EG0800331

Computational Study of Couette Flow Between Parallel Plates for Steady and Unsteady Cases

Y. Rihan

Atomic Energy Authority, Hot Lab. Centre, Egypt

ABSTRACT

Couette flow between parallel plates is a classical problem that has important applications in various industrial processing. In this investigation an analytical solution was obtained to predict the steady and unsteady Couette flow between parallel plates. One of the plates was stationary and the other plate moved with constant velocity. The governing partial differential equations were solved numerically using Crank-Nicolson implicit method to represent the flow behavior of the fluid.

***Key Words: Couette flow/ modeling/ parallel plates/ steady/ unsteady*.**

INTRODUCTION

Couette flow is a classical problem of primary importance in the history of fluid mechanics (1-4), which is a typical example of exact solutions for Navier-Stokes equation. Couette flow is perhaps the simplest of all viscous flows, while at the same time retaining much of the same physical characteristics of a more complicated boundary-layer flow. One of the most important process in industry is extrusion. Since the gap between the barrel and the screw of extruder is small, assuming a fluid flowing between parallel plates leads to representative results. There exist a large number of parameters in extrusion process which influence significantly the production rate and the quality of the final product.

Couette flow between parallel plates is a classical problem that has important applications in power generators and pumps, etc. Several investigations have been done in this type of flow (5-10). Etemad et al. (11) solved the simultaneously developed case of the motion and energy equation for power law fluid between parallel stationary plates when the variation of viscosity with temperature and viscous dissipation could not be neglected. They solved the problem numerically using finite element method and, as a special case, calculated the flow and heat transfer characteristics for fully developed conditions. The Couette flow with slip boundary conditions was studied by Thompson and Troian (12) using molecular dynamics simulations for steady-state flows. The slip length dependence on the shear rate at the wall was quantified in their work. However, there is few reports about the unsteady behavior of Couette flow. In the current work, the unsteady and steady velocities profile for Couette flow will be presented and solved exactly using the numerical method.

THE GOVERNING EQUATIONS AND NUMERICAL SOLUTION

Consider the viscous flow between two parallel plates separated by the vertical distance D, as shown in Fig. 1. The upper plate is moving at the velocity u_e, and the lower plate is stationary. The flow field between the two plates is driven exclusively by the shear stress exerted on the fluid by the moving upper plate, resulting in velocity profile across the flow.

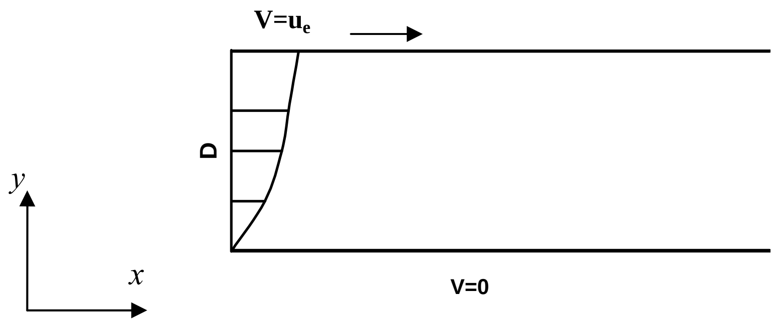

Fig. 1: Schematic diagram of flow domain.

The governing equation for this flow is the x-momentum equation, given by

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho f_x \qquad (1)$$

when applied to Couette flow, this equation is greatly simplified, as follows. The model for Couette flow stretches to plus and minus infinity in the x direction. Since there is no beginning or end of this flow, the flow-field variables must be independent of x; that is, $\partial/\partial x = 0$ for all quantities. Moreover, from the continuity equation for steady flow,

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \qquad (2)$$

for Couette flow becomes

$$\frac{\partial(\rho v)}{\partial y} = \rho\frac{\partial v}{\partial y} + v\frac{\partial \rho}{\partial y} = 0 \qquad (3)$$

The boundary and initial conditions are:

$u(0,t) = 0$ for all t,

$u(D,t) = u_e$ for t > 0,

$u(y,0) = 0$ for $0 \le y < D$

where D is the distance between two boundaries and it will be considered as the length scale of the flow.

The variables in equations governing the Couette flow are non-dimensionalized as follows:

$$\bar{u} = u/u_e, \qquad \bar{v} = v/u_e, \qquad \bar{\rho} = \rho/\rho_0$$

$$\bar{x} = x/D, \qquad \bar{y} = y/D, \qquad \bar{\tau}_{ij} = \tau_{ij}/p_0$$

where the subscript 0 denotes in the reference state.

After the non-dimensionalization, the Navier-Stokes equations for steady flow become

$$\frac{\partial(\overline{\rho u})}{\partial \bar{x}} + \frac{\partial(\overline{\rho v})}{\partial \bar{y}} = 0$$

for Couette flow becomes

$$\frac{\partial(\overline{\rho v})}{\partial \bar{y}} = \bar{\rho}\frac{\partial \bar{v}}{\partial \bar{y}} + \bar{v}\frac{\partial \bar{\rho}}{\partial \bar{y}} = 0$$

The boundary and initial conditions are:

$\bar{u}(0,t) = 0$ for all t,

$\bar{u}(D,t) = u_e$ for t > 0,

$\bar{u}(y,0) = 0 \quad \text{for } 0 \le \hat{y} < 1$

Major advances in computer technology and numerical techniques have made it possible to propose an alternative or at least a complementary approach to the classical and analytical techniques used in laboratory experiments. Computational fluid dynamics becomes part of the design process. The numerical technique that we will employ for the solution of the Couette flow is the Crank-Nicolson implicit method [13]. The use of an implicit technique allows a much larger marching step size than would be the case for an explicit solution.

RESULTS AND DISCUSSIONS

To examine the flow behavior, the velocity of the moving plate is set at $\bar{u} = 1$, and the dimensionless value of pressure and density are set equal 1. In order to get physical insight into the problem; numerical calculations are carried out for velocity field. Figure 2 represents the predicted flow dimensionless velocities profiles in the channel at different times and at the steady state condition. The transient velocities profiles at different times are shown at t = 25-1100 sec. The steady state velocity profile is illustrated in Fig. 2 where the time = ∞. As shown in the Figure 2 the velocity distribution is depicted for different times and the velocity decreases with increasing time.

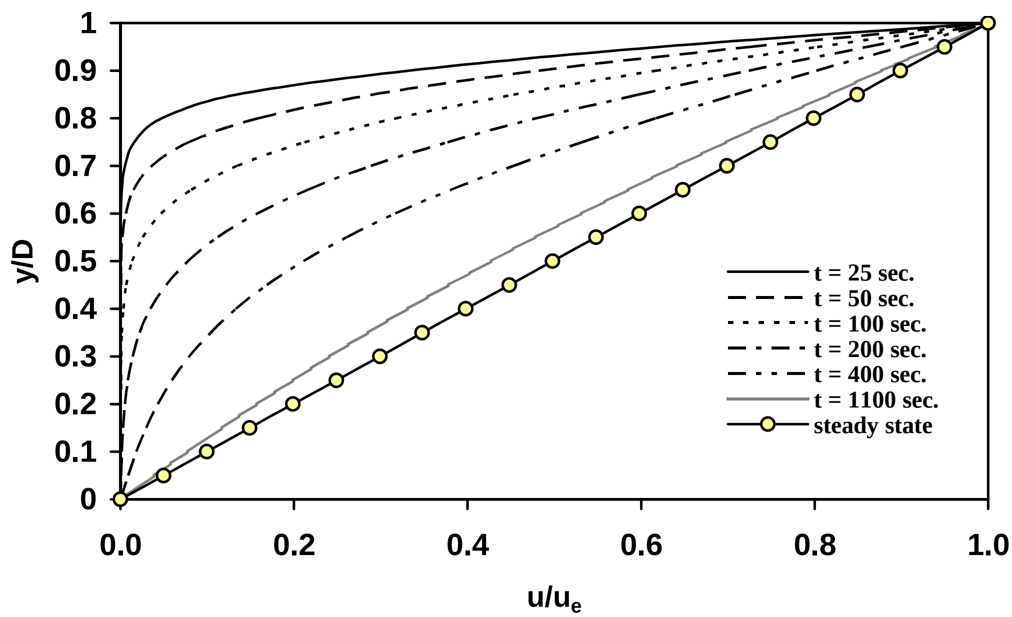

Fig. 2: Velocity profile for steady and unsteady Couette flow.

CONCLUSION

In this investigation a mathematical model was obtained in order to predict the steady and unsteady Couette flow between two parallel plates one of them was stationary and the other plate moved with constant velocity. Couette flow is a classical problem that has important applications in various industrial processing. The Crank-Nicolson implicit method was used to represent the flow behavior of the fluid. The use of an implicit technique allows a much larger marching step size than would be the case for an explicit solution.

NOMENCLATURE

D	distance between two parallel plates
p	pressure
t	time
u	velocity component in the x-direction
v	velocity component in the y-direction
V	plate velocity
x	axial coordinate
y	lateral coordinate

Greek symbols

ρ	density
τ	stress tensor

REFERENCES

(1) H. Schlichting and K. Gresten "Boundary layer theory", 8th revised and enlarged edition, Springer, New York, Berline, Heidelberg (2000).

(2) P. Panton "Incompressible flow", 2nd edition J Wiley, New York (1996).

(3) D. Rogers "Laminar flow analysis", Cambridge University Press, New York (1992).

(4) K. Rajagopal and P. Kaloni "Continuum mechanics and its applications", Hemisphere Press, Washington, DC, (1989).

(5) H. A. Attia; Arch. Appl. Mech.; 75, 268 (2006).

(6) L. A. Borag, O. G. Chkhetiani, M. Fröhner, and V. Myrnyy; Journal of Fluids and Structures; 20, 621 (2005).

(7) C. E. Siewert; European J. of Mechanics B/Fluids; 21, 579 (2002).

(8) K. D. Singh; Z. angew. Math. Phys.; 50, 661 (1999).

(9) Tiegang Fang and C. F. Lee; Heat Mass Transfer; 42, 255 (2006).

(10) H. Xue, H. M. Ji, and C. Shu; Int. J. Heat and Mass Transfer; 44, 4139 (2001).

(11) S. Gh. Etemad, A. S. Majumdar, and B. Huaung; Int. J. Heat Fluid Flow; 15, 122 (1994).

(12) P. Thompson and S. Troian; Nature; 389, 360 (1997).

(13) J. D. Anderson "Computational fluid dynamics", McGraw-Hill, Inc., (1995).