ABSTRACT
Numerical predictions of three-dimensional flow and heat transfer are presented for non-rotating and rotating turbine blade cooling passages with or without the rib turbulators. A multi-block Reynolds-averaged Navier-Stokes method was employed in conjunction with a near-wall second-moment closure to provide detailed velocity, pressure, and temperature distributions as well as Reynolds stresses and turbulent heat fluxes in various cooling channel configurations. These numerical results were systematically evaluated to determine the effect of blade rotation, coolant-to-wall density ratio, rib shape, channel aspect ratio and channel orientation on the generation of flow turbulence and the enhancement of surface heat transfer in turbine blade cooling passages. The second-moment solutions show that the secondary flow induced by the angled ribs, centrifugal buoyancy, and Coriolis forces produced strong nonisotropic turbulent stresses and heat fluxes that significantly affected flow field and surface heat transfer coefficients.

INTRODUCTION
Advanced gas turbine engines operate at high temperatures (1200-1500°C) to improve thermal efficiency and power output. As the turbine inlet temperature increases, the heat transferred to the turbine blade also increases. The level and variation in the temperature within the blade material, which cause thermal stresses, must be limited to achieve reasonable durability goals. The operating temperatures are far above the permissible metal temperatures. Therefore, there is a critical need to cool the blades for safe operation. The blades are cooled with extracted air from the compressor of the engine. Since this extraction incurs a penalty on the thermal efficiency and power output of the engine, it is important to understand and optimize the cooling technology for a given turbine blade geometry under engine operating conditions. Gas turbine cooling technology is complex and varies between engine manufacturers. Figure 1 shows the common cooling technology with three major internal cooling zones in a turbine blade with strategic film cooling in the leading edge, pressure and suction surfaces, and blade tip region. The leading edge is cooled by jet impingement with film cooling, the middle portion is cooled by serpentine rib-roughened passages with local film cooling, and the trailing edge is cooled by pin fins with trailing edge injection. This paper focuses on the detailed flow and heat transfer distributions in turbine blade cooling passages with rib turbulators. Interested readers are referred to several recent publications that address state-of-the-art reviews of turbine blade cooling and heat transfer. These include rotational effect on the turbine blade coolant passage heat transfer by Dutta and Han [1], recent developments in turbine blade film cooling by Han and Ekkad [2], and recent developments in turbine blade internal cooling by Han and Dutta [3]. A recent book focusing entirely on the range of gas turbine heat transfer issues and the associated cooling technology is available by Han et al. [4]. A symposium volume dealt with heat transfer in gas turbine systems is recently edited by Goldstein [5]. A detailed review of convective heat transfer and aerodynamics in axial flow turbines is now available by Dunn [6].
transfer in the internal coolant passages from the non-rotating channels. The presence of rib turbulators adds a further complexity since these ribs produce complex flow fields such as flow separation, reattachment and secondary flow between the ribs, which produce a high turbulence level that leads to high heat transfer coefficients.

1.2 Literature Review: Experimental Studies. The complex coupling of the Coriolis and buoyancy forces with flow separation/reattachment by the sharp 180° turn and by the ribs has prompted many investigators to study the flow and temperature fields generated in non-rotating rectangular ducts with smooth walls. Using the near-wall second-moment closure model accurately predicted the complex three-dimensional flow and heat transfer characteristics resulting from the rotation and strong wall curvature. The authors investigated the effect of channel orientation on rotating ribbed two-pass rectangular channel with channel aspect ratio of 2:1. Griffith et al. [20] studied the effect of channel orientation on rotating smooth and ribbed rectangular channels with channel aspect ratio of 4:1.

1.3 Literature Review: Numerical Studies

1.3.1 Smooth Surfaces. In addition to the experimental studies mentioned above, several studies have been made to predict numerically the flow and heat transfer in radially rotating smooth and ribbed ducts. Iacovides and Lauder [21], Prakash and Zerkle [22], Dutta and Han [12], Soong et al. [13] and Azad et al. [14] investigated rotating ducts with smooth walls. Wagner et al. [11], Han and Park [7], Han et al. [8] Ekkard and Han [9] and Liou et al. [10] and the references cited there. Experimental studies on rotating ducts have been less numerous. Wagner et al. [11], Dutta and Han [12], Soong et al. [13] and Azad et al. [14] investigated rotating two-pass square channel with smooth and angled ribs. Azad et al. [14] also investigated the effect of channel orientation on rotating ribbed two-pass rectangular channel with channel aspect ratio of 2:1. Griffith et al. [20] studied the effect of channel orientation on rotating smooth and ribbed rectangular channels with channel aspect ratio of 4:1.

1.3.2 Ribbed Surfaces. Stephens et al. [32, 33] studied inclined ribs by the same second-moment closure of Chen et al. [29, 30]. Interested readers for other second-order Reynolds stress turbulence models are referred to recent papers mentioned in the previous sections.

The present paper will examine first the anisotropic turbulent stresses and heat fluxes resulting from the Coriolis and centrifugal buoyancy forces and the 180° turn for a rotating two-pass square channel with smooth walls (Chen et al. [29, 30]). The secondary flow and Reynolds stresses in stationary two-pass square channels with 90°, 60°, and 45° ribs (Jang et al. [37-39]) will then be presented to evaluate the effects of rib shape and flow-attack-angle on the turbulent flow production and heat transfer enhancements. For the sake of brevity, however, we will present only the numerical results obtained by our research group at Texas A&M University using the near-wall second-moment closure model.

Using the same model and method of Chen et al. [29, 30], Jang et al. [37, 38] studied flow and heat transfer behavior in a non-rotating two-pass square channels with 60° and 90° ribs, respectively. Their results were in good agreement with Eckad and Han’s [9] detailed heat transfer data which validated their code and demonstrated the second-moment closure model superiority in predicting flow and heat transfer characteristics in the ribbed duct. In a later study, Jang et al. [39] predicted flow and heat transfer in a rotating square channel with 45° angled ribs by the same second-moment closure model. Heat transfer coefficient prediction was well matched with Johnson et al. [16] data for both stationary and rotating cases. Al-Qahtani et al. [40] predicted flow and heat transfer in a rotating two-pass rectangular channel with 2:1 channel aspect ratio and 45° angled ribs by the same second-moment closure model of Chen et al. [29, 30]. More recently, Al-Qahtani et al. [41] studied the effect of rotation number, coolant-to-wall density ratio and channel orientation on rotating one-pass smooth and ribbed rectangular channels with channel aspect ratio of 4:1. Their heat transfer coefficient predictions for the 2:1 and 4:1 channels were compared with the data of Azad et al. [14] and Griffith et al. [20], respectively, for both the stationary and rotating cases. It predicted fairly well the complex three-dimensional flow and heat transfer characteristics resulting from the angled ribs, sharp 180° turn, rotation, centrifugal buoyancy forces, channel orientation and aspect ratio.

The aforementioned studies affirmed the superiority of the second-moment closure model compared to simpler isotropic eddy viscosity turbulence models. This model solves each individual Reynolds stress component directly from their respective transport equations. The primary advantage of this model is that it resolves the near-wall flow all the way to the solid wall including the viscous sublayer, buffer layer as well as the fully turbulent flow in the outer region without the wall-function approximation. With this near-wall closure, surface data like heat transfer coefficients and friction coefficients can be evaluated directly from velocity and temperature gradients on the solid wall.

1.4 Objective: Even though the second-order Reynolds stress turbulence models have been used recently for internal cooling applications, these advanced turbulence models were able to provide very detailed three-dimensional velocity, pressure, turbulence, Reynolds stresses, and turbulent heat fluxes that were not previously available in most of the experimental studies. In this paper, we will systematically examine the second-moment predictions for various turbine blade cooling passages to facilitate a detailed investigation of the effects of blade rotation, rib geometry, channel aspect ratio and channel orientation on the turbulent flow production and the associated heat transfer enhancements. For the sake of brevity, however, we will present only the numerical results obtained by our research group at Texas A&M University using the near-wall second-moment closure model of Chen et al. [29, 30]. The present paper will examine first the anisotropic turbulent stresses and heat fluxes resulting from the Coriolis and centrifugal buoyancy forces and the 180° turn for a rotating two-pass square channel with smooth walls (Chen et al. [29, 30]). The secondary flow in rotating rectangular ducts with smooth wall and angled ribs (Jang et al. [37-39]) will then be presented to evaluate the effects of rib shape and flow-attack-angle on the turbulent flow production and heat transfer enhancements. For the sake of brevity, however, we will present only the numerical results obtained by our research group at Texas A&M University using the near-wall second-moment closure model of Chen et al. [29, 30].

The present paper will examine first the anisotropic turbulent stresses and heat fluxes resulting from the Coriolis and centrifugal buoyancy forces and the 180° turn for a rotating two-pass square channel with smooth walls (Chen et al. [29, 30]). The secondary flow and Reynolds stresses in stationary two-pass square channels with 90°, 60°, and 45° ribs (Jang et al. [37-39]) will then be presented to evaluate the effects of rib shape and flow-attack-angle on the turbulent flow production and heat transfer enhancements. The combined effects of the rotation and rib turbulators will also be investigated for a rotating one-pass square channel with 45° ribs of rounded cross section (Jang et al. [39]). Finally, the effects of channel aspect ratio and channel orientation will be evaluated for both the 2:1 and 4:1 rotating rectangular channels with 45° ribs (Al-Qahtani et al. [40,41]) to provide a critical assessment on the overall performance of the present near-wall second-moment closure model.

**NOMENCLATURE**

\( D_s, D \quad \) hydraulic diameter, \( m \)

\( e \quad \) rib height, \( m \)

\( h \quad \) heat transfer coefficient, \( W/m^2K \)

\( k \quad \) thermal conductivity of coolant, \( W/m^2K \)
2. GOVERNING EQUATIONS

All the numerical results presented here were obtained using either the near-wall second-order Reynolds stress closure model of Chen [42, 43] or the two-layer eddy viscosity model of Chen and Paté [44]. Both models were developed originally for incompressible flows in non-rotating coordinates. They have been generalized in Chen et al. [29, 30] to include the rotation and buoyancy terms, and also the energy equation for the heat transfer prediction. For completeness, we shall summarize the generalized near-wall second-moment closure and the two-layer eddy viscosity model in the following:

2.1 Second-moment closure model

Consider the non-dimensional Reynolds-Averaged Navier-Stokes equations in general curvilinear coordinates (ξ, η, i=1,2,3, for unsteady incompressible flow:

\[ \rho \frac{\partial U^i}{\partial t} + \rho U^i U^j + \frac{\partial}{\partial \xi^j} \left( \rho U^i U^j \right) = 0 \]  \hspace{1cm} (1)

\[ \rho \frac{\partial U^i}{\partial t} + \rho U^i U^j + \frac{\partial}{\partial \eta^j} \left( \rho U^i U^j \right) = -\frac{\partial \overline{p}}{\partial \xi^j} - \frac{\partial}{\partial \eta^j} \left( \rho \overline{u^i u^j} \right) \]  \hspace{1cm} (2)

where \( \overline{u^i u^j} \) is the ensemble Reynolds averaging and the \( \rho \) is the density of medium and \( \overline{u^i u^j} \) is the pressure-strain correlation of Speziale, Sarkar and Gatski [46] based on extensive numerical optimizations performed in the present calculations.

The Reynolds stress tensor \( R^i_j = \overline{u^i u^j} \) is the solution of the transport equations

\[ \frac{\partial R^i_j}{\partial t} + U^m R^i_m = P^i + D^i + D^m_{ij} + D^m_{ji} + \Phi^i_j - \epsilon^i_j \]  \hspace{1cm} (5)

where

- Production \( P^i = (\overline{R^i_j U^j} - \overline{R^i_j U^j}) - 2 \epsilon_{ijk} (\overline{R^k_i R^j_m} + \overline{R^k_j R^i_m}) \)
- Diffusion by \( u^m \) \( D^i = -(u^m \overline{u^i u^m}) \)
- Diffusion by \( p' \) \( D^i_j = -g^{mn} (u^m \overline{u^n p'}) \)
- Viscous Diffusion \( D^i_j = V g^{mn} R^m_{ijn} \)
- Pressure-Strain \( \Phi^i_j = (p'/p)(g^{mn} u^n_j + g^{mn} u^n m) \)
- Dissipation \( \epsilon = 2 \nu g^{mn} (u^m u^n) \)

To solve these equations, appropriate closure models must be provided for the pressure-strain, diffusion and dissipation terms. In the present study, the pressure-strain correlation of Speziale, Sarkar and Gatski [46] was combined with the near-wall Reynolds stress closure of Chen [42, 43] for detailed resolution of three-dimensional boundary layer flow all the way up to the solid walls. For the sake of completeness, we will briefly summarize the present near-wall second-moment closure model in the following:

1. Diffusion \( D^i_j = D^i_{m} + D^i_{p} \) (Daly and Harlow [47])

\[ D^i_j = C_i^j (R^m_{ijn}) ; C_i^j = 0.22 \]  \hspace{1cm} (6)

2. Pressure-Strain and Dissipation (Speziale et al. [45]; Chen [42, 43])

\[ \Phi^i_j - \epsilon^i_j = \Phi^i_j + \Phi^i_j + \Phi^i_j - \frac{2}{3} g^i_j \epsilon \]  \hspace{1cm} (7)

where

\[ \Phi^i_j = -C_i (1 - (1 + \overline{C_i}) f_n) u^i \]

\[ \Phi^i_j = (C_i - C_i^j) \overline{u^i} \overline{u^j} + C_i^j (g^{mn} b^m b^n - \frac{2}{3} g^i_j II) \]

\[ \Phi^i_j = f_n (0.45 (P^i - \frac{2}{3} g^i_j P) - 0.03 (Q^i - \frac{2}{3} g^i_j P) + 0.08 k (2S^i)) \]

and

\[ b^i = \frac{R^i_j}{2k} \frac{1}{3} g^i_j ; H = g^{mn} g^m_b b^n \]

\[ S^i = \frac{1}{2} (g^{mn} U^m + g^{mn} U^n) ; V^i = \frac{1}{2} (g^{mn} U^m + g^{mn} U^n) \]

\[ P = \frac{1}{2} g^{mn} p_{mn} ; Q^i = -g_{im} (g^{mn} R^i + g^{mn} R^i) U^n \]

\[ \overline{C_i} = C_i + \overline{C_i} \]

where the model coefficients \( C_i, C_i^j, C_i^j, C_i, C_i, C_i \) are equal to

(3.4, 1.80, 2.40, 0.8, 1.30, 1.25, 0.40). A more detailed description of the present near-wall second-moment closure is given in Chen [42, 43].

In general, the transport equations for turbulent heat fluxes \( u T^m \) may also be derived using second-order closure models such as those shown in Launder [3]. In this study, however, we will use the generalized gradient diffusion hypothesis (GDDH) given in Bo et al. [24];

\[ \overline{u T^m} = -C_0 \frac{k}{\varepsilon} R^m_{T} \]

where \( C_0 = 0.225 \) is the Co value used here is somewhat lower than that proposed by Bo et al. [24] based on extensive numerical optimizations performed in the present calculations.
To complete the Reynolds stress closure, the rate of turbulent kinetic energy dissipation $\varepsilon$ must also be modeled. This study adopted the low Reynolds number model of Shima [48] with minor modifications as follows:

$$\frac{\partial e}{\partial t} + \frac{U^m}{\varepsilon} = \left( \frac{v g^m + C_k}{\varepsilon} - \frac{R^m}{\varepsilon} \right) \frac{e}{\varepsilon} + C_\varepsilon (l + C_{\varepsilon}) \frac{e}{k}$$

$$- C_{\varepsilon} f_b \frac{\varepsilon e}{k} + \frac{\varepsilon}{\sigma}$$

where the model coefficients are $(C_{\varepsilon}, C_{\varepsilon}'_1, C_{\varepsilon}'_2, C_{\varepsilon}'_4) = (0.15, 1.35, 1.8, 1.0)$. The near-wall damping function $f_b, f_{\varepsilon}$ and the source terms $\xi$, $\varepsilon$, $\nu_*$ are given in Chen [42, 43].

2.2 Two-Layer $k-\varepsilon$ model

In order to detail a detailed assessment of the present near-wall second-order Reynolds stress closure for turbulent flow and heat transfer predictions, calculations were also performed using the two-layer isotropic eddy viscosity model of Chen and Patel [44]. In this two-layer approach, the Reynolds stresses are related to the mean rate of strain by:

$$- \rho \varepsilon = 2 \mu_s S^{\prime \prime} - \frac{2}{3} \frac{\varepsilon}{\sigma}$$

where $\mu_s$ is the eddy viscosity and $k = g_{\varepsilon} \mu_s n'/2$ is the turbulent kinetic energy. $S^{\prime \prime}$ is the contravariant components of the rate of strain tensor given in Equation (12). Similarly, the turbulent heat fluxes can be related to the mean temperature gradient as follows:

$$- \rho \varepsilon T^{\prime} = \frac{\mu_s}{Pr} g^{m} T^{\prime} ; \quad Pr_r = 0.9$$

where $Pr_r$ is the turbulent Prandtl number. Substitution into (2) and (3) yields momentum and energy equations for eddy viscosity turbulence modeling:

$$\rho \left( \frac{\partial U^m}{\partial t} + U^m U^m \right) + \rho g^m \Omega^m U^m + \rho g^m \Omega^m \varepsilon - \Omega^m \varepsilon = -g^m (p + \frac{2}{3} \rho k^m) + 2 \mu_s S^{\prime} m^m (\mu + \mu_s) g^m U^m$$

$$\rho \left( \frac{\partial T^m}{\partial t} + U^m T^m \right) = g^m \left( \mu + \mu_s \right) T^m$$

where $Pr$ is the Prandtl number. Equations (19) and (20) are closed using the two-layer turbulence model of Chen and Patel [44]. The approach utilizes a two-equation $k-\varepsilon$ model for most of the flow field, but a one-equation $k-\ell$ model in the viscous sublayer and buffer zone. The prescribed length scale $l$ circumvents numerical problems often encountered with near-wall dissipation calculations and reproduces the universal law-of-the-wall profiles in the laminar sublayer, buffer layer and logarithmic region.

In the fully turbulent region, the conservation equations for turbulent kinetic energy and its dissipation rate can be written:

$$\rho \left( \frac{\partial k}{\partial t} + U^m k_m \right) = g^m \left[ (\mu + \mu_s) k_m \right] + P + P_b - \rho \varepsilon$$

$$\rho \left( \frac{\partial \varepsilon}{\partial t} + U^m \varepsilon_m \right) = g^m \left[ (\mu + \mu_s) \varepsilon_m \right] + \frac{\varepsilon}{\sigma} (C_\varepsilon P + C_\varepsilon P_b - C_\varepsilon \rho e)$$

where

$$P = 2 g_{\varepsilon} \mu_s S^{\prime} m^m P_b = \frac{\mu_s}{Pr} T g^m \left( \Omega^m \varepsilon - \Omega^m \varepsilon \right)$$

The buoyancy generated turbulence production $P_b$ was proposed by Snider and Andrews [49] and the model coefficients $(C_{\varepsilon}, C_{\varepsilon}'_1, C_{\varepsilon}'_2, C_{\varepsilon}'_4)$ are fixed constants equal to (0.09, 1.44, 1.92, 0.9, 1.0, 1.3).

3. CHIMERA RANS METHOD

In the present study, the chimera RANS method of Chen [43] and Chen and Chen [50] has been further extended to include the effects of rotation and buoyancy. The present method solves the mean flow and turbulence quantities in arbitrary combination of embedded, overlapped, or matched grids using a chimera domain decomposition approach. In this approach, the solution domain is first decomposed into a number of smaller blocks that facilitate efficient adaptation of different block geometries, flow solvers and boundary conditions for calculations involving complex configurations and flow conditions.

Within each computational block, the finite-analytic numerical method of Chen et al. [45] was employed to solve the unsteady RANS equations on a general curvilinear, body-fitted coordinate system. The coupling between the pressure and velocity is accomplished using a hybrid PISO/SIMPLER algorithm given in Chen and Patel [51]. The method satisfies continuity of mass by requiring the contravariant velocities to have a vanishing divergence at each time step. Pressure is solved using the concept of pseudo-velocities, and when combined with the finite-analytic discretization gives the Poisson equation for pressure. The overall solution procedure consists of an outer loop over time and an inner loop that iterates over the blocks of the grid. The discretization equations for pressure, velocity, and turbulence quantities form a system of tridiagonal matrices that was solved using an iterative ADI scheme. To ensure the proper conservation of mass and momentum between linking grid blocks, the grid-interface conservation techniques of Hubbard and Chen [52] were used to eliminate unphysical mass source resulting from the interpolation errors between the chimera grid blocks. More detailed descriptions of the chimera RANS method were given in Hubbard and Chen [52] and Chen and Chen [50].

4. RESULTS AND DISCUSSION

The chimera RANS method was employed recently by the authors’ research group at Texas A&M University to study the fluid flow and heat transfer in non-rotating and rotating channels of square and rectangular cross-sections under various flow conditions. Very encouraging results were obtained for a wide range of cooling channel configurations under various combinations of rotation number and coolant-to-wall density ratio. In the present paper, we shall summarize the most important results obtained from these numerical investigations to facilitate a detailed understanding of the turbulent flow characteristics induced by the channel bend curvature, turbine blade rotation, coolant-to-wall density ratio, channel aspect ratio and channel orientation.

4.1 Smooth Channel

Chen et al. [29, 30] investigated the effects of the channel rotation and the 180° bend on the flow turbulence and the associated heat transfer for the multi-pass square channel with smooth walls as tested by Wagner et al. [11]. Figure 2 shows the geometry and an enlarged view of the numerical grids around the 180° bend. The length of both the first pass and second pass are 14 $D_p$. The inner radius of curvature of the bend is 1.25 $D_p$ and the radius from axis of rotation is 42 $D_p$. All walls
are heated to a constant temperature. In their study, the Reynolds number was fixed at 25,000 which is the typical operating conditions of medium size gas turbines. Comparisons between the calculations and measurements were made for three different rotation numbers of 0, 0.18, and 0.24, and four coolant-to-wall density ratios of 0, 0.07, 0.13 and 0.22.

Chen et al. [29, 30] presented two-dimensional plots of the mean velocity, temperature, Reynolds stresses and turbulent heat flux contours for the non-rotating and rotating cases at several axial stations as defined in Figure 2. In order to facilitate a more detailed understanding of the general flow and heat transfer characteristics induced by the rotation and the 180° bend, we shall present the three-dimensional developments of the secondary flow, streamwise velocity, and temperature fields along the cooling channel as shown in Figures 3 thru 6. For the non-rotating case shown in Figure 3(a), the centrifugal forces and the associated pressure gradients (low pressure at inner surface, high pressure at outer surface) in the bend produced two symmetric counter-rotating vortices which convected fluid from the core toward the outer surface. This secondary flow decreased after the 180° turn and vanished almost completely at the end of the second passage.

Figure 3(b) shows the cross-stream velocity vectors for the rotating cases with rotation number Ro = 0.24 and density ratio $\Delta\rho/\rho = 0.22$. In the first passage, the Coriolis forces produce a secondary flow which pushes the cold fluid from the core towards the trailing surface and then returns along the side walls (i.e., inner and outer surfaces) where the fluid is heated. In the bend, the secondary flow structure formed in the first passage is completely destroyed. The rotation-induced radially outward flow, as it enters the bend section of the duct, is accelerated asymmetrically in the cross section. The heavier cold fluid near the trailing surface is first accelerated and then followed by the lighter fluid near the leading surface in the duct cross section. This causes the fluid near the trailing surface to be thrown towards the outer side wall, resulting in the clockwise (viewing from upstream) circulation in the middle of the bend region. In the second passage, the Coriolis force acts in the opposite direction, compared to the one in the first passage, which pushes the cold fluid towards the leading surface. This has led to the formation of two large vortices downstream of the bend with the larger one near the leading surface and the smaller one near the trailing surface. This secondary flow structure is produced by the interaction of the circulation generated in the bend and the Coriolis force due to the duct rotation. Farther downstream, the secondary flows are due primarily to the Coriolis force while the effect of bend diminishes gradually in the second passage.

The secondary flow caused by channel rotation also distorted the axial velocity profiles as shown in Figures 4 and 5. For the non-rotating case, Figure 4(a) shows that the axial velocity profiles shift towards outer surface in the bend, but return quickly to a fairly flat profile in the second passage. A detailed examination of the solutions reveals no axial flow reversal in this stationary duct. For the rotating case shown in Figure 4(b), the Coriolis and centrifugal buoyancy forces produced a region of axial flow reversal at the end of the first passage near the inner surface. The Coriolis forces push the cold fluids towards the trailing surface so that the centrifugal buoyancy force tends to slow down the lighter fluid, producing thicker boundary layer near the leading surface and accelerates the heavier fluid near the trailing surface. Thus, it causes flow reversal in the streamwise direction on the leading surface as shown in Figures 5(b) and 5(c). The size of the reverse flow region depends on the magnitude of the buoyant force but, so far, no measurement or computation provides the magnitude or extent of the reverse flow.

Figure 4. Velocity vectors midway between leading and trailing surfaces (Chen et al. [30]).
Figure 6(a) shows the isothermal contours for the non-rotating duct. Before the bend, the cooler fluids are located in the core region. After the bend, however, the cooler fluid is pushed toward the outer surface by the centrifugal force induced by the streamline curvatures. This leads to steep temperature gradients and hence high heat transfer coefficients on the outer wall after the bend. For the rotating cases shown in Figures 6(b) and 6(c), the Coriolis forces push the cold fluids toward the trailing surface so that the centrifugal buoyancy force tends to slow down the lighter fluid, producing a thicker boundary layer near the leading surface and accelerates the heavier fluid near the trailing surface. Thus, it causes flow reversal in the streamwise direction on the leading surface as shown earlier in Figure 5. In general, the reverse flow region in the first passage increases with increasing coolant-to-wall density ratio and buoyancy. On the other hand, the Coriolis force in the second passage acts in the opposite direction and pushes the cold fluids toward the leading surface. Thus, the centrifugal buoyancy forces accelerate the lighter fluid near the trailing surface and, consequently, flatten the axial velocity profile.

In addition to the mean velocity and temperature fields, the present second moment closure model also provide detailed Reynolds stresses and heat fluxes which were not available in most of the measurements. In order to facilitate a through evaluation of the rotation and bend effects on smooth wall heat transfer, we shall present the secondary flow vectors, streamwise velocity contours, and dimensionless temperatures (first column), normal Reynolds stresses (second column), turbulent shear stresses (third column), and turbulent heat fluxes (fourth column) at three selected planes; $Z/D_0 = 11.28$ in the first passage (location $B^*$ in Figure 2), midsection of bend (location $D$), and $Z/D_0 = 11.94$ (location $E^p$) in the second passage for both the non-rotating and rotating case. In the first passage, the second-moment results in Figure 7(a) clearly indicate the presence of four pairs of counter-rotating vortices that were absent in the two-layer $k$-$\varepsilon$ solutions (not shown). It is well known that these corner vortices were produced as a result of Reynolds stress anisotropy in the straight duct. At this station, the levels of turbulence intensity ($\sqrt{\langle \overline{v'v'} \rangle}$) has been observed close to channel walls. Figure 7(b) shows a dramatic change in the pattern of Reynolds stresses in the midsection of bend. This is clearly caused by the pressure-driven cross-stream flow with the presence of two strong counter-rotating vortices. Relatively high levels of turbulence intensity (10% to 18%) arise near the outer surface and side wall surfaces, whereas lower values were observed along the inner surface. The shear stresses $\overline{vw}$ near the outer surface are more than 20 times higher than that near the inner surface. Stabilizing curvature, occurring on convex walls (inner surface of the bend), has the effect of lowering Reynolds shear stresses and turbulence energy levels. Destabilizing curvature appears on concave walls (outer surface of the bend) and results in high levels of turbulent shear stresses and turbulent kinetic energy. It is also noted that the turbulent heat fluxes ($\overline{w'\overline{w}T}$) are negative near the outer surface and positive near the inner surface. This means that the heat transfer is enhanced near the outer surface, whereas it is reduced near the inner surface. In the second passage, as shown in Figure 7(c), the flow pattern is modified further with $\overline{f'w'/W}$ returning to lower values near the outer surface but increasing near the inner surface. This suggests that the high turbulence produced near the outer surface of the bend have been convected toward the side walls and inner surface by the pressure-driven cross-stream flow. The turbulent heat fluxes ($\overline{w'\overline{w}T}$) near the outer surface is about two times higher than that of the inner surface.

The secondary flow pattern shown in Figure 8(a) for the rotating case is completely different from the stationary case shown earlier. The Coriolis forces and the attendant pressure gradients produce the two counter-rotating vortices adjacent to the trailing surface. The normal-stress driven corner vortices observed in the non-rotating duct are almost completely powered by the pressure-driven vortices, except in the corners of the leading surface. Near the trailing surface and middle section of the duct, the turbulence intensities are very high (14% to 20%)
Figure 7. Mean flow, Reynolds stresses and turbulent heat fluxes for non-rotating smooth duct (Chen et al. [30]).

Figure 8. Mean flow, Reynolds stresses and turbulent heat fluxes for rotating smooth duct (Chen et al. [30]).
and 14% to 30%, respectively), as a result of the high shear on the trailing surface and the flow reversal near the duct center. However, a higher degree of anisotropy (2.4 ≤ \( \frac{\tilde{u}'\tilde{w}'}{\tilde{u}'\tilde{u}'} \) or \( \frac{\tilde{w}'\tilde{w}'}{\tilde{w}'\tilde{w}'} \leq 4.3 \)) was observed on the leading surface in comparison with that seen on the trailing surface. This indicates that the flow at the leading surface is more anisotropic than that at the trailing surface. At this station, the turbulent heat flux \( \frac{\tilde{u}'\tilde{w}'}{\tilde{u}'\tilde{w}'} \) also shows the similar tendency: it is 7.5 times higher on the trailing surface than that of leading surface. In the bend, the level of turbulence intensity shown in Figure 8(b) increases to 17% ~ 26% on all four side walls due to the combined effects of pressure-driven flow in the bend and the secondary flow induced by the Coriolis and centrifugal buoyancy forces. In the second passage, however, the turbulence intensities shown in Figure 8(c) reduced to 8% ~ 15% and the degree of anisotropy is also lower (1.2 ≤ \( \frac{\tilde{u}'\tilde{w}'}{\tilde{u}'\tilde{u}'} \) or \( \frac{\tilde{w}'\tilde{w}'}{\tilde{w}'\tilde{w}'} \leq 1.8 \)). It is interesting to note that the turbulence intensities in the second passage are actually lower than those in the first passage, even though the turbulent shear stresses are higher in the second passage. The turbulent heat fluxes are also lower in comparison with those observed in the first passage and in the bend.

With the present near-wall second-moment closure, surface data like heat transfer coefficients and friction factors can be evaluated directly from velocity and temperature gradient on the solid wall as shown in Chen et al. [29, 30]. They found that the predicted heat transfer coefficients are in close agreement with the experimental data of Wagner et al. [11] on both the leading and trailing surfaces for several different rotating numbers and coolant-to-wall density ratios. The results clearly demonstrated the superiority of the near-wall second-moment closure model over simpler isotropic eddy viscosity models in the prediction of heat transfer characteristics resulting from the rotation and 180° turn effects.

4.2 Stationary Ribbed Channel

As noted earlier, advanced gas turbines often use rib turbulators on two opposite walls of internal coolant passages to augment heat transfer. In this section, we will present the second-moment results for two-pass square channels with 90° and 60° angled ribbed walls of Jang et al. [37, 38] as tested by Ekkad and Han [9] to examine the effects of the normal and angled ribs on the mean velocity, temperature, Reynolds stresses, and turbulent heat fluxes. Figure 9 shows the geometries for the one side ribbed channels. A total of nine ribs were simulated. For the 60° inclined ribs, four ribs in the first passage were angled away from the divider wall and four ribs in the second passage were angled toward the divider wall. There was a 90° rib in the turn region for both channel configurations.

The length of the 60° ribbed duct was 8.6105D₀. The length from the inlet to the first rib (L₁) was 3.5D₀ and the length from the last rib in the first passage to the outer surface in the bend (L₂) was 0.6395D₀. The length from the divider wall to the outer surface in the bend (L₃) was 1D₀. The inner wall thickness (d) was 0.25D₀. The rib height-to-hydraulic diameter ratio \( \frac{h}{D₀} \) was 0.125 and the rib pitch-to-height ratio \( \frac{P}{h} \) was 10. The physical dimensions of the 90° ribbed channel is almost identical to the 60° case except that \( L₁ = 8.375D₀ \) and \( L₂ = D₀ \). A fully developed turbulent boundary layer profile was used at the inlet of the duct in the present calculations. All walls including the rib surfaces were heated to a constant temperature. The coolant fluid at the entrance of the duct was air, at a uniform temperature, \( T_e = (T - T_w)(T_e - T_w) = 0 \) and the wall temperature, including the ribs, was kept constant at \( T = T_w \) (\( \theta = 1 \)). The total grid point used are approximately 1,020,000 and 1,060,000 points for the 60° and 90° cases, respectively.

Figures 10(b) and 10(c) show the velocity vector distributions in the planes midway between the top and bottom surfaces for the 90° and 60° ribbed channels, respectively. Comparisons were also made with the smooth duct results of Jang et al. [38] as shown in Figure 10(a) to quantify the effects of angled ribs on the flow field. For the smooth channel case, a large separation bubble existed near the tip of the divider wall due to the inability of the flow to follow the sharp turn. The predicted reattachment length was about 1.8D₀ away from the divider wall tip which is consistent with the measurement of Liou et al. [10]. For the normal ribbed channel, however, no separation bubble was present near the divider wall tip in the bend region. Even immediately downstream of the bend, there was no separation bubble. This can be attributed to the presence of ribs in three locations; immediately upstream of the bend, in the middle of the bend and immediately downstream of the bend. Those ribs reduced the centrifugal effect in the bend. Similar to the normal rib case, the separation bubble was also absent in Figure 10(c) for the 60° angled ribbed duct case. This phenomenon was due to two factors. First, it is attributed to the effect of the rib-induced secondary flow upstream of the bend, which pushed the fluid in the core toward the divider wall and thus, weakened the development of the curvature-induced radial velocity component in the bend. Second, the centrifugal effect was reduced due to the presence of the ribs in the bend.
Figure 11(a) shows the streamwise velocity profiles for the 90° normal ribbed duct case at both the first and second passages in the planes midway between the inner and outer surfaces. The reversal flow occurred immediately downstream and upstream of the ribs. In the first passage, the reattachment length was about 3.4 times the rib height, which was in good agreement with the Ekkad and Han [9] data. After the bend, a strong flow impingement occurred on the bottom surface between ribs 6 and 7 due to the bend effect. For the 60° angled ribbed case shown in Figure 11(b), the reversal flow also occurred immediately downstream of the ribs. In the first passage, the reattachment length was about 3.1 times the rib height, which was also consistent with the Ekkad and Han [9] data. Immediately upstream of the ribs, however, there was no separation, such as that found in the 90° normal ribbed channel case. After the sharp 180° bend, a strong flow impingement was also observed on the bottom surface between ribs 6 and 7. For both the 90° and 60° ribbed channels, the temperature fields on the bottom surface were disturbed by the presence of the ribs. The periodic ribs produced local wall turbulence due to the flow separation and reattachment between the ribs.

Figure 11. Axial velocities midway between the inner and outer walls (Jang et al. [37, 38]).

Figure 12 shows the cross-stream velocity vectors for 90° normal ribbed duct at selected planes. It is noted that the 90° parallel ribbed channel produced periodically up-and-down flow movement and simultaneously generated the small vortex near the bottom surface on the both inner and outer surfaces, which was measured by Liou et al. [53] using LDV. On the other hand, secondary flow occurred near the top surface was not generated by ribs but by non-isotropic turbulence. The secondary flow pattern in section C was completely different with section A and B due to the bend effect. In the bend, a strong vortex was generated near the top of the rib 5 and a smaller one occurred near the top and outer surfaces due to the combined effect of centrifugal-induced vortex in the bend and rib-induced vortex in the upstream. The secondary flow pattern in the second passage is more complicated due to the combined effect of the bend and ribs. In section E, three vortices were generated and velocity magnitude was very high compared to the one in the first passage. It could be seen that the bend effect persisted farther downstream of the rib 7. The secondary flow structure generated in the first passage transported the cooler fluid from the core toward the top surface. In the bend, the cooler fluid is pushed toward the outer surface due to the combined effect of centrifugal-induced vortices in the bend and ribs.

Figure 12. Secondary flow in 90° ribbed channel (Jang et al. [37]).

Figure 13 shows the cross-stream velocity vectors for the 60° angled ribbed channel at selected planes. Sections I and J in Figure 13 show that the 60° angled one-side ribbed channel produced one large vortex in the first passage. This figure also shows that the angled ribs induced the fast flow near the bottom surface between the ribs, which impinged on the outer surface and then returned along the top surface. Instead of the two counter-rotating vortices in the bend as was observed in the smooth channel, one strong vortex was generated near the top of rib 5 due to the combined effect of centrifugal induced vortex in the bend and rib-induced vortex upstream. The secondary flow in the second passage was more complicated due to the bend effect. In section L, between ribs 6 and 7, three vortices were generated. It can be seen that the velocity magnitude was very high near all four surfaces. Farther downstream (section M), rib-induced secondary flow tried to overcome the bend effect. Thus, one large vortex started to appear and simultaneously the other vortices began to disappear. The secondary flow structure generated in the first passage transported the cooler fluid from the core toward the top and inner surfaces. In the bend the cooler fluid was pushed toward the outer surface due to the secondary flow characteristic in this region. In the second passage, the secondary flows pushed the cooler fluid toward the outer surface, which was opposite to that seen in the first passage.

Figure 13. Secondary flow in 60° ribbed channel.

In addition to the streamwise and secondary flow vectors, it is desirable to examine the turbulent flow field induced by the 90° and 60° ribbed ducts. Figure 14 shows the Reynolds stress components for the 90° normal ribs at selected cross sections. In the first passage, high turbulence intensities (15% - 25%) occurred on the top of the ribs and in the regions of reattachment between the ribs. The degree of anisotropy was about 1 ≤ wvw/ww or wvw/vv ≤ 2. In the bend, turbulence level was still high on the top of rib: where the turbulence intensity was about 27%
(~ 41%). The secondary flow in the bend transported the high Reynolds stresses toward the inner and top surfaces. The degree of the anisotropy was as high as 2.3. In the second passage, the peak turbulence intensity (~35%) occurred around the bottom and outer surfaces in section E (between ribs 6 and 7). The reason for this is high shear layer in that location, which was caused by the bend effect. Farther downstream, the general turbulence level was down to about 10% ~ 30%. The degree of anisotropy (ww/\(u'v'\) or \(\nu\nu/\nu\nu\)) in the second passage was about 1.5 ~ 2.

Figure 14. Reynolds stresses in 90° ribbed channel.

Figure 15 shows the Reynolds stress components for the 60° angled ribbed channel at several selected cross sections. In the first passage, high turbulence intensities (15% - 26%) occurred on the top of the ribs and in the regions of reattachment between the ribs. However, unlike the symmetric flow patterns produced by the 90° ribs, the secondary flow induced by the inclined 60° ribs transported the Reynolds stresses asymmetrically toward the outer surface and then to the top surface. Thus, the turbulence level was relatively high near the bottom and outer surfaces and low near the top and inner surfaces. In the bend, turbulence level was still high on the top of rib, where the turbulence intensity was about 30% ~ 41%. The secondary flow in the bend transported the high Reynolds stresses toward the inner and top surfaces. In the second passage, the peak turbulence intensity (~42%) for \(\sqrt{\nu\nu/\nu\nu}\) occurred around the core region in section L (between ribs 6 and 7). The reason for this is high shear layer in that location, which was caused by the bend effect. For the \(\sqrt{\nu\nu/\nu\nu}\) case, high turbulence intensity (40%) occurred near the bottom surface due to the rib-induced secondary flow between the ribs. Farther downstream, the general turbulence level was down to about 10% ~ 30%. The degree of the anisotropy (\(\nu\nu/\nu\nu\) or \(\nu\nu/\nu\nu\)) in the second passage was about 1 ~ 2.

Figure 15. Reynolds stresses in 90° ribbed channel.

Figures 16 and 17 show the detailed Nusselt number ratio distributions in the two-pass channel with 90° and 60° parallel ribs, respectively. For the 90° case, the highest Nusselt number ratios were obtained on the top of ribs in both passages of the channel. Heat transfer distributions between adjacent ribs appeared periodic in the first passage. Nusselt number ratio was high in the middle region between two ribs, and very low immediately before and after the ribs. In the turn region, heat transfer was enhanced greatly due to the combination of the sharp 180° turn and the 90° ribs. The presence of the ribs appeared to reduce the effect of centrifugal forces on the secondary flow and caused lesser impingement on the outer surfaces. Locally high heat transfer region was obtained immediately downstream of the ribs in the turn and second passage. Nusselt number ratios decreased with the reduction in the effect of the turn. Two-layer calculations predicted well the heat transfer patterns in the entire channel. However, it underpredicted the level of Nusselt number ratio in the entire channel. On the other hand, the second-moment calculation results are in close agreement with the Ekkad and Han [9] data.
For the 60° parallel ribs, the highest Nusselt number ratios were also obtained on the top of ribs in both passages. In the first passage, the heat transfer distribution between the ribs was nearly periodic in Ekkad and Han [9]. However, the calculation results did not show the periodic Nusselt number distributions in the first passage. The reason was that the flow in the passage with 60° parallel ribs arrangement required at least eight ribs to achieve a fully developed flow conditions because the angled rib-induced secondary flow was developed and became stronger along the channel, as indicated by Han and Park [7]. For this reason, the heat transfer results, even for the second-moment calculations showed slightly lower values and the reattachment point between the ribs was a little farther away from the ribs compared to Ekkad and Han [9] data. In the second passage, the Nusselt number ratios were higher near the outer surface and decreased toward the inner surface.

4.3 Rotating Ribbed Channels
After examining the effects of the channel rotation and rib turbulators separately in previous sections, it is desirable to determine the combined heat transfer enhancement due to both the rotation and ribs in a rotating ribbed channel. In this section, we shall present second-moment results obtained recently by Jang et al. [39] for the first pass of a rotating square channel with 45° angled ribbed wall (Figure 18) as tested by Johnson et al. [16]. The leading surface is roughened with thirteen equally spaced ribs of rounded cross section, and the trailing surface with twelve equally spaced ribs of rounded cross section. The ribs on these two walls were staggered relative to each other, with ribs on the leading surface offset upstream from those on the trailing surface by a half pitch (P). All ribs were inclined at an angle (α) of 45° with respect to the flow. The rib height-to-hydraulic diameter ratio (e/Dh) was 0.1 and the rib pitch-to-height ratio (P/e) was 10.

Figure 18. Geometry of 45° ribbed channel (Jang et al. [39]).

Computations were performed for three rotation numbers of 0.0, 0.12, and 0.24, at a Reynolds number of 25,000 and, an inlet coolant-to-wall density ratio of 0.13. Figure 19 shows the three-dimensional particle traces for non-rotating case around the ninth, tenth, and eleventh rib near the leading surface. Unlike the prediction of Stephens et al. [33] that there was no separation around the ribs, the present prediction shows that the rib-induced separations (vortices) were generated downstream of each rib and next to the inner surface (i.e., rib leading corner), then pushed away diagonally between the angled ribs and then dissipated by the spiral motion of the streamwise flow (not shown). Figure 20(a) shows partial views of the streamwise velocity vector distribution for non-rotating case at three locations: in the middle of the rib, near the inner surface (0.018Dh from the inner surface), and near the outer surface (0.018Dh from the outer surface). This figure helps show the three-dimensional flow characteristics of this configuration. Flow separation was present near the inner surface but not at the middle of the ribs or near the outer surface. The rounded rib shape tends to reduce the extent of flow separation as compared to the sharp rectangular ribs shown earlier in Figure 11. Figure 20(b) shows the streamwise velocity vector distribution for the entire section of the plane between the leading and trailing surfaces in the middle of the ribs at a rotation number of 0.24 and inlet density ratio of 0.13. Unlike the non-rotating case, flow separation was present on the leading surface due to the centrifugal
buoyancy force, which tended to slow down the lighter fluid (lower density fluid) near the leading surface. It is also observed that high momentum cooler fluid was pushed toward the trailing surface by the Coriolis force.

where the flow tends to separate. Both the leading and trailing surfaces have a similar heat transfer pattern. In the regions between the ribs, the heat transfer was high next to the inner surface because the flow reattachment resulted in a thinner thermal boundary layer. Due to the rib-induced secondary flow characteristics of this configuration, the heat transfer decreased diagonally downstream of the rib leading edge to upstream of the next rib trailing edge. Another high heat transfer spot is seen on the outer surface next to both the leading and trailing surfaces and is caused by the impingement of the secondary flow. It is quite obvious that the rounded rib shape produced a smoother and more uniform heat transfer pattern in comparison with those shown earlier in Figures 16 and 17 for sharp rectangular ribs.

The Nusselt number distribution for the rotating case is drastically different from the stationary case. On the leading surface, the high heat transfer next to the inner surface can no longer be seen because the Coriolis force pushes the core fluid toward the trailing surface, and the Coriolis-induced vortex prevents flow reattachment on the leading surface, which usually leads to a high heat transfer coefficient. The high heat transfer location (Figure 20(c)) on the outer surface and next to the trailing surface was caused by the secondary flow impingement, which was higher than the non-rotating case. The surface heat transfer on the trailing surface (Figure 20(d)) looks similar to the non-rotating case. However, the heat transfer level was higher than the non-rotating case due to the stronger vortex induced by the Coriolis force. The heat transfer patterns on both the inner and outer surfaces were reversed as opposed to the non-rotating case. In other words, the heat transfer on the inner surface was less than that of the outer surface.

The Nusselt number ratio contour plots on both the leading and trailing surfaces for the non-rotating case. The entrance and exit regions were cut off so that the rib regions could be seen more clearly. The Nusselt numbers between the ribs increased along the duct until the flow approached the ninth rib and then decreased after the eleventh rib. This was because the heat transfer in this region was strongly affected by the secondary flows, which became stronger as the flow moved downstream. This feature was a unique characteristic for the
angled rib arrangement, which was not found in the normal rib case. Another feature was that in every section between the ribs, the Nusselt number ratio decreased for a higher number ratio. The inner first and then decreased to the outer surface, which was consistent with the experimental data (Ekkad and Han [9]). This was because of the flow reattachment next to the inner surface. The flow started to redevelop in this location and caused the thermal boundary layer to become thinner. Figures 22(b) and 22(d) show the Nusselt number ratio contours on both the leading and trailing surfaces at a rotation number of 0.12 and inlet density ratio of 0.13. On the leading surface (Figure 22(b)), the Nusselt number ratios decreased from the first rib to the last rib due to the Coriolis force, which pushed the cooler fluid from the leading surface toward the trailing surface. The Nusselt number ratios on the trailing surface (Figure 22(d)) increased from the first rib and reached a maximum between the seventh and ninth ribs and decreased after that. Figures 22(c) and 22(e) show the Nusselt number ratio contours on the leading and trailing surfaces at a rotation number of 0.24 and inlet density ratio of 0.13. In general, the higher rotation number induced stronger Coriolis and centrifugal buoyancy forces. As expected, the Coriolis forces further pushed the cooler fluid toward the trailing surface, where the heat transfer became higher as seen compared with the low rotation number (Ro = 0.12). On the leading surface, the heat transfer further decreased due to the presence of hotter fluid.

Figure 22. Detailed Nusselt number ratio distribution (Jang et al. [39]).

A detailed comparison of the calculated and measured spanwise-averaged and regional-averaged heat transfer coefficients were presented in Jang et al. [39] for both the non-rotating and rotating cases. They found that the overall predicted Nusselt number behavior was relatively close to the data of Johnson et al. [16] except on the leading surface of rotating channel case.

4.4 Rectangular Channels

Most of the previous studies investigated coolant channels that have square cross sections and are perpendicular to the axis of rotation. However, the orientation of the cooling channel in the leading and trailing edge regions of the turbine blade may be at an angle β from the direction of rotation and its cross section may not be square. It is not well known how this affects the flow field and heat transfer characteristics. Al-Qahtani et al. [40, 41] recently computed the flow and heat transfer for two rotating rectangular ducts (aspect ratios AR = 2 and 4) with 45° angled ribs using the present near-wall second-moment closure model. In this section, we will present first the second-moment results for a two-pass rectangular channel (AR = 2) as tested by Azad et al. [14]. The geometry around the sharp 180° turn is shown in Figure 23. The channel hydraulic diameter, $D_h$, is 1.69 cm and the radius of curvature of the 180° sharp turn is $r/D_h = 0.375$. In the ribbed section, the leading and trailing surfaces for both the first and second passages are roughened with nine equally spaced ribs of square cross section. The rib height-to-hydraulic diameter ratio ($h/D_h$) is 0.094 and the rib-pitch-to-height ratio ($P/e$) is 10. All ribs are inclined at an angle $\alpha = 45^\circ$ with respect to the flow. Two channel orientations are studied: $\beta = 90^\circ$ corresponding to the mid-portion of a turbine blade and $\beta = 135^\circ$ corresponding to the serpentine passages in the trailing edge region of a blade. In this study, the Reynolds number was 10,000 and the inlet coolant-to-wall density ratio was 0.115. Three cases were studied: (1) non-rotating channel (Ro = 0.0), (2) rotating channel (Ro = 0.11) with channel orientation angle $\beta = 90^\circ$ and (3) rotating channel (Ro = 0.11) with channel orientation angle $\beta = 135^\circ$. A uniform velocity profile was used at the inlet of the duct. The unheated length ($L_u$) was long enough for the velocity profile to be fully developed turbulent profile before the heating start-point. A 336x1x804 numerical grid (804 in the streamwise direction) was used with a total approximately 1,100,000 grid points.

Figure 23. Geometry for two-pass rectangular channel (AR = 2) with 45° ribs (Al-Qahtani et al. [40]).

Before discussing the detailed computed velocity field, a general conceptual view about the secondary flow patterns induced by angled ribs and rotation is summarized and sketched in Figure 24. The parallel angled ribs in the non-rotating duct (case 1, Figure 24(a)) produce symmetric counter rotating vortices that impinge on the inner surface in the first passage and on the outer surface in the second passage. The Coriolis force in the 90° rotating duct (case 2, Figure 24(b)) produces two additional counter-rotating vortices that push the cooler fluid from the core to the trailing surface in the first passage, and to the leading surface in the second passage. For the 135° rotating duct (case 3, Figure 24(c)), the Coriolis force produces secondary flow that migrates diagonally away from the corner of the inner-leading surfaces toward the center of the channel in the first passage, and from the corner of the inner-trailing surfaces towards the center of the channel in the second passage.

Figure 25 is a plot of the secondary flow for non-rotating duct in the first passage. The parallel angled ribs in the non-rotating duct (case 1, Figure 25(a)) produce symmetric counter rotating vortices that impinge on the inner surface in the first passage and on the outer surface in the second passage. The Coriolis force in the 90° rotating duct (case 2, Figure 25(b)) produces two additional counter-rotating vortices that push the cooler fluid from the core to the trailing surface in the first passage, and to the leading surface in the second passage. For the 135° rotating duct (case 3, Figure 25(c)), the Coriolis force produces secondary flow that migrates diagonally away from the corner of the inner-leading surfaces toward the center of the channel in the first passage, and from the corner of the inner-trailing surfaces towards the center of the channel in the second passage. The ribs induce secondary flow from the outer surface (where it is the strongest) to the inner surface (where it is the weakest). Note that this secondary flow pattern is the same in all inter-rib areas. The consequence of this fast secondary flow is explained in Figure 24.
Figure 24. Conceptual view of the secondary flow induced by angled ribs and rotation (Al-Qahtani [40]).

Figure 25. Velocity vectors and temperature contours at 1/10 rib height from the leading and trailing surfaces for non-rotating case.

Figure 26 is a plot of the cross-stream velocity vectors and temperature contours for the Ro = 0.11 and β = 135° case. As noted earlier, the Coriolis force produces secondary flow that migrates diagonally away from the corner of the inner-leading surfaces toward the center of the channel. As the flow approaches the first rib, this Coriolis force induced secondary flow distorts the secondary flow started by the inclined ribs. However, from rib 1 on, this rotation induced secondary flow is dominated by the rib induced secondary flow. A careful comparison between the secondary flow and temperature fields with the β = 90° given in Al-Qahtani et al. [40] indicated that the rotation induced secondary flow changes the rib induced secondary flow for β = 135° case. This change appears more clearly in the temperature field. In the first passage, the cooler fluid is pushed back toward the leading surface, reducing the steep temperature gradients on the trailing surface. The temperature contours do not change much in the bend. In the beginning of the second passage, the cooler fluid is pushed back slightly toward the leading surface, while the temperature field in the rest of the second passage is the same as in the β = 90° case.

Figures 27(a) and 27(b) show the local Nusselt number ratio contours on the leading and trailing surfaces (identical due to symmetry) for the non-rotating case. The highest Nusselt number ratios were obtained on the top of the ribs, and the lower Nusselt number ratios were obtained right before and after the ribs. Between any two ribs in the first passage, the Nusselt number ratios are highest near the inner surface and decrease as we move toward the outer surface. This is due to the rib induced secondary flow moving from the rib leading to the trailing side as shown in Figure 25(a). Moreover, the Nusselt number ratios between the ribs increased gradually along the first passage until the flow approaches the seventh rib, where it decreases gradually until the ninth rib. Nusselt number ratios in the turn are higher in the region next to the divider wall tip while lower at the first corner. In the second passage (between any two ribs), the Nusselt number ratios are higher near the outer surface and decrease as we move toward the inner surface. Again, this is a result of the rib induced secondary flow in the second passage shown in Figure 25(b). Figures 27(c) and 27(d) show the Nusselt number ratio contours on the leading side for case 2 (Ro = 0.11 and β = 90°) and case 3 (Ro = 0.11 and β = 135°), respectively. Comparing these figures with the non-rotating leading side, we notice that the Nusselt number ratios decrease in the first passage, in both cases, with the decrease in case 3 being higher. In the second passage, the Nusselt number ratios in both cases increase with respect to case 1. Figure 27(e) and 27(f) show the Nusselt number ratio contours on the trailing surface for case 2 (Ro = 0.11 and β = 90°) and case 3 (Ro = 0.11 and β = 135°), respectively. Comparing these figures with the non-rotating trailing side, we notice that the Nusselt number ratios increase in the first passage, for both cases, with the increase in case 2 being higher. In the second passage, the Nusselt number ratios in both cases decrease, with the decrease in case 2 being higher. The Nusselt number in the bend is much higher for both cases when compared to the non-rotating case.

Figure 27. Detailed Nusselt number ratios for AR = 2 rectangular duct (Al-Qahtani et al. [40]).
The present method was also employed for a parametric study of single-pass rectangular ducts with a higher channel aspect ratio (AR) of 4:1 as shown in Figure 28. The channel hydraulic diameter, $D_h$, is 0.8 in (2.03 cm). In the ribbed section, the leading and trailing surfaces are roughened with nine equally spaced ribs of square cross section. The rib height-to-hydraulic diameter ratio ($e/D_h$) is 0.078 and the rib-pitch-to-height ratio ($P/e$) is 10. All ribs are inclined at an angle $\alpha = 45^\circ$ with respect to the flow. Two channel orientations are studied: $\beta = 90^\circ$ corresponding to the mid-portion of a turbine blade and $\beta = 135^\circ$ corresponding to the trailing edge region of a blade.

In general, the strength of the rotation-induced secondary flow increases and gradually overcomes the rib induced secondary flow as we increase the rotation number and density ratio. By reaching a rotation number of 0.28 and a density ratio of 0.40, the rotation-induced secondary flow is found to be dominant over the rib induced secondary flow especially downstream of the channel. The rib induced secondary flow is not any more able to drive the secondary flow from the ribs leading side (near the top surface) to the ribs trailing side (near the bottom surface). On the contrary, the rotation induced secondary flow moves the cold fluid from the bottom surface along the ribbed surfaces with the secondary flow along the leading surface is much stronger than the one on the trailing surface. The temperature contours in Figure 29 indicate that the cold fluid is moved toward the bottom surface which is in contrary to the low rotation cases shown in Al-Qahtani et al. [41].

Figure 30 shows the local Nusselt number ratio contours for various rotation numbers. The non-rotating case in Figures 30(a) and 30(b) will be used as a baseline for comparison and discussion. In Figure 30(a), the highest Nusselt number ratios were obtained on the top of the ribs, and the lower Nusselt number ratios were obtained right before and after the ribs. Between any two ribs, the Nusselt number ratios are highest near the top surface and decrease as we move towards the bottom surface. This is due to the rib induced secondary flow that moves from the top surface (and parallel to the ribbed walls) to the bottom surface.

For fixed rotation number and density ratio ($Ro = 0.14$ and $\Delta\rho/\rho = 0.122$), Figures 30(c) and 30(k) show the Nusselt number ratios contours on the leading side for $\beta = 135^\circ$ and $90^\circ$, respectively. Comparing these figures with the non-rotating leading side, it is noticed that the Nusselt number ratios decreased in both cases with the decrease in the $\beta = 135^\circ$ case being the most (a 19% decrease compared to a 10% decrease in the $90^\circ$ case). Figures 30(d) and 30(l) show the Nusselt number ratios contours on the trailing side for $\beta = 135^\circ$ and $90^\circ$, respectively. Comparing these figures with the non-rotating trailing side, it is noticed that the Nusselt number ratios in the $\beta = 135^\circ$ case decreased more on the leading side and increased less on the trailing side compared to $\beta = 90^\circ$ case can be understood in light of the conceptual secondary flow diagram in Figure 31. The rotation induced vortex in the $\beta = 135^\circ$ configuration move along the full face of the leading or trailing surfaces. However, the rotation induced vortex in the $\beta = 90^\circ$ configuration moves along only one half the face of the leading or trailing surfaces. With this in mind, we notice in Figure 31 that the two secondary flows produced by rotation and angled ribs for the rotating $\beta = 135^\circ$ duct combine destructively (opposite direction) and thus reduce heat transfer on both the leading surface and the trailing surface. On the other hand, the two secondary flows produced by rotation and angled ribs for the rotating $\beta = 90^\circ$ duct combine to (i) constructively (same direction) enhance heat transfer for only one half of each of the leading and trailing surfaces and (ii) destructively (opposite direction) reduce heat transfer for the other half of each of the leading and trailing surfaces.

When the rotation number was increased from 0 to 0.14 (Figure 30(c)), the Nusselt number ratios were decreased by 19%. But when the rotation number was further increased to 0.28 (Figure 30(e)), the Nusselt number ratios were decreased only by 10% compared to the non-rotating case. Moreover, it is noted that the high Nusselt number ratios regions are shifted to the middle of the ribbed surface. This is because of the rotation induced secondary flow getting stronger and gradually overcomes the rib induced secondary flow. In Figure 30(g), the rotation number is kept fixed at 0.28 while the density ratio is increased to 0.20. It is seen from this figure that the high Nusselt number ratios regions are moved further toward the bottom surface. Increasing the density ratio further to 0.40 (Figure 30(i)), we notice that the high Nusselt number ratios regions are now existing next to the bottom surface with a total decrease of only 4% compared to the non-rotating case.

For the trailing surface, the Nusselt number ratios were increased only by 1% when the rotation number changes from 0 to 0.14 (Figure 30(d)). In Figure 30(f), the rotation number was increased further to 0.28 while the density ratio is kept fixed at 0.122. This causes the Nusselt number ratios to increase by 6% compared to the non-rotating number of 10,000 with rotation numbers ranging from 0 to 0.28 and inlet coolant-to-wall density ratios $\Delta\rho/\rho$ ranging from 0.122 to 0.40. Detailed secondary flow vectors, temperature contours, and Nusselt number distributions were given in Al-Qahtani et al. [41] for all nine cases considered. Here, we will present the results for the highest rotation number ($Ro = 0.28$) and highest coolant-to-wall density ratio ($\Delta\rho/\rho = 0.40$) case as shown in Figure 29 to illustrate the general flow characteristics for high aspect ratio (AR = 4) cooling passage in the trailing edge region ($\beta = 135^\circ$).
case. Also, it is seen from this figure that the high Nusselt number ratios regions are spreading toward the bottom surface. In Figure 30(h), the rotation number is kept fixed at 0.28 while the density ratio is increased to 0.20. It is seen in this figure that the high Nusselt number ratios are moved slightly more toward the bottom surface. Increasing the density ratio further to 0.40 (Figure 30(j)) causes the Nusselt number ratios to increase by 12% compared to the non-rotating case. It is also seen from this figure that, upstream of the channel, the high Nusselt number ratios are pushed slightly more toward the bottom surface while downstream they dominate most of the inter-rib regions.

![Figure 30. Leading and trailing surfaces Nusselt number ratios in AR = 4 rectangular duct (Al-Qahtani et al. [41]).](image)

![Figure 31. Conceptual view of the secondary flow induced by angled ribs and rotation for AR = 4 rectangular duct (Al-Qahtani et al. [41]).](image)

**CONCLUDING REMARKS**

A multiblock RANS method was employed for the calculation of three-dimensional flow and heat transfer in stationary and rotating cooling passages with various combinations of rotation number, coolant-to-wall density ratio, bend geometry, rib configuration, channel aspect ratio, and channel orientation. The method solved Reynolds-Averaged Navier-Stokes equations in conjunction with a near-wall second-order Reynolds stress closure model for accurate resolution of the turbulent flow and thermal fields produced by rotation and buoyancy effects. In general, the second-moment solutions exhibit a significant level of anisotropy in turbulent stress and heat flux distributions. For the rotating turbine blade cooling passages considered here, the Coriolis and centrifugal buoyancy forces produced strong non-isotropic turbulence that significantly influenced the development of momentum and thermal boundary layers along the turbine blade cooling passages. The present near-wall second-moment closure model accurately predicted the effects of blade rotation, 180° sharp turn, rib-induced flow separation and reattachment, rib-induced secondary flow, channel aspect ratio, and channel orientation on the generation of near-wall turbulence and the enhancement of surface heat transfer.

**REFERENCES**


