International Journal of Latest Engineering and Management Research (IJLEMR)
ISSN: 2455-4847
www.ijlemr.com || Volume 02 - Issue 04 || April 2017 || PP. 36-40

Time Advancement solution to 2-DIncompressible lid-driven Cavity using the Explicit Euler method of a fluid flow

K Subbanna[1], S Gouse Mohiddin[2],R Bhuvana Vijaya[3]and M.Sasikala[4]
[1,2,4]*Dept. of Mathematics, Madanapalle Institute of Technology&Science, Madanapalle A.P, India*
[3]*Dept. of Mathematics, JNTUA College of Engineering Anantapur, Anantapuramu, A.P, India*

Abstract: The present work is concerned with the computations of two- 2-dimensional lid-driven cavity flows to obtain for a 1x1 surface with given boundary conditions using a Time-Advanced multi-grid code based on the EXPLICIT EULER METHOD on a collocated grid arrangement. The transport equations are discretized using the explicit Euler Time-Advanced method scheme. Velocity fields on a collocated grid can be found using numerical approximations by calculating terms for advection, viscosity, and pressure. The system is solved explicitly, except for the pressure equations, which are taken implicitly. Numerical methods used for calculations include both backward and forward difference schemes, the resulting flow fields show that solution exists for all Reynolds numbers.

Keywords: Incompressible flows; lid-driven cavity; finite-difference method; explicit Euler Time-Advanced method.

Nomenclature

dx = Length of Grid spacing
Hi = Advective and Viscous Term
n = Time step
p = Pressure
u = Velocity Component in x-direction
v = Velocity Component in y-direction
t = Time
ρ = Density / Thickness
τ = Viscosity

I. INTRODUCTION

For a 1x1 2-dimensional surface, limits for a lid-driven cavity flow are calculated. The numerical model for this problem includes changes to the Navier Stokes equations that solve for velocity while satisfying continuity. Pressure is found with a Poisson equation, and a new velocity value is then found. Boundary conditions for velocity are given and stay the same throughout the problem, while others fluctuate.

II. GOVERNING EQUATIONS AND NUMERICAL METHOD

The Navier Stokes equations for steady-state, incompressible systems are shown by the following:

$$\frac{\partial U}{\partial X}+\frac{\partial U}{\partial Y}=0 \qquad (1)$$

$$\frac{\partial u}{\partial t}+U\frac{\partial u}{\partial x}+V\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+V(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}) \qquad (2)$$

$$\frac{\partial v}{\partial t}+U\frac{\partial u}{\partial x}+V\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+V(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}) \qquad (3)$$

Where Equation (1) is the continuity equation, and Equations (2) and (3) are the momentum equations for the x and y-directions. For unsteady equations, a numerical Poisson equation, Equation (4), is created for pressure to enforce continuity, and is discretized in space but not time.

$$\frac{\partial(\rho u)}{\partial t}=\frac{\delta(\rho u_i u_j)}{\delta x}-\frac{\delta(p)}{\delta x_i}+\frac{\delta(\tau_{i,j})}{\delta x_i}=H_I-\frac{\delta(p)}{\delta x_i} \qquad (4)$$

For this equation ($\delta / \delta x$) is a discretized spatial derivative. Solving Equation 4 for the Euler method for time advancement, Equation (5) is found.

$$u_p^{n+1} = u_p^* - (\frac{\delta^F p^n}{\delta x})\Delta t \tag{5}$$

Because the velocity terms do not satisfy the continuity equation, the numerical divergence is found by applying ($\delta / \delta x$) to Equation (5) and making the LHS (right-hand-side) equal to zero for the divergence of the velocity field. This reveals the discrete Poisson's equation for pressure, given by:

$$\frac{\delta}{\delta x_i}(\frac{\delta p^n}{\delta x_i}) = \frac{\delta H_i^n}{\delta x_i} \tag{6}$$

With divergence operators in place for the continuity and momentum equations, the velocity field at the $n+1$ time step will be divergence free as long as p satisfies this equation. H can be found by the following equations:

$$H_x^n = v(\frac{\delta^2 u_x^n}{\delta x \delta x} + \frac{\delta^2 u_x^n}{\delta y \delta y}) - \frac{\delta}{\delta x}(u_x u_x)^n - \frac{\delta}{\delta y}(u_x u_y)^n \tag{7}$$

$$H_y^n = v(\frac{\delta^2 u_y^n}{\delta x \delta x} + \frac{\delta^2 u_x^n}{\delta y \delta y}) - \frac{\delta}{\delta x}(u_y u_x)^n - \frac{\delta}{\delta y}(u_y u_y)^n \tag{8}$$

Also, initial values for the velocity field can be found by using the following equations:

$$u_p^* = u_p^n + H_{x/p}^n \Delta t \tag{9}$$

$$v_p^* = v_p^n + H_{y/p}^n \Delta t \tag{10}$$

Where p is the nodal point in consideration. Equation (6) can be expanded with the backwards difference and forward difference schemes, which reveals the following equation:

$$\frac{\delta^B}{\delta x}(\frac{\delta^F p^n}{\delta x})\downarrow_P - \frac{\delta^B}{\delta x}(\frac{\delta^F p^n}{\delta y})\downarrow_P = \frac{\delta^B}{\delta x}(H_x^n) - \frac{\delta^B}{\delta y}(H_y^n) \tag{11}$$

Equations (9) and (10) do not satisfy continuity. Once Poisson's equation is resolved for pressure, the final velocity fields can be found by:

$$u_p^{n+1} = u_p^* - (\frac{\delta^F p^n}{\delta x})\Delta t \tag{12}$$

$$u_p^{n+1} = u_p^* - (\frac{\delta^F p^n}{\delta x})\Delta t \tag{13}$$

III. TECHNICAL APPROACH

Boundary conditions are given as 0 for the u and v vectors, except for u=1 across the top of thecavity. These boundaries remain constant, while the velocity field for the inner nodes changeswith each iteration of calculations until the system reaches steady-state.

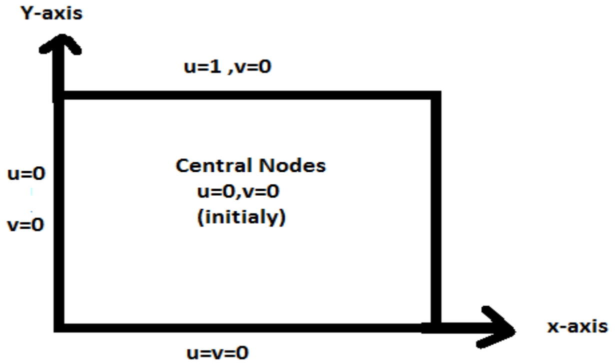

Figure:1 Representation of surface and boundary conditions

Mesh sizes are appropriated according to convergence.
The algorithm for the solution is as follows:

1. Start with a given u_i at time tn. Make sure it is divergence free.
2. Calculate the combination H_i, and its divergence. This will be different for each iteration.
3. Solve the Poisson equation for pressure p .
4. Compute the new divergence free velocity field at the next time step.
5. Repeat.

The calculations start with the given velocities and time equal to zero. Then the H_i terms for x and y is found by using Equations (7) and (8) with the central difference method.

$$H_x^n = \frac{-1}{2x}[(u_E^n) + u_N^n v_N^n - u_S^n v_S^n - (u_W^n)^2] + \frac{v}{(\Delta x)^2}[u_E^n + u_W^n + u_N^n + u_S^n - 4u_P^n] \tag{14}$$

$$H_y^n = \frac{-1}{2x}[(u_N^n) + u_E^n v_E^n - u_W^n v_W^n - (u_S^n)^2] + \frac{v}{(\Delta x)^2}[u_E^n + u_W^n + u_N^n + u_S^n - 4u_P^n] \tag{15}$$

Outside nodes for advection and viscosity terms are extrapolated. After an Hx *and* Hy value is foundfor each node, a B matrix of constants is obtained by the right side of Equation (11), expanded for this problem to be:

$$B(i,j) = \frac{1}{(\Delta x)}[Hx(i,j) - Hx(i,j-1)] + \frac{1}{(\Delta y)}[Hy(i,j) - Hy(i-1,j)] \tag{16}$$

The LHS of Equation (11) is used to discretize a sparce matrix A of pressure coefficients, and an iterative successive over-relaxation solver is used to solve a system of equations where $Ax = B,$ where x is used to represent the variables for pressure. Once pressure is obtained from the Poisson's equation, the new divergence free velocity is obtained for the next time step.

IV. RESULTS AND DISCUSSION:

Results for the velocity fields are given below. Figures 2 and 3 show plots for the velocitycomponents and pressures across the cavity *at* $x = 0.5$ *and at* $y = 0.5$. Figure 4 shows a contour plotof the pressure profile. Figures 5 and 6 show velocity profiles for u and v, respectively.

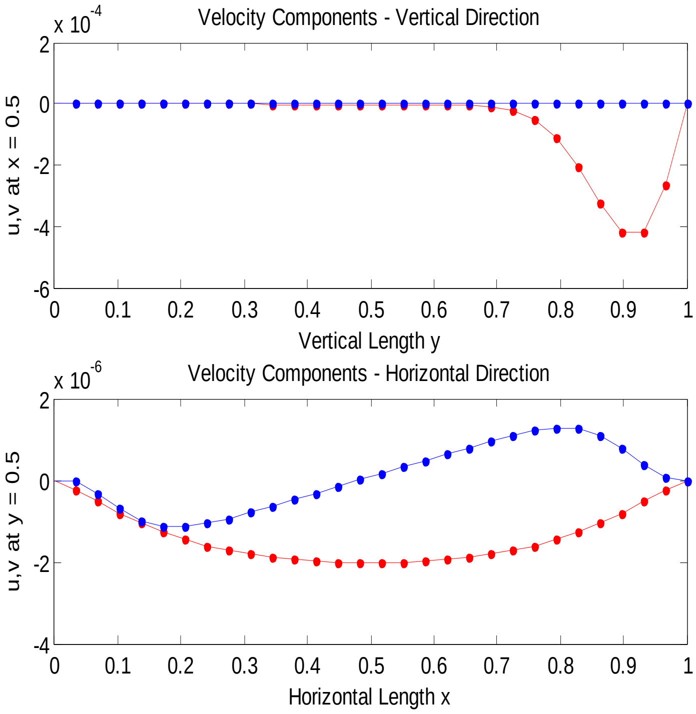

Fig 2 show plots for the velocity components.

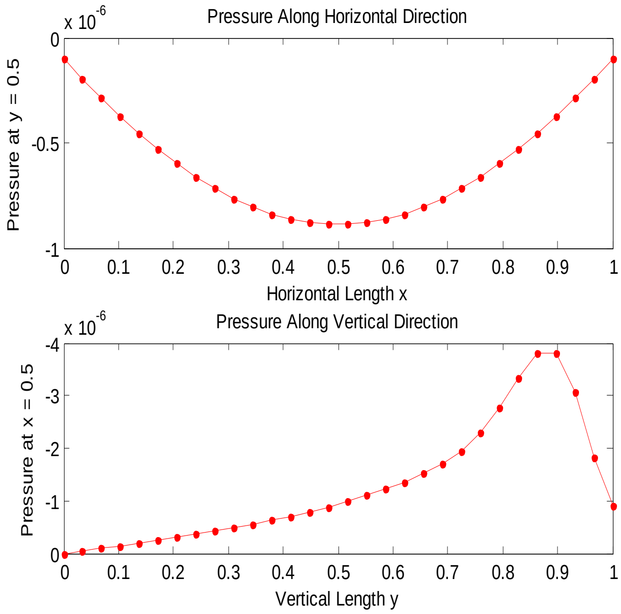

Fig 3 show plots for the Pressure components.

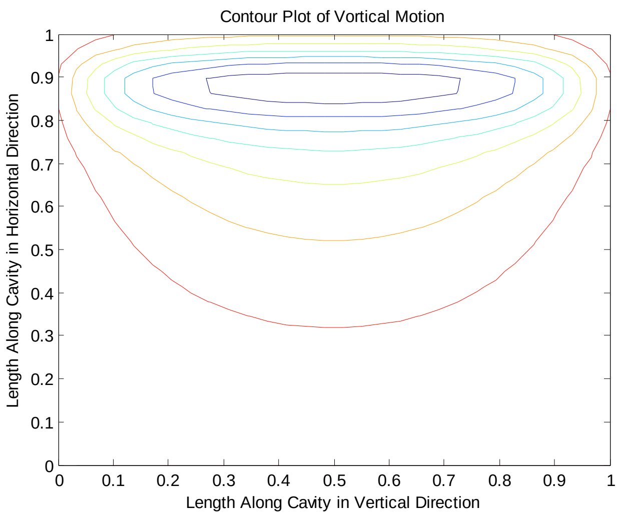

Fig.4 Contour plot for Pressure

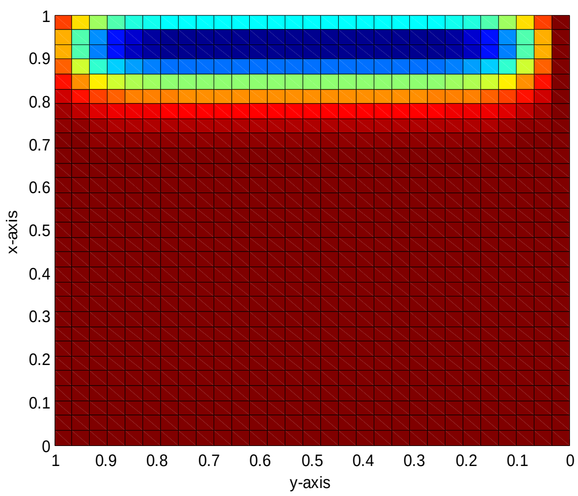

Fig:5 Profile of Velocity u

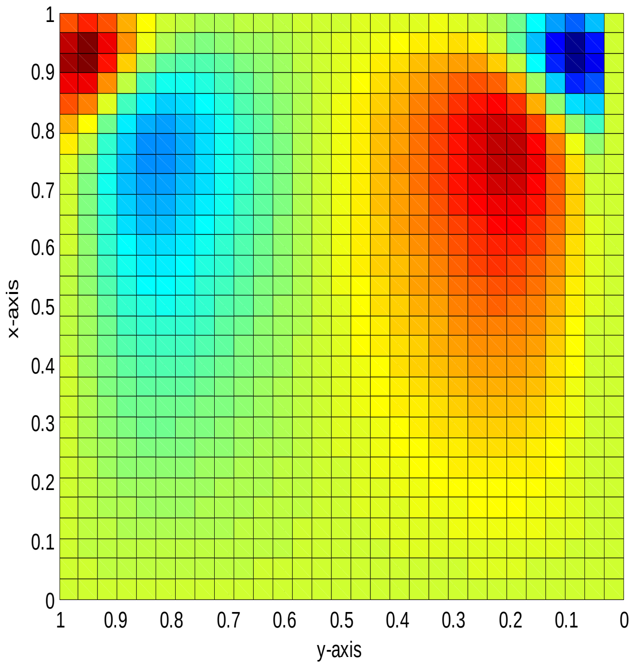

Fig:6 Profile of Velocity v

V. Summary

The code using the finite difference method and explicit Euler Time-Advanced method scheme on a collocated grid arrangement is developed to obtain stable multiple solutions for two- dimensional lid-driven cavities. Since the advection terms in the Navier-Stokes equations are discretized using the explicit scheme and using numerical approximations by calculating terms for advection, viscosity, and pressure. The system is solved explicitly, except for the pressure equations, which are taken implicitly. Numerical methods used for calculations include both backward and forward difference schemes. To set up the credibility and performance of the code, it is first used to compute the flow in a standard 2-D single-lid-driven cavity to demonstrate that the results closely match with the corresponding highly reliable existing results. The flow configurations show many interesting flow features which include multiple stable solutions. It is established that the solutions valid and stable, this model is perfect, as it had to be manipulated to solve for pressure, while also satisfying continuity.

VI. References:

[1]. John D. Anderson, Jr. 'Computational Fluid Dynamics: The Basics with Applications', McGraw-Hill Inc, 1995.
[2]. RyszardGrybos, 'Podstawy mechanikiplynow' (Tom 1 i 2), PWN 1998.
[3]. David Potter 'Metody obliczeniowe fizyki', PWN 1982.
[4]. James D. Bozeman, Charles Dalton, 'Numerical Study of Viscous Flow in Cavity', Journal of Computational Physics, vol. 12, 1973.
[5]. J.C. Tannehill, D.A. Anderson, 'Computational Fluid Mechanics and Heat Transfer, Second Edition', Series in Computational and Physical Processes in Mechanics andThermalSciences.
[6]. C.A.J. Fletcher, 'Computational Techniques for Fluid Dynamics, Volume 2', Springer .
[7]. F. H. Harlow, John P. Shannon, 'The Splash of Liquid Drop', Journal of Applied Physics (vol. 38, n.10 Sept.1967).
[8]. J. Welch, F. Harlow, J. Shannon, 'The MAC Method', Los Alamos Scientific Laboratory of The University of California (1965).
[9]. N.Foster, D.Metaxas, 'Realistic Animation of Liquids', Center for Human Modeling and Simulation.