Flow and Heat Transfer in a Ribbed U-Duct under Typical Engine Conditions

T. I.-P. Shih, Y.-L. Lin, M. A. Stephens, and M. K. Chyu
Department of Mechanical Engineering, Carnegie Mellon University
Pittsburgh, Pennsylvania 15213-3890

K.C. Civinskas
NASA – Lewis Research Center, Cleveland, Ohio 44135

ABSTRACT

Computations were performed to study the three-dimensional flow and heat transfer in a ribbed U-shaped duct of square cross section under operating conditions that are typical of industrial gas turbines. Basically, all walls were maintained at a temperature of 800 K, and the coolant air at the duct inlet had a temperature of 550 K and a pressure of 10 atm. Both rotating and non-rotating cases were investigated. When rotating, the angular speed was 3,600 rpm. The Reynolds number based on the duct hydraulic diameter was set at 350,000, which represents an upper limit in coolant flow. The results obtained in this study were compared with those from previous numerical studies with a lower Reynolds number, namely 25,000, which represents a lower limit in coolant flow.

This computational study is based on the ensemble-averaged conservation equations of mass, momentum (compressible Navier-Stokes), and energy. Turbulence is modelled by two low-Reynolds number k-ω models: an SST version with isotropic eddy diffusivity and a nonlinear version with anisotropic eddy diffusivity from an explicit algebraic Reynolds stress model. Solutions were generated by using a cell-centered finite-volume method, that is based on flux-difference splitting and a diagonalized alternating-direction implicit scheme with local time-stepping and V-cycle multigrid.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_h$</td>
<td>hydraulic diameter of duct</td>
</tr>
<tr>
<td>$h$</td>
<td>heat transfer coefficient ($q_w(T_w - T_b)$)</td>
</tr>
<tr>
<td>$M_i$</td>
<td>Mach number ($V_i/\sqrt{T_b R_i}$)</td>
</tr>
<tr>
<td>$Nu$</td>
<td>Nusselt number ($hD_h/\kappa$)</td>
</tr>
<tr>
<td>$Nu_s$</td>
<td>Nusselt number for smooth duct ($0.023 Re^{0.8} Pr^{0.4}$)</td>
</tr>
<tr>
<td>$q_w$</td>
<td>wall heat transfer rate per unit area</td>
</tr>
<tr>
<td>$R_i$, $R_o$</td>
<td>inner and outer radius of 180° bend (Fig. 1)</td>
</tr>
<tr>
<td>$R_p$, $R_t$</td>
<td>radius from axis of rotation (Fig. 1)</td>
</tr>
<tr>
<td>$R$</td>
<td>gas constant for air</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number ($\rho V_i D_h/\mu$)</td>
</tr>
<tr>
<td>$Ro$</td>
<td>rotation number ($\Omega D_h/V_i$)</td>
</tr>
<tr>
<td>$T$</td>
<td>temperature</td>
</tr>
<tr>
<td>$V_i$</td>
<td>average velocity at duct inlet</td>
</tr>
<tr>
<td>$x, y, z$</td>
<td>coordinate system (Fig. 1)</td>
</tr>
</tbody>
</table>

Greek

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\Delta \rho$</td>
<td>density rate per unit $k$</td>
</tr>
<tr>
<td>$\omega$</td>
<td>dissipation rate per unit $k$</td>
</tr>
</tbody>
</table>

Subscripts

- $c$ = coolant air
- $i$ = coolant inlet conditions
- $w$ = wall

INTRODUCTION

Internal cooling is an effective method for keeping the temperature of turbine blades and vanes within acceptable limits for structural integrity. Since the air used to cool the blades and vanes is extracted from the compressor, efficiency considerations demand effective cooling with minimal cooling air. This need for efficiency has led numerous investigators to study fluid flow and heat transfer processes inside internal coolant passages and to develop and evaluate design concepts that enhance heat transfer with minimal drag.

Computational studies on internal cooling have mostly been on non-rotating smooth and ribbed ducts and smooth rotating ducts. In recent years, Prakash & Zerkle (1995), Bonhoff, et al. (1997), and Stephens & Shih (1997) studied rotating ducts with ribs. Prakash & Zerkle (1995) used a k-ω model with wall functions to study the flow and heat transfer in a straight duct with square ribs. In their study, the flow was taken to be incompressible, and periodicity in the streamwise direction was assumed. Bonhoff, et al. (1997) studied the flow and heat transfer in a U-duct (i.e., a duct with two straight sections and a 180-degree bend) in which the leading and trailing walls were lined with inclined ribs. In their study, a Reynolds stress equations model with wall functions were used. Stephens & Shih (1997) also studied a U-duct with inclined ribs lining two opposite walls. Their study was based on a low Reynolds number k-ω turbulence model (SST version). By using a low Reynolds number turbulence model, their study was able to model with greater fidelity the actual geometry of the ribs. When wall functions are used, the rib geometry can be compromised since boundary conditions are applied at one grid point away from the boundary, and the location of that grid point is typically at a $y^+$ of 20 to 60. The rib geometry must be resolved correctly because they exert considerable influence on the flow.

Despite concerns in modelling of turbulence, one major advantage of computational studies is its ability to match all dimensionless parameters associated with actual engine conditions. This feat is difficult to do experimentally especially for rotating ducts because of scaling problems and hardware limitations. Nevertheless, all computational studies that are cited above for rotating ducts did not do so. Instead, they simulated conditions that are similar to those in experimental studies, which differed from realistic engine conditions in wall and coolant temperatures as well as rotational speeds.

Presented at the International Gas Turbine & Aeroengine Congress & Exhibition
To investigate computationally the three-dimensional flow and heat transfer in a ribbed U-duct under operating conditions that are typical of an industrial gas turbine. The Reynolds number of 350,000, however, represents an upper limit since a lower Reynolds number limit of 25,000 was computed by Stephens & Shih (1997), that simulation is compared with the ones obtained in this study under rotating and non-rotating conditions at the higher Reynolds number limit. Note that when the temperature and pressure at the duct inlet is fixed as is done in this study, Reynolds number effectively determines the coolant flow rate. The higher the Reynolds number, the higher is the flow rate and the lower is the rotation number for a fixed angular speed.

**DESCRIPTION OF PROBLEM**

A schematic diagram of the problem investigated is shown in Fig. 1, and is identical to the one studied by Stephens & Shih (1997). It consists of a U-shaped duct of square cross section made up of two straight sections and a 180° bend. The geometry of the straight section is the same as that reported in Wagner, et al. (1991a,b). The dimensions of the duct are as follows (see Fig. 1). The duct hydraulic diameter is \( D_h = 1.27 \text{ cm} \) \((0.5 \text{ in})\). The radial position of the duct relative to the axis of rotation is given by \( R/R_h = 41.85 \) and \( R/R_h = 56.15 \) so that the mean radius is \( R_m/D_h = (R_0 + R_l)/2D_h = 49 \). The length of the duct’s straight section is \( L/D_h = 14.3 \). The curvature of the 180° bend is given by \( R/R_h = 0.22 \) and \( R/R_h = 1.44 \).

The leading and trailing walls of the straight sections are each roughened with five equally-spaced ribs. The ribs on those two walls are staggered relative to each other with ribs on the leading wall offset upstream from those on the trailing wall by a half pitch (\( p \)). The ribs are located just upstream or downstream of the 180° bend. All ribs are inclined with respect to the flow at an angle of 45°. The cross section of the rounded ribs is made up of three circular arcs of radius 0.0365 cm (0.025 in) so that the rib height (\( e \)) is 0.127 cm (0.05 in) and the rib-height to hydraulic-diameter (\( e/D_h \)) is 0.1. The pitch-to-height ratio (\( p/e \)) was set to 5 (same as UTRC experiments). With only five ribs on each face, a considerable length of the duct has smooth walls. See Stephens & Shih (1997) for details on the ribs.

For this problem, the four walls of the duct including the rib surfaces were maintained at a constant temperature of \( T_w = 800\,\text{K} \). The coolant flow entering the duct is air, and had a uniform temperature of \( T_c = 550\,\text{K} \) and an average static pressure of 10 atm. Unlike the temperature profile, the velocity profile at the inlet was not uniform but taken to be “fully developed” in the sense that any changes in the profile are due to pressure changes needed to overcome drag. The process used to generate the velocity profile is given in Stephens, et al. (1996a). The Reynolds number at the inlet was \( Re = 350,000 \). The angular speed of the rotating duct was 3,600 rpm (376.99 radians per second). These conditions correspond to a rotation number of \( Ro = 0.039 \); an inlet Mach number (relative to duct) of \( M_i = 0.26 \); and a coolant-to-wall temperature ratio at the duct inlet of 0.68 giving rise to an inlet density ratio of \( \Delta \rho/\rho = 0.3125 \).

The above geometry and operating conditions are typical of an industrial gas turbine. The Reynolds number of 350,000, however, represents an upper limit. Since a lower Reynolds number limit of 25,000 was computed by Stephens & Shih (1997), that simulation is compared with the ones obtained in this study under rotating and non-rotating conditions at the higher Reynolds number limit. Note that when the temperature and pressure at the duct inlet is fixed as is done in this study, Reynolds number effectively determines the coolant flow rate. The higher the Reynolds number, the higher is the flow rate and the lower is the rotation number for a fixed angular speed.

**PROBLEM FORMULATION**

The flow and heat transfer problem described in the above section and depicted in Fig. 1 is modeled by the ensemble-averaged conservation equations of mass (continuity), momentum (compressible Navier-Stokes), and total energy for a thermally and calorically perfect gas with Sutherland's model for viscosity and thermal conductivity. These equations are written in a coordinate system that rotates with the duct so that steady-state solutions with respect to the duct can be computed. The form of these equations is well known. See, Steinthorsson, et al. (1991), Prakash & Zerkle (1995), and Stephens & Shih (1997) for details.

The aforementioned ensemble-averaged conservation equations were closed by two different low Reynolds number \( k-\omega \) turbulence models. One model is the SST version with isotropic eddy diffusivity, the other model is a nonlinear version with anisotropic eddy diffusivity from an explicit algebraic stress model. Abid, et al. (1995). The SST version was selected because it satisfies many constraints required of turbulence models, such as asymptotic consistency near walls without the need for ad hoc corrections. Also, it does not require knowledge about normal distance from the wall in its implementation. Note that no changes were made to modify this turbulence model for rotation. The nonlinear version was selected because it can account for some of the anisotropic effects of turbulence such as those that arise in corners of square ducts and streamline curvature from bends and rotation.

The essence of the nonlinear model employed is as follows (see Abid, et al. (1995) for details). First, invoke the two assumptions needed to obtain algebraic Reynolds stresses. That is, assume (1) convection minus diffusion in the Reynolds stress transport equations is proportional to convection minus diffusion in the turbulent kinetic energy transport equation and (2) the anisotropic Reynolds stress is constant along a streamline. Second, model the pressure-strain correlation according to the one proposed by Launder, et al. (1975). This produces an implicit algebraic equation for the Reynolds stresses. Third,
employ the method of Gatski & Speziale (1993) to obtain an explicit instead of implicit algebraic equation for the Reynolds stresses. This method is a three-dimensional extension of the technique developed by Pope (1975) and is based on a first-order polynomial expansion of the intensity basis. An explicit expression for the Reynolds stresses greatly facilitates numerical studies because a major source of the numerical instability is eliminated.

In order to obtain solutions to the aforementioned conservation equations, boundary and initial conditions are needed. At the inflow boundary (duct entrance), a "fully" developed profile was specified for velocity (see Stephens, et al. (1996a)), but the temperature profile was taken to be uniform at 550 K. Turbulence quantities (k and $\omega$) were also specified in a manner that is consistent with the velocity profile (average turbulent intensity was 5%). Only pressure was extrapolated. At the outflow boundary where the cooling air exits the duct, an average back pressure was imposed but the pressure gradients in the two spanwise directions were extrapolated. This is important because secondary flows induced by inclined ribs, the bend, and centrifugal/Coriolis forces cause considerable pressure variations in the spanwise directions. At all walls, the no-slip condition was imposed, and the cross-stream plane is 65 x 65. From Fig. 2, it can be seen that grid lines are clustered near all solid surfaces to resolve the sharp gradients there, are smooth as grid spacings change from fine to coarser, and are orthogonal everywhere except along rib surfaces. Along rib surfaces, the grid lines are aligned with the rib inclination. The grid system used satisfy a set of "rules of thumb" that are needed to obtain grid independent solutions such as having five grid points within $y^+$ of 5. The rules and the process can be found in Stephens, et al. (1996a), and will not be repeated here.

On a Cray C-90 computer, where all computations were performed, the memory requirement for each run is 155 megawords (MWs). The CPU time required depends on the turbulence model used. It is 40 hours for the SST model and 50 hours for the nonlinear k-c model. The CPU time given is for one converged steady-state solution, which typically involved 3,000 iterations. Note that the "path" to steady-state from the initial condition given by Eq. (2) is implemented as follows: Run laminar for 200 time steps, then run SST to convergence if SST model is of interest. If nonlinear k-o model is of interest, then SST is run for 1000 time steps before switching to nonlinear k-o.

At this point, noted that the code used in this study has been validated for two problems. The first problem is flow in a non-rotating square duct in which two opposite walls are lined with inclined ribs. In that problem, the geometry of the duct and rib is identical to that studied here for the up-leg portion of the U-duct. The computed heat-transfer enhancement for that problem was compared with measurements obtained using liquid crystal thermography (see Stephens, et al. (1996a)). The comparisons showed that the code used can predict the qualitative features very well. The predicted quantitative features were also found to be reasonable (peak values are within 10% of measurements). The second problem is flow in a rotating U-duct with smooth walls. The geometry of that problem is the same as that studied here except for the absence of ribs. For that problem, the computed Nusselt numbers on the leading and trailing walls for the up-leg part of the U-duct were compared with measured ones by Wagner, et al. (1991a). Those comparisons, described in Stephens, et al. (1996b), showed very good agreements. The good comparisons obtained in those two studies give some confidence to the code for studying the present problem.

NUMERICAL METHOD OF SOLUTION

Solutions to the governing equations just described were obtained by using a cell-centered finite-volume code called CFL3D (Thomas, et al. (1990) and Rumsey & Vatsa (1993)). In this study, the CFL3D code was slightly modified so that it can account for steady-state solutions in a rotating frame of reference by adding source terms that represent centrifugal and Coriolis forces in the momentum and energy equations.

The CFL3D code contains a number of different algorithms. The one used in this study is as follows. All inviscid terms were approximated by the second-order accurate flux-difference splitting of Roe (1981). Flux-difference was used so that numerical diffusion is much smaller than physical diffusion. All diffusion terms were approximated conservatively by differencing derivatives at cell faces. Since only steady-state solutions were of interest, time derivatives were approximated by the Euler implicit formula. The system of nonlinear equations that resulted from the aforementioned approximations to the space- and time-derivatives were solved by using a diagonalized alternating-direction scheme (Pulliam & Chaussee (1981)) with local time-stepping (local Courant number always set to unity) and three-level V-cycle multigrid (Anderson, et al. (1988)).

The domain of the problem was replaced by a single-block grid system that was assembled by patching together 37 blocks of H-H structured grids with C^2 continuity at all block interfaces (see Fig. 2). For this grid system, the total number of grid points in the streamwise direction from inflow to outflow is 761. The number of grid points in the cross-stream plane is 65 x 65. From Fig. 2, it can be seen that grid lines are clustered near all solid surfaces to resolve the sharp gradients there, are smooth as grid spacings change from fine to coarser, and are orthogonal everywhere except along rib surfaces. Along rib surfaces, the grid lines are aligned with the rib inclination. The grid system used satisfy a set of "rules of thumb" that are needed to obtain grid independent solutions such as having five grid points within $y^+$ of 5. The rules and the process can be found in Stephens, et al. (1996a), and will not be repeated here.

RESULTS

As noted in the Introduction, the objective of this study is to investigate the details of the three-dimensional flow and heat transfer in a ribbed U-shaped duct with and without rotation under typical engine conditions for industrial gas turbines. The focus is to shed light on how to evaluate data (experimental or computational) gathered under more idealized conditions (e.g., stationary or low-speed rotating ducts)
to actual engine conditions (e.g., rotation at 3,600 rpm).

To meet this objective, computations were performed for two cases which represent an upper limit in coolant flow rate: (1) non-rotating \((Ro = 0.0)\) with \(Re = 350,000\) and (2) rotating at 3,600 rpm \((Ro = 0.599)\) with \(Re = 350,000\), in which \(T_w = 800\) K and \(T_c = 550\) K for both cases. These results are compared with a computation generated by Stephens & Shih (1997) for the same duct and rib geometry which represents a lower limit in coolant flow: rotating at 3,132 rpm \((Ro = 0.24)\) with \(Re = 25,000\), \(T_w = 344.83\) K, and \(T_c = 300\) K. Note that there is a small difference in the angular speed and a large difference in Rotation number \(Ro\) and the wall and coolant temperatures. All other parameters not mentioned are identical between the two studies including the 10 atm static pressure at the duct inlet. The differences must be considered while comparing results.

The numerical data generated in this study are given in Figs. 3 to 5. Since details are hard to discern from these figures, the complete set of numerical data in PLOT3D format can be made available by contacting the first author.

At this point, note that the two turbulence models investigated gave similar results in terms of the major features of the flow. The differences are in the values of the maxima and minima as well as their relative locations (up to 20%). Since the models used do not affect the interpretations and conclusions made in this section, further details on the effects of the turbulence models used are not given. A detailed quantitative study on the effects of these two turbulence models is given in another paper.

**Flow Field**

The flow field in the U-duct with ribs and a coolant-to-wall temperature that is less than unity can be inferred from Figs. 3 and 4. These figures show the cross-stream velocity vectors projected onto several \(Y-Z\) planes (in the straight portions of the duct) and one \(X-Y\) plane (in the middle of the bend). These figures also show normalized temperature defined by \((T - T_c)/(T_w - T_c)\) in each of the planes. Note that all vector plots are looking in the +\(x\) direction. Also, vectors in certain planes were magnified more in order to show more clearly the flow patterns there. Planes that require magnification were those far upstream of ribs (magnified by a factor of 6 relative to other planes).

Before discussing the results in Figs. 3 and 4, it is important to review the nature of the flow in a smooth U-duct under non-rotating and rotating conditions. For a non-rotating U-duct with \(Re = 25,000\), Stephens, et al. (1996b) showed the velocity profile to have a maximum about the center of the duct-cross-section in the up-leg portion. Around the bend, a large streamwise separation region forms on the convex side of the bend but none on the concave side. The bend also forms a symmetric pair of secondary flows (Dean type), which shifts the velocity towards the outer wall in the down-leg portion. When there is rotation \((Ro = 0.24, Re = 25,000)\), Stephens & Shih (1997) showed two symmetric counter-rotating flows to form because of the Coriolis force. With radially outward flow, this pair flows from the trailing face to the leading face along the inner and outer walls, which transports cooler air from near the center of the duct cross section to the trailing wall. Since the thermal boundary layer starts on the wall and coolant temperatures are lower than that near the leading wall. With higher temperature and hence lower density near the leading wall, centrifugal buoyancy accelerates the flow there, and this can cause flow separation to take place on that surface. If there is flow separation on the leading wall, then additional secondary flows form next to that surface. This is because the reverse-flow in the separated region causes the Coriolis force to reverse in direction. In the down-leg part of the U-duct, centrifugal buoyancy accelerates the lower density fluid.

With the above as a backdrop, the results of this study given in Figs. 3 and 4 are described.

**Rotation in the U-Duct at High Reynolds Number** Consider first the case involving non-rotating U-duct with \(Re = 350,000\). For this case, the following observations can be made (most but not all can be inferred from Fig. 3). The first observation is that with a nonlinear \(k-\omega\) model, the vortex-pairs in each of the four corners of the duct induced by anisotropy in the turbulence can now be resolved. The magnitude of the velocity components in the cross-stream plane, however, is extremely small when compared to those generated by inclined ribs. Their existence is eliminated once the flow approaches the much stronger secondary flows induced by the ribs. Thus, these vortices play a very minor role in the flow field when there are ribs.

The second observation is that inclined ribs induce a pair of secondary flows (Fig. 3). The secondary flows originate from the inner wall to the outer wall along the two side walls in the up-leg part and from the outer wall to the inner wall in the down-leg part. In the up-leg part of the duct, the pair of secondary flows is nearly symmetric when they reached the top of the fourth rib (or the second rib before the bend) on the leading and trailing faces despite the staggered arrangement of the ribs. Note that leading and trailing is not relevant here except to label the surfaces. In the down-leg part, the flow is much more complex because of interactions with flows induced by the ribs in the up-leg part and the bend as explained below.

The third observation is that there is a large separated region around the inner wall of the bend that extends all the way to the first rib downstream of the bend. This separated region reduced the effectiveness of the first rib downstream of the bend in creating secondary flows. This streamwise separated flow will adversely affect the surface heat transfer in that region. Since the flow separation about the bend attaches just upstream of the first rib in the down-leg part of the U-duct, placing a broken rib further upstream could minimize or eliminate this separated region.

The fourth observation is that though the flow does not separate next to the outer wall of the bend, there is a region of minimum velocity next to the leading and trailing walls along a strip between the inner and outer side walls. The source of these two strips of highly retarded flows can be attributed to the interactions between the secondary flows that emerge from the up-leg part and the pressure gradient created by the bend.

The fifth observation is that the secondary flows induced by inclined ribs dominate the flow field. As mentioned earlier, they are much stronger than the corner vortices. It turns out that with a \(Re = 350,000\), they are also much stronger than the secondary flows induced by curvature in the bend. In fact, Fig. 3 shows that Dean type secondary flows never seem to form! Dean-type secondary flows should flow from the outer wall to the inner wall along the leading and trailing walls. In Fig. 3, it can be seen that in the middle of the bend, the secondary flows are flowing from the inner wall to the outer wall instead of vice versa. These secondary flows are generated by inclined ribs that are located in the up-leg part of the U-duct. Though Fig. 3 does show two vortices that rotate in the sense of Dean-type vortices in the two corners next to the outer wall in the middle of the bend, these vortices would also be created by the rib-induced secondary flows from the up-leg part of the duct. Along the bend, the secondary flows induced by the ribs in the up-leg part of the duct become stronger because the acceleration in the streamwise direction stretched the vortices. Downstream of the bend, the flow field is dominated by the interactions between the secondary flows generated by ribs in the up-leg part and those generated in the down-leg part. Just upstream of the ribs in the down-leg part of the duct (see plane denoted by just downstream of the bend in Fig. 3), the secondary flow induced in the up-leg part of the duct are pushed towards the outer wall. The secondary flows induced by the ribs in the down-leg part are confined near the inner wall. After the fourth rib tops on the leading and trailing faces downstream of the bend, the secondary flows created in the up-leg part continue to persist. In fact, this persistence continued even \(4\delta_y\) downstream of the last rib in the down-leg part of duct. The net result of these secondary flows and their interactions is to transport cooler fluid to the inner wall in the up-leg part and to the outer wall in the down-leg part.

The above results indicate that if the rib inclination was \(-45^\circ\) instead of \(+45^\circ\) in the up-leg part of the duct, then Dean-type secondary flows in the bend would be damped by secondary flows. To continue this reinforcement in the down-leg part of the duct, the rib inclination there should remain at \(+45^\circ\). Whether this reinforcement is good or not for heat transfer require further study.

**Effects of Rotation at High Reynolds Number** Now, consider the case involving the rotating U-duct with \(Re = 350,000\) and \(Ro = \)
0.039. For this case, the following observations can be made (most but not all can be inferred from Figs. 3 and 4). The first observation is that Coriolis induces two counter-rotating vortices that transport cooler fluid from near the center of the duct to the trailing wall in the smooth part of the U-duct. Note that the corner vortices induced by the anisotropy in the turbulence are essentially non-existent.

The second observation is that once the flow approaches the inclined ribs, the flow induced by the ribs quickly dominate those induced by the Coriolis force. The fourth observation is that the leading and trailing faces, the secondary flows induced by the ribs are again nearly symmetric in a manner similar to that for the non-rotating case. The temperature field, however, do differ because the secondary flows from the Coriolis force brought cooler fluid to the trailing face and hotter fluid to the leading face in the smooth part of the duct. Just downstream of the fifth and last rib in the up-leg part of U-duct (see plane labeled just before of bend in Fig. 4), the temperature field also appears to be nearly symmetric. Thus, at very high Reynolds numbers such as the one investigated here (namely, Re = 350,000), the ribs dominate the flow in the up-leg part of duct.

The third observation is that though centrifugal buoyancy decelerates the flow next leading and outer walls in the up-leg part of U-duct because of lower densities there, flow separation never took place. Thus, flow separation from centrifugal buoyancy is of concern only at lower Reynolds number or flow rates where rotation number becomes higher.

The fourth observation is that the two secondary flows formed in the up-leg part of U-duct are rotated as a pair along the bend in a clockwise direction (i.e., the vortex next to the trailing face moved to the outer wall, and the vortex next to the leading face moved next to the inner wall; see plane labeled middle of bend in Fig. 4). This motion is induced by a combination of Coriolis force and pressure gradient induced by the curvature in the bend — all of which strengthen the vortex next to the trailing wall and weaken the vortex next to the leading wall. Again, traditional Dean-type secondary flow do not form. The net effect of this flow is to transport cooler fluid to the inner and trailing walls.

The fifth observation is that streamwise flow separation next to the inner wall of the bend is essentially undistorted by rotation. This is consistent with the other observations at high Reynolds numbers.

The sixth observation is that the two strips of retarded flow on the leading and trailing faces about the bend are no longer symmetric as in the non-rotating case. On the leading face, this strip only extends half way into the bend but is thicker. On the trailing face, this strip still extends across the entire bend.

The seventh observation is that in the down-leg part of the duct, the vortices formed by ribs on the leading face in the up-leg and the down-leg parts merge. Thus, in the down-leg part of duct, the vortex next to the leading face dominate over the one next to the trailing face. The net effect of this flow is to transport cooler fluid to the outer and leading walls. At 4Dh after the fifth or last rib in the down-leg part, only the vortex formed by the leading face persists.

Thus, rotation still affects the nature of the flow in the down-leg part of the U-duct despite the very high Reynolds number at least in the bend and the first four to five ribs after the bend. Based on these results, one could conjecture that results obtained for the up-leg part of the U-duct under non-rotating conditions are also applicable to rotating ducts if the Reynolds number is sufficiently high as far as the flow field is concerned.

Effects of Reynolds and Rotation Numbers in Rotating Duct

At this point it is again emphasized that when the temperature and pressure at the duct inlet is fixed as is done in this study, the higher the Reynolds number, the higher is the flow rate and the lower is the rotation number for a fixed angular speed. Thus, Reynolds and rotation numbers are not independent of each other under such circumstances. To examine the effects of Reynolds or rotation number in a rotating duct, the data obtained in this study for Re = 350,000 (Ro = 0.039) are compared against those generated by Stephens & Shih (1997) for Re = 25,000 (Ro = 0.24).

When comparing results from Stephens & Shih (1997) with Fig. 4, the following observations can be made. First, the strength of the secondary flows setup by the Coriolis force is clearly stronger when Re is higher. Second, the secondary flow induced by inclined ribs is also stronger when the Re is higher. Third, the secondary flows induced by Coriolis force affect the secondary flows induced by the inclined ribs when the Re is low and Ro is high. Fourth, when the Re is low, bend induced Dean-type secondary flows definitely form. However, the pair is asymmetric with the one next to the leading face being much larger. This is because the secondary flows formed by ribs on the trailing face were strengthened by the pressure gradient induced by the curvature in the bend. Fifth, at the lower Re, there is no streamwise flow separation around the bend. Also, there is only small strip of retarded flow on the leading face in the bend. Sixth, downstream of the bend and into the region with ribs, the aforementioned dominant secondary flows continues to dominate.

Based on the above, rotation can be seen to play a very important role on the fluid flow when the Re is low because Ro becomes high.

Surface Heat Transfer

In this section, the heat transfer characteristics in a ribbed U-duct with and without rotation are described with focus on effects of rotation and Reynolds number. Figure 5 shows the heat transfer results in terms of the Nusselt number that is normalized by the following well-known correlation for flow in ducts (Kays & Perkins (1993)):

$$N_u = 0.023 Re^{0.8} Pr^{0.4}$$

where $Re = 350,000$ or 25,000 and $Pr = 0.72$. Note that though $Nu/N_u$ can exceed 5, it is plotted in the range between 0.5 and 3.0 to more clearly show the variations on the leading and trailing surfaces.

From Fig. 5, the following four observations can be made about the heat transfer characteristics with and without rotation at $Re = 350,000$. First, whether there is rotation or not, the maximum heat transfer rate occurs at the ribs. The lowest heat transfer rates take place at the following locations on the leading and trailing faces: just downstream of the rib tops and towards the outer wall for the up-leg part; just downstream of the rib tops and towards the inner wall for the down-leg part; in the streamwise separated region just downstream of the bend; and in the thin strips on the leading and trailing faces in the bend where the flow is highly retarded.

Second, when there is no rotation, the surface heat transfer on the leading and trailing faces are nearly identical with the average heat transfer rate in the up-leg part higher than that in the down-leg part. The slight difference in the heat transfer rates on the leading and trailing faces is due to the staggered arrangement of the ribs. The reduction in heat transfer rate in the down-leg part is due to the separated region downstream of the bend, which reduced the strength of the secondary flows induced by the ribs there.

Third, when there is rotation, the surface heat transfer on the trailing face is markedly higher than that on the leading face. On the leading face, the average heat transfer rate in the up-leg part is lower than that on the down-leg part.

Fourth, though the flow field in the up-leg part of the duct with and without rotation are similar at high Re (and hence low Ro), the heat transfer characteristic are quite different, especially on the leading face. This difference is created by the asymmetry in the temperature distribution induced by rotation in the smooth part of the duct. Thus, if heat transfer characteristics are of interest, results generated for non-rotating ducts cannot be readily extended to rotating ducts even when the Re is as high as 350,000 if the angular speed is 3,600 rpm.

From Fig. 5, the following three observations can also be made about the heat transfer characteristics in a rotating duct at high and low Re (or low and high Ro). First, the heat transfer enhancement created by ribs is higher at the low Re. This confirms a well-known fact. Second, the heat transfer enhancement is the highest at the lower Re, there is no streamwise flow separation around the bend and the two strips of highly retarded flow next to the leading and trailing walls that occur at the high Reynolds number. This suggests that ribs and/or pin fins are needed in the bend region to disrupt these adverse flow patterns and produce greater heat transfer enhancement. Third, at low Re, heat transfer enhancement is low just upstream of the ribs. This suggests that even at low Re, ribs
should be placed immediately after the bend.

SUMMARY

Computations were performed by using a validated CFD code to examine the flow and heat transfer in a ribbed U-shaped duct with and without rotation under operating conditions that are typical of industrial gas turbines. The Reynolds number investigated was 350,000, which represents an upper limit. The results generated were compared against those obtained by Stephens & Shih (1997) at a Reynolds number of 25,000 which represents a lower limit. Based on these results and their comparisons, the following conclusions can be made:

- At high Re and hence low Ro (Re = 350,000, Ro = 0.039), the two turbulence models (SST and explicit algebraic stress) investigated gave similar results for the general features of the flow and heat transfer. Quantitatively, they do differ up to 20% from shifts in the locations of maximums and minimums.
- At high Re (Re = 350,000), secondary flows induced by inclined ribs dominate over those induced by the bend but bend causes streamwise flow separation next to the inner wall as well as thin strips of highly retarded flow next to the leading and trailing walls.
- Ribs and/or pin fins are needed in the bend region to prevent flow separation and the strips of retarded flow in the bend region at high Re flows.
- At high Re and hence low Ro, rotation does not affect appreciably the flow but do affect the heat transfer characteristics in an appreciable manner, especially those on the leading face.
- At low Re and hence high Ro (Re = 25,000, Ro = 0.24), the secondary flows induced by inclined ribs are comparable in magnitude to the Dean-type secondary flow induced by the bend.
- At low Re, there are no streamwise flow separations in the bend so that heat transfer enhancement in the bend is reasonable except for a small strip on the leading face next to the inner wall.
- If flow is of interest, data generated from non-rotating ducts cannot be applied to rotating ducts in the bend and a short distance downstream of the bend (at least 5 ribs downstream) even when Re is as high as 350,000 with an angular speed of 3,600 rpm. At low Re, rotation has strong effects in the entire duct.
- If heat transfer are of interest, data generated from non-rotating ducts cannot be applied to rotating ducts, especially the leading face.

ACKNOWLEDGEMENT

This work was partially supported by NASA grant NAG 3-1727 from NASA - Lewis Research Center. The computer time was provided by the NAS facility. The authors are grateful for this support.

REFERENCES


Fig. 3. Cross-stream velocity vectors colored by normalized temperature (Re = 350,000, Ro = 0).
Fig. 4. Cross-stream velocity vectors colored by normalized temperature ($Re = 350,000, Ro = 0.039$).
Fig. 5. Normalized Nusselt number. Top: Ro = 0.24. Middle: Ro = 0.039. Bottom: Ro = 0.