However, in a geometrically complex channel such as a centrifugal impeller blade passage, the streamwise direction can be defined in several different ways, and each definition will yield a slightly different result for the secondary flow. This problem has been pointed out by many previous authors. The purpose of the following discussion is to document the procedure used to generate the secondary flow field results presented in this paper. The same procedure is applied to both the computational and experimental results.

The secondary flows presented in this paper are defined as the departure of the local relative velocity vector from the local streamwise grid direction. The secondary velocity vector is given by $V_T - V_p$, where $V_T$ is the relative velocity vector and $V_p$ is the projection of $V_T$ in the local streamwise grid direction, $g(r, \theta, z)$. Euclidean and pitchwise components of the secondary velocity vector, $V_p$ and $V_q$, are the projections of the secondary velocity vector in the local spanwise and pitchwise grid directions, as given by the two dot products

$$V_p = (V_T - V_q) \cdot \hat{g}_s$$

$$V_q = (V_T - V_p) \cdot \hat{g}_p$$

where $\hat{g}_s$ and $\hat{g}_p$ are unit vectors in the local spanwise and pitchwise grid directions. When secondary flow results are presented in the form of vector plots in a quasi-orthogonal plane, $V_p$ and $V_q$ are used to determine the magnitude and direction of the plotted secondary velocity vectors.

The procedure described above is applied at each measurement grid node when processing experimental data and at each CFD grid node when processing CFD results. Thus a flow field with no secondary flow components will appear as a point at each grid node, indicating that the flow is following the streamwise grid direction. Since the local streamwise grid direction is parallel to the blade, hub, and shroud surfaces, the definition of secondary flow presented above also insures that the secondary velocity is zero at all solid surfaces.

The quasi-meridional velocity component, $V_{qm}$, is the vector projection of the meridional velocity vector, $V_m$, in the local streamwise grid direction, $V_{gm} = V_m \cdot \hat{g}_m$. Since a quasi-orthogonal plane is nearly normal to the streamwise grid direction in the meridional plane at any station in the impeller, $V_{qm}$ is a close approximation to the throughflow velocity that crosses a quasi-orthogonal plane. $V_{qm}$ is also a close approximation to the streamwise velocity component measured in laser anemometer investigations published by previous authors (Krain, 1988; Ahmed and Elder, 1990; Fagan and Fleeter, 1991). In these investigations, the streamwise velocity component was defined as the velocity component in the direction tangent to the shroud meridional direction at each measurement station.

**DISCUSSION**

J. Moore and J. G. Moore

This paper is a welcome milestone in NACA/NASA studies of large-scale centrifugal compressor impeller flows. These experimental measurements in the Low-Speed Centrifugal Compressor Facility were made to provide "benchmark" data for the verification of three-dimensional viscous flow codes and to aid in the development of more sophisticated models of the various physical phenomena occurring in centrifugal compressors. This study with a 60-in. diameter wheel is the modern equivalent of the earlier NACA study with the 48-in. wheel (Hamrick, 1956).

To aid this study we performed a series of three computational investigations (Moore and Moore, 1988, 1990a, 1990b). First, we calculated the flow and performance of the NACA 60-in. impeller at design and off-design conditions. Next, we predicted the flow in the NASA 60-in. wheel at design conditions with a tip clearance gap varying from 2.6 percent of the hub-shroud blade height at the impeller inlet to 4.0 percent of the blade height at the exit. Then we made a computational prediction of the possible effects of curvature and rotation on turbulence and thus on the flow development in the NASA wheel.

Partly as a result of our predictions, the tip clearance gap for the tests in the present paper was reduced; it varies from 1.2 to 1.8 percent of local blade height, approximately one half of the clearance gap we used. Figure 14 shows results of our prediction of the effect of varying the tip clearance gap on the flow in the 60-in. impeller. The calculation method was the same as described by Moore and Moore (1990a). With tip leakage flow, the entropy contours show the high loss or wake fluid to be spread from the pressure-side/shroud corner to midshroud. Without tip leakage, the high loss fluid accumulates near the shroud toward the suction side. The tip leakage flow produces higher losses and convects them toward the pressure side.

To predict the effects of curvature and rotation on turbulence in the NASA wheel, one can calculate a gradient Richardson number based on rotation and local flow curvature and modify a mixing length for the turbulent flow. We used the following model, which applies a mean modification factor $\bar{F}$ in shear layers and the outer regions of boundary layers:

$$\mu_f = \rho L^3 \frac{du}{dy}$$

$L = \text{smaller of } 0.41\frac{\gamma}{\mu} \quad 0.085 \bar{F}$

Van Driest correction used in 0.41y region

$F = 1 - \beta \text{ Ri } \quad \text{for } \text{ Ri } < 0$

$F = 1/(1 + \beta \text{ Ri }) \quad \text{for } \text{ Ri } > 0$

$\beta = 4$
Fig. 15 Rotation/curvature modification factors, $F^2$, for the turbulent viscosity 74 percent of the way through the impeller: left: enhancement factor contours, $F^2 = 1, 2, 4, 8, 16$; right: reduction factor contours, $F^2 = 1/1.1, 1/2, 1/4, 1/8, 1/16$

Fig. 16 Entropy contours for calculations without and with the rotation/curvature turbulent viscosity modification

$$Ri = \frac{2u \times (u \cdot \nabla u - \Omega \times u) \cdot [u \times (\nabla \times u + 2\Omega)]}{|u \times (\nabla \times u + 2\Omega)|^2}$$

$Ri = (\text{curv-rot}) (\text{abs vorticity})$ deformation squared

Figures 15 and 16 show results from this analysis for the NASA impeller flow with the original large tip gap (2.6-4.0 percent). In Fig. 15, the modification factors $F^2$ for the turbulent viscosity are shown on a plane 74 percent of the meridional distance through the impeller, together with the individual contributions from rotation and curvature. The classical two-dimensional boundary layer modifications can be seen with enhancement due to rotation in the pressure side boundary layer, reduction due to rotation in the suction side boundary layer, and enhancement due to curvature on the concave hub wall. In the wake there are enhancements and reductions due to rotation where the entropy gradients are in the same sense as in the blade boundary layers. The wake also sees a reduction due to curvature near midheight where the entropy gradient is in the same sense as for a shroud wall (convex wall) boundary layer.

Overall, in Fig. 16, we see that the character of the flow development was the same with and without the turbulence modifications. The shape of the wake was slightly modified by changes in the turbulent mixing at the jet/wake boundary. There was increased mixing on the suction side of the wake leading to smaller entropy gradients, and reduced mixing on the hub and pressure sides of the wake resulting in larger entropy gradients.

We would expect these turbulence modifications, giving changes of the order of factors of two to four in turbulent viscosity, to be present and measurable in the large-scale NASA wheel. In addition, since there is significant backflow in the meridional direction along most of the shroud wall, especially with the large tip gap, we would expect enhanced turbulence near the shroud as this flow mixes to exit from the impeller. We encourage the authors and their NASA colleagues to study this interesting flow physics further.

References


Authors' Closure

The authors would like to thank the Moores for their continued interest in this research program and for their kind remarks regarding this paper. As previously reported (Hathaway et al., 1992), the Moores were awarded a NASA grant prior to initiation of the Low-Speed Centrifugal Compressor (LSCC) research program. This grant was to provide a calculation of the impeller flow field to determine whether any modifications might be required that would enhance the worth of the experiment to the turbomachinery community. Their predictions, together with experimental measurements, resulted in reduction of the LCSS tip clearance to avoid a shroud wall separation.

Due to the manner in which the LSCC laser anemometer data have been acquired to date, details of the three-dimensional turbulence field cannot be determined. Therefore, it is not currently possible to assess the validity of the Moores’ predictions of the effects of curvature and rotation on turbulence. However, further research is planned in the LSCC that will include measurements of the level of unsteadiness of the three-dimensional velocity field. Furthermore, the authors believe that the throughflow velocity wake, which largely dominates the flow field, is an instationary vortical structure that requires further study.