1988

Primitive numerical simulation of circular Couette flow

Jan Franciszek Hasiuk
Iowa State University

Follow this and additional works at: https://lib.dr.iastate.edu/rtd

Part of the Aerospace Engineering Commons

Recommended Citation
INFORMATION TO USERS

The most advanced technology has been used to photograph and reproduce this manuscript from the microfilm master. UMI films the original text directly from the copy submitted. Thus, some dissertation copies are in typewriter face, while others may be from a computer printer.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyrighted material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each oversize page is available as one exposure on a standard 35 mm slide or as a 17" × 23" black and white photographic print for an additional charge.

Photographs included in the original manuscript have been reproduced xerographically in this copy. 35 mm slides or 6" × 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI
Accessing the World's Information since 1938
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
Primitive numerical simulation of circular Couette flow

Hasiuk, Jan Franciszek, Ph.D.
Iowa State University, 1988
PLEASE NOTE:

In all cases this material has been filmed in the best possible way from the available copy. Problems encountered with this document have been identified here with a check mark \(\checkmark\).

1. Glossy photographs or pages ______
2. Colored illustrations, paper or print ______
3. Photographs with dark background ______
4. Illustrations are poor copy ______
5. Pages with black marks, not original copy ______
6. Print shows through as there is text on both sides of page ______
7. Indistinct, broken or small print on several pages ______\(\checkmark\)
8. Print exceeds margin requirements ______
9. Tightly bound copy with print lost in spine ______
10. Computer printout pages with indistinct print ______
11. Page(s) ________ lacking when material received, and not available from school or author.
12. Page(s) ________ seem to be missing in numbering only as text follows.
13. Two pages numbered ______. Text follows.
14. Curling and wrinkled pages ______
15. Dissertation contains pages with print at a slant, filmed as received ______
16. Other__________________________

____________________________________

____________________________________

______________________________

\UMI
Primitive numerical simulation of circular Couette flow

by

Jan Franciszek Hasiuk

A Dissertation Submitted to the
Graduate Faculty in Partial Fulfillment of the
Requirements for the Degree of
DOCTOR OF PHILOSOPHY

Major: Aerospace Engineering

Approved:

Signature was redacted for privacy.

In Charge of Major Work

Signature was redacted for privacy.

For the Major Department

Signature was redacted for privacy.

For the Graduate College

Iowa State University
Ames, Iowa

1988
<table>
<thead>
<tr>
<th>TABLE OF CONTENTS</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>FOREWORD</strong></td>
</tr>
<tr>
<td><strong>NOMENCLATURE</strong></td>
</tr>
<tr>
<td>Dimensional parameters</td>
</tr>
<tr>
<td>Nondimensional variables, constants, parameters and operators</td>
</tr>
<tr>
<td>Subscripts</td>
</tr>
<tr>
<td><strong>INTRODUCTION</strong></td>
</tr>
<tr>
<td><strong>COUETTE-TAYLOR FLOW</strong></td>
</tr>
<tr>
<td><strong>FLOW EQUATIONS</strong></td>
</tr>
<tr>
<td>The approximation</td>
</tr>
<tr>
<td>Vorticity-velocity equations</td>
</tr>
<tr>
<td>Boundary conditions</td>
</tr>
<tr>
<td>Roberts transformation: $r,z,t \rightarrow \eta,\xi$</td>
</tr>
<tr>
<td>Transformed equations of motion</td>
</tr>
<tr>
<td><strong>NUMERICAL METHOD</strong></td>
</tr>
<tr>
<td><strong>FLOW RESULTS</strong></td>
</tr>
<tr>
<td><strong>TURBULENT SOLUTIONS</strong></td>
</tr>
<tr>
<td>Turbulent flow equations</td>
</tr>
<tr>
<td>Solution method</td>
</tr>
<tr>
<td>Results comparison and discussion</td>
</tr>
<tr>
<td><strong>TURBULENT DEPARTURES</strong></td>
</tr>
<tr>
<td><strong>ENERGY-DISSIPATION TURBULENCE MODEL</strong></td>
</tr>
<tr>
<td><strong>CONCLUSIONS</strong></td>
</tr>
<tr>
<td><strong>APPENDIX</strong></td>
</tr>
<tr>
<td>Key to graphs</td>
</tr>
<tr>
<td><strong>ACKNOWLEDGEMENTS</strong></td>
</tr>
<tr>
<td><strong>LITERATURE</strong></td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Nonturbulent numerical solution convergence history / NAS 9160 computer</td>
<td>23</td>
</tr>
<tr>
<td>2.</td>
<td>Turbulent circular Couette flow experiments</td>
<td>64</td>
</tr>
<tr>
<td>3.</td>
<td>1-d turbulent numerical solution convergence history / NAS 9160 computer</td>
<td>76</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>Description</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Integration domain, coordinates and variables</td>
<td>13</td>
</tr>
<tr>
<td>2</td>
<td>Vorticity contours: a,b,c,d ~ Re=100,200,300,400: ( \Delta \omega = 0.1 )</td>
<td>25</td>
</tr>
<tr>
<td>3</td>
<td>Streamline contours: a,b,c,d ~ Re=100,200,300,400: ( \Delta \psi = 0.0005 )</td>
<td>26</td>
</tr>
<tr>
<td>4</td>
<td>Azimuthal velocity contours: a,b,c,d ~ Re=100,200,300,400: ( \Delta u = 0.1 )</td>
<td>28</td>
</tr>
<tr>
<td>5</td>
<td>Radial velocity contours: a,b,c,d ~ Re=100,200,300,400: ( \Delta v = 0.005 )</td>
<td>30</td>
</tr>
<tr>
<td>6</td>
<td>Axial velocity contours: a,b,c,d ~ Re=100,200,300,400: ( \Delta w = 0.005 )</td>
<td>31</td>
</tr>
<tr>
<td>7</td>
<td>Vorticity surfaces: a,b ~ Re=100,400</td>
<td>33</td>
</tr>
<tr>
<td>8</td>
<td>Streamfunction surfaces: a,b ~ Re=100,400</td>
<td>34</td>
</tr>
<tr>
<td>9</td>
<td>Azimuthal velocity surfaces: a,b ~ Re=100,400</td>
<td>35</td>
</tr>
<tr>
<td>10</td>
<td>Radial velocity surfaces: a,b ~ Re=100,400</td>
<td>38</td>
</tr>
<tr>
<td>11</td>
<td>Axial velocity surfaces: a,b ~ Re=100,400</td>
<td>39</td>
</tr>
<tr>
<td>12</td>
<td>Comparisons with exact Couette solution</td>
<td>40</td>
</tr>
<tr>
<td>13</td>
<td>Azimuthal velocity profiles: ( u(r), u(z) ) for Re=100</td>
<td>41</td>
</tr>
<tr>
<td>14</td>
<td>Radial velocity profiles: ( v(r), v(z) ) for Re=100</td>
<td>42</td>
</tr>
<tr>
<td>15</td>
<td>Axial velocity profiles: ( w(r), w(z) ) for Re=100</td>
<td>43</td>
</tr>
<tr>
<td>16</td>
<td>Vorticity profiles: ( \omega(r), \omega(z) ) for Re=100</td>
<td>44</td>
</tr>
<tr>
<td>17</td>
<td>Streamfunction profiles: ( \psi(r), \psi(z) ) for Re=100</td>
<td>45</td>
</tr>
<tr>
<td>18</td>
<td>Azimuthal velocity profiles: ( u(r), u(z) ) for Re=200</td>
<td>46</td>
</tr>
<tr>
<td>19</td>
<td>Radial velocity profiles: ( v(r), v(z) ) for Re=200</td>
<td>47</td>
</tr>
</tbody>
</table>
FIGURE 20. Axial velocity profiles: \( w(r), w(z) \) for Re=200 

FIGURE 21. Vorticity profiles: \( \omega(r), \omega(z) \) for Re=200 

FIGURE 22. Streamfunction profiles: \( \psi(r), \psi(z) \) for Re=200 

FIGURE 23. Azimuthal velocity profiles: \( u(r), u(z) \) for Re=300 

FIGURE 24. Radial velocity profiles: \( v(r), v(z) \) for Re=300 

FIGURE 25. Axial velocity profiles: \( w(r), w(z) \) for Re=300 

FIGURE 26. Vorticity profiles: \( \omega(r), \omega(z) \) for Re=300 

FIGURE 27. Streamfunction profiles: \( \psi(r), \psi(z) \) for Re=300 

FIGURE 28. Azimuthal velocity profiles: \( u(r), u(z) \) for Re=400 

FIGURE 29. Radial velocity profiles: \( v(r), v(z) \) for Re=400 

FIGURE 30. Axial velocity profiles: \( w(r), w(z) \) for Re=400 

FIGURE 31. Vorticity profiles: \( \omega(r), \omega(z) \) for Re=400 

FIGURE 32. Streamfunction profiles: \( \psi(r), \psi(z) \) for Re=400 

FIGURE 33. Turbulent profile comparison to experiments 

FIGURE 34. Turbulent profile comparison to experiments 

FIGURE 35. Turbulent profile comparison to experiments 

FIGURE 36. Turbulent profiles with correction 

FIGURE 37. Turbulent profiles with correction 

FIGURE 38. Turbulent profiles with correction 

FIGURE 39. Eddy viscosity profiles: \( a, b \sim \text{Baseline, Corrected} \)
FOREWORD

to

Freedom, Work and the Pursuit of Happiness
NOMENCLATURE

**Dimensional parameters**

- \( H \)  cylinder height
- \( K \)  1000
- \( R \)  cylinder radius
- \( U \)  azimuthal velocity

**Nondimensional variables, constants, parameters and operators**

- \( C \)  empirical turbulence constant
- \( C_c \)  empirical turbulence constant correction for streamline curvature
- \( C_1, C_2, C_3, C_4, C_5, C_6 \)  empirical turbulence constants
- \( C_y \)  energy-dissipation model turbulence function
- \( d \)  inter-cylinder gap, \( d = r_0 - 1 \)
- \( V \)  Laplacian
- \( \nabla^2 \)  transformed Laplacian
- \( e \)  isotropic turbulent kinetic energy dissipation rate
- \( f_1, f_2, f_3, f_4, f_5 \)  metric functions of the transformation
- \( G \)  tensorial gradient
- \( g_1, g_2, g_3, g_4 \)  metric functions of the transformation
- \( h \)  cylinder height
- \( k \)  turbulent kinetic energy
- \( 1-d \)  one-dimensional
- \( R_f \)  flux Richardson number
- \( Re \)  Reynolds number
- \( Ri \)  Richardson number
r, θ, z and t cylindrical coordinates and time
S source term
Ta Taylor number
3-d three-dimensional
u azimuthal velocity
v radial velocity
w axial velocity
a_v, a_w and a_ψ Poisson equation relaxation parameters
β transformation clustering parameter
δ boundary layer thickness
Δr, Δz mesh increments in the r, z directions
Δt time increment
e error tolerance
ξ transformed axial coordinate
η transformed radial coordinate
κ Von Karman's constant
Λ molecular kinematic viscosity (same as ν)
λ mixing length of turbulence
ν molecular kinematic viscosity
ν_t turbulent eddy viscosity
ξ transformed time
ρ fluid density
σ energy-dissipation model turbulence constant
T turbulent eddy viscosity (same as ν_t)
\( \tau \)  shear stress
\( \phi \)  any dependent variable
\( \psi \)  crossflow streamfunction
\( \omega \)  azimuthal vorticity

**Subscripts**
- \( c \)  corrected value
- \( \text{crit} \)  critical value for instability onset
- \( e \)  turbulent energy dissipation rate equation reference
- \( i \)  inner cylinder
- \( ij \)  integration domain location \( \sim (r,z) \)
- \( iw \)  inner wall value
- \( k \)  turbulent kinetic energy equation reference
- \( \text{max} \)  maximum value
- \( \text{min} \)  minimum value
- \( o \)  outer cylinder
- \( ow \)  outer wall value
- \( r \)  radial direction
- \( r\theta \)  perpendicular to \( r \) in the \( \theta \)-direction
- \( t \)  turbulent quantity
- \( w \)  wall location
- \( z \)  axial direction
- \( z\theta \)  perpendicular to \( z \) in the \( \theta \)-direction
- \( \omega \)  vorticity transport equation reference
INTRODUCTION

Circular Couette flow has been the subject of extensive study throughout the modern history of fluid mechanics. Early investigators found it to be a convenient flow to reproduce experimentally as contrasted to the difficulty in producing the flow of popular interest, namely, planar Couette flow. The convenience allowed the simple interpretation that, for large radii, small gap and long cylinders, circular Couette flow asymptotically approaches planar Couette flow.

The study of Couette flow is important for academic and practical reasons. Well-known, exact, laminar 1-d solutions of the Navier-Stokes equations for Couette flow exist (Couette, 1890). Couette flow analysis has applications to mechanical devices such as turbomachinery, generators, combustors, motors and bearings. Experimental verification of the early measurements of the coefficient of molecular viscosity by Poiseuille were made in a circular Couette flow apparatus (Mallock, 1888, 1896). These results, of course, have been exploited in the designs of modern viscometers.

More recently, a circular Couette flow machine has been proposed as a practical wind tunnel for use in space. The Carrousel Wind Tunnel (CWT) has been designed for use on board a space station to investigate the physics of aeolian processes such as the transport of sand, dust, and snow (Greeley and Iversen, 1983). The CWT offers a virtually infinite flow development length, zero pressure gradient in the streamwise, azimuthal direction (analogous to the flat plate boundary layer), and a well-known radial pressure distribution.
More generally, the CWT design offers a practical, working wind tunnel for use in space. In addition, the physical space economy advantage of the CWT may influence future designs for earth-based wind tunnels.

This work is motivated by interest in numerically solving 3-d Navier-Stokes equations to obtain laminar and turbulent flowfield predictions inside a finite-length CWT model, that is, for a real machine.
COUETTE-TAYLOR FLOW

Prevailing interest in the stability of a fluid to undergo transition from laminar to turbulent motion led Taylor to write down the instability mode mathematically and to verify the instability experimentally for a circular Couette flow with small gap and infinite cylinder length (Taylor 1923, 1936). The Taylor-vortex instability takes the form of an even number of steady, axisymmetric, counter-rotating, toroidal vortices filling the inter-cylinder gap with regular spacing in the axial direction. Taylor’s prediction of the instability onset is given by what has since become known as the critical Taylor number, that is a nondimensional function of the rotation speed and cylinder radii and is a form of scaled Reynolds number. Taylor-vortex flow has applications to the fluid mechanics of geo- and astrophysics, meteorology, gas separation in centrifuges, tribology, and to the phenomenon of laminar-turbulent transition (Buhler et al., 1986).

The visualization experiments of Coles (1965) documented the nonuniqueness of the transition process and identified two distinct kinds of transition in circular Couette flow:

1. Transition by spectral evolution that is characteristic of the motion when the inner cylinder has a larger angular velocity than the outer cylinder.

2. Catastrophic transition that is characteristic of the motion when the outer cylinder has a larger angular velocity than the inner cylinder.

The transition territory across the Taylor boundary has been fertile ground for the experimental observations of a wide variety of circular Couette flow transition regimes. New flows are continually being discovered (Andereck et al., 1986). Numerical problems associated with the nonlinearity of the transition process are complex and difficult. Thus, numerical analysis has moved forward at a more conservative rate. Though numerical excursions crossing the Taylor boundary have been few, valuable and accurate solutions exist.

de Roquefort and Grillaud (1978) obtained numerical solutions of Taylor-vortex flow for Re=10 to 3000 for which Re_{crit}=2569 and for Re=250 to 300 for which Re_{crit}=272.8. They found good agreement between their numerical results and theory for the occurrence of instability. Using a time-dependent, vorticity-streamfunction approach for a physically realistic finite-length model with rotating endwalls for a small gap and small aspect ratio (R_1/R_2=0.933, H/R_2=0.67), they experimented with 3 types of differencing for the convective terms: central, first-order upwind, and second-order upwind. Their problem diverged for Re > 9000 for central and first-order upwind differencing, whatever the time step and relaxation parameter, even with a fine mesh. The trouble reportedly came from the coupling of the vorticity and streamfunction equations via the streamfunction variable in the
convective terms, even though the first-order upwind scheme was diagonally dominant. The second-order upwinding permitted solutions up to Re=100,000. For higher Re, they noticed a plausible structure with toroidal vortices and with loss of symmetry with respect to the cylinder midheight suggesting an axisymmetric numerical turbulence, though they recognized that such a result may have been due to round-off errors.

Fasel and Booz (1984) obtained fourth-order, spatially accurate numerical solutions for supercritical Taylor-vortex flow for a wide gap and large aspect ratios by imposing periodic boundary conditions and fixing the size of the expected Taylor flow cell. The available range of solutions was extended to 100 Ta_{crit} observing drastic flowfield changes as the Taylor number was increased from just supercritical to 100 Ta_{crit}. They found sharp increases in the number of harmonics contributing to the total solution, a jetlike or shocklike flow structure, boundary layer development along the inner and outer cylinder walls, and inviscid flow inside the Taylor cell core region.

The wide gap problem was also explored by Hamidi and Georgini (1986), who integrated the Navier-Stokes equations in Fourier space form as nonlinearly coupled ordinary differential equations dependent on time and radius. They solved for the Fourier coefficients of the streamfunction, azimuthal velocity, and azimuthal vorticity using 16 modes in the axial direction and 49 grid points in the radial direction for R_1/R_0=0.5 and for a supercritical Re=150.
Neitzel (1984) discusses the transient development of the Taylor-vortex structure based on a time-dependent computation for finite-length geometry. For a supercritical Re and large aspect ratio, Neitzel numerically integrates the Navier-Stokes equations using a circulation-vorticity-streamfunction method. He found elongated endplate streamline cells due to Ekman pumping for the steady-state solution and found good agreement between the numerical results and the laboratory experiments of Burkhalter and Koschmieder (1974) for \( \text{Re}/\text{Re}_{\text{crit}} < 4 \). For greater Re, differences were thought to be due to wavy mode effects not modelled in the axisymmetric numerical calculations.
FLOW EQUATIONS

The approximation

The azimuthal-invariant, Navier-Stokes Equations have been a popular choice for numerical analysis of circular Couette flow problems because there is no pressure gradient in the azimuthal or circumferential direction. A circular Couette flow machine such as the CWT represents a wind tunnel with an effective infinite draw, analogous to the zero pressure gradient flat plate boundary layer, so there is no physical mechanism for large-scale property variance in the azimuthal direction. The above approximation results in large computer time savings, thus is important from a practical standpoint. Elimination of pressure via a vorticity approach, results in additional computer time savings.

Vorticity-velocity equations

The nondimensional, azimuthal-invariant, cylindrical, incompressible, 3-d Navier-Stokes equations are written in a vorticity-velocity formulation:

\[
\frac{\partial \omega}{\partial t} + \frac{\partial \omega}{\partial r} + \frac{\partial \omega}{\partial z} - \frac{1}{r} \left( \nu \omega + 2u \frac{\partial \omega}{\partial z} \right) = \frac{1}{\operatorname{Re}} \left( \nabla^2 \omega - \frac{\omega}{r^2} \right)
\]

\[
\frac{\partial u}{\partial t} + \frac{\partial u}{\partial r} + \frac{\partial u}{\partial z} + \frac{uv}{r} = \frac{1}{\operatorname{Re}} \left( \nabla^2 u - \frac{u}{r^2} \right)
\]

\[
\nabla \cdot v = \frac{\partial \omega}{\partial z}
\]

\[
\nabla \cdot w = - \frac{\partial \omega}{\partial r} - \frac{\omega}{r}
\]
where the Laplacian is given by: $\nabla^2 = \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} + \frac{\partial^2}{\partial z^2}$

where the azimuthal vorticity is: $\omega = \frac{\partial v}{\partial z} - \frac{\partial w}{\partial r}$

where coordinates, parameters, and variables are given by:

$$r = \frac{r}{R_i} \quad z = \frac{z}{R_i} \quad t = \frac{t}{R_i} \quad \text{Re} = \frac{U_i R_i}{\nu}$$

$$u = \frac{u}{U_i} \quad v = \frac{v}{U_i} \quad w = \frac{w}{U_i} \quad \omega = \frac{R_i}{U_i} \omega$$

where dimensional quantities are denoted by overbars.

The above equation set involves two transport equations for the primary flow (azimuthal velocity and vorticity) and two Poisson equations for the secondary crossflow (radial and axial velocities).

An additional Poisson equation for the stream function may be obtained via continuity such that:

$$\frac{\partial^2 \psi}{\partial r^2} - \frac{1}{r} \frac{\partial \psi}{\partial r} + \frac{\partial^2 \psi}{\partial z^2} = rw$$

where $v = \frac{1}{r} \frac{\partial \psi}{\partial z}$, $w = -\frac{1}{r} \frac{\partial \psi}{\partial r}$

where continuity is given by: $\frac{1}{r} \frac{\partial}{\partial r}(rv) + \frac{\partial w}{\partial z} = 0$

The equations may, in principle, be solved subject to appropriate boundary conditions where the integration domain is defined so that:

$$u, v, w, \omega, \psi \text{ are } f(r, z)$$

The integration domain showing variables and coordinates is diagrammed in Figure 1.
FIGURE 1. Integration domain, coordinates and variables
Boundary conditions

Boundary conditions are no-slip, except for inner cylinder rotation:

\[ u(l, z) = 1 \quad u(r_o, z) = u(r, 0) = u(r, h) = 0 \]
\[ v(l, z) = v(r_o, z) = v(r, 0) = v(r, h) = 0 \]
\[ w(l, z) = w(r_o, z) = w(r, 0) = w(r, h) = 0 \]
\[ \psi(l, z) = \psi(r_o, z) = \psi(r, 0) = \psi(r, h) = 0 \]
\[ \omega(l, z) = -\frac{\partial w}{\partial r}(l, z) \quad \omega(r_o, z) = -\frac{\partial w}{\partial r}(r_o, z) \]
\[ \omega(r, 0) = \frac{\partial v}{\partial z}(r, 0) \quad \omega(r, h) = \frac{\partial v}{\partial z}(r, h) \]

Roberts transformation: \( r, z, t \longrightarrow \eta, \xi, \xi \)

\[
\eta = \frac{1}{2} \left[ 1 + \log_e \left( \frac{\beta_{r} + \frac{2(r - 1)}{r_o - 1} - 1}{\beta_{r} - \frac{2(r - 1)}{r_o - 1} + 1} \frac{\beta_{z} - 1}{\beta_{z} + 1} \frac{\beta_{z} + \frac{2z - 1}{h} - 1}{\beta_{z} - \frac{2z - 1}{h} + 1} \right) \right]
\]

\[
\xi = \frac{1}{2} \left[ 1 + \log_e \left( \frac{\beta_{z} - \frac{2z - 1}{h} + 1}{\beta_{z} + \frac{2z - 1}{h} - 1} \frac{\beta_{z} + 1}{\beta_{z} - 1} \frac{\beta_{z} - 1}{\beta_{z} + 1} \right) \right]
\]

\( \xi = t \)
The clustering parameter \((\beta)\) is specified by estimates of the boundary layer thickness \((\delta)\) such that:

\[
\beta_r = \left[ 1 - \frac{\delta_r}{(r_0 - r_1)} \right]^{-0.5} \quad \beta_z = \left[ 1 - \frac{\delta_z}{h} \right]^{-0.5}
\]

**Transformed equations of motion**

\[
\frac{\partial \omega}{\partial t} + f_2 v \frac{\partial \omega}{\partial \eta} + f_4 v \frac{\partial \omega}{\partial \xi} - \frac{1}{\xi} \left( \nu \omega + 2 f_4 u \frac{\partial u}{\partial \xi} \right) = \frac{1}{Re} \left( \nabla^2 \omega - \frac{\omega}{\xi^2} \right)
\]

\[
\frac{\partial u}{\partial t} + f_2 v \frac{\partial u}{\partial \eta} + f_4 v \frac{\partial u}{\partial \xi} + \omega v = \frac{1}{Re} \left( \nabla^2 u - \frac{u}{\xi^2} \right)
\]

\[
\nabla^2 v - \frac{v}{\xi^5} = f_4 \frac{\partial \omega}{\partial \xi}
\]

\[
\nabla^2 w = -f_2 \frac{\partial \omega}{\partial \eta} - \frac{\omega}{\xi^5}
\]

\[
\xi_1 \frac{\partial^2 \psi}{\partial \eta^2} + \left[ g_2 - \frac{2 f_2}{\xi^5} \right] \frac{\partial \psi}{\partial \eta} + g_3 \frac{\partial^2 \psi}{\partial \xi^2} + g_4 \frac{\partial^2 \psi}{\partial \xi \partial \eta} = f_5 \omega
\]

where \(f_2\), \(f_4\), and \(f_5\) are metric functions of the transformation such that:

\[
f_5 = r(\eta)
\]

\[
f_2 = \frac{\partial \eta}{\partial \xi} = \frac{K_1}{[\beta_r - f_1]}
\]

\[
f_4 = \frac{\partial \xi}{\partial \eta} = \frac{K_3}{[\beta_z - f_3]}
\]

\[
f_3 = \frac{r(\eta)}{r_0 - 1} - 1
\]

\[
f_1 = \frac{2[r(\eta) - 1]}{r_0 - 1} - 1
\]
where the transformed Laplacian is given by:

\[ \nabla^2 = g_1 \frac{\partial^2}{\partial \eta^2} + g_2 \frac{\partial}{\partial \eta} + g_3 \frac{\partial}{\partial \xi} + g_4 \frac{\partial^2}{\partial \xi^2} \]

where the inverse transformation is given by:

\[
\begin{align*}
K_1 &= \frac{2\beta_r}{(r_0 - 1) \log \left[ \frac{\beta_r + 1}{\beta_r - 1} \right]} \\
K_3 &= \frac{2\beta_z}{h \log \left[ \frac{\beta_z + 1}{\beta_z - 1} \right]}
\end{align*}
\]

where

\[
g_1 = \frac{f_2}{2} \\
g_2 = \frac{f_2}{f_5} + \frac{K_2 f_1}{[\beta_r^i - f_1^i]^2} \\
g_3 = \frac{K_4 f_3}{[\beta_z^i - f_3^i]^2} \\
g_4 = f_4
\]
Thus, the logarithmically stretched physical domain with unequal grid spacing and wall grid clustering is mapped into a unit square in the computational plane providing equal spacing in either direction (or both directions).
Numerical Method

Interpret spatial derivatives were differenced by second-order central differencing. Endwall derivatives were differenced by second-order one-sided differences. Time derivatives were differenced by first-order forward differencing. The equations were linearized and uncoupled by evaluating coefficients and unknowns at the previous time iteration. A zero-flow initial condition was applied.

Each equation was solved in turn by an alternating-direction implicit (ADI) method leading to tridiagonal matrices that were solved by a Thomas algorithm (see Anderson et al., 1984). Azimuthal velocity and vorticity were advanced one-half time step by a row sweep in the radial direction, followed by another half time step column sweep in the axial direction. After each time step, the Poisson equations for the secondary flow, axial and radial velocities, were solved by the same ADI method and iterated until convergence using over-relaxation. After the equation set had been advanced a full time step, the vorticity boundary conditions were updated.

The solution was marched in time until steady-state. Steady state convergence criteria were established as when changes in the variables between time iterations became sufficiently small. During the solution process, tridiagonal matrix coefficients were checked for diagonal dominance and correct viscous fluid properties. After each time step, continuity was checked by calculation of the continuity equation at each grid point.
Two methods of solution were investigated:

1. Vorticity-streamfunction formulation
2. Vorticity-velocity formulation

For the vorticity-streamfunction approach, boundary vorticity was updated based on Taylor series expansions from the wall for the streamfunction as in the methods of Woods and Thom (cited by Roache, 1976). Applying Woods' second-order method results in the following nondimensional forms:

\[
\omega_w(r_i, r_o) = \frac{6\psi_{w+1} - r(\Delta r)^2 \omega_{w+1}}{2(\Delta r)^2[r + \Delta r]} + O(\Delta r)^2
\]

\[
\omega_w(z_{min}, z_{max}) = \frac{6\psi_{w+1} - r(\Delta z)^2 \omega_{w+1}}{2r(\Delta z)^2} + O(\Delta z)^2
\]

These same expressions derived in the nondimensional transformed case are given by:

\[
\omega_w(r_i, r_o) = \frac{6\psi_{w+1} - f_2 r(\Delta r)^2 \omega_{w+1}}{(3 - f_2)(\Delta r)^2r + 2(\Delta r)^2} + O(\Delta r)^2
\]

\[
\omega_w(z_{min}, z_{max}) = \frac{6\psi_{w+1} - f_4 r(\Delta z)^2 \omega_{w+1}}{(3 - f_4) r(\Delta z)^2} + O(\Delta z)^2
\]

Thom's first-order method resulted in the following expressions for both nondimensional and transformed cases:

\[
\omega_w(r_i, r_o) = \frac{2\psi_{w+1}}{r(\Delta r)^2} + O(\Delta r)
\]
\[
\omega_w \bigg|_{z_{\text{min}}, z_{\text{max}}} = \frac{2\psi_{w+1}}{r(\Delta z)^2} + O(\Delta z)
\]

The Woods-type update led to numerical instability in the solution procedure and no converged solutions were obtained for the Re=100 case. Thom's method, on the other hand, resulted in a converged solution that contained many of the essential flow features, however, near the wall boundaries, continuity was satisfied to a rather poor degree resulting in "infinite" velocity gradients in the preferred boundary layer directions for the secondary flow. Corresponding inadequacy in the streamfunction solution was visible as a nonzero wall gradient or no-slip violation.

For the vorticity-velocity approach, boundary vorticity was updated based on explicit second-order representations of the wall derivatives for the secondary flow via the definition of the azimuthal vorticity as given previously in the Boundary conditions section under Flow Equations. The vorticity-velocity method produced a converged solution that satisfied continuity everywhere to a reasonable degree and a streamfunction that satisfied no-slip at the boundaries. Computer times for both methods of solution were roughly comparable.

Shortcomings in the vorticity-streamfunction method can be attributed to the boundary condition treatment of the streamfunction equation. A correct streamfunction for a viscous fluid at an impenetrable wall must satisfy no-slip and no-penetration boundary
conditions. However, the numerical solution cannot have it both ways mathematically because of overspecification. Thus, no-penetration is usually applied in the streamfunction equation and no-slip is imposed by the clever rendering of a Woods or Thom vorticity update. But as shown here, this can lead to inaccurate solutions or no solution at all. de Roquefort and Grillaud (1978) point out that Briley (1970) has shown, in some cases, that accuracy and stability can be increased, if near-wall velocities are evaluated using higher order expressions. They also note that under-relaxation sometimes helps. However, such procedures may not be successful or conclusive and worthwhile, especially in view of the vorticity-velocity alternative.

The vorticity-velocity method ensures that no-slip is satisfied via the Dirichlet boundary conditions for the crossflow Poisson equations. After a steady-state has converged, the streamfunction may be calculated using the usual no-penetration boundary conditions resulting in an accurate streamfunction satisfying both necessary boundary conditions. Thus, the vorticity-velocity approach is shown to provide a formally more natural and accurate treatment of realistic boundary conditions leading to more consistent and accurate solutions.

An absolute convergence criteria was found to be adequate for all results presented here and defined as when the change in any point variable between two iterations was less than $1 \times 10^{-6}$: such that for any variable $\phi$:

$$|\Delta \phi| < 1 \times 10^{-6}$$
A relative convergence criterion was developed for more stringent accuracy requirements. For a specified error tolerance $\varepsilon$ for every grid point, define a new $\varepsilon$, called $\varepsilon_{ij}$ as:

$$
\varepsilon_{ij} = \varepsilon |\phi_{ij}^n - \phi_{ij}^n|
$$

so that if $|\phi_{ij}^{n+1} - \phi_{ij}^n| > \varepsilon_{ij}$, then iterate.

This criterion works well provided that: $\phi_{ij}^n \neq \varepsilon$

This apparent difficulty can be overcome by the following conditional (for double precision arithmetic):

$$
\text{if } |\varepsilon_{ij}| < 1 \times 10^{-16}, \text{ then set } \varepsilon_{ij} = 1 \times 10^{-15}.
$$

This amounts to calling zero, zero when it is computationally necessary to do so, providing the problem scale allows.

A grid size of 20x20 was used to compute the first solution for $Re=100$. For the same $Re$, a 40x40 grid size solution was found to be virtually identical to the 20x20 solution. For $Re=200-400$, it was necessary to use 40x40 grid sizes. Double precision arithmetic was used for all calculations.

There was some concern that the analytic computation of the transformation metrics may not be as accurate as a numerical calculation of the metrics. Taylor series analysis implies that the metric derivative should be numerically represented for second-order accuracy. It was found that analytic and numerical metric computations produced virtually identical solutions for $Re=100$, which suggests the second-order maintenance of the Roberts transformation.
The following relaxation parameters were found to be most efficient:

- Radial velocity equation: \( \alpha_r = 1.5 \)
- Axial velocity equation: \( \alpha_w = 1.5 \)
- Streamfunction equation: \( \alpha_\psi = 1.0 \)

An upwinded scheme based on the vorticity-velocity formulation using first-order, weather differencing of the convective terms resulted in a solution that was virtually identical to the centrally differenced scheme for Re=100 and both 20x20 and 40x40 grids.

Table 1 outlines the convergence history for the nonturbulent numerical solutions. A maximum error in continuity of 0.01 corresponds to an error estimate for the secondary flow velocities of \( 1/12,000 \).

<table>
<thead>
<tr>
<th>Re</th>
<th>Grids</th>
<th>CPU time (sec)</th>
<th>Time steps (( \Delta t = 0.01 ))</th>
<th>max</th>
<th>Continuity</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>20^2</td>
<td>260</td>
<td>1694</td>
<td>0.309 ( \times 10^{-1} )</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>40^2</td>
<td>1158</td>
<td>1902</td>
<td>0.790 ( \times 10^{-2} )</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>40^2</td>
<td>2196</td>
<td>3106</td>
<td>0.121 ( \times 10^{-1} )</td>
<td></td>
</tr>
<tr>
<td>300</td>
<td>40^2</td>
<td>2547</td>
<td>3532</td>
<td>0.126 ( \times 10^{-1} )</td>
<td></td>
</tr>
<tr>
<td>400</td>
<td>40^2</td>
<td>3128</td>
<td>4388</td>
<td>0.131 ( \times 10^{-1} )</td>
<td></td>
</tr>
</tbody>
</table>
FLOW RESULTS

Flow results are presented for a numerical model geometrically similar to the CWT model at Iowa State University. The relevant dimensions are:

\[ R_l = 356 \text{mm} \]

\[ R_o = 531 \text{mm} \]

\[ H = 260 \text{mm} \]

The contour plots of flow variables in Figures 2-6 clearly reveal the changes in the flow field for the various Reynolds number test cases: \( \text{Re}=100,200,300,400 \). The contour increment for each variable was constant for all Re.

The strength of the vorticity grows strongly with increasing Re (Figure 2) as the number of lines increases and the spacing between lines decreases. Vorticity symmetry is maintained but the extrema migrate toward the center and toward the outer cylinder from the inner cylinder corners. The building of large vorticity gradients is noticed at the walls and at cell interfaces, the most predominant interface being the flow center jet location.

A vivid description of the secondary flow in Figure 3 is given by the crossflow streamline pattern of isolines. The increase in line density indicates the growth of the cellular vortex intensity with increasing Re. Streamline extrema migrate toward the flow center and toward the outer cylinder for increasing Re.
FIGURE 2. Vorticity contours: a, b, c, d ~ Re=100, 200, 300, 400: $\Delta \omega = 0.1$
FIGURE 3. Streamline contours: a, b, c, d ~ Re=100, 200, 300, 400; Δψ=0.0005
Streamline gradient growth at the center jet location indicates the strengthening of the radial jet with increasing Re. This major restructuring of the flow with increasing Re was noted by Fasel and Booz in their numerical solutions as confirming the experimental observations of the jetlike structure by Snyder and Lambert (1966) and Burkhalter and Koschmieder (1973). The Fourier-space integration of Hamidi and Georgini also revealed the presence of the jetlike structure, exhibited by a disproportionate growth in radial velocity in the source regions relative to the sink regions, as in the experiments of Burkhalter and Koschmieder.

The primary flow, azimuthal velocity contours are given in Figure 4. For the subcritical case, Re=100, the velocity head at the inner cylinder cascades smoothly down to zero flow at the outer walls in elliptical balance with the Couette secondary flow. For greater Re (200-400), u-contours become warped. At the cylinder midheight, the lines are bent increasingly outward for greater Re, while above and below the midheight, the lines are kinked inward toward the inner cylinder. This interesting result suggests that the primary flowfield is reshaped for higher Re by the accompanying vortex intensification in the correct mathematical direction by the associated vectored growth in the secondary flowfield. This is in agreement with the strict Taylor-vortex cell computations of Fasel and Booz who reported that u-contours were attracted to the inner wall by the radial jet and that the cell centers became depleted of isolines, meaning that u varies in the cell
FIGURE 4. Azimuthal velocity contours: a, b, c, d ~ Re=100, 200, 300, 400; \( \Delta u=0.1 \)
core very little relative to strong change near the walls so that the large Re fluid in the cell centers moves with almost constant velocity in the azimuthal direction. Hamidi and Georgini also found inner wall u-contour attraction and constant u-velocity in the cell centers. Clustering of the u-contours, at the inner and outer cylinder walls for increasing Re indicates boundary layer development which was also observed in the solutions of Fasel and Booz and of Hamidi and Georgini.

Further clarification of the secondary flowfield is given by the radial velocity contours in Figure 5 and the axial velocity contours in Figure 6.

In Figure 5, v-contour density in the central cell jet location increases for higher Re as the radial jet strength increases while the central cell size diminishes giving way to larger endcell sizes. Associated with the vortex growth, v-extrema move outward from the inner cylinder and toward the center. Radial velocity boundary layer development is indicated by v-contour clustering at cylinder endplates.

Similar Re effects of contour growth and extrema shift can be observed for the axial velocity contours in Figure 6. W-isoline growth along the inner cylinder shows how the axial flow necessarily pumps the radial jet region. Axial velocity boundary layer development is noticed on the inner and outer cylinder walls.

Three-dimensional Figures 7-11 provide scenic descriptions of the flowfield and its change with Re where the dependent variables are drawn as surfaces over the integration domain.
FIGURE 5. Radial velocity contours: a, b, c, d ~ Re=100, 200, 300, 400; \( \Delta v=0.005 \)
FIGURE 6. Axial velocity contours: a, b, c, d - Re=100, 200, 300, 400: \( \Delta w=0.005 \)
If there is a driving function in the numerical solution of the governing equations, it is the vorticity since it formally drives the crossflow Poisson equations and is present in each equation. In addition, the boundary vorticity update is crucial to the stability, convergence, and accuracy of any proposed solution method.

It is interesting that vorticity makes the most dramatic series of 3-d surface plots (Figure 7). For the laminar Re=100, the vorticity surface exhibits large gradients at the inner cylinder energy source and at the cylinder endplates, relatively mild gradients at the outer cylinder, and peaked average field extrema in the central cells. The supercritical result for Re=400 bears a marked contrast in that large gradients exist at all boundaries and the average vorticity canopy is full and well-rounded.

Streamfunction surfaces are given in Figure 8, where the higher Re effect appears as larger magnitude and steeper gradients for the average field. The valuable utility of the 3-d streamfunction observation is that it clearly shows how accurately the secondary flow problem has been solved in that both necessary boundary conditions on \( \psi \) are met. Streamfunction symmetry demonstrates the conservative elliptical balance that mathematically governs the secondary flow.

In Figure 9, Re effects on the tangential velocity surface are observed as the pushing of the \( u \)-contours outward toward the outer cylinder and upward in magnitude along the cylinder midheight associated with magnitude suppression on either side of the center ridge.
FIGURE 7. Vorticity surfaces: a,b - Re=100, 400
FIGURE 8. Streamfunction surfaces: a, b ~ Re=100,400
FIGURE 9. Azimuthal velocity surfaces: a, b, ~ Re=100,400
Figure 10 shows how the radial velocity field changes from magnitude symmetry between cells for the laminar flow to the heightened central jet flow for the Taylor vortex flow.

Differences in magnitude of the axial velocity field cells between the inner and outer cylinder pairs are lessened for the higher Re in Figure 11.

Figure 12a shows that near midheight azimuthal velocity profiles are smaller in magnitude or damped when compared to the 1-d analytic Couette flow solution (labelled U-Couette); presumably, this is a small aspect ratio effect. Figure 12b shows azimuthal velocity as a function of the radial coordinate at sequential axial locations (see key in Appendix) for Re=100 and h=2, or about 3× longer than for the solutions given above, where h=0.73. Compared to Figure 12a, the axially-stretched model solution of Figure 12b indicates that the near midheight solution approaches the analytic Couette solution. The endwall effect is observed for near endplate profiles for the axially-stretched case of Figure 12b, in that less magnitude damping occurs than for the shorter cylinder of Figure 12a.

The transition from laminar flow at Re=100 to Taylor-vortex flow for higher Re is noticed when comparing azimuthal velocity profiles in Figures 13a and 18a. In Figure 18a for Re=200, the profiles near the midheight take on reflex curvature and several profiles take on larger magnitudes than the 1-d analytic laminar Couette flow solution.
Thus, transition from laminar to Taylor-vortex flow appears to occur between Re=100 and Re=200. This observation is in agreement with a critical Re calculation (Re_{crit}=133 for d=0, Re_{crit}=227 for d≠0) based on Taylor's results, even though the Taylor calculations are not strictly applicable for the large gap case here. Detailed plots of the dependent variables as functions of radial and axial coordinates where explicit magnitudes and directions are given can be found in Figures 13-32 (graph key in Appendix).
FIGURE 10. Radial velocity surfaces: a, b ~ Re=100,400
FIGURE 11. Axial velocity surfaces: a, b ~ Re=100,400
FIGURE 12. Comparisons with exact Couette solution
FIGURE 13. Azimuthal velocity profiles: $u(r)$, $u(z)$ for $Re=100$
FIGURE 14. Radial velocity profiles: v(r), v(z) for Re=100
FIGURE 15. Axial velocity profiles: \( w(r), w(z) \) for \( Re=100 \)
FIGURE 16. Vorticity profiles: $\omega(r)$, $\omega(z)$ for $Re=100$
FIGURE 17. Streamfunction profiles: $\psi(r), \psi(z)$ for Re=100
FIGURE 18. Azimuthal velocity profiles: $u(r)$, $u(z)$ for $Re=200$
FIGURE 19. Radial velocity profiles: $v(r)$, $v(z)$ for $Re=200$
FIGURE 20. Axial velocity profiles: $w(r), w(z)$ for Re=200
FIGURE 21. Vorticity profiles: $\omega(r), \omega(z)$ for Re=200
FIGURE 22. Streamfunction profiles: $\psi(r)$, $\psi(z)$ for $Re=200$
FIGURE 23. Azimuthal velocity profiles: $u(r)$, $u(z)$ for $Re=300$
FIGURE 24. Radial velocity profiles: $v(r), v(z)$ for $Re=300$
FIGURE 25. Axial velocity profiles: $w(r), w(z)$ for $Re=300$
FIGURE 26. Vorticity profiles: $\omega(r), \omega(z)$ for $Re=300$
FIGURE 27. Streamfunction profiles: $\psi(r)$, $\psi(z)$ for Re=300
FIGURE 28. Azimuthal velocity profiles: $u(r)$, $u(z)$ for $Re=400$
FIGURE 29. Radial velocity profiles: \( v(r) \); \( v(z) \) for \( Re=400 \)
FIGURE 30. Axial velocity profiles: $w(r)$, $w(z)$ for $Re=400$
FIGURE 31. Vorticity profiles: $\omega(r)$, $\omega(z)$ for $Re=400$
FIGURE 32. Streamfunction profiles: $\psi(r), \psi(z)$ for $Re=400$
TURBULENT SOLUTIONS

Turbulent flow equations

Assuming long cylinders, the Navier-Stokes equations may be reduced to a single transport equation for the azimuthal velocity that is appropriate for a turbulent solution based on an effective isotropic eddy viscosity hypothesis. The governing equations in nondimensional form may be written as:

\[ \nu = w = \frac{\partial}{\partial z} = \frac{\partial}{\partial \theta} = 0 \]

\[ \frac{\partial u}{\partial t} = \frac{1}{r^2} \frac{\partial}{\partial r} \left[ (\nu + \nu_t) r^2 \frac{\partial}{\partial r} (u) \right] \]

where \( \nu = \frac{\nu}{U_i R_i} \) and \( \nu_t = \frac{\nu_t}{U_i R_i} \)

Subsequent to the Roberts transformation, the transport equation may be written as:

\[ \frac{\partial u}{\partial \xi} = \frac{f_2}{r^2} \frac{\partial}{\partial \eta} \left[ f_2 r^2 (\nu + \nu_t) \frac{\partial}{\partial \eta} (u) \right] \]

The closure method to model the effective turbulent stress is based on a Prandtl-Van Driest algebraic turbulence model formulation. The relevant modelling equations may be written in dimensional form, neglecting overbars, such that:

\[ \nu_t = \lambda^2 \left| \frac{\partial u}{\partial r} - \frac{u}{r} \right| \]

\[ \lambda_i = \lambda (r - r_i) [1 - \exp(-D_i)] \]

\[ \lambda = C (r_o - r_i) \]
\[ \lambda_o = \kappa [r_o - r][1 - \exp(-D_o)] \]

where

\[ D_i = \left[ \frac{[r - r_i]}{26\nu} \right] \left[ \frac{\tau_{w,i}}{\rho} \right]^{0.5} \]

\[ \tau_{w,i} = \mu \left| \frac{\partial u}{\partial r} - \frac{u}{r} \right|_{iw} \]

and

\[ D_o = \left[ \frac{[r_o - r]}{26\nu} \right] \left[ \frac{\tau_{w,o}}{\rho} \right]^{0.5} \]

\[ \tau_{w,o} = \mu \left| \frac{\partial u}{\partial r} - \frac{u}{r} \right|_{ow} \]

where the nomenclature has been generalized so that the subscripts (i,o) for mixing lengths and Van Driest damping functions refer to near-wall regions for the respective cylinders. Here, the mixing length is assumed to vary linearly with distance with suitable damping from the wall in the wall regions (\( \lambda_i, \lambda_o \)) and to be constant in the outer wall regions (\( \lambda \)) as in the usual practice of algebraic turbulence modelling.

For the transformed case, the modelling equations take the following nondimensional form:

\[ \nu_t = \lambda^2 \left| \frac{\partial u}{\partial \eta} - \frac{u}{r} \right| \]

\[ \lambda_i = \kappa [r - r_i][1 - \exp(-D_i)] \]

\[ \lambda = C[r_o - r_i] \]

\[ \lambda_o = \kappa [r_o - r][1 - \exp(-D_o)] \]
Solution method

The transport equation was differenced using second-order central space differences and first-order forward time differences. The solution was marched in time to steady-state using Dirichlet boundary conditions of inner cylinder rotation and outer cylinder no-slip, such that \( u_i = 1 \) and \( u_o = 0 \). The numerical method employed was implicit and analogous to the ADI scheme used for the nonturbulent solutions resulting in tridiagonal matrices. Numerical solutions were obtained for 100 mesh points in the radial direction for the same convergence criterion as in the nonturbulent case.

Conservative differencing of the viscous term was crucial to the diagonal dominance of the matrix coefficients and the stability of the numerical convergence procedure. Turbulent numerical solutions were obtained to model the geometry and Re for several known experiments of turbulent flow in circular Couette flow machines. The outer region constant was selected to best fit the trends of the available data. By strict analogy to external boundary layers, the empirical constant (C) should be taken as 0.0895. However, for circular Couette flow it is not clear how to interpret \( \delta \). In a computational study of annular,
turbulent flow with rotating core, Sharma, Launder and Scott (1976) used \((r_o-r_i)/2\) as the relevant length scale and used a constant of 0.14 for essentially an axial pipe flow with swirl. In this study, it was assumed that an appropriate scale may be \((r_o-r_i)\). The empirical constant that best fit the data trends was found to be 0.11.

**Results comparison and discussion**

Numerical solutions were obtained and compared to experiment for six cases. Table 2 outlines salient aspects of the experiments.

**TABLE 2. Turbulent circular Couette flow experiments**

<table>
<thead>
<tr>
<th>Name</th>
<th>Fluid</th>
<th>Re</th>
<th>(r_o)</th>
<th>h</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frank (1987)</td>
<td>air</td>
<td>509K</td>
<td>1.49</td>
<td>0.73</td>
</tr>
<tr>
<td>Greeley and Iversen (1983)</td>
<td>air</td>
<td>462K</td>
<td>1.49</td>
<td>0.73</td>
</tr>
<tr>
<td>Ustimenko et al. (1972)</td>
<td>water</td>
<td>2830K</td>
<td>1.46</td>
<td>unknown</td>
</tr>
<tr>
<td>Wendt1 (1933)</td>
<td>water</td>
<td>180K</td>
<td>1.47</td>
<td>5.0</td>
</tr>
<tr>
<td>Wendt2 (1933)</td>
<td>water</td>
<td>281K</td>
<td>1.18</td>
<td>4.0</td>
</tr>
<tr>
<td>Wendt3 (1933)</td>
<td>water</td>
<td>341K</td>
<td>1.07</td>
<td>3.64</td>
</tr>
</tbody>
</table>

Comparisons between numerical and experimental results are given in Figures 33-38. In Figure 33a, the numerical solution for the primary velocity profile is seen to overpredict the data of Greeley and Iversen. In Figure 33b, the discrepancy is more prominent for the data of Frank. In both Figures 33a and 33b, the curvature of the high-gradient boundary layer regions seems to be well represented in the numerical solutions.
FIGURE 33. Turbulent profile comparison to experiments
For the data of Ustimenko et al. in Figure 34a, the boundary layer comparison is much worse so that the numerical solution trend in the near-wall region does not reflect the data, though in the core region a reasonable comparison is observed. A similar comparison is made for the data of Wendt1 in Figure 34b, where excellent agreement was obtained in the core and rather poor agreement in the boundary layers. However, for the small gap data of Wendt2 and Wendt3 in Figures 35a and 35b, there is extremely poor comparison between numerical and experimental results.

The effects of streamline curvature and of system rotation on turbulent boundary layers may be important and various correction methods have been employed to model these effects (Bradshaw, 1973, 1978 and Nallasamy, 1987). Usually, the corrections take the form of a parameter that is used to multiply the normal mixing length. Frequent reference has been made to the curvature correction formula of Bradshaw (1973) where a form of gradient Richardson number is used. In Turbulence (1978, ed. P. Bradshaw), Johnston reports that Eide and Johnston (1974) successfully used a similar parameter to correct for rotation effects based on a rotation number and developed a two parameter expression that models both effects. Unfortunately, such correction methods involve the specification of additional empirical constants.

Both of the above mentioned methods were attempted in the present study and were found to cause divergence or clearly unacceptable
FIGURE 34. Turbulent profile comparison to experiments
FIGURE 35. Turbulent profile comparison to experiments
solutions (solutions that appeared more like nonturbulent solutions). Since it was possible that the above methods were inappropriate for the flow of current study, a more empirical approach was followed. It was assumed that the normal mixing length may be scaled by a simple function of a relevant geometric length scale, namely, the radius of curvature.

Various functional representations of the radius as a mixing length multiplier were tested including: \( r, \sqrt{r}, \log r, \log r, \sqrt{\log r}, \sqrt{\frac{1}{r}}, \sqrt{\frac{0.5}{r}}, \sqrt{\frac{0.25}{r}}, \sqrt{\frac{0.375}{r}} \). It was found that the best agreement with the available data could be obtained using a multiplier of \( r^{0.5} \) such that:

\[
\lambda_c = \lambda r^{0.5}
\]

Figures 36-38 contain comparisons based on numerical solutions where the above correction factor multiplier had been used to modify the mixing length expressions.

Compared to the baseline results, much better comparison between the numerical solution and the data are observed for the small aspect ratio cases of Greeley and Iversen in Figure 36a and of Frank in Figure 36b, though there is some tradeoff here in that the outer cylinder boundary layer is less well represented. A somewhat better comparison was obtained for the data of Ustimenko et al. in Figure 37a, again with greater discrepancy in the outer boundary layer region. The effect of the correction worsens the comparison to the data of Wendt_1 in Figure 37b. Little difference to the baselines results is observed in the solution comparisons to the data of Wendt_2 and Wendt_3 in Figures 38a and 38b.
FIGURE 36. Turbulent profiles with correction
Re=2830K, rout=1.46
Ustimenko et al. ○
Algebraic model △

Re=180K, rout=1.47
Wendt ○
Algebraic model △

FIGURE 37. Turbulent profiles with correction
FIGURE 38. Turbulent profiles with correction
Correct interpretation of the results comparison is imperative and difficult in that questions about the experimental error and testing must be addressed as well as the limitations of the numerical solution theory.

Numerical solutions were obtained for a 1-d model equation that assumes long cylinders. The normal or baseline mixing length turbulence model adequately predicts the core flow for the large aspect ratio data of Wendt. Similar agreement was obtained for the data of Ustimenko et al., which suggests experiments with long cylinders. Thus, the core region is adequately modelled without correction for large aspect ratios and large gap. For the small aspect ratio data of Greeley and Iversen and of Frank, however, the numerical solutions overpredict the experimental data. For all available data, the normal 1-d model does not adequately represent the near-wall region boundary layers where secondary flow, curvature and rotation effects are not modelled and may be important.

The applied mixing length multiplier correction appears to improve the baseline prediction of the small aspect ratio data for the core region and for the inner cylinder boundary layer with slight loss of resolution in the outer cylinder boundary layer. The correction worsens the baseline prediction for the large aspect ratio experimental data.

Based on the limited data studied here, it appears that the mixing length multiplier adequately corrects for the effect of small aspect
ratio. The correction has little or no influence in modelling the
effects of curvature, rotation and secondary flow. The correction may
worsen the core region representation given by the normal mixing length
model. Another interesting possibility may be that the presence of the
measuring probes permitted large experimental errors near walls for all
data and in the core for small aspect ratio and small gap experiments.
Peculiarities in the data of Wendt have been remarked on previously in
the literature. It would seem to be very difficult to make accurate
pressure measurements using a pitot probe in a small gap. The accuracy
of the small gap data of Wendt₂ and Wendt₃ may be questionable when
compared to the numerical and experimental agreement found in all other
cases studied here. This may point to experimental measurement errors
associated with pitot probe presence for the small gap cases of Wendt.

Figure 39 displays typical eddy viscosity profiles from the 1-d
numerical turbulent solutions. Table 3 outlines the convergence
history for the 1-d turbulent numerical solutions.
FIGURE 39. Eddy viscosity profiles: a, b ~ Baseline, Corrected
TABLE 3. 1-d turbulent numerical solution convergence history / NAS 9160 computer

<table>
<thead>
<tr>
<th>Re</th>
<th>$r_o$</th>
<th>Grid</th>
<th>CPU time (sec)</th>
<th>Time steps ($\Delta t=1.0$)</th>
<th>CPU time (sec)</th>
<th>Time steps ($\Delta t=1.0$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>180K</td>
<td>1.47</td>
<td>100</td>
<td>1.47</td>
<td>10</td>
<td>1.368</td>
<td>9</td>
</tr>
<tr>
<td>281K</td>
<td>1.18</td>
<td>100</td>
<td>1.18</td>
<td>6</td>
<td>0.691</td>
<td>6</td>
</tr>
<tr>
<td>341K</td>
<td>1.07</td>
<td>100</td>
<td>1.07</td>
<td>3</td>
<td>0.274</td>
<td>3</td>
</tr>
<tr>
<td>462K</td>
<td>1.49</td>
<td>100</td>
<td>1.49</td>
<td>19</td>
<td>0.2739</td>
<td>16</td>
</tr>
<tr>
<td>509K</td>
<td>1.49</td>
<td>100</td>
<td>1.49</td>
<td>21</td>
<td>0.2935</td>
<td>17</td>
</tr>
<tr>
<td>2830K</td>
<td>1.46</td>
<td>100</td>
<td>1.46</td>
<td>67</td>
<td>10,050</td>
<td>55</td>
</tr>
</tbody>
</table>
TURBULENT DEPARTURES

Several 3-d equation sets to model turbulent circular Couette flow have been derived and tested. Though the methods show promise, to date no converged solutions have been obtained. Work is continuing on these methods. The 3-d azimuthal-invariant turbulent equations of motion and turbulence modelling equations will be presented here. There will be a discussion of techniques attempted in the solution of the equations followed by possible recommendations for additional study.

Analytic extension of the 3-d nonturbulent equations of motion permits the following analogous set of 3-d turbulent governing equations written in a nondimensional vorticity-velocity formulation:

Let \( \Lambda = \nu = \frac{\nu}{U_i R_i} \) and \( T = \nu_t = \frac{\nu_t}{U_i R_i} \)

\[
\frac{\partial \omega}{\partial t} + v \frac{\partial \omega}{\partial r} + w \frac{\partial \omega}{\partial z} - \frac{1}{r} \left( v \omega + 2 w \frac{\partial u}{\partial z} \right) = (\Lambda + T) \left( \nabla^2 \omega - \frac{\omega}{r} \right) + S_{\omega_t}
\]

\[
\frac{\partial u}{\partial t} + v \frac{\partial u}{\partial r} + w \frac{\partial u}{\partial z} + \frac{u v}{r} = \frac{1}{r^2} \frac{\partial}{\partial r} \left[ (\Lambda + T) r \frac{\partial u}{\partial r} \right] + \frac{\partial}{\partial z} \left[ (\Lambda + T) \frac{\partial u}{\partial z} \right]
\]

\[\nabla^2 v - \frac{v}{r^2} = \frac{\partial \omega}{\partial z}\]

\[\nabla^2 w = - \frac{\partial \omega}{\partial r} - \frac{\omega}{r}\]

where

\[
S_{\omega_t} = \left[ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial r} \right] \left[ \frac{\partial^2 T}{\partial z^2} - \frac{1}{r} \frac{\partial T}{\partial r} - \frac{\partial^2 T}{\partial r^2} \right] - 2 \frac{\partial T}{\partial r} \left[ \frac{\partial^2 w}{\partial z^2} + \frac{\partial^2 w}{\partial r^2} \right] + 2 \frac{\partial^2 T}{\partial z \partial z} - \frac{v}{r^2} - 2 \frac{\partial w}{\partial z} \frac{\partial^2 T}{\partial r \partial z}
\]
The isotropic eddy viscosity hypothesis was employed in an algebraic mixing length turbulence model. The following Prandtl-Van Driest formulation of modelling equations was developed:

\[ \nu_t = \lambda^2 |G| \]

Where it was assumed that the mixing length may be averaged following a suggestion by Patankar et al. (1979) such that:

\[ \frac{1}{\lambda} = \frac{1}{\lambda_r} + \frac{1}{\lambda_z} \]

where:

\[ G = \left[ G_r \theta + G_z \theta \right]^{0.5} \]

\[ G_r \theta = \frac{\partial u}{\partial r} - \frac{u}{r} \]

\[ G_z \theta = \frac{\partial u}{\partial z} \]

Where the mixing lengths in the radial direction are given by:

\[ \lambda_{r,i} = \kappa [r - r_i] [1 - \exp(-D_{r_i})] \]

\[ \lambda_{r,O} = \kappa [r_O - r] [1 - \exp(-D_{r_O})] \]

\[ \lambda_r = C [r_O - r_i] \]

\[ D_{r_i} = \left[ \frac{[r - r_i]}{26} \right] \left[ \text{Re} \left| \frac{\partial u}{\partial r} - \frac{u}{r} \right| \right]^{0.5} \]

\[ D_{r_O} = \left[ \frac{[r_O - r]}{26} \right] \left[ \text{Re} \left| \frac{\partial u}{\partial r} - \frac{u}{r} \right| \right]^{0.5} \]
Where the mixing lengths in the axial direction are given by:

\[
\lambda_{z_i} = \kappa z [1 - \exp\left(-\frac{D_{z_i}}{D_{z_i}}\right)]
\]

\[
\lambda_{z_0} = \kappa [h - z] [1 - \exp\left(-\frac{D_{z_0}}{D_{z_0}}\right)]
\]

\[
\lambda_z = Ch
\]

\[
D_{z_i} = \left[ \frac{z}{26} \right] \left[ \text{Re} \left| \frac{\partial u}{\partial z} \right|_{iw} \right]^{0.5}
\]

\[
D_{z_0} = \left[ h - \frac{z}{26} \right] \left[ \text{Re} \left| \frac{\partial u}{\partial z} \right|_{ow} \right]^{0.5}
\]

where the nomenclature has been generalized to admit the following interpretation for turbulence quantities in the axial direction:

- \( i \) inner endplate region near \( z = 0 \)
- \( o \) outer endplate region near \( z = h \)
- \( iw \) inner endplate wall value
- \( ow \) outer endplate wall value

In order to maintain numerical stability at the high Reynolds numbers of turbulence, a hybrid central-upwind differencing scheme was employed for spatial derivatives. Whenever possible, space derivatives were differenced using second-order central differences. Convective terms were differenced using first-order weather differencing whenever a central difference representation produced a diagonalization or viscous modelling problem in any matrix row or column operation. As
for the nonturbulent solutions, an implicit ADI numerical method was used.

To date, no converged solutions have been obtained. The numerical calculations remain stable but oscillate about what appears to be appropriate steady-state values. For a mid-domain azimuthal velocity point, the solution oscillates between values of 0.8 and 0.0. A correct value based on data should be around 0.4. In addition, azimuthal velocity profiles (u(r)) taken inside the oscillation when u<0.4 appear very similar to the form of expected turbulent profiles. Thus, the numerical solutions appear to correctly predict the primary flow in an average sense. Sometimes, such transient solutions may be correct solutions of the Navier-Stokes equations. Clearly, here, this is not the case since it does not make sense that the primary flow velocity would cyclically damp to zero flow in the center of a Couette flow machine. Thus, the oscillations appear to stem from some numerical problem associated with the specification of the turbulent eddy viscosity (as the code has been run with zero turbulent viscosity for low-Re producing accurate, converged nonturbulent solutions). The problem may be with the turbulent viscosity update or with the nonconservative form of the source term in the vorticity transport equation.

Various techniques have been applied to modify the method in attempts to obtain a converged solution. The following list outlines the major areas of such numerical experimentation:

1. Refining the mesh.
2. Adjusting the time step size.
3. Starting from a laminar solution.
4. Implicitizing source terms by appropriate inclusion in diagonal terms.
5. Smoothing or averaging of the eddy viscosity after each time step.
6. Under- and over-relaxing the eddy viscosity.
7. Only updating the eddy viscosity periodically.

Since the above reasonable methods do not influence the oscillatory solution behavior, the problem may be more fundamental and possibly associated with the 3-d mixing length theory attempted here or with the nonconservative form of the source terms in the vorticity transport equation. The analysis and work continue in progress, including an extension via an energy-dissipation turbulence model.
A low-Reynolds number, energy-dissipation turbulence model that includes corrections for the effects of streamline curvature and rotation has been developed following the suggestions of Launder and Spalding (1974). The additional transport equations for the isotropic turbulent energy dissipation rate ($e$) and the turbulent kinetic energy ($k$) may be given by the following nondimensional equations:

\[
\frac{\partial e}{\partial t} + \nabla \cdot \left( \nabla e \right) = \frac{1}{r} \frac{\partial}{\partial r} \left[ r \left( \Lambda + \frac{T}{\sigma_e} \right) \frac{\partial e}{\partial r} \right] + \frac{\partial}{\partial z} \left[ \left( \Lambda + \frac{T}{\sigma_e} \right) \frac{\partial e}{\partial z} \right] + S_{et}
\]

\[
S_{et} = C_1 P_k T \frac{e}{k} - C_2 \frac{e^2}{k} - C_3 \Lambda T \left[ \frac{\partial}{\partial r} \left( \sqrt{P_k} \right) \right]^2 - C_4 \Lambda T \left[ \frac{\partial}{\partial z} \left( \sqrt{P_k} \right) \right]^2
\]

\[
\frac{\partial k}{\partial t} + \nabla \cdot \left( \nabla k \right) = \frac{1}{r} \frac{\partial}{\partial r} \left[ r \left( \Lambda + \frac{T}{\sigma_k} \right) \frac{\partial k}{\partial r} \right] + \frac{\partial}{\partial z} \left[ \left( \Lambda + \frac{T}{\sigma_k} \right) \frac{\partial k}{\partial z} \right] + S_{kt}
\]

\[
S_{kt} = P_k T - e - C_5 \Lambda \left[ \frac{\partial}{\partial r} \left( \sqrt{k} \right) \right]^2 - C_6 \Lambda \left[ \frac{\partial}{\partial z} \left( \sqrt{k} \right) \right]^2
\]

where $e = \frac{R_i}{U_i} - e$ and $k = \frac{K}{U_i}$

where $T$ is given by the Kolmogorov - Prandtl relation such that:

\[
T = \frac{C_\nu}{R_i U_i} \left( \frac{k^2}{e} \right)
\]

where $C_\nu = 0.09 \exp \left[ \frac{-3.4}{\left[ 1 + \frac{Re_t}{50} \right]^2} \right]$
where the turbulent Reynolds number is defined such that:

\[ \text{Re}_t = \text{Re} \frac{k^2}{\varepsilon} \]

where constants and parameters are given by:

\[ C_1 = 1.44[1 + 0.9R_f] \]

\[ C_2 = 1.92[1 - 0.3\exp\left(-\text{Re}_t^2\right)][1 - C_c R_i] \]

\[ R_f = \frac{2u \frac{\partial r}{\partial r}(r)}{p_k} \]

\[ R_i = \frac{k^2 u}{e^2} r \frac{\partial}{\partial r}(ru) \]

where \( \sigma_k = 1.0 \) and \( \sigma_e = 1.3 \)

\[ C_3 = C_4 = 2.0 \]

\[ C_5 = C_6 = 2.0 \]

\[ C_c = 0.2 \]

where the production term is given by:

\[ p_k = 2 \left[ \left( \frac{\partial w}{\partial z} \right)^2 + \left( \frac{\partial v}{\partial r} \right)^2 + \left( \frac{\partial v}{r} \right)^2 \right] + \left( \frac{\partial w}{\partial r} + \frac{\partial v}{\partial z} \right)^2 + \left( \frac{\partial u}{\partial z} \right)^2 + \left[ r \frac{\partial}{\partial r}(ru) \right]^2 \]

The above modelling equations were assembled and extended from suggestions in the literature of turbulence modelling. The extra destruction terms in the transport equations have been included to simulate the low-Re effect in near-wall regions following the work of
Launder and Spalding (1974), Launder et al. (1977) and Sharma (1979). The gradient Richardson number has been used by Launder et al. (1977) to correct for the effect of streamline curvature. The flux Richardson number has been used by Rodi (1979), as reported by Nallasamy (1987), to correct for the effect of system rotation. The production term appears as given by Gupta (1984), Ramos (1981) and Nallasamy (1987). The remaining constants are the usual ones prescribed in the literature.

Boundary conditions for the energy-dissipation transport equations depend in general on the particular formulation of the equations. For the usual high-Re forms of the equations, $k$ and $e$ are usually prescribed at the near-wall from considerations based on the law of the wall as in Kubo and Gouldin (1975) and in Rodi and Scheurer (1983). For low-Re formulations, the boundary conditions are $k = e = 0$ at the wall as given by Sharma (1979), Launder and Spalding (1974) and Jones and Launder (1972).
CONCLUSIONS

A primitive method for the Navier-Stokes equations has been developed and has been shown to provide accurate 3-d Couette-Taylor flow solutions modelling plausible physical realizations for finite-length models at modest cost.

A vorticity-velocity formulation has been shown to be preferable to a vorticity-streamfunction method.

The number of Taylor-vortex cells remained at two for subcritical and supercritical Taylor numbers tested here. This may suggest that the cells do not mutate to a multiple-cell steady-state Taylor motion that has more than just two cells for increasing Re for the finite-length case of large gap and small aspect ratio.

A Prandtl-Van Driest turbulence model has been applied to circular Couette flow resulting in accurate 1-d turbulent flow solutions for the core flow for long cylinders. The normal model does not adequately resolve the inner and outer cylinder boundary layers, where the effects of streamline curvature, system rotation and secondary flow may be important.

A small aspect ratio correction has been applied to the normal turbulence model resulting in much better numerical solution comparison to experiment for the core and some improvement in boundary layer regions. The small aspect ratio correction is a mixing length multiplier that is the square root of the radius of curvature such that: $\lambda_c = \lambda r^{0.5}$. 
A 3-d turbulence model based on the Prandtl-Van Driest formulation, which extends the governing equations and numerical method of the nonturbulent solutions, has been developed and shows promise. Additional analysis and numerical work is necessary to produce solutions that model realistic physical simulations.

Additional transport equations for an energy-dissipation 3-d turbulence model that include corrections for streamline curvature, system rotation and low-Re effects have been developed.
APPENDIX

Key to graphs

For \( f(r) \) in Figures 13-32, graph symbols 2-11 correspond to increasing sequential \( z \)-locations from the cylinder endplate (\( z=0 \)) to the approximate cylinder midheight (\( z=h/2=0.3652 \)) such that:

\[
\begin{align*}
2 & \sim 0.003925 \\
3 & \sim 0.01416 \\
4 & \sim 0.02853 \\
5 & \sim 0.04842 \\
6 & \sim 0.07544 \\
7 & \sim 0.1112 \\
8 & \sim 0.1568 \\
9 & \sim 0.2125 \\
10 & \sim 0.2770 \\
11 & \sim 0.3472
\end{align*}
\]
For f(z) in Figures 13-32, graph symbols 2-11 correspond to increasing sequential r-locations from the inner cylinder (r=1) to the approximate annulus center (r=[r_1+r_0]/2=1.246) such that:

2 ~ 1.003
3 ~ 1.010
4 ~ 1.019
5 ~ 1.033
6 ~ 1.051
7 ~ 1.075
8 ~ 1.106
9 ~ 1.143
10 ~ 1.186
11 ~ 1.234

Note: For continuity of the graphical presentations, the 40x40 solution for Re=100 was used.
ACKNOWLEDGEMENTS

The author gratefully acknowledges the thoughtful time, collegial discussions and suggestions of the members of his doctoral committee: A. Abian, R. G. Hindman, J. D. Iversen, W. D. James, A. K. Mitra and R. H. Fletcher. It has been a privilege to have studied under the supervision of J. D. Iversen, who has served as the author's major professor and academic advisor over the years at Iowa State. Discussions on this work with fellow graduate students are gratefully acknowledged. Editorial improvements to the manuscript by L. F. Bishop are noted with special thanks. The presentation of the results has benefited from technical consultations with M. S. Beck, P. E. Fox and K. A. Plagge. The author is thankful for professional courtesies extended by the staff of the Office of Editorial Services at Iowa State University. Resources that made this research possible have been made available by the Engineering Research Institute and Iowa State University, for which the author is deeply grateful.


C-S Lin. "Diffusion-deposition of particles from a continuous line source into a disturbed atmospheric boundary layer." Diss., Iowa State Univ., 1986.


