Paper presented at the 16th Annual Northeast Bioengineering Conference, March 26-27, 1990, Penn State University, University Park, PA

COMPUTER SIMULATION OF CONVECTIVE COOLING EFFECTIVENESS INSIDE TURNING PASSAGES

Seungsoo Lee and George S. Dulikravich
Department of Aerospace Engineering
233 Hammond Building
Pennsylvania State University
University Park, PA 16802

Abstract

A complete mathematical model for laminar steady incompressible flows including heat transfer was presented. It was demonstrated that accurate and efficient prediction of detailed flow field and thermal field is computationally feasible for arbitrary passages. Fluids with higher Prandtl numbers offer clearly better cooling characteristics.

Incompressible Navier-Stokes Equations with Heat Transfer

Incompressible laminar flow Navier-Stokes equations can be written in a general curvilinear ξ, η non-orthogonal coordinate system as

$$\frac{\partial\tilde{Q}}{\partial t} + \frac{\partial\tilde{E}}{\partial\xi} + \frac{\partial\tilde{F}}{\partial\eta} = D^2(J\tilde{Q}) \qquad (1)$$

where

$$\tilde{Q} = \frac{1}{J}\begin{bmatrix} \frac{p}{\beta} \\ u \\ v \\ \theta \end{bmatrix} \quad \tilde{E} = \frac{1}{J}\begin{bmatrix} U \\ Uu+\xi_x p \\ Uv+\xi_y p \\ U\theta \end{bmatrix} \quad \tilde{F} = \frac{1}{J}\begin{bmatrix} V \\ Vu+\eta_x p \\ Vv+\eta_y p \\ V\theta \end{bmatrix}$$

where u, v are the velocity vector components, p is the sum of hydrostatic pressure and hydrodynamic pressure, $\theta = \frac{T - T_c}{\Delta T_c}$ is the non-dimensional temperature. Notice that the artificial compressibility term, $\frac{p}{\beta}$ has been added in the continuity equation [1]. Here, J is the Jacobian determinant, while U and V are the contravariant velocity vector components normal to constant ξ and η grid line, respectively.

The physical viscous terms are contained in

$$D^2(J\tilde{Q}) = \left[\frac{S}{J}g_{ij}\left(J\tilde{Q}\right)_{,j}\right]_{,i} \qquad (2)$$

where g_{ij} is the contravariant metric tensor, $g_{ij} = \nabla x'_i \nabla x'_j$.

$$S = \frac{1}{Re}\begin{bmatrix} 0 \\ 1 \\ 1 \\ \frac{1}{Pr} \end{bmatrix} \qquad (3)$$

Here, Re is the Reynolds number, $Pr = \frac{\kappa}{\nu}$ is the Prandtl number, κ is the thermal diffusivity, and ν is the kinematic viscosity. Equations (1) do not have to be solved simultaneously. However, in this work they were solved simultaneously.

Results

Based on the above mathematical formulation, a computer program was written in Fortran using explicit finite differencing. Numerical results were obtained on NASA Ames NAS CRAY-YMP by using $\beta = 1.0$. The CFL number was 2.8 and our new acceleration algorithm [2,3,4,5,6] in conjunction with Runge-Kutta time stepping method [7] was used to reduce the CPU time which was typically ten minutes until machine accuracy was achieved.

The first test case represented developing flow of a cold fluid through a hot-wall straight channel with Re = 10 and Pr = 1.0. Numerical results (Figs. 1 and 2) show excellent agreement with analytic results thus verifying the code.

The second test case was a U-shaped channel having hot walls and a steady developing flow of an initially cold fluid with Re = 10 and Pr = 0.7 which is typical for gases. Figs. 3 and 4 depict the developing flow field, while Fig. 5 demonstrates that the fluid has been completely heated soon after the entrance.

The third test case was the same U-shaped channel and a developing flow with Re = 5 and Pr = 13 which is typical for water at room temperature. Fig. 6 demonstrates a rapidly growing thermal boundary layer on the hot walls of the channel. The fluid now continues to cool the channel over a longer distance.

The fourth test case was the same U-shaped channel and a developing flow with Re = 5 and Pr = 153 which is typical for certain heavier oils. Fig. 7 demonstrates very slowly growing thermal boundary layers on the hot walls of the channel. This type of fluid continues to cool the channel over a very large distance.

References

[1] Chorin, A. J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," Journal of Computational Physics, Vol. 2, 1967, pp. 12-26.

[2] Lee, S., Dulikravich, G. S. and Dorney, D., "Distributed Minimal Residual (DMR) Method for Explicit Algorithms Applied to Nonlinear Systems," presented at the Conference on Iterative Method for Large Linear Systems, Austin, Texas, Oct. 19-21, 1988.

[3] Lee, S. and Dulikravich, G. S, "Acceleration of Iterative Algorithms for Euler Equations of Gasdynamics," AIAA Paper 89-0097, Reno, NV, Jan. 1989 a; Also, accepted for publication in AIAA Journal.

[4] Lee, S. and Dulikravich, G. S., "Accelerated Computation of Viscous, Steady Incompressible Flows," ASME paper 89-GT-45, Gas Turbine and Aeroengine Congress and Exposition, Toronto, Canada, June 4-8, 1989 b.

[5] Lee, S. and Dulikravich, G. S., "A Fast Iterative Algorithm for Incompressible Navier-Stokes Equations," Proceedings the 10th Brazilian Congress of Mechanical Engineering, Rio de Janeiro, Dec. 7-10, 1989 c.

[6] Lee, S., Acceleration of Iterative Algorithms for Euler and Navier-Stokes Equations, Ph. D. Thesis, The Department of Aerospace Engineering, The Pennsylvania State University, May 1990.

[7] Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Scheme," AIAA paper 81-1259, Palo Alto, CA,

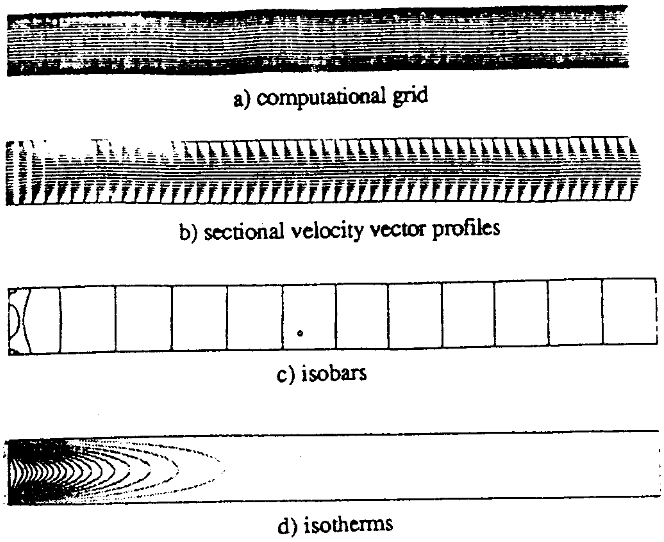

Fig. 1 Developing flow in a straight channel

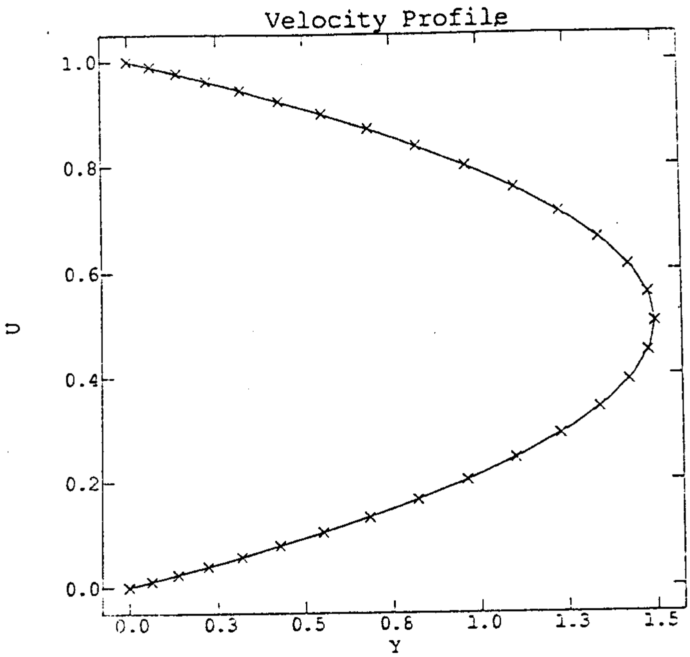

Fig. 2 Fully developed velocity profiles in a straight channel (---) analytical; (x-x-x) numerical.

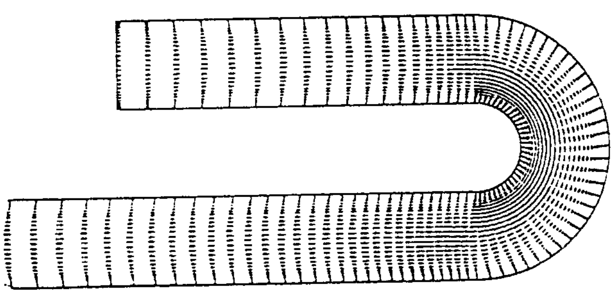

Fig. 3 Velocity vector profiles in a developing flow in a U-shaped channel with Re = 10.

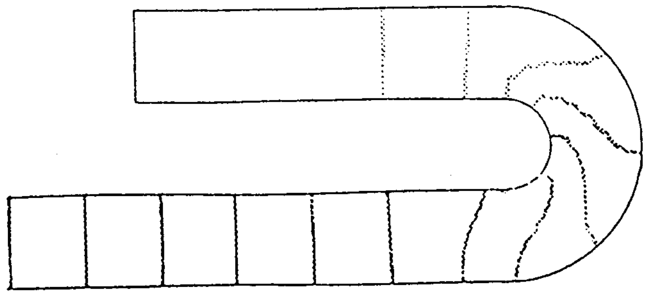

Fig. 4 Isobars in a developing flow in a U-shaped channel with Re = 10.

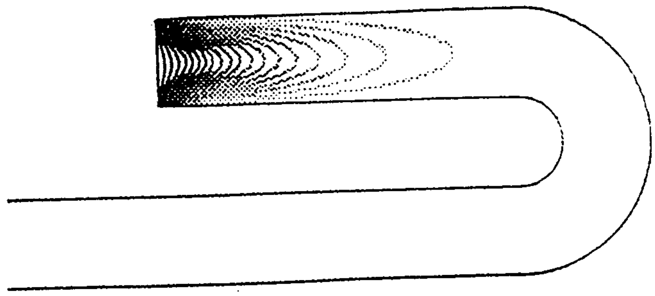

Fig. 5 Isotherms in a developing flow in a U-shaped channel with Re = 10 and Pr = 0.7.

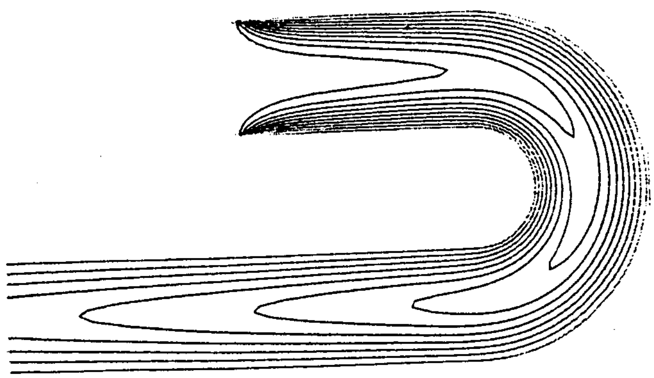

Fig. 6 Isotherms in a developing flow in a U-shaped channel with Re = 5 and Pr = 13.

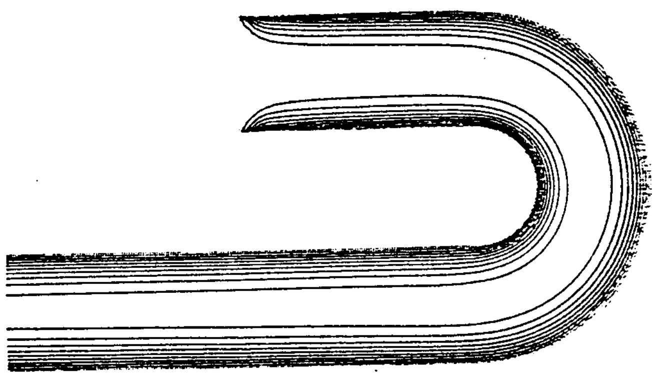

Fig. 7 Isotherms in a developing flow in a U-shaped channel with Re = 5 and Pr = 153.