Direct numerical simulations of turbulent flow through a stationary and rotating infinite serpentine passage

Gregory M. Laskowski
Stanford University

Paul A. Durbin
Iowa State University, durbin@iastate.edu

Follow this and additional works at: http://lib.dr.iastate.edu/aere_pubs
Part of the Aerospace Engineering Commons, and the Mechanical Engineering Commons

The complete bibliographic information for this item can be found at http://lib.dr.iastate.edu/aere_pubs/53. For information on how to cite this item, please visit http://lib.dr.iastate.edu/howtocite.html.
Direct numerical simulations of turbulent flow through a stationary and rotating infinite serpentine passage

Gregory M. Laskowski and Paul A. Durbin

Citation: Physics of Fluids 19, 015101 (2007); doi: 10.1063/1.2404940
View online: http://dx.doi.org/10.1063/1.2404940
View Table of Contents: http://scitation.aip.org/content/aip/journal/pof2/19/1?ver=pdfcov
Published by the AIP Publishing

Articles you may be interested in

Lagrangian statistics of turbulent channel flow at Reτ = 950 calculated with direct numerical simulation and Langevin models

Direct numerical simulation of a high-Froude-number turbulent open-channel flow

Direct numerical simulation of turbulence in injection-driven plane channel flows
Phys. Fluids 20, 105103 (2008); 10.1063/1.2963137

Interface-resolved direct numerical simulation of vertical particulate channel flow in the turbulent regime

Comparison of direct numerical simulations and particle-image velocimetry data of turbulent channel flow rotating about the streamwise axis
Phys. Fluids 19, 085114 (2007); 10.1063/1.2760278

This article is copyrighted as indicated in the article. Reuse of AIP content is subject to the terms at: http://scitation.aip.org/termsconditions. Downloaded to IP: 129.186.176.40 On: Wed, 26 Aug 2015 15:14:13
Direct numerical simulations of turbulent flow through a stationary and rotating infinite serpentine passage

Gregory M. Laskowski and Paul A. Durbin
Department of Mechanical Engineering, Stanford University, Stanford, California 94305

(Received 14 February 2006; accepted 1 November 2006; published online 4 January 2007)

Serpentine passages are found in a number of engineering applications including turbine blade cooling passages. The design of effective cooling passages for high-temperature turbine blades depends in part on the ability to predict heat transfer, thus requiring an accurate representation of the turbulent flow field. These passages are subjected to strong curvature and rotational effects, and the resulting turbulent flow field is fairly complex. An understanding of the flow physics for flows with strong curvature and rotation is required in order to improve the design of turbine blade cooling passages. Experimental measurements of certain turbulence quantities for such configurations can be challenging to obtain, especially near solid surfaces, making the serpentine passage an ideal candidate for a direct numerical simulation (DNS). A DNS study has been conducted to investigate the coupled effect of strong curvature and rotation by simulating turbulent flow through a fully developed, smooth wall, round-ended, isothermal serpentine channel subjected to orthogonal mode rotation. The geometry investigated has an average radius of curvature $R_c/\delta = 2.0$ in the curved section and dimensions $12\pi\delta \times 2\delta \times 3\pi\delta$ in the streamwise, transverse, and spanwise directions. The computational domain consists of periodic inflow/outflow boundaries, two solid wall boundaries, and periodic boundaries in the spanwise direction. The simulations were conducted for Reynolds number, $Re_\delta = 5600$, and rotation numbers, $Ro_{\beta z} = 0$ and 0.32. Differences observed between the stationary and rotating cases are discussed in terms of the mean velocity, secondary flow, and Reynolds stresses. © 2007 American Institute of Physics. [DOI: 10.1063/1.2404940]

I. INTRODUCTION

Gas turbine thermal efficiency is a strong function of the turbine inlet temperature. The temperature entering the turbine is typically much higher than the melting point of the turbine blade metal. In order to ensure that the structural integrity of turbine blades is not compromised at high temperatures, elaborate cooling strategies are required. One approach is to extract relatively low-temperature fluid from the compressor and circulate it through the turbine blade internal serpentine passages. The design of effective and efficient turbine blade thermal management systems is strongly rooted in the ability to predict the turbulent flow field that exists in the internal passage. Analysis of these designs is typically accomplished with Reynolds-averaged Navier-Stokes (RANS) models and can be quite challenging due to strong curvature and rotational effects. For even simple flow geometries, curvature and rotation can result in very complex flow physics, which can prove challenging to model. For example, consider two variants of the spanwise periodic plane channel flow:1 moderate curvature and spanwise rotation. Curved channel flow may give rise to secondary flow or longitudinal vortex pairs,2,3 due to curvature-induced centrifugal instabilities, and spanwise rotating straight channel flow may give rise to similar vortical structures4 due to rotational-induced Coriolis instabilities. This secondary flow can impact the Reynolds stresses. Independently, moderate curvature or rotation can have a stabilizing effect on the flow near the convex surface or suction side, respectively, and a destabilizing effect on the flow near the concave surface or pressure side, respectively. For a curved channel subjected to spanwise rotation, the coupled effect of centrifugal forces due to curvature and Coriolis forces due to rotation can have a stabilizing or destabilizing effect on the flow field.5

In order to better understand the effects of strong curvature and separation, numerous experiments investigating the turbulent flow field in stationary U-bends have been conducted.6-9 These experiments have served as validation databases for RANS computations with eddy viscosity models (EVM), algebraic stress models (ASM), and full second moment closure models (SMC).10-19 While curvature-corrected EVM’s have met with some success,17,20 it is generally recognized that SMC models are required to capture the strong anisotropy and secondary flow that exists in such flows. The success of a particular turbulence model depends on available databases for their design, calibration, and validation.

Fewer investigations of flow subjected to both strong curvature and rotation for smooth wall U-bends or serpentine passages have been reported in the literature.8,9,21,22 In one computational study,11 it was reported that low-Re ASM simulations predicted good agreement when the curvature and Coriolis forces reinforce each other and resulted in poor

---

aPresent address: Computational Heat Transfer and Aeroacoustics Lab, GE Global Research, Niskayuna, NY 12309. Electronic mail: laskowski@research.ge.com

bPresent address: Department of Aerospace Engineering, Iowa State University, Ames, IA 50011.
agreement when the curvature and Coriolis forces oppose one another.

In order to gain additional insight into the coupled effect of rotation and curvature, direct numerical simulations for a fully developed, smooth wall, isothermal, serpentine passage have been conducted. While the mode of rotation is a deviation from the true operating environment of turbine blade cooling passages, the present configuration provides an opportunity to study the coupled effect of curvature and rotation. This is due in part to the geometric simplicity and periodicity between inflow/outflow boundary conditions. Because of the latter, the channel is an infinite serpentine passage. The geometry under investigation is $12/\delta$, with an average curvature ratio $R_c/\delta = 2.0$ based on the channel half-width $\delta$. The nondimensional parameters based on the bulk velocity and channel width are the Reynolds number $Re_b = 5600$ and $Re = 180$, and the rotation numbers $Ro_b = 0$ and $0.32$, where $Ro_b = 2\theta U_b/\nu$ and $Ro = 2\theta \Omega/\nu$. Results are presented for mean velocity field, Reynolds stresses, secondary flow, and skin friction coefficient for the stationary and rotating cases. The momentum, turbulent kinetic energy, and Reynolds stress budgets were collected and will be the focus of a separate paper.

**II. NUMERICAL DETAILS**

The governing equations are the incompressible, isothermal Navier-Stokes equations,

\[
\frac{\partial u_i}{\partial x_i} = 0,
\]

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial u_i}{\partial x_j} \right) + \frac{1}{\rho} f_i,
\]

where $\nu$ is the kinematic viscosity, $\nu = \mu/\rho$, and $f_i$ is a body force. The Coriolis force is

\[
\vec{f} = -2\rho \Omega \times \vec{u}
\]

and a conservative, centripetal term is absorbed into the pressure gradient. Following Krisoffersen and Anderssen, $p$ in Eq. (1b) is now the effective pressure. Equations (1) are non-dimensionalized by the channel half-width $\delta$ and average friction velocity $u_f$.

**TABLE I. Computational grids for the $Re_b=5600$ case.**

<table>
<thead>
<tr>
<th>C1</th>
<th>M1</th>
<th>F1</th>
<th>F2</th>
<th>F3</th>
</tr>
</thead>
<tbody>
<tr>
<td>192 $\times$ 32 $\times$ 64</td>
<td>384 $\times$ 64 $\times$ 128</td>
<td>768 $\times$ 128 $\times$ 256</td>
<td>1536 $\times$ 128 $\times$ 256</td>
<td>768 $\times$ 256 $\times$ 256</td>
</tr>
</tbody>
</table>

**FIG. 1.** Stationary turbulent channel flow results in wall units. (a) Mean velocity profile. (b) Mean rms velocity fluctuation profiles.

**FIG. 2.** Rotating turbulent channel results. (a) Mean velocity profile. (b) Mean turbulent kinetic energy profile.
\[ u_{\text{mp}} = \frac{1}{S} \int_{s_0}^{s_{\text{max}}} u_{\text{mp}}(s) \, ds; \quad u_{\text{m}} = \frac{1}{S} \int_{s_0}^{s_{\text{max}}} u_{\text{m}}(s) \, ds \]

and

\[ u_{\text{m}}^2 = \frac{1}{2} (u_{\text{m}}^2 + u_{\text{mp}}^2), \]

(3)

resulting in the nondimensional Reynolds and rotation numbers,

\[ \text{Re}_r = \frac{u_\tau \delta}{\nu}, \quad \text{Ro}_r = \frac{2 \Omega \delta}{u_\tau}. \]

(4)

The nondimensional form of Eqs. (1a) and (1b) can now be expressed as

\[ \frac{\partial \overline{u}_i}{\partial \overline{x}_i} = 0, \]

(5a)

\[ \frac{\partial \overline{u}_i}{\partial \overline{t}} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial \overline{x}_j} = - \frac{\partial \overline{p}}{\partial \overline{x}_i} + \frac{1}{\text{Re}_r} \overline{u}_j \frac{\partial^2 \overline{u}_i}{\partial \overline{x}_j \partial \overline{x}_j} + \overline{f}_i, \]

(5b)

where \( \overline{f}_i = -e_{ijk} \text{Ro}_r \overline{u}_k \) and \( \overline{p}_{\text{eff}} = \overline{p} - \frac{1}{8} (\text{Ro}_r^2 \overline{r}^2) \) for system rotation about the span axis. Equations (5a) and (5b) are solved with a modified version of the DNS code originally developed by Wu and Durbin, which is based on the fractional step, generalized coordinate finite volume algorithm of

FIG. 3. (a) Computational grid for M1 Re=5600 serpentine passage simulations. (b) Definitions of geometric terminology used throughout.

FIG. 4. Grid resolution downstream of the U-bend inner wall.
The dependent variables are the rotated volume fluxes across each face of the computational cell. Spatial derivatives are discretized with second-order central differences. The convection term is advanced via second-order Adams-Bashforth time integration and the diffusion term is advanced using the second-order Crank-Nicolson scheme. The pressure Poisson equation is solved with a V-cycle multigrid in the xy plane and a fast Fourier transform (FFT) in the spanwise direction.

**Boundary conditions**

The serpentine passage geometry under investigation is analogous to channel flow simulations commonly found in the literature. However, unlike channel flow simulations, the pressure gradient in the streamwise direction is unknown. Typically in plane channel flow simulations, the body force term in Eq. (1) is taken to be the average pressure gradient, which must balance the average wall shear. The pressure that is solved for is the pressure fluctuation. This allows implementation of truly periodic boundary conditions in the streamwise direction. Alternatively, inflow-outflow boundary conditions have been used successfully for channel flow simulations. The outflow boundary condition has been used with much success in backward-facing step simulations, flow past a cube, and was investigated thoroughly by Pauley et al. Pauley showed that the effect of using a convective boundary condition only degrades the overall solution locally, within a few grid points of the boundary, and allows for smooth convection of vortical structures out of the computational domain. Numerical tests were conducted using a backwards differenced convective outflow boundary condition.

Rosenfeld et al. The dependent variables are the rotated volume fluxes across each face of the computational cell. Spatial derivatives are discretized with second-order central differences. The convection term is advanced via second-order Adams-Bashforth time integration and the diffusion term is advanced using the second-order Crank-Nicolson scheme. The pressure Poisson equation is solved with a...
\[
\frac{\partial \overline{u}_i}{\partial t} + \overline{U}_b \frac{\partial \overline{u}_i}{\partial x} = 0,
\]

where \( \overline{U}_b \) is the time-averaged bulk velocity and a pseudo-periodic inflow boundary condition,

\[
[\overline{u}_i(i=1)]^{n+1} = [\overline{u}_i(i = N_z - m)]^n,
\]

where \( m \) is the \( i \)th grid point to ensure symmetry with the first grid point in the \( i \)th direction. This class of boundary conditions has been verified, and results will be presented for stationary and rotating channel flow simulations. Data are copied from the outflow to inflow recycle planes. This allows the same turbulence structures to develop in the overlap zones, further avoiding spurious effects of the convective outflow condition. The grid is such that identifying the outflow and inflow recycle planes will create a smooth variation of mesh stretching. The spanwise direction is homogeneous with periodic boundary conditions for the velocity and pressure fields.

Validation data for RANS models often suffer from ambiguous inflow conditions. That makes DNS data with periodic conditions in the flow direction attractive. All turbulence variables are generated by the solution of the Navier-Stokes equations, with no help from inflow specifications.

Plane channel flow has become a standard benchmark. That case is statistically homogeneous in \( x \). The present DNS is also periodic, but not homogeneous in \( x \). It acquires the attractiveness of requiring no inflow specification. A RANS computation would be two-dimensional, periodic in the serpentine direction, with no-slip outer walls: topologically equivalent to a plane channel, but fluid dynamically rather more complex, introducing curvature, rotation, and separation.

### III. VERIFICATION CASES

The code was extensively verified before initiating the serpentine DNS studies. Of particular interest are direct numerical simulations that were undertaken for a stationary and rotating plane channel. The domain, Reynolds, and rotation numbers were selected based on published DNS data, and these same nondimensional numbers were then later applied to the serpentine passage simulations. Additional verification results can be found in Laskowski.\(^{23}\)

#### A. DNS of a stationary turbulent channel

A simulation was conducted for \( \text{Re}_c=180 \) on a 128\(^3\) grid for a domain of \( 4 \pi \delta \times 2 \delta \times 2 \pi \delta \). The grid is stretched in the...
wall normal direction according to a hyperbolic-tangent function. The results are compared to the spectral DNS data of Moser et al.\textsuperscript{30} and Kawamura’s second-order finite-difference results.\textsuperscript{31} Figure 1 presents the mean velocity profile and rms of the velocity fluctuations in wall units. Both the viscous sublayer and log layer are clearly evident and the results agree well with the published data.

B. DNS of a rotating turbulent channel

In order to verify the implementation of the Coriolis term and streamwise boundary conditions, a rotating turbulent channel simulation was conducted. The stationary case results served as the initial condition. The Reynolds number (Re\textsubscript{\infty}=180) and rotation number (Ro\textsubscript{\infty}=5.0) are comparable to the DNS studies of Kristoffersen and Andersson.\textsuperscript{4} Figure 3 presents the mean flow and turbulent kinetic energy at Ro\textsubscript{\infty}=3.0 and 7.6. Again, agreement is good. While the stationary turbulent channel velocity profile is symmetric about the channel center line, rotation causes a strong asymmetry via the Coriolis force (Fig. 2). The flow in the core of the channel is irrotational with \( dU/dy = -2\Omega \) and the flow near each channel wall is distinctly different. When rotation and shear are aligned, as is the case near the pressure side, the turbulence is destabilized (augmented). When rotation and shear are opposed, as is the case near the suction side, the turbulence is stabilized (damped).

IV. DNS SERPENTINE PASSAGE RESULTS

The serpentine passage simulations were conducted for the same Reynolds and rotation numbers used in the channel verification cases, namely Re\textsubscript{\infty}=5600 and Ro\textsubscript{\infty}=0 and 0.32. The geometry under investigation has dimensions of \( 12\pi \delta \times 2\delta \times 3\pi \delta \) in the streamwise, transverse, and spanwise directions, with curvature ratio \( r_s/\delta=0.33 \) and an average curvature ratio \( R_c/\delta=2.0 \). The computational grid consists of a Cartesian and cylindrical mesh in the straight and curved passages, respectively, utilizing the same stretching function as in the plane channel simulation. Doing so allows for a constant grid spacing in the streamwise direction for both the straight and curved sections of the passage, thus eliminating any global mesh stretch error in the streamwise direction,\textsuperscript{32} although this approach may reduce the formal order accuracy locally. To test grid independence, five different grid levels, listed in Table I, were investigated. The geometry and com-

FIG. 8. Time averaged mean-flow contours for rotating and stationary cases.
Computational grid of the serpentine passage is depicted in Fig. 3. Stations 1 and 5 are at the same “x” location and represent the inflow into U-bend 1 and U-bend 2, respectively. Station 5 is also the exit of U-bend 1. Stations 2 and 6 represent the inflow planes into the bend region corresponding to $\theta=0^\circ$. Stations 3 and 7 are located at $\theta=90^\circ$ for U-bends 1 and 2. Finally, stations 4 and 8 are located at the exit of the bend section, $\theta=180^\circ$.

For the F1 case, 128 grid points were evenly distributed in the streamwise direction for the straight section from 0 $\leq X \leq 2 \pi$. Thus, the streamwise distribution of grid points is twice as fine as that of the channel verification cases pre-

FIG. 9. Time-averaged velocity and Reynolds stress profiles. (a) Stations 1 and 5, (b) stations 2 and 6, (c) stations 3 and 7, (d) stations 4 and 8.
Another 128 grid points were evenly distributed in the circumferential direction in the bend section. For the geometry presented in Figs. 3 and 4, this corresponds to a streamwise spacing at the channel midspan that is coincident with the streamwise distribution in the straight section. The streamwise distribution near the inner wall is finer than the midspan distribution and, conversely, the distribution near the outer wall is coarser than the midspan. Finally, another 128 grid points were evenly distributed in the straight section after the bend to the symmetry point. In the wall normal direction, 128 grid points were distributed based on the same stretching function described in the channel validation section. In the spanwise direction, 256 grid points were evenly distributed. The second U-bend, which completes the serpentine passage, has the same grid point distribution as the first U-bend.

Simulations were initially conducted for C1-180-0 (grid-Reynolds number-rotation number), which was initialized with a converged 2D RANS simulation based on the mass flow obtained from the plane channel verification simulations. A divergence-free random perturbation field was imposed on the mean flow. The flow was then allowed to develop until a statistical steady state was reached. Statistics were gathered for grid independence study. The flow field was then interpolated onto the M1-180-0 and M1-180-5 grids and the F1-180-0 and F1-180-5 grids, and the flow was allowed to develop. Once the F1 case reached a statistical steady state, the flow field was interpolated onto the F2 and

FIG. 10. (a) Instantaneous and time-averaged turbulent kinetic energy contours. (b) Turbulent kinetic energy profiles at different station locations.
F3 grids to ensure adequate grid resolution in the streamwise and wall normal directions, respectively. This methodology is depicted in Fig. 4. Extensive verification of grid resolution is described by Laskowski.23

The simulations for F1-180-0 and F1-180-5 were conducted with a time step of \( \Delta \tau = 1.25 \times 10^{-4} \). Statistics were gathered over 75000 iterations, which corresponds to approximately 3.75 flow-through times. Due to computer limitations, only 25000 iterations were feasible for cases F2 and F3; that was sufficient to converge first-order statistics.

Figure 5 presents the instantaneous and time-averaged spanwise vorticity for the stationary and rotating F1 cases. The flow is highly turbulent, particularly in the downstream region of the U-bend inner wall, and the effect of rotation is apparent in the strong asymmetry that now exists between the upstream and downstream U-bends.

### A. Grid independence

Considerable effort was taken to ensure adequate grid resolution of the turbulent structures down to an order of the Kolmogorov scale. Figure 6 depicts the level of grid independence achieved for the stationary and rotating cases at station 4. This is the location of flow separation and slight grid discontinuity discussed earlier. At station 4, the primary flow direction is negative in the \( x \) direction and the separated region is noted where \( U > 0 \). Figure 6 depicts the streamwise mean velocity obtained on grids C1, M1, F1, F2, and F3. First consider the stationary case. The difference between the M1-180-0 and F1-180-0 (uniform grid refinement) results is significant. However, cases F1-180-0 and F2-180-0, where the grid points are doubled in the streamwise direction, are nearly indistinguishable. This indicates that the resolution in the streamwise direction is sufficiently fine. The F2-180-0 case corresponds to \( \Delta x^+ = 4; \Delta y^+ = 0.5 − 5; \Delta z^+ = 6 \) based on the average friction velocity, where \( x, y, \) and \( z \) are the streamwise, transverse, and spanwise directions. Similar findings were noted for the rotating case and a representative result is provided in Fig. 6(b). The resolution in the spanwise direction for F1, F2, and F3 is the same, and is consistent with the DNS simulations of Le et al.27 and Ikeda.33

### B. Domain independence

Spanwise correlations were computed to ensure independence of the lateral domain size. This is the direction of statistical homogeneity. The two-point correlation coefficient,

\[
r_{ij} = \frac{u_i'(z)u_j'(z + \Delta z)}{\overline{u_i'^2}} ,
\]

was computed and is plotted in Fig. 7 for the stationary case at station 1. Three different transverse positions are plotted, at \( 2/3 \delta, \delta, \) and \( 4/3 \delta \) measured from the inner wall. The correlations all fall nearly to zero for large separation, indicating that the domain width is sufficient for the use of periodic boundary conditions. Similar findings were noted for the rotating case, and additional details concerning domain independence can be found in Laskowski.23

### C. Mean velocity and Reynolds stresses

The mean flow field is represented qualitatively in Fig. 8 and quantitatively in Fig. 9. The coordinate system in Fig. 9 is a local coordinate system in order to align the inner and outer walls at the same location along the abscissa for all station locations. These figures allow for a comparison between the flow field in U-bend 1 and U-bend 2. Turbulence statistics in the two bends should be identical in the stationary case, and the streamwise component, \( U \), should be equal and opposite. The addition of orthogonal mode rotation results in a strong asymmetry between U-bend 1 and U-bend 2. It is informative to compare the flow field for the stationary and rotating cases at the same station locations for U-bend 1.
and U-bend 2. Station 1 is both the entrance of U-bend 1 and the exit of U-bend 2. Thus the flow into U-bend 1 is strongly influenced by the curvature in U-bend 2. Likewise the flow entering U-bend 2 at station 5 is strongly influenced by the curvature of U-bend 1. This detail is not important in the stationary case but plays a significant role in understanding the rotating case.

First, consider the stationary passage. The flow undergoes strong acceleration near the inner wall as the flow approaches the curved section and continues to accelerate and eventually separates. The streamwise Reynolds stress decreases near the convex surface (station 3 just upstream of separation), and results in very large values in the separated region at station 4. Fairly large values of \( \overline{v'v'} \) are observed very close to the convex surface and in the core of the channel between stations 3 and 4. Finally, large values of \( \overline{w'w'} \) are seen in the separated region and near the concave walls.

Focusing attention now on the rotating case, the streamwise Reynolds stresses are substantially reduced near the outer wall, compared to the stationary case. The large values of \( \overline{v'v'} \) noticed in the stationary case near the convex surface have been reduced and are very nearly the same for stations 3 and 7, yet the value has increased in the separated region in U-bend 1 when compared to the stationary case. In the core of the channel, the \( \overline{v'v'} \) component between stations 3 and 4 has been reduced when compared to the stationary case. Conversely, in the core of the channel, the \( \overline{w'w'} \) component between stations 7 and 8 has substantially increased. The effect of rotation has been to reduce \( \overline{w'w'} \) near the convex walls for both U-bend 1 and U-bend 2 when compared to the stationary case. Finally, the flow at station 8 is nearly identical for both the stationary and rotating cases.

Figure 10 presents the instantaneous and time-averaged turbulent kinetic energy, \( k = (\overline{u'u'} + \overline{v'v'} + \overline{w'w'})/2 \). Corresponding to Figs. 8 and 9. Again, the effect of rotation is to increase \( k \) in the separated region of U-bend 1 and decrease it in U-bend 2. The level of \( k \) near the convex surface at the onset of separation has been substantially reduced with the addition of rotation. For the rotating case, the turbulent ki-
netic energy is typically higher near the outer wall in U-bend 2 than in U-bend 1. Conversely, the turbulent kinetic energy is typically lower near the inner wall in U-bend 2 than in U-bend 1.

Figure 11 presents the ratio between the stationary and rotating normal stresses and turbulent kinetic energy. It is now clearly evident that the overall effect of rotation is to augment the levels of turbulent kinetic energy in the core of U-bend 2 and, conversely, to suppress the levels of $k$ in the core of U-bend 1. The opposite effect can be seen in the separated regions of U-bend 1 and U-bend 2 since the local flow direction opposes the primary flow.

D. Secondary flow

Flow structures not aligned with the primary fluid motion, often referred to as secondary flows, have been studied extensively for a wide range of problems. The generation of such counter-rotating vortex pairs in a channel can be attributed to centrifugal or Coriolis forces, due to curvature or rotation, which turns the shear layer perpendicular to its main vorticity direction.\(^3\) Direct numerical simulations have identified secondary flow structures in periodic curved channel flow\(^3\) and spanwise rotating plane channel flow.\(^4\) In the present study, the Reynolds number is comparable to both, the curvature is much stronger than that studied by Moser and Moin,\(^3\) and the strength of rotation is comparable to that of Kristoffersen and Andersson.\(^4\) Consider the following nondimensional numbers, in particular the Dean number for curvature effects and the rotation number (or inverse Rossby number) for rotational effects:

$$D_{e_c} = Re_{c} \left( \frac{2 \delta}{R_c} \right)^{1/2} = \frac{\text{inertial centrifugal}}{\text{viscous inertial}},$$

$D_{e_c}$
observed for $Ro=5.0$ but they were seen in coarse grid simulations at lower rotation numbers of $Ro=3.0$ and 1.0. Hence they are created by curvature and suppressed by Coriolis forces. The other panes show how spanwise features are discernible in instantaneous fields.

For laminar flow in curved channel and spanwise rotation, the effect of rotation is to either enhance or counteract the effect of curvature based in part on the direction and strength of the local vorticity and the direction and strength of rotation. The present simulations involve strong rotation, strong curvature, and separated flow. The Coriolis force has destroyed the vortex pair that existed in the stationary case throughout the entire passage. No secondary flow was generated along the convex surface or in the straight channel due to rotation.

Figure 13 presents the Reynolds stresses for the stationary and rotating cases at station 1, and contours of the ratio $\beta=(u_i' u_j')_{Avg}/(u_i' u_j')_{Avg}$ based on the previously defined averaging procedures. It can be seen that the effect of the secondary flow vortices at station 1 is substantial, and comprises up to 25% of the stresses for the stationary case near the outer wall. Again, no secondary flow is observed for the rotating case.

### E. Skin friction and separation

The skin friction coefficient is shown in Fig. 14. In both cases the skin friction initially rises along the inner wall as the boundary layer becomes thinner (approaching the convex surface) and, conversely, the boundary layer thickens along the outer wall (approaching the concave surface). Between stations 2 and 3 and 6 and 7, the skin friction reaches a maximum along the inner surface. The flow undergoes strong acceleration and subsequently separates upstream of station 4. The effect of rotation is to cause separation to occur sooner in both U-bends. In U-bend 1 the flow reattaches further downstream, and in U-bend 2 the flow reattaches further upstream compared to the stationary case. In the stationary case, the flow remains attached along the outer wall for both U-bend 1 and U-bend 2. Rotation has resulted in separation along the outer wall near station 6.

The dynamics of the separated flow downstream of the inner wall for the stationary and rotating cases are presented in Table II and shown in Fig. 15. The extent of the separation is dramatically increased in U-bend 1 when comparing the rotating case to the stationary case. As expected, the converse is true for U-bend 2, where the separated region is smaller than in U-bend 1.

### Table II. Mean separation details for stationary and rotating cases.

<table>
<thead>
<tr>
<th>Ro=0 (U-bend 1 and 2)</th>
<th>Ro=5 (U-bend 1)</th>
<th>Ro=5 (U-bend 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Separation</td>
<td>$\theta=143.3^\circ$</td>
<td>$\theta=116.6^\circ$</td>
</tr>
<tr>
<td>Attachment</td>
<td>$x=5.02$</td>
<td>$x=3.22$</td>
</tr>
<tr>
<td>Bubble height</td>
<td>5.26%</td>
<td>13.95%</td>
</tr>
</tbody>
</table>
V. CONCLUSIONS

Direct numerical simulations for a stationary and rotating serpentine passage were conducted for \(Re_b = 5600\) \((Re_r = 180)\) and \(Ro_b = 0\) and 0.32 \((Ro_r = 0\) and 5.0). The simulations provide insight into the behavior of turbulence in a geometry consisting of strong convex curvature, strong concave curvature and the coupling of curvature with rotation. The spatially evolving flow, with pressure gradients, curvature/inertia-based flow separation, secondary flow, and the effect of rotation on these qualitative aspects of the flow, have been identified. Secondary flow was evident in the stationary case and played a significant role on the Reynolds stresses. However, it was noted that as the rotation number was increased to \(Ro_b = 0.32\) the secondary flow was no longer evident. The flow in U-bend 1 is identical to U-bend 2 for the stationary case due to geometric symmetry. The effect of rotation causes a large asymmetry between U-bend 1 and U-bend 2 and has either a stabilizing or destabilizing effect on the Reynolds stresses depending on the local velocity and pressure field. We anticipate that these data will make a good test case for turbulence modeling validation.

ACKNOWLEDGMENTS

This work was sponsored by the Department of Energy Advanced Strategic Computing (ASC) program. The simulations were conducted on the SGI Origin 2000 Theta cluster at Los Alamos National Laboratory, New Mexico.

16H. Iacovides, D. C. Jackson, G. Kelemenis, and B. E. Lauder, “The


