PREDICTION OF TURBULENT FLOW AND HEAT TRANSFER IN A RIBBED RECTANGULAR DUCT WITH AND WITHOUT ROTATION

C. Prakash
General Electric Company – Aircraft Engines
Mail Drop A322; P.O. Box 156301
One Neumann Way, Cincinnati, OH 45215-6301

R. Zerkle
General Electric Company – Aircraft Engines
Mail Drop A322; P.O. Box 156301
One Neumann Way, Cincinnati, OH 45215-6301

ABSTRACT

The present study deals with the numerical prediction of turbulent flow and heat transfer in a 2:1 aspect ratio rectangular duct with ribs on the two shorter sides. The ribs are of square cross-section, staggered and aligned normal (90-deg) to the main flow direction. The ratio of rib height to duct hydraulic diameter equals 0.063, and the ratio of rib spacing to rib height equals 10. The duct may be stationary or rotating. The axis of rotation is normal to the axis of the duct and parallel to the ribbed walls (i.e., the ribbed walls form the leading and the trailing faces). The problem is three-dimensional and fully elliptic; hence, for computational economy, the present analysis deals only with a periodically-fully-developed situation where the calculation domain is limited to the region between two adjacent ribs. Turbulence is modelled with the k-epsilon model in conjunction with wall-functions. However, since the rib height is small, use of wall-functions necessitates that the Reynolds number be kept high. (Attempts to use a two-layer model that permits integration to the wall did not yield satisfactory results and such modelling issues are discussed at length). Computations are made here for Reynolds number in the range (30,000-100,000) and for Rotation number=0 (stationary), 0.06, and 0.12. For the stationary case, the predicted heat transfer agrees well with the experimental correlations. Due to the Coriolis induced secondary flow, rotation is found to enhance heat transfer from the trailing and the side walls, while decreasing heat transfer from the leading face. Relative to the corresponding stationary case, the effect of rotation is found to be less for a ribbed channel as compared to a smooth channel.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>a, b</td>
<td>length of the shorter (ribbed) and longer (smooth) walls</td>
</tr>
<tr>
<td>T, ( \tilde{T} )</td>
<td>temperature (equation 3)</td>
</tr>
<tr>
<td>u,v,w</td>
<td>velocity components in the x,y,z directions</td>
</tr>
<tr>
<td>B, C, ( C_1, C_2, C_\mu )</td>
<td>pressure and temperature gradients in the flow direction (Eqs. 2,3)</td>
</tr>
<tr>
<td>( c_p )</td>
<td>turbulence model constants</td>
</tr>
<tr>
<td>( c_p )</td>
<td>specific heat</td>
</tr>
<tr>
<td>D</td>
<td>hydraulic diameter ([=2ab/(a+b)])</td>
</tr>
<tr>
<td>E</td>
<td>wall roughness parameter</td>
</tr>
<tr>
<td>f</td>
<td>friction factor ([= BD/(2 \rho \bar{u}^2)])</td>
</tr>
<tr>
<td>G</td>
<td>rate of generation of turbulent kinetic energy</td>
</tr>
<tr>
<td>h</td>
<td>heat transfer coefficient</td>
</tr>
<tr>
<td>k</td>
<td>turbulent kinetic energy</td>
</tr>
<tr>
<td>m</td>
<td>mass flow rate through the duct</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number ([= hD/\lambda])</td>
</tr>
<tr>
<td>P</td>
<td>rib pitch, i.e. inter-rib spacing</td>
</tr>
<tr>
<td>Pr, ( Pr_t )</td>
<td>laminar and turbulent Prandtl numbers</td>
</tr>
<tr>
<td>q</td>
<td>heat flux</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number ([= \bar{u}D/\nu])</td>
</tr>
<tr>
<td>Ro</td>
<td>Rotation number ([= \Omega D/\nu])</td>
</tr>
<tr>
<td>s</td>
<td>local friction coefficient</td>
</tr>
<tr>
<td>St</td>
<td>local Stanton number</td>
</tr>
</tbody>
</table>

Copyright © 1993 by ASME
**1.0 INTRODUCTION**

1.1 Motivation

Effective turbine blade cooling is necessary to enhance the efficiency of advanced aircraft engines. In general, film cooling is imposed on the external surfaces of the blades, while forced-convection cooling is employed inside the blades by means of cooling passages as shown in Figure 1. The present study pertains to the latter, i.e., flow and heat transfer in the internal blade cooling passages.

To trip the boundary layers for promoting turbulence and thereby enhancing heat transfer, various kinds of turbulators (ribs) are usually provided on the walls of the blade cooling passages. An example of a ribbed passage is shown in Figure 2. The presence of the ribs leads to a complex flow field with regions of flow separation before and after the ribs. Further, since the turbine undergoes high rotation, the flow is strongly affected by the Coriolis and centrifugal forces. Needless to say, therefore, the flow field, and hence the heat transfer at the walls, is quite complex.

Detailed information about the flow distribution in a ribbed rotating passage can be very valuable for a designer engaged in deciding where, how many, and what kind of ribs to employ. However, obtaining such information experimentally can be very expensive, difficult, and often impossible.
ments with low-__Re__ model can be so demanding that several efforts are currently underway to develop the so-called two-layer models which seek to provide the benefits of a low-__Re__ model but on much coarser grid. Several two-layer models have been reviewed by Rodi [4] and it appears that for three-dimensional elliptic flows even these may require very fine grids.

If centrifugal–buoyancy is small, then the flow can become periodically–fully–developed (i.e., repeat cyclically). In such a situation, the computational domain is limited to the region between two adjacent ribs and it may be possible to provide an adequate grid for a low-__Re__ or a two-layer model. However, in practical situations, centrifugal buoyancy can be significant and it can lead to large flow reversals near the walls [1]. In such cases, the flow is not cyclically developed and one must analyze the entire channel with several ribs. For such a complete analysis, the grid required for the low-__Re__ and the two-layers models can indeed be very large, preventing economical computations on a routine basis.

When the ribs are large (high blockage ratio), and/or the Reynolds number is high, then the above difficulties do not arise and one may be able to use wall-functions and work with reasonable grid sizes. However, two additional considerations become important: (a) Secondary cross-stream flows due to three-dimensional effects can become significant. These flows are caused by turbulence anisotropic effects and require advanced turbulence models (of Reynolds-stress-type) for prediction. Such secondary flows, it should be pointed out though, may be less important in rotating channels as compared to stationary channels because of the swamping effect of the Coriolis flows in the former. (b) Multiple length scale effects can become important necessitating the use of advance turbulence models that acknowledge the presence of two or more length scales.

On the basis of the key issues discussed above, four categories of problems can be defined:

1. Low Reynolds number/low blockage ratio
2. Low Reynolds number/high blockage ratio
3. High Reynolds number/low blockage ratio
4. High Reynolds number/high blockage ratio

Categories 1 and 2 require a low-Reynolds-number turbulence model while for 3 and 4 the wall-function procedure may be adequate. Further, categories 2 and 4 may require Reynolds stress models to account for anisotropic effects and, possibly, multiple-length-scale models. Clearly, problems of type 2 are the most demanding, those of type 3 least demanding, while types 1 and 4 are of moderate difficulty.

Finally, rotation, and associated buoyancy effects, may require refinements to the turbulence models and possibly the near-wall treatments.

### 1.3 Scope of the Present Study

The purpose of the previous section is to highlight that the problem of predicting flow in turbine blade cooling passages is by no means easy. Hence, it would be unreasonable to expect a single effort that would yield a turbulence model that can address all the difficulties outlined above. Instead, progress will have to come in several steps, with each effort representing an advance over the previous one. There is no doubt, of course, that the intermediate steps, though limited in scope, will continue to add to our understanding of the various physical aspects of the flow. The work described in this paper is to taken in this spirit and regarded as a building block.

In terms of the classification provided in the previous section, the simulations presented in this paper correspond to category 3 (high Reynolds number/low blockage ratio . . . least demanding). Our intent here is to see how well the standard (isotropic) k-epsilon model performs in conjunction with the wall functions.

### 1.4 Literature Review

The literature pertaining to flow in rotating smooth ducts (no ribs) was reviewed extensively by the present authors in [1] and is not repeated here.

#### 1.4.1 Ribbed Passages – Theoretical

The original theoretical analyses dealt with the development of heat transfer and friction correlations using concepts similar to those used for rough tubes [5].

Due to the difficulties discussed in Section 1.2, numerical predictions of turbulent flow in ribbed passages are few and recent. Chang and Mills [6] employed a low-__Re__ turbulence model for a two-dimensional situation involving flow in a stationary circular tube with repeated rectangular ribs. This

---

**FIGURE 2. TURBULATED PASSAGE.**

- **Flow In**
  - **Rotating Axis Parallel to **y** Axis**
  - **Root Radius = 38.90 in.**

- **Flow Out**
  - **Rotating Duct**
    - **a = 0.429 in.**
    - **b = 0.855 in.**
  - **Ribs of Square Cross Section; Each Side e = 0.036 in.**
  - **Rib Spacing P = 0.36 in.**
  - **Hydraulic Diameter D = 0.571 in.**
  - **e/D = 0.63**
  - **Blockage Ratio = e/b = 0.0421**
  - **Duct Aspect Ratio = b/a = 2.0**
  - **System Pressure = 27.02 psi**
  - **Inlet Temperature = 135°F**
  - **Wall Temperature = 335°F**
  - **Re = 2 x 10⁴ - 1 x 10⁵**
  - **Ro = 0.12**

---
study provides an indication of the large grids that may be required by a low-Re model. Arman and Rabas [7] subsequently predicted flow and heat transfer in a stationary circular tube with repeated ribs using a two-layer model.

Recently Cunha [8] predicted flow and heat transfer in rotating circular tubes with ribs. A one equation (k – mixing length) turbulence model is employed. Near the wall, a wall-function type treatment is employed but use is made of experimentally determined roughness function expressions for ribbed channels. In this sense Cunha’s study is not fundamental – i.e., it uses experimentally determined velocity/temperature defect correlations at the walls instead of establishing them from the equations of fluid motion.

1.4.2 Ribbed Passages – Experimental

Due to the practical importance of the problem, there have been several experimental investigations aimed at determining the overall heat transfer and pressure drop in ribbed passages. Earlier efforts dealt with the stationary ducts while some of the recent studies examine the effect of rotation.

Stationary

Burgraff’s study [9] is an example of one of the earlier works. For the last several years, Prof. J.C. Han’s laboratory at Texas A & M has been engaged in obtaining heat transfer and pressure drop data in ducts of varying aspect ratio, rib height and orientation. A sampling of Han’s studies is provided by references [10–12]. Professor Taslim’s group at Northeastern University has been another source of experimental data as exemplified by reference [13].

Rotating

Experimental studies including the effect of rotation are more recent and fewer. The currently active groups are those of Dr. Johnson at United Technology Research Center [14–15] and of Dr. Taslim at Northeastern [16–17].

2.0 ANALYSIS

2.1 Background

The present study is directed by our goal to predict flow and heat transfer in the ribbed channel shown in Figure 2. This configuration is the result of discussions with turbine cooling designers, and includes parameters of practical interest. As shown, the channel consists of a long, smooth, rotating (but unheated) entrance length on top of which is perched the ribbed section. Clearly, within the ribbed part of the channel, the flow is in a developing mode both hydrodynamically and thermally. Further, in the range of operating parameters, both Coriolis and centrifugal–buoyancy effects are important.

2.2 The Problem Solved: Assumptions and Implications

Our final goal is to simulate the flow in the channel shown in Figure 2. However, the present analysis is an intermediate step, a rationale for which may be found in the discussion of sections 1.2 and 1.3. Here we focus our attention on a periodically–fully–developed situation which assumes that the flow repeats itself cyclically from one rib to the next. This assumption allows the calculation domain to be limited to the region between two adjacent ribs and the computational grid is reasonable. The calculation domain, and grid used, are shown in Figure 3.

As for a fully–developed duct flow, a periodically–fully–developed situation is intended to imply that conditions within a module are isolated from happenings outside the module either upstream or downstream. This requirement could be violated by centrifugal buoyancy effects if they lead to large scale flow reversals as was found in [1] for a smooth channel. For this reason, centrifugal–buoyancy effects are neglected in this analysis (i.e. density is assumed constant), and only Coriolis effects are considered. In the same spirit, though with lesser implications, is the requirement of constant thermo–physical properties (viscosity, conductivity, etc.) so that a periodically–fully–developed situation can be defined.

FIGURE 3. PROBLEM SCHEMATICS AND GRID (36 X 32 X 10 = 11,520 NODES).

If buoyancy effects are weak and do not lead to large scale flow reversals, then, under certain thermal boundary conditions (e.g. uniform wall heat flux), it may be possible to account for buoyancy within a periodically–fully–developed framework if certain assumptions are made (e.g. Boussinesq in conjunction with relative temperature differences within the computational module etc.). However, based on experience with a smooth channel [1], it is felt that centrifugal–buoyancy effects are best handled via the simulation of the flow in the complete channel with several ribs including careful modelling of the exit boundary conditions. Such a complete analysis is currently underway.
2.3 Governing Equations

The coordinate system shown in Figure 3 rotates with the duct. In such a system the flow is steady but account must be taken of the Coriolis and centrifugal forces which appear as additional source terms in the equations of motion. However, as already discussed, the fluid is assumed incompressible and buoyancy effects neglected. Hence, the centrifugal terms only modify the pressure and do not affect the flow field. As a result, as long as the pressure is interpreted as being the modified pressure, the centrifugal force terms need not be included in the governing equations. Further, as is common practice, we solve the time-averaged form of turbulent flow equations in conjunction with a suitable turbulence model [2,3,4]. The following equations pertain to the mean flow.

A comprehensive discussion of the periodically-fully-developed analyses is provided by Patankar, Liu, and Sparrow in [18] and all details are not repeated here. Basically, all flow variables repeat cyclically over the length of the computational module (the ribs spacing \( P \)). Thus:

\[
\phi (x,y,z) = \phi (x+P,y,z) \quad (1)
\]

where \( \phi \) could be any velocity component \((u,v,w)\), the turbulent kinetic energy \((k)\) or the turbulent dissipation rate \((\epsilon)\). Further, the pressure can be decomposed as

\[
p = -Bx + \tilde{p} \quad (2)
\]

where \( \tilde{p} \) is cyclic (i.e. abides by equation (1)) while the term \(-Bx\) is related to the net pressure loss (due to friction, form-drag, etc.) over the computational module.

In a periodic analysis, the flow rate is not known a priori. Instead, one specifies \( B \) and iteratively adjusts it to get the desired flow rate.

Patankar et al. [18] also discuss the formulation of the thermal problem in a periodic framework. The case of uniform wall temperature is somewhat difficult because it leads to an eigenvalue problem. Therefore, taking advantage of the fact that boundary conditions have a lesser effect on turbulent heat transfer coefficients than laminar ones, we consider the simpler case of uniform heat flux at all wall surfaces (ribs as well as the walls). For such a case, the temperature field can be expressed as

\[
T = Cx + \tilde{T} \quad (3)
\]

where \( \tilde{T} \) is cyclic (i.e., follows equation (1)) while the first term is related to the net heat gain and can be obtained from an overall heat balance, i.e.

\[
C = Q / (m c_p P) \quad (4)
\]

where \( Q \) is the total heat input over the module, \( m \) the through flow rate, and \( c_p \) the specific heat.

With the above considerations, the governing equations are:

**Continuity:**

\[
\text{div} (\vec{u}) = 0 \quad (5)
\]

**x-momentum:**

\[
\rho \text{ div} (\vec{u} u) = -\frac{\partial \tilde{p}}{\partial x} + B - 2\rho \Omega w + \text{div} [(\mu + \mu_t) \text{ grad} (u)] \quad (6)
\]

**y-momentum:**

\[
\rho \text{ div} (\vec{u} v) = -\frac{\partial \tilde{p}}{\partial y} + \text{div} [(\mu + \mu_t) \text{ grad} (v)] \quad (6)
\]

**z-momentum:**

\[
\rho \text{ div} (\vec{u} w) = -\frac{\partial \tilde{p}}{\partial z} + 2\rho \Omega u + \text{div} [(\mu + \mu_t) \text{ grad} (w)] \quad (8)
\]

**energy:**

\[
\rho \text{ div} (\vec{u} T) = -\rho u C + \text{div} \left[ \left( \frac{\mu}{Pr_t} + \frac{\mu_t}{Pr_t} \right) \text{ grad} (T) \right] \quad (9)
\]

In the above, the symbols (div) and (grad) designate the divergence and the gradient operators. The variable \( \mu_t \) represents the turbulent viscosity and \( Pr_t \) designates the turbulent Prandtl number.

To close the above system of equations, expressions need to be provided for the turbulence viscosity \( \mu_t \) and the turbulent Prandtl number \( Pr_t \). The turbulent Prandtl number, \( Pr_t \), is usually constant and is taken = 0.86 here. However, turbulent viscosity varies throughout the flow field and requires a more elaborate turbulence model. This issue will be discussed next.

2.4 Turbulence Model I: The Two-Layer Model

A very popular and practical turbulence model is the two equation \( k-\epsilon \) model in conjunction with the wall functions [2]. This model will be discussed in the next section. A difficulty
with this model is that grid points should not get too close to the wall. Specifically, this model requires that the y+ corresponding to the near-wall nodes should be greater than ~ 30.

In our case the ribs are small, and our intention is to capture the details of the flow around the ribs. Hence, the grid around the ribs has to be of some modest size. The problem then arises that unless the Reynolds number is sufficiently high the y+ corresponding to the near-wall nodes become too small (i.e. < 30). Indeed, it is this issue which was discussed at length in section 1.2.

Recognizing this difficulty up-front, we started this project using a two-layer model of turbulence which is free of the above near-wall problems. Like the low-Re-models [3], the two-layer models permit integration to the wall, but require a much coarser grid. We employed a two-layer model similar to the one used by Rodi’s group [4].

The predictions with the two-layer model looked quite good for a two-dimensional stationary ribbed channel, i.e. for a case with no side-wall in the y-direction (recal Figure 2). Indeed, the predicted heat transfer agreed quite favorably with Han’s correlation obtained from experiments on two-dimensional (a/b – 12) parallel plate channels [10].

However, when the two-layer model was applied to the three-dimensional situation, it did not yield satisfactory results. In particular, the heat transfer from the shorter side (the ribbed side) was much under-predicted. The flow field also showed some undesirable features including too long a recirculation zone behind the ribs.

We are currently exploring ways of improving the two-layer model predictions. In the interim, the two-layer model results are not included in this paper. Instead, we focus here on the predictions with the standard k-ε model with wall functions, and stay at the higher end of Reynolds numbers for which the near-wall y+ problem is less severe.

2.5 Turbulence Model II: The Two-Equation k-ε Model

2.5.1 Core Equations The results presented in this paper were obtained using the k-ε model where the turbulence viscosity, µt, is obtained from the turbulent kinetic energy (k) and rate of dissipation of turbulent energy ε [2]. Specifically:

\[ \mu_t = \rho C_\mu k^{2/\epsilon} \]  \hspace{1cm} (10)

k and ε are obtained from the following transport equations.

\[ \text{div} (\rho \mathbf{u} k) = \text{div} \left( \left( \mu_t / \sigma_k \right) \text{grad} (k) \right) + G - \rho \varepsilon \]  \hspace{1cm} (11)

\[ \text{div} (\rho \mathbf{u} \varepsilon) = \text{div} \left( \left( \mu_t / \sigma_\varepsilon \right) \text{grad} (\varepsilon) \right) + C_1 \varepsilon G/k - C_2 \rho \varepsilon^{3/2} k \]  \hspace{1cm} (12)

where:

\[ G = \mu_t \left\{ 2 \left[ \left( \frac{\partial u}{\partial x} \right)^2 + \left( \frac{\partial v}{\partial y} \right)^2 + \left( \frac{\partial w}{\partial z} \right)^2 \right] + \left[ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right]^2 \right\} \left[ \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right]^2 \]  \hspace{1cm} (13)

The various constants above have the following values:

\[ \sigma_k = 1.0; \quad \sigma_\varepsilon = 1.314; \quad C_\mu = 0.09 \]

\[ C_1 = 1.44; \quad C_2 = 1.92 \]

2.5.2 Wall-Functions The k-ε equations provided above apply in the fully turbulent regions away from the walls where laminar effects can be neglected. To relate this outer solution to the wall, the relevant equations have to be integrated across the viscous sublayer. The wall-functions, e.g. the log-law of the wall, are the outcome of such semi-empirical integrations. The purpose of these functions is to relate the wall shear and heat flux to the velocity and the temperature difference between the wall and the near-wall node [2].

We use the generalized wall function treatment of Rosten and Worrell [19] details of which are provided in Appendix A. The following features of the method may be noted:

a. The friction velocity (u_f = V/\sqrt{s}) is replaced by a velocity scale calculated from the local turbulent kinetic energy at the near wall point. Here u_f designates the friction coefficient and V the magnitude of the tangential velocity at the near wall point.

b. The kinetic energy at the near wall point is deduced from the regular transport equation with zero normal gradient at the wall. The generation term for the wall cells is calculated by an analytical integration based on the wall shear stress.

c. The dissipation rate for the near wall cells is fixed to an average obtained from analytical integration.

d. The wall heat transfer is expressed using Jayatillika’s [20] Stanton number correlation.

e. Most importantly: if the near-wall point lies in the viscous sublayer, laminar expressions are used for the friction coefficient and the Stanton number.

2.6 Boundary Conditions

No inlet and exit conditions are required for a periodically-developed analysis. At the walls, the no slip condition is used in conjunction with the above wall-functions. For the
temperature equation, since the heat flux is prescribed at the wall, the wall–functions are used to determine the wall temperature from the computed near-wall temperature. The difference between the wall temperature so determined, and the local bulk temperature, is then used to compute the local heat transfer coefficient.

2.7 Dimensionless Parameters

The geometrical parameters were kept fixed corresponding to Figure 2. The Prandtl number was taken = 0.7 corresponding to air. The Reynolds number was varied in the range (30,000 – 100,000) and the Rotation number had values = 0 (stationary), 0.06 and 0.12. (see Nomenclature for the definition of the parameters).

2.8 Numerical Details

The control–volume–based finite difference method described by Patankar [20] was employed in the present study. It uses the primitive variables (u,v,w,p) as unknowns, a staggered grid, and the pressure correction algorithm. The iterative line-by-line TDMA was used to solve the discretized equations with a cyclic–TDMA employed in the x–direction [18].

The grid deployed is shown in Figure 3. There were 36 nodes in the main flow direction (x) of which 6 extend over the rib and 30 lie in the inter-rib space. In the direction normal to ribbed walls (z), there is a total of 32 cells disposed symmetrically (16 each) around the mid-height (z=b/2); of these, 6 cells extend over the rib height e, 4 cover an additional height e over the top of the rib, and 6 cover the remaining distance up to the mid-height. In the direction normal to the smooth side wall (y), a total of 10 cells is used. In total, there were 11,520 nodes in this simulation.

Numerical computation of periodically-fully-developed flow is rendered difficult by the fact that no boundary information is available in the main flow direction along which the discretization coefficients are largest. Partly due to this reason, the code took ~1500 iterations of the SIMPLER algorithm [20] for convergence. On Cray–YMP, this translated to ~30 minutes of CPU time.

Grid Refinement: In the present problem, most of the real action is around the ribbed walls. However, due to the conflicting requirement of keeping near wall y+ reasonably large, the grid could not be refined excessively around the ribs. Far from the walls, i.e. in the core, the variables do not vary much and, hence, an excessively fine grid is not necessary. The grid shown in Figure 3 is a result of all these practical considerations. Limited numerical experimentation suggested the grid used to be fairly adequate.

2.9 Smooth Channel Predictions

For a smooth channel, the periodically-fully-developed analysis is same as the simple fully-developed flow in a duct. For each ribbed case considered, the corresponding smooth channel case was also computed for comparison.

3.0 RESULTS I: THE STATIONARY CASE

3.1 The y+ Issue

As already indicated, for the use of the k–ε model with wall functions, the near-wall nodes should be sufficiently far from the wall; specifically, y+ > 30 is recommended. However, in most practical flows, this condition cannot be guaranteed at all points in the computational domain. For instance, in recirculating flows, the y+ values become quite small around the reattachment point. Also, in three-dimensional duct flows, the velocities are quite small near the corners where, again, y+ becomes small. The stance normally taken in these situations is that use of wall–functions is probably not very harmful as long as the locations of low y+ are not too many.

In the present problem, with small ribs, even a modest grid around the ribs leads to the potential for low y+ at the near-wall nodes. To keep this issue in focus, the range of y+ encountered on different surfaces is listed below:

<table>
<thead>
<tr>
<th>Surface</th>
<th>y+ Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top face of ribs</td>
<td>15–20</td>
</tr>
<tr>
<td>Back face of ribs</td>
<td>7–10</td>
</tr>
<tr>
<td>Front Face of Ribs</td>
<td>10–20</td>
</tr>
<tr>
<td>Inter-rib Floor Space</td>
<td>5–15</td>
</tr>
<tr>
<td>Smooth Side-Wall</td>
<td>20–50</td>
</tr>
</tbody>
</table>

The above numbers are representative of the values over most of the listed surfaces. As already noted, lower values cannot be avoided in local regions such as the reattachment line and the corners.

The reader's attention is again drawn to the laminar limits imposed on the wall–function expressions (Appendix A) to take account of the possibility of low y+. Also it should be mentioned that y+ > 30 is general recommendation though values of y+ of ~15 are considered not too objectionable.

Our experience has been that if the y+ is pushed too low, then the effect is generally to overpredict both friction and heat transfer.

3.2 The Flow Field

Figure 4 shows the flowfield in the vicinity of the ribs. As can be noted, the flow is complex and consists of recirculation zones before and after the ribs.

3.3 Heat Transfer

3.3.1 Definition The local heat transfer coefficient is defined as:

\[ h = \frac{q}{(T_w - T_b)} \] (14)

where \( q \) is the heat flux (uniform), \( T_w \) the wall temperature (varying over the surface and computed using the wall–functions and near–wall node temperature), and \( T_b \) the bulk temperature at the same location.
The average heat transfer coefficient over a surface is defined as:

$$h = \frac{1}{A_t} \int h \, dA$$

where $A_t$ is the true heat transfer area, while $A_p$ (not necessarily equal to $A_t$) is the projected base area.

All surfaces of the ribs as well as the inter-rib floor space are used in computing the average heat transfer over the ribbed faces, i.e., $A_t = (aP + 2e_a)$. However, the projected base area $A_p$ is taken to be just the base area, i.e. $A_p = aP$.

Likewise, in the smooth side-wall, $A_t = (bP - 2e_2)$ while $A_p = bP$.

Note that due to the use of the projected base area, the above heat transfer coefficient embodies within it heat transfer augmentation due to both factors, namely (i) additional heat transfer area provided by the ribs, and (ii) enhancement caused by the ribs by perturbing the flow and the turbulence fields.

### 3.3.2 Heat Transfer Augmentation By The Ribs

Computed heat transfer results for the stationary duct are shown in Figure 5.

Consider the smooth channel case. Since the longer sides (side-b) are closer to each other and the core, it is expected that the temperature gradients will be higher on these compared to the shorter sides (side-a). Hence, the heat transfer coefficient on the longer side is expected to be higher than on the shorter side as the results of Figure 5 indicate. It should be noted though that the predicted difference (~ 30%) is higher than expected. The expected difference is ~ $2^{0.2} - 1 = 15\%$, which is based on the application of the correlations $Nu_b - Re_b^{0.8}$ and $Nu_a - Re_a^{0.8}$ to the sides individually. Apparently, the $k - \varepsilon$ model over-predicts heat transfer on the longer sides and under-predicts it on the shorter side (with the fortunate consequence that the average for the channel is predicted well as Figure 6 will show). It would be appropriate to mention here that with the two-layer model of turbulence, the difference in average heat transfer coefficient for the two sides was found to be even larger.

For a ribbed channel, the above situation changes. The effect of ribs is to significantly enhance (more than double) the heat transfer from the face on which they are employed (the shorter side-a). The ribs also increase the heat transfer from the smooth side-walls (the longer side-b) but the enhancement (~ 20-30%) is much less than for the ribbed wall. As a result, for the ribbed channel, the shorter side (the ribbed wall) has a higher heat transfer coefficient than the longer side (the smooth side-wall).
3.3.3 Comparison  The computed results for the smooth and the ribbed channels are compared, in Figures 6–9, with the correlations proposed by Han et al. [11] on the basis of extensive experimental data. As can be noted, the agreement is good.

FIGURE 7. STATIONARY RIBBED CHANNEL; VARIATION OF AVERAGE NUSSELT NUMBER FOR THE RIBBED SIDE.

FIGURE 8. STATIONARY RIBBED CHANNEL; VARIATION OF AVERAGE NUSSELT NUMBER FOR THE SMOOTH SIDE–WALL.

It must be pointed out that care was taken in using Han’s correlation [11] to ensure that the expressions were consistent with the use projected area in the definition of the average heat transfer coefficient (Equation 15).

3.4 Friction Factor

The computed friction factors are presented and compared with Han’s [11] correlation in Figure 10. The predictions correctly indicate the increase of friction by the ribs. The levels of enhancement are also correctly predicted.

For the ribbed case, agreement with Han is only reasonable within $\pm 25\%$. The present predictions indicate a friction factor that is nearly independent of the Reynolds number which is typical of a rough pipe when the wall friction losses are small compared to losses due to form–drag etc.

4.0 RESULTS II: EFFECT OF ROTATION

4.1 Comments

Attention will now be turned on examining the effect of rotation on flow and heat transfer. However, since the present cyclic analysis only considers Coriolis effects (i.e. buoyancy and entrance effects are not accounted for), a meaningful comparison with data is not possible and hence not made.

4.2 Flow Field

Recall Equation (8) and note the Coriolis term $(2p\Omega u)$ on the right hand side. Since the through–flow velocity $u$ is higher in the core and smaller near the walls, the consequence of this body force is to create a cross–stream flow which is directed from the leading to the trailing wall near the core and from the trailing to the leading wall near the side–wall.

Figures 11(a) and 11(b) show the Coriolis induced cross–stream flow for a smooth channel. At low Rotation numbers, the flow is characterized by two symmetric vortices while four–vortex solutions arise at higher rotation numbers. These two– and four–vortex solutions were well discussed by Launder and Iacovides [21]. For $Ro=0.06$, two–vortex solu-
tions occurred at lower Reynolds numbers with a switch to four-vortex solution at higher Reynolds numbers. For \( Ro = 0.12 \), four-vortex solutions were found for all Reynolds numbers considered.

The Coriolis induced cross-stream flow for the ribbed channel are shown in Figures 12(a) and 12(b). The non-rotating cases are included for comparison. The vectors near the ribs also indicate the state of flow as it crosses over the ribs. For the range of parameters, no four-vortex solutions were found for a ribbed passage.

### 4.3 Heat Transfer

The Coriolis induced secondary flow transports cooler, faster moving, fluid from the core to the trailing face and hence heat transfer there is enhanced compared to a non-rotating case. For similar reasons, heat transfer is also enhanced at the side-wall. However, the rotating fluid gets warmer and slower by the time it gets to the leading wall with the result that heat transfer is decreased relative to a non-rotating case.

From Figures 11–12, and general intuition, it is to be expected that the Coriolis effects are more pronounced in a smooth channel as compared to a ribbed channel. The reason is simply that in a ribbed channel there is much going on already in the inter-rib space (separation, re-attachment, etc.) which prevents Coriolis effects from reaching the walls.

The above features are evidenced by Figures 13–16 where the ratio

\[
\frac{Nu_{\text{rotating}}}{Nu_{\text{stationary}}}
\]

is presented on the different walls.

From Figure 13, the heat transfer enhancement at the trailing wall can be noted. Also, as discussed above, the effect of rotation, relative to the stationary case, is less for a ribbed

**FIGURE 11.** ROTATING SMOOTH CHANNEL; CORIOLIS INDUCED CROSS-STREAM FLOW; \( RE = 3.232 \times 10^4 \); (A) \( RO = 0.06 \), TWO-VORTEX SOLUTION; (B) \( RO = 0.12 \), FOUR-VORTEX SOLUTION.

**FIGURE 12.** (A) RIBBED-CHANNEL; CROSS-STREAM FLOW OVER THE RIB ON THE LEADING WALL. \( RE = 3.232 \times 10^4 \). (B) RIBBED-CHANNEL; CROSS-STREAM FLOW OVER RIB ON THE TRAILING WALL. \( RE = 3.232 \times 10^4 \).
FIGURE 13. EFFECT OF ROTATION ON AVERAGE NUSSLET NUMBER AT TRAILING WALL.

channel compared to a smooth channel. For a fixed rotation number, increasing the Reynolds number decreases the Nusselt number ratio (rotating/stationary) at the trailing face. The only exception to this statement is the smooth channel case for Ro = 0.06 where the rise occurred due to the switching of the solution from a two-vortex to four-vortex flow at higher Reynolds numbers [recall Figure 11] with associated increase in heat transfer [21].

The effect of rotation in reducing heat transfer at the leading face may be noted from Figure 14. Compared to the trailing face, the heat transfer ratio (rotating/stationary) for the leading face appears to be less sensitive to the parameters over the range considered — the ratio remaining largely around ~ 0.8.

Rotation increases the heat transfer at the smooth side-wall also as is evidenced by Figure 15. The increase, again, is more for a smooth channel as compared to a ribbed channel.

Since the heat transfer is enhanced at the trailing face and decreased at the leading face, there is a cancelling effect when the average heat transfer for the entire channel is computed. This can be seen in Figure 16 where the ratio (rotating/stationary) for the average Nusselt number is found to be closer to unity.

4.4 Friction Factor

The ratio of rotating to stationary friction factors is presented in Figure 17.

For the smooth channel, wall friction is the primary loss mechanism and hence rotation has a noticeable effect on the friction factor.

For the ribbed channel, wall friction is less important since form—drag etc. make a major contribution to the pressure loss. Hence, since over the range of Rotation number considered the Coriolis flow has a rather weak influence on the separated flow

FIGURE 14. EFFECT OF ROTATION ON AVERAGE NUSSLET NUMBER AT THE LEADING WALL.

around the ribs, rotation is found to have a small effect on the friction factor.

5.0 CONCLUDING REMARKS

5.1 Summary of Results

One purpose of the present paper is to highlight the turbulence modeling issues related to the numerical prediction of flow in turbine blade cooling passages. It is discussed that the problems can be categorized into four groups, namely (1) low Reynolds number/low blockage ratio, (2) low Reynolds number/high blockage ratio, (3) high Reynolds number/low blockage ratio, and (4) high Reynolds number/high blockage ratio. Categories 1 and 2 require a low-Reynolds-number turbulence model while for 3 and 4 the wall-function treatment may be adequate. Likewise, an isotropic k-epsilon model may be adequate for types 1 and 3 while a Reynolds-stress turbulence model may be needed for 2 and 4 in order to account
Rotating

a. The Coriolis induced secondary flows and their effect on heat transfer at the different walls (enhancement at the trailing and side walls, and decrease at the leading wall) is correctly predicted. It is also shown that the effect of rotation on the heat transfer is less for a ribbed channel compared to a smooth channel [16, 17].

b. Unfortunately, since the present cyclic analysis does not include entrance and buoyancy effects, a meaningful comparison with the data has not been possible. For such a comparison, the entire channel (including several ribs) must be simulated, and such a study is currently in progress.

5.2 What's Next

a. The present analysis needs to be performed over a wide range of duct aspect ratios and rib heights to determine the range of blockage ratios over which the isotropic k-epsilon model with wall-functions may be adequate.

b. The exercise should be extended for a full channel analysis including several ribs so that the effect of buoyancy can be studied.

c. As discussed, a computationally affordable low-Reynolds number model is deemed to be necessary, and efforts towards development of such models need no defending.

d. Finally, to capture anisotropic effects, a Reynolds stress model is required.

We are currently involved in working on all of the aspects listed above.

ACKNOWLEDGEMENTS

The authors wish to express their appreciation to Dr. Mikio Suo for his guidance and support to our CFD methods development activities in gas turbine heat transfer at GE Aircraft Engines. We also wish to thank Mr. John Starkweather for his critical review and analysis of our work from the perspective of a turbine cooling designer.

REFERENCES


APPENDIX A

Generalized Wall Functions [19]

Consider Figure A-1, which shows a near-wall grid point P located at a distance δ from the wall. Let $V_p$ and $k_p$ represent the tangential velocity and the turbulent kinetic energy at the point P. The shear stress at the wall is obtained as:

$$\frac{T_w}{\rho} = s V_p^2 \quad \text{(A1)}$$

where $s$ is the friction coefficient given by

$$s = \text{greater of } \left\{ \begin{array}{ll}
k \frac{C_{1/4}^{1/2} k_p^{1/2}}{V_p \ln (E \delta^{1/4} k_p^{1/2}/\nu)} & \text{turbulent limit (S_{turb})} \\
\nu / V_p \delta & \text{laminar limit (S_{lam})} \end{array} \right. \quad \text{(A2)}$$

In the above,

$k$: Von-Karman constant = 0.435

and

$E$: Wall roughness parameter = 9 (smooth wall)
The turbulent kinetic energy, $k_p$, is obtained via the differential equation for $k$ with the zero-gradient condition imposed at the wall.

The generation rate of $k$ at the point $p$ is computed as:

$$G_p = \rho \left( s V_p^2 \right) V_p$$  \hspace{1cm} (A3)

The dissipation rate at the point $P$ is fixed as:

$$G_p = \frac{C_p^{3/4} k_p^{3/2} \ln \left( E \delta \frac{C_p^{1/4} k_p^{1/2}/\nu}{k_p} \right)}{k_p}$$  \hspace{1cm} (A4)

The heat flow at the wall is given by

$$q_w = (St p V_p) c_p (T_w - T_p)$$  \hspace{1cm} (A5)

where $T_w$ and $T_p$ designate the temperature at the wall and the point $P$ respectively, and $St$ is the Stanton number given by

$$St = \text{greater of}$$

$$\left[ \frac{St_{\text{tur}}}{Pr_t \left( 1 + \frac{s_{\text{tur}} V_p}{C_p^{1/4} k_p^{1/2}} \right)} ; \frac{V_p}{V_p \delta (Pr)} \right]$$

$$\text{turbulent} \hspace{2cm} \text{laminar limit}$$

(A6)

In the above, the P-function is given by

$$P = 9 \left( \frac{Pr}{Pr_t} - 1 \right) \left( \frac{Pr}{Pr_t} \right)^{1/4}$$  \hspace{1cm} (A7)