Acknowledgments

The authors appreciate the assistance of A. Kondo, a former undergraduate student of Keio University, in the present study. Thanks are also due to Diamond Air Service, Nagoya, Japan, for their faithful cooperation in the course of the parabolic flight experiments.

References

[1]Tilton, D. E., Ambrose, J. H., and Chow, L. C., "Closed-System, High-Flux Evaporative Spray Cooling," Society of Automotive Engineers, SAE-892316, Sept. 1989.

[2]Choi, K. J., and Yao, S. C., "Mechanism of Film Boiling Heat Transfer of Normally Impacting Spray," *International Journal of Heat and Mass Transfer*, Vol. 30, No. 2, 1987, pp. 311–318.

[3]Oka, T., Abe, Y., Mori, Y. H., and Nagashima, A., "Pool Boiling of *n*-Pentane, CFC-113 and Water Under Reduced Gravity," *Transactions of the American Society of Mechanical Engineers Journal of Heat Transfer* (to be published).

Simulation of Turbulent Heat Transfer in a Rotating Duct

Sandip Dutta,* Malcolm J. Andrews,† and Je-Chin Han‡

Texas A&M University, College Station, Texas 77843-3123

Introduction

WE consider the turbulent heat transfer in a rotating duct, typical of cooling channels in a turbine blade. Figure 1 shows the geometry of the problem. Howard et al.[1] formulated a Coriolis turbulence production term for use in two-equation turbulence models without heat transfer. The predictions of Prakash and Zerkle[2] and Tekriwal[3] without a Coriolis turbulence production term in a k-ε turbulence model predict Nusselt numbers from trailing and leading surfaces that compare unevenly with experimental data. Dutta et al.[4] improved the predictions with the Coriolis modified turbulence term of Howard et al.[1] This Note presents further improvement of the work by Dutta et al.[4] and uses the published experimental heat transfer data of Wagner et al.[5] as a basis for comparison. Detailed consideration of the inlet effects, asymmetric distribution of turbulence due to rotation, and buoyancy effects are included.

Analysis

This work uses the high Reynolds number k-ε model of Launder and Spalding[6] incorporated into the PHOENICS computer code, and the nonequilibrium wall functions of Launder and Spalding[6] applied to near wall nodes. However, Coriolis and centrifugal buoyancy terms are added to the base model. Our modified model includes a Coriolis turbulence production, $P_C = 9\Omega\mu_t\cdot(\partial v_z/\partial x)$, and a buoyancy generated turbulence production, $P_b = \mu_t/Pr_t\cdot(\Omega^2 z)/T\cdot(\partial T/\partial z)$. Typical governing equations are given in Prakash and Zerkle,[2] while detailed equations are available in Dutta et al.[4]

Howard et al.[1] found that the Coriolis modified turbulence model increases the flow velocity and turbulence near the trailing side for radial outward flow in rotation, but reduces it near the leading side and stabilizes the flow. Their predictions were verified with experimental data in an unheated duct. We found that besides the Coriolis effect, in a heated duct rotational buoyancy due to temperature gradients accelerates the cooler fluid near the trailing edge and decelerates the warmer fluid near the leading surface. In the presence of a strong buoyancy force, the flow may separate near the leading side and the velocity and turbulence distribution may be entirely different from that in an unheated rotating duct.

The experimental setup of Wagner et al.[5] had a plenum chamber at the inlet of the test section. Their nonrotating flow measurements showed a fully developed turbulent flow at the inlet of the test section. However, no measurements with rotation were presented. The hydraulic diameter of the plenum chamber was larger than the hydraulic diameter of the actual test section. An increased hydraulic diameter means a larger cross-sectional flow area. Therefore, the axial flow velocity through the plenum chamber would be less than the axial flow velocity through the test section with an increase of the rotation number (Fig. 1) in the plenum chamber. Previous numerical predictions did not include the effect of the plenum chamber at the inlet. Prakash and Zerkle[2] and Tekriwal[3] started their computation domain from the start of heating and used fully developed turbulent profiles as the inlet condition. This approximation at the inlet condition resulted in a computation of a leading side Nusselt number that was higher than the trailing side, which is contrary to the experimental data. Rotating ducts are short in length and so inlet conditions need to be carefully prescribed. This work extended the computation domain upstream of the heated test section to allow flow development.

The cross-sectional area of the plenum chamber is 2.5 times that of the test section. To account for the area increase we have correspondingly increased the rotation speed by a factor of 4 along the unheated section of the duct ($z_0 < 0$ in Fig. 1) to maintain the same rotation number as in the plenum. Thus, we have a reasonable representation of the plenum chamber prior to the test section that better captures inlet profiles to the actual test section. This modification of the inlet condition

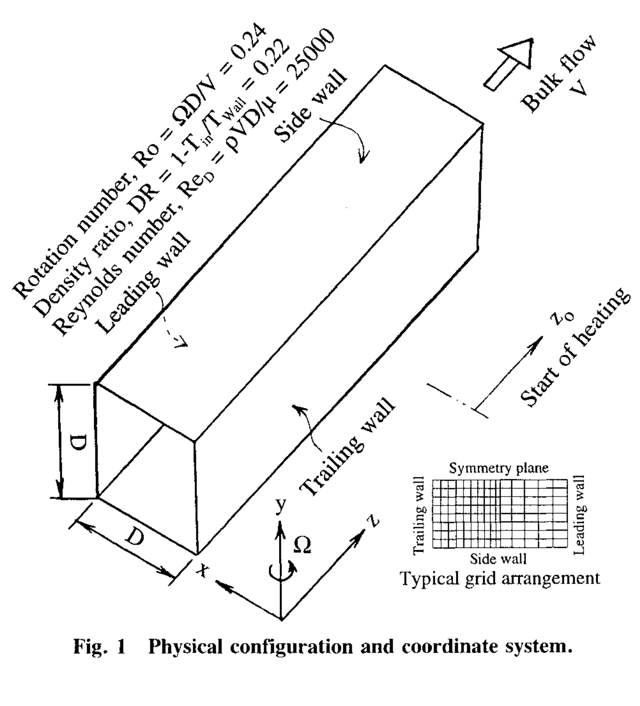

Fig. 1 Physical configuration and coordinate system.

Received Oct. 18, 1993; revision received July 5, 1994; accepted for publication Oct. 13, 1994.

*Graduate Student, Department of Mechanical Engineering. Student Member AIAA.

†Assistant Professor, Department of Mechanical Engineering. Member AIAA.

‡HTRI Professor, Department of Mechanical Engineering. Associate Fellow AIAA.

coupled with a Coriolis modification in turbulence gives improved predictions.

Figure 1 shows a schematic of the problem and computational domain as well as the trailing and leading walls. Inlet is at the center of rotation ($z = 0$) and the turbulent rotating flow develop through an unheated section. Outlet flow conditions are taken as zero gradient and outflow only. Such an outlet condition near the end of the test section would give poor results, especially with flow reversal, and so the computational domain was extended five hydraulic diameters beyond the experimental domain to ensure that outlet conditions do not affect the comparison with experimental data. Figure 1 shows the cross-sectional computational grid. Successful grid independence tests similar to Dutta et al.[4] and Tekriwal[3] were performed. The wall function formulation requires that the near wall nodes be at $y^+ > 12$. Low turbulence in the leading half restricts the grid compaction towards the wall. The computation takes advantage of the symmetry present in the flow so that only one-half of the channel is modeled. The heated walls are at a uniform temperature. The flow and heating conditions are $Re_D = 25{,}000$, $Ro = 0.24$, and $DR = 0.22$, also indicated in Fig. 1.

Results

Figure 2 shows the improvements obtained from the modified model over the standard k-ε model. The Nusselt numbers are normalized by the Nusselt numbers for fully developed pipe flow and plotted against the distance measured from the start of heating nondimensionalized by the hydraulic diameter of the duct. With the improved inlet condition and with the Coriolis modified turbulence model, the predictions compare satisfactorily with the experimental results. Turbulence production due to the Coriolis term increases turbulence k in the trailing half and decays turbulence near the leading half. This Coriolis effect increases the separation of the Nusselt number profiles between leading and trailing sides and the buoyancy effect causes an increase of Nusselt number in the latter part of the channel.

Figure 3 shows the velocity and temperature distributions in the channel. One-half of the square duct is shown in the figures because of symmetry. The axial velocity contour of Fig. 3a shows that the flow near the trailing wall is higher than the flow near the leading wall. As a consequence, the increase in temperature of the fluid near the trailing wall is less compared with that near the leading wall (Fig. 3b). This asymmetric distribution of temperature coupled with a large centrifugal force makes the buoyancy effects significant. Figure 3a shows that flow has separated at the leading wall (negative velocity contours) due to a strong rotational buoyancy effect.

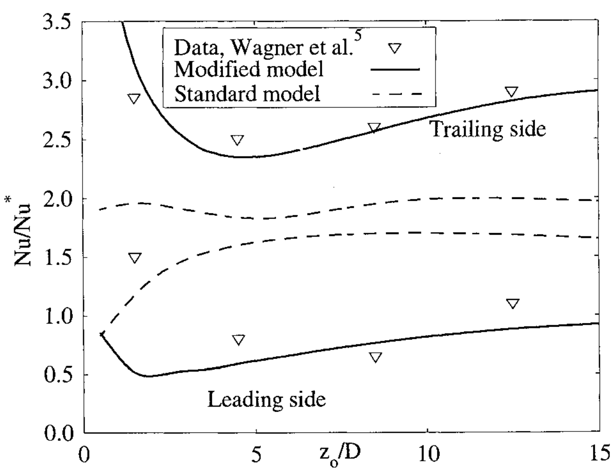

Fig. 2 Surface-averaged local Nusselt number distributions.

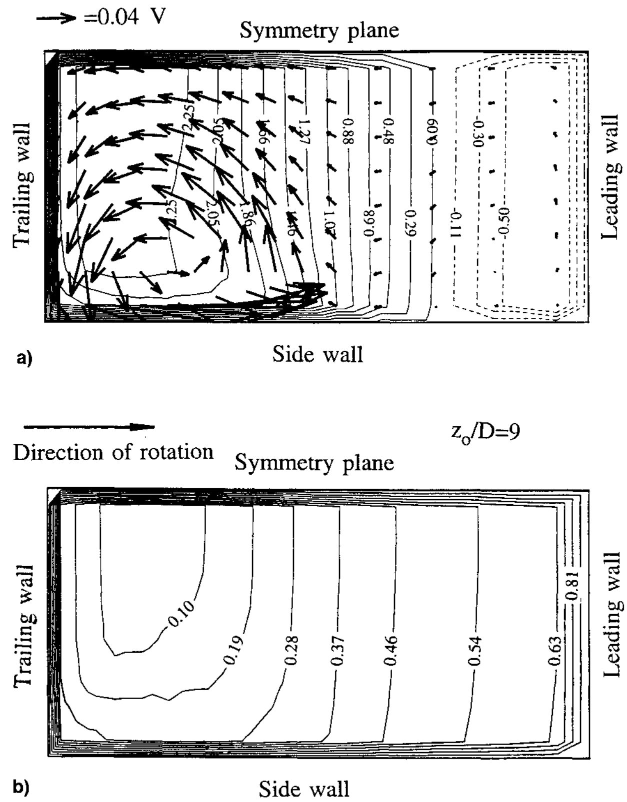

Fig. 3 Velocity and temperature distribution in the rotating square duct: a) axial velocity/V contours and secondary flow vectors and b) $(T - T_{in})/(T_{wall} - T_{in})$ contours.

The standard k-ε model includes the Coriolis and rotational buoyancy effects in the momentum equations, however, there are no corresponding terms in the k and ε transport equations. Our modified model adds turbulence production and dissipation from the Coriolis and buoyancy effects. The results show that these modifications are necessary to obtain reasonable predictions of rotational heat transfer effects in the cooling channels of a turbine blade.

Acknowledgment

Support from the Supercomputing Center of Texas A&M University is gratefully acknowledged.

References

[1]Howard, J. H. G., Patankar, S. V., and Bordynuik, R. M., "Flow Prediction in Rotating Ducts Using Coriolis-Modified Turbulence Models," *Journal of Fluids Engineering*, Vol. 102, Dec. 1980, pp. 456–461.

[2]Prakash, C., and Zerkle, R., "Prediction of Turbulent Flow and Heat Transfer in a Radially Rotating Square Duct," *Journal of Turbomachinery*, Vol. 114, Oct. 1992, pp. 835–846.

[3]Tekriwal, P., "Heat Transfer Predictions with Extended k-ε Turbulence Model in Radial Cooling Ducts Rotating in Orthogonal Mode," *Journal of Heat Transfer*, Vol. 116, May 1994, pp. 369–380.

[4]Dutta, S., Andrews, M. J., and Han, J. C., "Numerical Prediction of Turbulent Heat Transfer in a Rotating Square Duct with Variable Rotational Buoyancy Effects," *Proceedings of the AIAA/ASME 6th Thermophysics and Heat Transfer Conference* (Colorado Springs, CO), *General Papers in Heat and Mass Transfer, Insulation, and Turbomachinery*, HTD Vol. 271, June 1994, pp. 161–170.

[5]Wagner, J. H., Johnson, B. V., and Hajek, T. J., "Heat Transfer in Rotating Passages with Smooth Walls and Radial Outward Flow," *Journal of Turbomachinery*, Vol. 113, Jan. 1991, pp. 42–51.

[6]Launder, B. E., and Spalding, D. B., "The Numerical Calculation of Turbulent Flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, No. 2, 1974, pp. 269–289.