**

Because of the requirements of both small size and light weight, as well as high efficiency and high loading (which are here opposing technical constraints), the blade row geometry of modern gas-turbine engines has become a three dimensional design consideration to minimize the losses caused by secondary flow. We therefore need to understand the complex flow field in such blade rows in detail. However, it seems that accurate prediction of the losses is not enough, as seen in results (Dent, 1996) of a blind test case organized under the auspices of ASME/IGTI (Wisler and Denton, 1994). There are probably several possible reasons for discrepancies between measurements and calculations. One of them could be attributed to the turbulence model. Shabbir et al. (1996) have assessed the turbulence models including a Baldwin-Lomax model, a standard k-ε model, and a NASA improved k-ε model. Their study had concluded that the improved k-ε model gave better solutions than the other models, because it better modeled a non-equilibrium flow region by using a local flow dependent parameter. Numerical studies taking account of the Reynolds stress anisotropy using an algebraic Reynolds stress model (ARSM) for a compressible flow have been performed by Ganeses and Lakshminarayana (1984). Kunz and Lakshminarayana computed the flow field inside a transonic centrifugal impeller (Krain, 1988) at the design point and compared the results computed using a k-ε model and two-layer k-ε/algebraic Reynolds stress models with and without a production tensor by frame rotation. They have stated that the results which incorporate the ARSM show significant, though not dramatic, differences, when compared with the k-ε model.

On the other hand, the effect of the secondary flow on the performance of a centrifugal compressor has been numerically investigated by many authors (e.g.; Krain and Hoffman, 1989; Hah and Krain, 1990; Moore and Moore, 1990; Kunz and Lakshminarayana, 1992; Hathaway et al., 1993; Hirsch et al., 1996). Hah and Krain have investigated the secondary flows inside the NASA low-speed centrifugal compressor, which was designed to simulate a flow field in high-speed centrifugal compressors, using laser measurements and Dawes’s three-dimensional viscous flow code. However, the effect of the turbulence model on their numerical investigation was not described. Hirsch et al. concluded that the effects of rotation and curvature on the turbulent flow were not predominant in the establishment of the main flow. The evidence indicating this conclusion, however, was not concretely shown in their paper. Moore and Moore proposed to modify a mixing length for the turbulent flow using a gradient of Richardson number based on rotation and local curvature. As described previously, computations taking into account Reynolds stress anisotropy can be seen in the study of Kunz and Lakshminarayana. However, to our knowledge, secondary flows in transonic compressors under various operating conditions have not been investigated via the ARSM.

The objectives of this study have been: (1) to assess the performance of the ARSM for both transonic axial and centrifugal compressors, (2) to ascertain whether rotation and curvature influence turbulent flow, and (3) to study the secondary flow, using the ARSM, in a high-speed centrifugal impeller, for three operating conditions (i.e., design, choke, and near surge).

**GOVERNING EQUATIONS AND NUMERICAL METHOD**

The basic equations governing the viscous flow through blade rows are the three-dimensional, Reynolds-averaged, Navier-Stokes equations, which can be written, in a normalized form in a generalized curvilinear coordinate system fixed to a rotating blade.

\[\tau_{ij} = \mu \left( u_{i,j} + u_{j,i} - \frac{2}{3} \delta_{ij} u_{ll} \right) - \rho u_i u_j \]

(1)

Here \(-\rho u_i u_j\) is the Reynolds stress tensor. To complete the system of governing equations, the Reynolds stress tensor is now defined for the different models.

**Eddy Viscosity Model**

The Reynolds stress tensor, \(-\rho u_i u_j\), can be evaluated using the Boussinesq approximation. The Reynolds stress tensor is assumed to be linearly proportional to the strain rate tensor.

\[-\rho u_i u_j = \mu_t \left( u_{i,j} + u_{j,i} - \frac{2}{3} \delta_{ij} u_{ll} \right) - \frac{2}{3} \delta_{ij} \mu_k \]

(2)

where \(\mu_t\) is evaluated using the Prandtl-Kolmogorov relation, i.e.:

\[\mu_t = C_{\mu} f_\mu \rho \frac{k^2}{\epsilon} \]

(3)

C\(_\mu\) and \(f_\mu\) are the constant and function proposed by Chien (1983) and defined by the expression,

\[C_{\mu} = 0.09, \quad f_\mu = 1 - \exp(-0.0115y^+)\]
Algebraic Reynolds Stress Model

The Reynolds stress tensor, $-\bar{\rho}u_i'u_j'$, has been evaluated via non-linear algebraic equations based on the Rodi's approximation (1976) to the transport equations of the Reynolds stress components. In this study, a compressible extension to the high-Reynolds number form by Galmes and Lakshminarayana (1984) has been adopted:

$$-\rho u_i'u_j' = -\rho k T_{ij} - \frac{2}{3} \delta_{ij} \rho k,$$  \hspace{1cm} (4)

where

$$T_{ij} = \frac{R_{ij} (2 - C_2) / 2 + (P_{ij} - 2PS_{ij} / 3)(1 - C_2)}{P + \rho e(C_1 - 1)},$$  \hspace{1cm} (5a)

$$R_{ij} = -2\Omega_k (e_{mn} + \rho u_i' u_m') + e_{jm} \rho u_i' u_m'),$$  \hspace{1cm} (5b)

$$P_{ij} = (-\rho u_i' u_j') + (\rho u_i' u_{j,k} - \rho u_j' u_{i,k}) \hspace{1cm} P = 1 - P_{ii},$$  \hspace{1cm} (5c)

and $C_1 = 1.5$; $C_3 = 0.6$.

This model does not include a wall reflection term, the so-called "echo effect", in the pressure-strain correlation terms. Therefore, a wall function must be applied in the near wall region, because the model is only valid in the fully turbulent region away from the wall. In this study, the Reynolds stress tensor near the wall is calculated from the Boussinesq's eddy viscous model described by Eq. (2) using the following transport equation for $k$ and $e$. The matching point for the present hybrid model of turbulence was chosen to be $y^{+}\text{max}=200$ as discussed by Kunz and Lakshminarayana (1992).

Transport Equations for $k$ and $e$

The low-Reynolds number formulation, of the transport equations for $k$ and $e$ proposed by Chien (1982) are solved to provide the values which appear in Eq. (2) and (4). They are represented in the conservative form in a generalized curvilinear coordinate system, as

$$\frac{\partial Q_T}{\partial t} + \frac{\partial F_{k}}{\partial x_i} = \frac{\partial G}{\partial x_j} + H_T$$  \hspace{1cm} (6)

where

$$Q_T = \frac{1}{J} \left( \frac{\rho k}{\rho e} \right),$$

$$F_{k} = \frac{1}{J} \left( \frac{\rho k U_k}{\rho e U_k} \right),$$

$$S_{\theta} = \frac{1}{J} \left( \begin{array}{c} \mu + \nu \frac{\partial^2}{\partial x_i} \\
\frac{\sigma_k}{\sigma_e} \frac{\partial}{\partial x_i} \\
\frac{\mu + \nu}{\sigma_e} \frac{\partial}{\partial x_i} \end{array} \right),$$

$$H_T = \frac{1}{J} \left( \begin{array}{c} G - \rho e - \frac{2\mu k}{y^2} \\
C_{e1}G - C_{e2}f_2 \rho e \frac{e}{k} - \frac{2f_3 \mu e}{y^2} \\
C_{e1}G - C_{e2}f_2 \rho e \frac{e}{k} - \frac{2f_3 \mu e}{y^2} \end{array} \right).$$  \hspace{1cm} (7)

Here, the rate of production of turbulent energy, $G$, is given by

$$G = -\rho u_i'u_j'\mu_{ij},$$  \hspace{1cm} (8)

where the components of the Reynolds stress tensor, $-\bar{\rho}u_i'u_j'$, are evaluated as the last iterated values calculated using Eq. (2) or Eq. (4).

The following constants and functions proposed by Chien are used:

$$\sigma_k = 1.0, \quad \sigma_e = 1.3, \quad C_{e1} = 1.35, \quad C_{e2} = 1.8, \quad f_2 = 1 - \exp(-0.5y'), \quad f_3 = 1 - \frac{2}{9} \exp\left(-\frac{R_i'}{36}\right),$$

Numerical Procedure

The governing equations are solved using an implicit time-marching finite-difference scheme to obtain a steady-state solution. The implicit approximate factorized (IAF) scheme based on the diagonal ADI (alternating direction implicit) scheme developed by Pulliam and Chaussee (1977) has been applied for the time integration. The high-accuracy total-variation-diminishing (TVD) formulation proposed by Chakravarthy and Osher (1985) has been used for the inviscid terms. The viscous terms have been evaluated using standard second-order, central-difference formulae.

The turbulence quantities are obtained by lagging, once the mean flow equations have been updated. The detailed numerical procedures have been previously reported (Arima et al., 1997). When the ARSM is used to model the turbulence flow, the nonlinear algebraic equations, Eq. (4), for the unknown six components of the Reynolds stress tensor are linearized using values of the previous time-step and are solved by the Gaussian elimination method at each grid point for each time-step.

Boundary Conditions

Along the blade surfaces, the no-slip condition is imposed on the velocities. The pressure is extrapolated from the adjacent grid points and the density is computed using the adiabatic wall condition. The turbulent kinetic energy and isotropic dissipation rate are set to zero along the solid boundaries. At the inflow boundaries, the total pressure and temperature profiles and two inlet-flow angles are specified; and the static pressure is extrapolated from the interior flow field domain. The turbulent kinetic energy and dissipation rate imposed at the inflow boundary are derived from the specified free-stream turbulence intensity and the unit eddy viscosity at the inflow boundary. At the outflow boundaries, the static pressure is specified. The density and velocity components, $k$ and $e$ are extrapolated from those at interior points. At the periodic boundaries upstream of the leading edge and downstream of the trailing edge, the values at two corresponding points should be equal. The tip clearance region is also handled by imposing periodic conditions across the blade and smoothly reducing the blade thickness to zero at the tip.

COMPUTATION OF TRANSONIC AXIAL COMPRESSOR

In order to validate the implementation of the ARSM and to assess the performance of turbulence models, the flow field of the transonic axial compressor, NASA Rotor 37, which has comparatively small streamline curvatures, has been computed using the ARSM. For comparisons, computations using the Baldwin-Lomax model and the low-Reynolds number $k$-$e$ model have also been performed. Rotor 37 had been originally designed as a test compressor for a core compressor of an aircraft engine at the NASA Lewis Research Center (Reid and
Figure 1. Meridional view of Rotor 37 geometry showing locations at which experimental measurements were made (Straziser, 1996)

Figure 2. Computational grid for Rotor 37

Figure 3. Comparisons of rotor performance for the different models

Figure 4. Comparisons of the spanwise distributions of the total pressure ratio and the total temperature ratio at Station 4 at 98% mass flow rate. We note that the ARSM is very similar to the low-Reynolds number k-ε model in the total pressure and total temperature profiles. In the total pressure profile, the Baldwin-Lomax model is qualitatively different from the other two models. The ARSM has a slightly better fit to the experimental data than the low-Reynolds number k-ε model. In the total temperature profile, similar trends for the three models have been found. There are troughs at approximately 10% and 70% span-height in the temperature profile. The troughs for the ARSM are closer to the experimental data than those for the other models although there are even discrepancies in this case close to the tip. The Baldwin-Lomax model considerably underestimates the total temperature in the midspan region.
Figure 4. Spanwise distributions of pressure ratio and temperature ratio at Station 4

Figure 5 shows the relative Mach number wake profile at 30%, 50%, and 70% span-height at Station 3 at 98% mass flow rate. The computed wake profiles for all three models are deeper than those observed experimentally at the three span-heights. The Baldwin-Lomax model profile is sharper than the other models at 30% and 70% span. At 50% span, the wake profiles for the three models are almost indistinguishable. This similarity in the wake profile predicted by the different turbulence models may be suggestive that some other factor than the turbulence model needs to be included to describe the observed results.

COMPUTATION OF SUBSONIC CENTRIFUGAL IMPELLER

To determine the relative predictions of the turbulence models for flow fields with large streamline curvature and rotation, the flow field of the subsonic centrifugal compressor impeller has been computed using the ARSM model, the Baldwin-Lomax model, and the low-Reynolds number k-e model. The centrifugal compressor chosen for this detailed numerical study here, is the one designed and tested by Krain (1988,1989). The rotor had been designed for a mass flow rate of 4.0 kg/s and a shaft speed of 22360 rpm. This rotor has 24 blades,
which are straight blade surfaces from hub to tip to allow flank milling manufacture and a 30 degree back-sweep. The design characteristics of this rotor are shown in Table 1. Figure 6 shows the locations for L2F measurements in the meridional plane of the impeller. All measurement planes are perpendicular to the shroud contour and they are located at meridional shroud lengths of 0, 20, 40, 60, 80, and 100.4 %.

The computational grid used in this analysis is shown in Figure 7. The computational grid has been generated algebraically using the rotor geometry provided by Crain (1997). The experimentally measured rotor had been fitted with a spherical nose cone. For the computations, therefore, the inflow boundary has been positioned in front of the spinner cone to take into account the acceleration due to the curvature on the spinner wall, and the outflow boundary has been set at the exit of the diffuser. Crain has reported that the measured tip clearance during operation was 0.5 mm at the inlet and 0.2 mm at the exit, respectively. The tip clearance has been varied linearly along the shroud from inlet to exit to generate the computational grid. The grid consists of 171 nodes in the streamwise direction, 49 nodes in the spanwise direction, and 49 nodes in the blade-to-blade direction. Ten nodes in the spanwise direction are used to describe the tip clearance. The grid clustering yielded as the nearest grid points to the solid wall satisfy the wall coordinate, \( y^+ \), at these positions is less than one.

**Validation and Assessment of Turbulence Model**

**Overall performance.** The computations had been performed to produce a performance map at design speed. Predictions from the Baldwin-Lomax model, the low-Reynolds number \( k-\varepsilon \) model, and the ARSM are compared with the experimental data to assess the performance of these turbulence models. Figure 8 shows a plot of total pressure rise characteristics of this impeller for the three models and the measured experimental results for comparison as a function of mass flow rate. Computed mass flow rates have been normalized in each case using the mass flow rate at which the total pressure ratio obtained using the low-Reynolds number \( k-\varepsilon \) model had been 4.7, i.e., the design point for the impeller. (The value of the predicted mass flow rate at which the total pressure ratio is equal to 4.7 was 4.25 kg/sec, while the measured one was 4.0 kg/sec.) The computed ARSM results are found to be very similar to those found for the low-Reynolds number \( k-\varepsilon \) model. The results for both the ARSM and the low Reynolds \( k-\varepsilon \) model are in good agreement with the experimental data. The Baldwin-Lomax model, however, underestimates the total pressure ratio.

**Impeller exit velocity profile.** In Figure 9, impeller-exit velocity profiles for the three turbulence models are compared and the underlying reason for the Baldwin-Lomax model's underestimating the total pressure becomes readily apparent. The velocity profile for the Baldwin-Lomax model has a higher gradient from hub to shroud than that for either the low-Reynolds number \( k-\varepsilon \) or the ARSM models. Differences between the low-Reynolds number \( k-\varepsilon \) model and the ARSM are not significant.

**Meridional Velocity and Flow Angle.** In order to validate the ARSM results, the meridional velocity and flow angle are compared with the experimental data. The calculated and measured meridional velocity contours at six measured planes at the design point are shown in Figure 10 in the form of contour lines of \( V_m/U_z \). The isolines of relative flow angle are shown in Figure 11. On each plane, the ARSM solution was interpolated in the streamwise direction to obtain predicted values of the velocity components at the experimental measurement planes shown in Figure 6. At plane 1, positive velocity...
gradients from the pressure side to suction side and from the hub to shroud caused by negative pressure gradients from the pressure side to suction side and from the hub to shroud can be observed in the experimental data. This tendency is also seen in the calculated results as the velocity contour lines. The computed results indicate velocity contours including a small concentric circle near the hub. This feature

![Figure 7. Computational grid of the Impeller of Krain](image)

![Figure 8. Comparisons of mass flow rate / total pressure ratio of the impleller of Krain for different models (Experimental data is taken from Krain, 1988)](image)

![Figure 9. Comparisons of Impeller exit velocity profile for the different models](image)
can not be observed in the experiment, because this region is located mostly outside of the optical measurement domain. Although similar features had been seen in calculations by Hah and Krain (1989), it has not been found in any of our Baldwin-Lomax calculations (not shown here) nor in the Baldwin-Lomax calculations reported by Hirsch et al. (1996). For the flow angle, good agreement between measurement and

Figure 10. Meridional velocity at planes I, II, III, IV, V, and VI
Calculation with ARSM model

Figure 11. Relative flow angle at planes I, II, III, IV, V, and VI
Calculation with ARSM model
calculation is also obtained at this plane. At plane II, the growth of the boundary layer along the blade surfaces causes the flow to begin to be dominated by viscous effects. Along the suction surface, the flow angle rapidly increases from the shroud toward the hub as seen in both the experimental and calculated results. At plane III, both the computed and measured results indicate the presence of low-momentum fluid near the pressure surface of the shroud. This low-momentum fluid is transported along the suction surface by the secondary flows arising from the axial-to-radial turning of the blade-boundary layer and is forced toward the middle of the channel by the tip-clearance flow. Along the suction surface, the flow angle increases toward the hub from the shroud as well as for plane II. At plane IV, the low-momentum fluid accumulates at the middle of the channel of the shroud. At plane V, the flow feature is similar to the flow field for plane IV except for the relative extent of the low-momentum fluid region on account of diminishing blade height. The location of the low-momentum fluid is very similar to that measured. At plane VI, the computed results are in qualitative agreement with the measurements, although the location of the computed minimum velocity is offset slightly to the pressure side away from the measurement. The low-momentum fluid migrating from the boundary layers is mixed out and the distribution of the meridional velocity becomes smooth. The calculated results are in relatively good agreement with the experimental data at six sections sampled.

**Secondary Flows Inside Impeller**

As discussed by previous authors (e.g., Cumpsty, 1989, Kunz, 1992, Hirsch, 1996), curvature, rotation and viscous physical phenomena strongly influence the impeller flow fields. Prior to discussion of the secondary flows inside the impeller, secondary flow motion caused by centrifugal forces, turning of the blade and endwall boundary layers, and Coriolis forces is reviewed for clarity and each type of flow is illustrated schematically in Figure 12.

**Secondary flow due to centrifugal force.** In Figure 12, the "centrifugal" secondary flow effect is illustrated in (a). The centrifugal force arises as a tangential component of the frame rotation to the absolute velocity vector inside the inducer of the impeller. As a consequence, a positive pressure gradient is generated from hub to shroud. This gradient extends in the blade boundary layers. However, the tangential component of velocity vector increases in the boundary layers on account of a decrease in the relative velocity. Therefore, the centrifugal force increases in the blade boundary layers; and secondary flows, migrating toward the shroud, occur.

**Secondary flow due to flow turning.** The "turning" secondary flow effect is illustrated in (b). A passage of a centrifugal impeller produces a convex surface at the shroud and a concave surface at the hub causing the flow to turn from axial to radial. Because the blade suction surface is convex and a pressure surface is concave, an easy inflow condition will be produced inside the inducer. In such a curved passage, a "centrifugal force" due to streamline curvature is generated and a positive pressure gradient from the suction surface to pressure surface is produced. While this gradient extends in the blade boundary layers, the Coriolis force is reduced in the boundary layers because of the small relative velocity in the boundary layers. Therefore, secondary flows from the pressure surface to suction surface are produced.

**Computed Secondary Flows Inside Impeller**

In general, the secondary flow effect is defined as the vector component perpendicular to the primary flow direction. As discussed by previous authors (e.g., Hathaway, 1993), given that the primary flow direction is the local streamwise grid direction, the computed secondary flow is just a measure of the departure of the local relative velocity vector from the local streamwise grid direction. The computed secondary flows inside the centrifugal impeller are shown in the form of limiting streamlines on the streamwise surface (for six L2F planes) in Figure 13, 14, and 15. For better clarity, the relative helicity contours are also shown in each figure. The sign of relative helicity indicates the rotational direction of a vortex and the position of its maximum value indicates the vortex core. These three figures correspond to the calculations for the design point, choke, and near surge conditions, respectively. All results presented here have been computed using the ARSM.

**Design point.** The computed results, for the points midway between plane I and II are shown in Figure 13(a). The computed results show the secondary flow effect calculated from the pressure side to the suction side, while moving inward. This flow moves radially outward along the blade surfaces because of the centrifugal force. A vortical flow due to the tip-leakage flow (very slight) has already been seen at the suction surface/shroud corner.

At plane II, shown in Figure 13(b), the calculation predicts a counterclockwise vortex on the suction side and two small clockwise vortices in the pressure side. The two vortices are smaller vortex at the pressure surface/hub corner and a larger vortex at the pressure surface/shroud corner. These vortices are caused by the centrifugal force in the blade boundary layer as illustrated by the flows labeled "a" and "b" in Figure 12. A spanwise flow along the suction surface migrates outward toward the shroud, is entrained in the tip-clearance flow, and migrates toward the pressure surface. This flow meets the flow from the pressure surface at the channel near the pressure surface; and they turn, moving inward together. The turning point is approximately at the same location as the low-momentum fluid region indicated in Figure 10(c). It is likely that these secondary flows transport low-momentum fluid from the passage boundary layers into the mainstream of the impeller.

The secondary flow results obtained at plane III are shown in Figure 13(c). While flowing toward plane III, the small vortex at the pressure/shroud corner disappears. The vortices on the suction surface and at the pressure surface/hub corner grow. These vortices form a large pair of counterclockwise and clockwise vortices. This vortex structure produces the typical "turning" secondary flows illustrated and labeled "b" in Figure 12.

At plane IV, the vortex on the pressure side continues growing. It becomes larger than the vortex on the suction side. This tendency of vortex motion may be caused by the "Coriolis" secondary flow labeled "c" in Figure 12 that begins to influence the main flow at this section. The flow migrating from the suction surface along the shroud turns, goes back to the suction side and moves inward toward the hub. The
Figure 12. Schematic secondary flows inside a centrifugal impeller

- a: Secondary flows due to centrifugal force
- b: Secondary flows arising from axial-to-radial turning of blade boundary layers
- c: Secondary flows due to Coriolis force
- d: Leakage flows from tip clearance

Figure 13. Computed secondary flows in form of limiting streamlines and relative helicity contours on planes I, II, III, IV, V, and VI at the design point
flow turning region shifts to the middle of the channel and begins to spread on the shroud wall. This feature may indicate that the flow migrating from the suction side rolls up at this area. The low-momentum fluid region also shifts to the same location, as seen in Figure 10(d).

At plane V, the vortex on the pressure side continues growing. It stretches in the blade-to-blade direction, while the passage height diminishes, and approaches the suction surface/hub corner at the inner passage. The core of the suction vortex is pushed up toward the suction surface/shroud corner. The flow turning region on the shroud becomes larger than that at plane IV.

At the position of 96% meridional distance, we see a dramatic change in the two vortices. The two vortices, the pressure side vortex and the suction side vortex, are stretched in the streamwise direction and they both decrease in size and shift to the shroud corners. The flow migrating from the suction side is still turning inward in the passage outer region. At plane VI, the vortices do not appear. However, from the relative helicity contours the extent of the clockwise vortex can be seen in the region from the pressure surface/shroud corner to the center of the passage. It seems to be difficult to define the computed secondary flow at this plane, because of a slip of the outflow at the exit of the blade and a vagueness of the primary flow direction.

**Choke condition.** The secondary flows for the choke condition are shown in Figure 14 in a manner similar to that for the design point. The result at the middle section between planes I and II is shown in Figure 14(a). The computed results are similar to those at the design point in the region from the pressure surface to the inner about 50% span of the suction surface. However, the secondary flow moving radially outward along the suction surface cannot be seen in the outer about 60% span of the suction surface. The tip-leakage flow cannot also be seen at the suction surface/shroud corner. At plane II, shown in Figure 14(b), the calculation predicts the two counterclockwise vortices on the suction side and a vortex at the pressure/shroud corner. One of the two suction vortices is at the shroud corner and the other is at the hub corner. The vortex at the suction surface/shroud corner may be caused by the tip-leakage flow. The tip-leakage flow starts at the downstream position relative to that at the design point shown in Figure 13(b). The vortex at the suction surface/hub corner induced by the migration of the passage boundary layer along the blade surface is smaller than that at the design point. In contrast, the growth of the clockwise vortex at the pressure surface occurs a little earlier than for the corresponding at the design point. At plane III shown in Figure 14(c), the growth of the counterclockwise vortex at the suction surface is slow relative to that at the design point and its core is at the middle of the blade span. At plane IV shown Figure 14(d), the flow migrating from the suction surface along the shroud turns, goes back toward the suction side and moves inward toward the hub in a manner similar to that observed at the design point. The flow turning region is closer to the shroud than that at the design point. This feature may indicate that the low-momentum fluid migrating from the boundary layer on the blade suction surface is rolling up with the tip-leakage flow close to the shroud. At plane V, the flow turning toward the suction side also occurs close to the shroud similar to that at plane IV. From the 96% meridional shroud distance to the plane VI, the secondary flow feature is very similar to that at the design point.

In summary, the results shown in Figure 14 indicate that the development of the counterclockwise vortex on the suction side is slower than that at the design point. The roll-up of the passage boundary layer and tip-leakage flow occurs closer to the shroud than that at the design point.

**Near surge.** The secondary flows for the near-surge condition are shown in Figure 15. The results at the middle section between plane I and II are shown in Figure 15(a). The computed results indicate that the counterclockwise vortex caused by the upward migration of the blade-boundary layer and the tip-leakage flow has already been formed at the suction surface/shroud corner of the passage. At plane II, shown in Figure 15(b), the calculation predicts the early development of the counterclockwise vortex on the suction side. There is a small vortex at the pressure surface/hub corner of the passage. At plane III shown in Figure 15(c), the pressure vortex grows rapidly and becomes the same size as that on the suction side. The flow migrates from suction surface along the shroud, turns toward the suction side, and moves inward toward the hub just as for the choke condition. At plane IV, shown in Figure 15(d), there is a dramatic change in the pressure vortex; it covers most of the passage. The position at which the flow, migrating along the shroud, turns toward the suction surface, is correspondingly closer to the shroud than that for the other two cases. The inward flow, migrating toward the hub between the pressure vortex and the suction vortex, approaches the suction surface. The suction vortex, developed earlier, is pushed to the blade suction surface and decreases in size. Its core can be seen at the suction surface/shroud corner. This feature suggests that the low-momentum fluid also migrate near the suction surface/shroud corner. At plane V, the pressure vortex enlarges further and its core approaches the center of the passage. The suction vortex reduces further in size. In the computed secondary flow, the vortices disappear between the plane at 96% meridional shroud distance and plane VI. However, we find from relative helicity contours that the counterclockwise vortex occupies almost the whole passage as it does at plane V.

In summary, although the counterclockwise vortex develops in the beginning, it decreases in size and its core shifts to the suction surface/shroud corner, while migrating downstream. In contrast, the clockwise vortex at the pressure surface enlarges rapidly and occupies most of the passage.

**Impeller Exit Wake Profile**

Figure 16 shows the computed wake profiles at the impeller exit for the three conditions. The wake profile at the choke condition has a higher gradient from hub to shroud than at the design and surge conditions. As Hab and Krain (1989) point out, this might be because the swirling vortex is superimposed on the mainstream and hence the low-momentum flow does not have time to mix out into the mainstream (on account of the increasing mass flow rate). In other words, the vortices, caused by the secondary flow between the clockwise and the counterclockwise vortices, play an important role in the transport of low-momentum fluid into the mainstream and in making the impeller exit velocity profile uniform. Figure 17 demonstrates the trajectories of particles released at the inlet/hub surface and inlet/suction surface for the design point (a), choke (b), and near surge (c) conditions. The tracers illustrate how low-momentum fluid along the blade surfaces migrates downstream. Taken all together, the blue tracers (released at the inlet/hub surface) move to the suction side on the hub wall due to the blade-to-blade pressure gradient, migrate along the blade surface toward the shroud, and then begin to roll up in the blade passage. The red tracers (released at the inlet/suction surface) migrate along the blade surface toward the shroud due to the centrifugal force, and then
Figure 14. Computed secondary flows in form of limiting streamlines and relative helicity contours on planes I, II, III, IV, V, and VI at the choke condition.

Figure 15. Computed secondary flows in form of limiting streamlines and relative helicity contours on planes I, II, III, IV, V, and VI at the surge condition.
Figure 16. Computed impeller exit velocity profiles for the three operating conditions

Figure 17. Perspective view of the paths of the tracers released at Inlet / hub (blue) and Inlet / suction (red) surfaces

roll up with the blue tracers. It might be reasonable to suppose that these trajectories indicate the motion of the low-momentum fluid region produced by the secondary flows shown in Figure 13 to 15. At the choke condition, the roll-up flows are more concentrated than those for the design point threshold, when it begins rolling up. In other words, the counterclockwise vortex on the suction side is smaller than that at the design point at plane II and III, as shown in Figure 14. At the surge condition, there are indications that the roll-up flow is strongly inhibited. The blue particles adhere to the suction surface and then roll up partly. The red particles rapidly migrate radially along the inlet part of the blade, and then move downstream along the shroud surface. There is a close correlation between the deficits of the wake profiles shown in Figure 16 and the low-momentum fluid regions demonstrated by observing the tracer trajectories.

CONCLUSIONS

Three-dimensional Navier-Stokes computations using an algebraic Reynolds turbulence stress model have been used to study the complex flow fields of a transonic axial compressor rotor and a transonic centrifugal compressor impeller. In order to assess the performance of the turbulence models, computations using an algebraic Reynolds stress model, a Baldwin-Lomax model, and a low-Reynolds number k-ε model were performed and were compared with experimental data. Furthermore, the secondary flows in the centrifugal impeller generated by the complex curvature, the rotation and the tip-clearance flow have been presented and investigated.

From the assessment of the turbulence models, the following conclusions have been drawn:

1. It is found that the Baldwin-Lomax model is less accurate than the other two turbulence models using transport equations.
2. It is found that the algebraic Reynolds stress model predicted better results than the Baldwin-Lomax model and the low-Reynolds number k-ε model for the prediction of overall rotor performance. However, differences between the algebraic
Reynolds stress model and the low-Reynolds number k-ε model are not significant.

3. The computed wake profiles for Rotor 37 were too deep for all three models. These deficiencies could not be attributed only to the turbulence model.

From the investigation of the secondary flow inside the centrifugal impeller, the following conclusions have been drawn:

1. At choke condition, the development of the counterclockwise vortex on the suction side is slower than that at the design point. The roll-up of the passage boundary layer and the tip-leakage flow occurs closer to the shroud for this case than at the design point.

2. At the surge conditions, although the counterclockwise vortex develops in the beginning, it decreases in size and its core shifts to the suction surface/shroud corner, while migrating downstream. In contrast, the clockwise vortex at the pressure surface enlarges rapidly and occupies most of the passage.

3. The secondary flows which form as counterclockwise and clockwise vortices may play an important role in transporting the low-momentum fluid into the mainstream and in making the impeller exit wake profile uniform.

ACKNOWLEDGEMENTS

The authors are grateful to Dr. Hartmut Krain of the German Aerospace Research Establishment for providing the blade geometry and experimental data. We would also like to thank Dr. Atsuo Tamura of National Aerospace Laboratory in Japan for his useful comments. Finally, we thank HONDA R&D Co., Ltd. for permission to carry out this work.

REFERENCES


