International Journal of Advance Research and Innovation
Vol. 5(2), Apr-Jun 2017, pp. 160-164
Doi: 10.51976/ijari.521728

Article Info
Received: 25 Jan 2017 | Revised Submission: 20 Feb 2017 | Accepted: 28 Feb 2017 | Available Online: 15 Jun 2017

Finite Difference Solver for Couette Flow with Applied Pressure Gradients

MrudangMathur, Nehal Amit Jajal** and Raj Kumar Singh****

ABSTRACT

A finite difference scheme was used to obtain solutions to a two dimensional CouetteFlow problem with applied pressure gradients. The solution to the non dimensionalizedCouette Flow problem was then obtained via Implicit methods and the results were subsequently validated with the analytical solution of the same problem. Finite difference scheme used returned satisfactory results under varying input conditions and was successfully validated on the Couette Flow problem through comparison with the analytical solution.

Keywords: *Coutte Flow; Pressure Gradient; Fluid Mechnics.*

1.0 Introduction

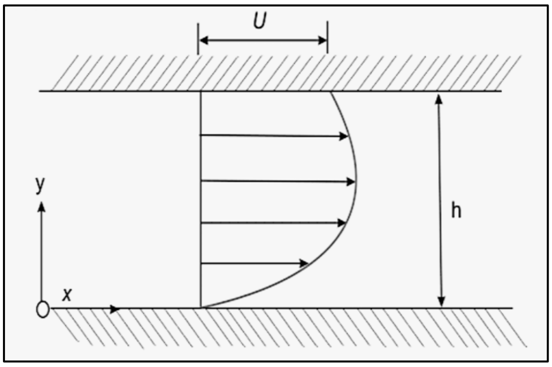
The foundation of modern fluid mechanics are the Navier-Stokes equation. It is an unsteady, nonlinear and second order partial differential equation. Thus making it an equation to which no analytical solution has been formulated to date. This has led to an alternate approach to solving the equation, this involves using numerical methods to obtain approximate solutions to the equation. The finite difference method is one such scheme developed which replaces all the partial derivatives and other terms in a partial differential equation by approximations. A numerical method for solving incompressible viscous flow problems was first introduced in 1947 by Alexandre Joel Chorin from Courant Institute. This method uses the velocities and the pressure as variables, and is equally applicable to problems in two and three space dimensions.Numerical methods and finite difference methods have been used in various applications such as modelling Navier-Stokes equations for vortex generation, pipe flow, couette flow, static structural analysis, wave analysis etc. It is seen that numerical methods have been adapted to solve a wide array of problems and prove to be a cheaper alternative to an experimental result. However it can't be ignored that numerical methods make use of approximations and thus the mathematical models have to be tested for various boundary conditions and governing equations for obtaining a reliable solution for the problem. A lot of research is being conducted on the stability of such numerical models as well as the reliability of the solution obtained by these methods.

2.0 Theory

Couette flow is the flow between two parallel plates. Here, one plate is at rest and the other is moving with a velocity U. Let us assume the plates are infinitely large in z direction, so the z dependence is not there. Figure 2.1 illustrates the flow between two plates.The governing equation is:

$$\frac{dp}{dx} = \mu \frac{\partial^2 y}{\partial x^2} \quad [1]$$

Fig 1: Couette Flow Between Two Pipes



*Mechanical Engineering Department, Delhi Technological University, Delhi 42, India
**Mechanical Engineering Department, Delhi Technological University, Delhi 42, India
***Coresponding Author: Mechanical Engineering Department, Delhi Technological University, Delhi 42, India
(E-mail: rajkumarsingh@dce.ac.in)

Upon integrating the given differential equation, we get

$$u = \frac{1}{2\mu}\frac{dp}{dx}y^2 + C_1 y + C_2 \quad ... [2]$$

Invoking the condition (at y = 0, u = 0), C_2 becomes equal to zero.

$$u = \frac{1}{2\mu}\frac{dp}{dx}y^2 + C_1 \quad ... [3]$$

Invoking the other condition (at y = h, u = U),

$$C_1 = \frac{U}{h} - \frac{1}{2\mu}\frac{dp}{dx}h \quad ... [4]$$

Inserting the constants into the original solution we obtain the expression for u, which is

$$u = \frac{y*U}{h} - \frac{h^2}{2\mu}\frac{dp}{dx}\frac{y}{h}(1 - \frac{y}{h}) \quad ... [5]$$

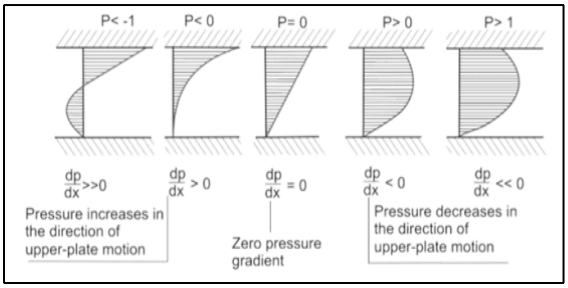
The above result can be non-dimensionalized to give

$$\frac{u}{U} = \frac{y}{h} + P\frac{y}{h}(1 - \frac{y}{h}) \quad ... [6]$$

$$P = -\frac{h^2}{2\mu U}\frac{dp}{dx} \quad ... [7]$$

When P > 0, i.e. for a negative or favorable pressure gradient in the direction of motion, the velocity is positive over the whole gap between the channel walls. For negative value of P (P < 0), velocity there is a positive or adverse pressure gradient in the direction of motion and the over a portion of channel width can become negative and back flow may occur near the wall which is at rest. Figure 2 shows the effect of dragging action of the upper plate exerted on the fluid particles in the channel for different values of pressure gradient.

Fig 2: Variation of Flow with Pressure Factor



3.0 Methodology

3.1 Mathematical Formulation

To formulate a Finite Difference Model a time marching scheme was used which approached a steady state solution in a certain number of time steps.

The Couette Flow equation for an unsteady flow can be given by [7]. The velocity, distance and time was then non-dimensionalized as [8-10] respectively and then substituted into the the unsteady flow equation [11].

The equation for Couette Flow with an applied pressure gradient was similarly non-dimensionalized [12] with P[13] being the non-dimensional pressure gradient. In all the equations $Re_D$is the Reynolds Number and $\mu$is the viscosity of the liquid.

$$\rho\frac{\partial u}{\partial t} = \mu\frac{\partial^2 u}{\partial y^2} \quad [1]$$

$$u' = \frac{u}{u_e} \quad [8]$$

$$y' = \frac{y}{D} \quad [9]$$

$$t' = \frac{t}{D/u_e} \quad [10]$$

$$\frac{\partial u\prime}{\partial t\prime} = \frac{\partial^2 u\prime}{Re_D \partial y'^2} \quad [11]$$

$$\frac{\partial u\prime}{\partial t\prime} = \frac{\partial^2 u\prime}{Re_D \partial y'^2} + \frac{2P}{Re_D} \quad [12]$$

$$P = \frac{-D^2}{2\mu u_e}\frac{\partial p}{\partial x} \quad [13]$$

The governing equations were then converted into Finite Difference form using the Crank-Nicolson method which is implicit and second order in time [14]. In order to simplify the solution procedure [14] can be written as [15] with the constants [16-18].

$$\frac{u_j^{n+1} - u_j^n}{\Delta t} = \frac{1}{Re_D}\frac{(u_{j+1}^{n+1} + u_{j+1}^n) + (-2u_j^{n+1} - 2u_j^n) + (u_{j-1}^{n+1} + u_{j-1}^n)}{2\Delta y^2} + \frac{2P}{Re_D} \quad [14]$$

$$Au_{j-1}^{n+1} + Bu_j^{n+1} + Au_{j+1}^{n+1} = K_j \quad [15]$$

$$A = -\frac{\Delta t}{Re_D 2(\Delta y)^2} \quad [16]$$

$$B = 1 + \frac{\Delta t}{Re_D(\Delta y)^2} \quad [17]$$

$$K_j = \left[1 - \frac{\Delta t}{Re_D(\Delta y)^2}\right]u_j^n + \frac{\Delta t}{Re_D 2(\Delta y)^2}\left(u_{j+1}^n + u_{j-1}^n\right) + \frac{2P}{Re_D} \quad [18]$$

A tri-diagonal matrix can now be constructed and a matrix equation can be developed for N nodes just as seen in the case without an external pressure factor.

The equations have to be solved using Thomas's algorithm for tri-diagonal matrix.The results generated from the implicit finite difference solver are validated by comparing them to analytical results for the same Couette flow problem.

The solver accurately predicts the velocity gradient even when an external pressure is applied to the Couette flow problem.

3.2 Code development

Both the methods, explicit as well as implicit, solvers are developed using the MATLAB programming language. Separate functions were made for each case and the results were then evaluated. MATLAB is a programming language with powerful functions and operators for matrix calculations and hence developing a code involving matrix calculations becomes relatively easier. MATLAB also has various features of obtaining graphical results from the numerical results generated. This makes it simple to compare and validate results with analytical results. The solver algorithm has been depicted in figure 3.

4.0 Results

All numerical solutions employ the following fixed parameter values:

$Re = 2000$ $\quad\quad\quad$ $\partial t = 0.1$

$N = 1000$

Solutions have been obtained against a varied value of *P*; the non-dimensional pressure:

$$P \in \{-3, -2, -1, 0, 1\}$$

4.1 Change in error with respect to time

As shown in figure 4.1 the error between the analytical solution and numerical solution reduce over time. This is consistent with our assumption that the Truncation Error decreases with an increase in time. The change in error for three distinct values of *P* have been depicted in the aforementioned figure 4.

Fig 3: Algorithm Used by the Solver

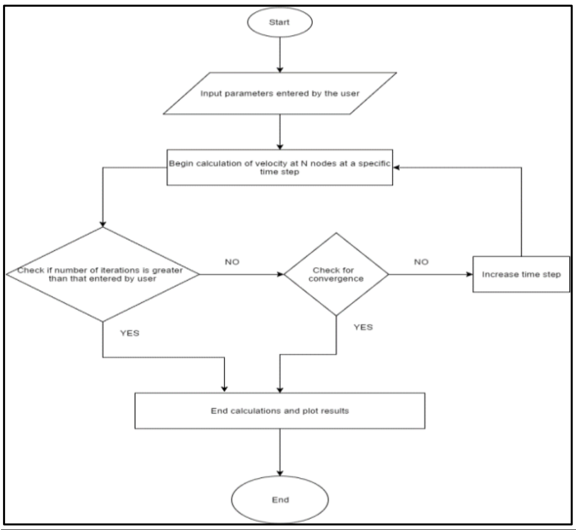


Fig 4: Error vs Time

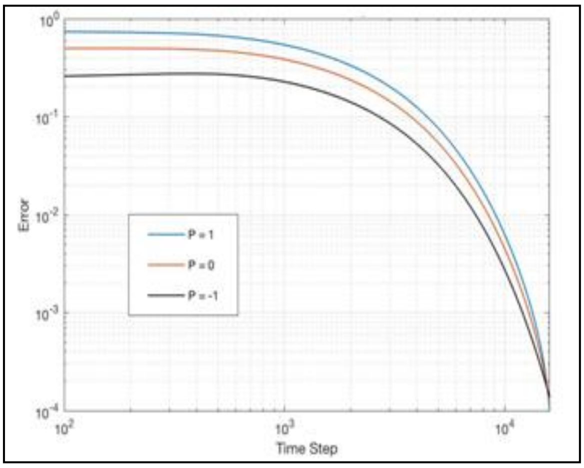


Fig 5(a): Normalized Velocity Profile

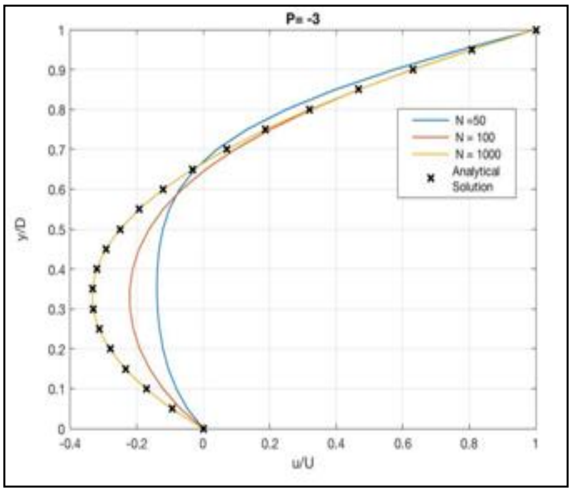
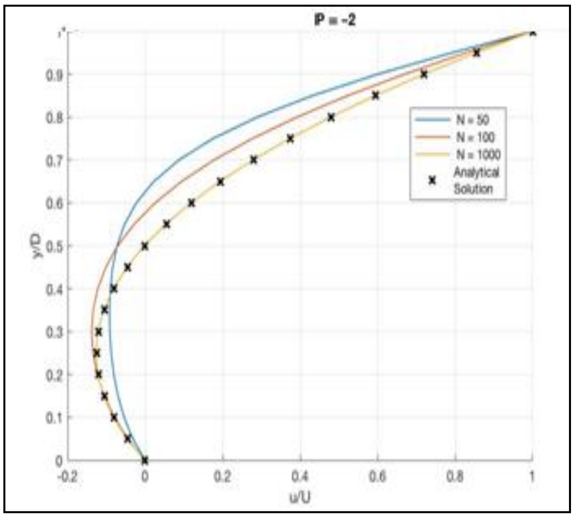


Fig 5(b): Normalized Velocity Profile



4.1 Convergence of velocity profiles

It can be observed from figures 4 to figure 8 that as the number of time steps increases i.e. the solution moves forward in the time domain; the transient velocity profile approaches the Steady-State velocity profile obtained from the analytical solution of the Couette Flow problem. The solution will continue to converge to the exact solution with the Truncation Error reducing with each advancement in time.

This process will continue till infinity. To break out of the infinite loop created we have fixed the number of time steps the code must simulate. This has allowed us to obtain an acceptable value of truncation error and gives us the freedom to choose the quality of results we require.\

Fig 5(c): Normalized Velocity Profile

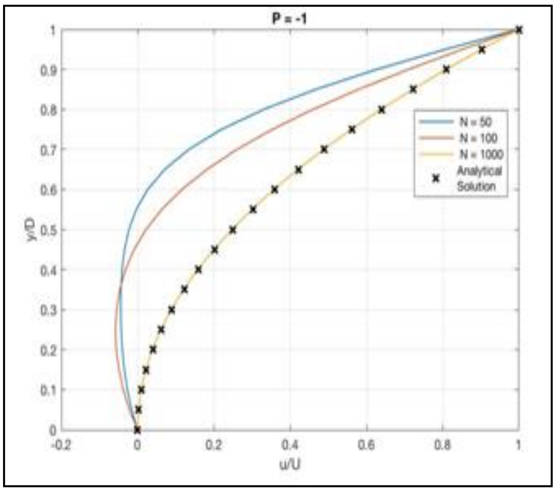


Fig 5(d): Normalized Velocity Profile

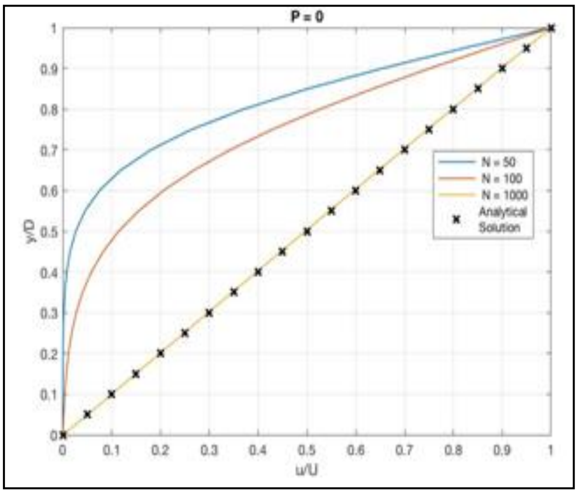
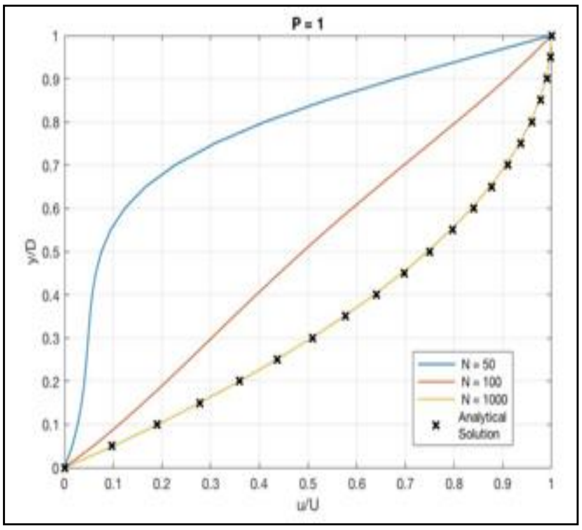


Fig 5(e): Normalized Velocity Profile



5.0 Conclusions

The Finite Difference Scheme used returned satisfactory results under varying input conditions and was successfully validated on the Couette Flow problem through comparison with the analytical solution. There is a large scope of work that can be done to further this solution strategy i.e. the scheme and formulation can be adapted to 2 and 3 dimensional flows taking into account the effects of turbulence as well as heat transfer in possible applications.

References

[1] Yunus A Çengel, John M Cimbala. Fluid Mechanics: Fundamentals and Applications, TMH-2013

[2] Alexandre Joel Chorin. A Numerical Method for Solving Incompressible Viscous Flow Problems. Journal of Computational physics, 135 (2), 1997, 118-125

[3] John D Anderson, JR.Computational Fluid Dynamics: The Basics with Applications.

[4] MA Maa Touga, M Ayadi. Numerical simulation of the second-order Stokes theory using finite difference method, Alexanderia Engineering Journal, 55(3), 2016, 3005-3013

[5] Hoffman JD, Frankel S. Marcel Dekker, Inc., Numerical methods for engineers and scientists, McGraw- Hill 1992.

[6] C Wayne Mastin. Implicit finite difference methods on composite grids, Journal of Computational and Applied Mathemitics, 20, 1987, 317-323