International Journal of Fluids Engineering.
ISSN 0974-3138 Volume 5, Number 1 (2013), pp. 21-28

http://www.irphouse.com

A Pressure based Solver for an Incompressible Laminar Newtonian Fluids

N. Sreenivasalu Reddy[1*], K. Rajagopal[2], P.H. Veena[3] and V.K. Pravin[4]

[1]*Department of Industrial Engineering and Management, DSCE, Bangalore, Karnataka, India*
nsreddysrsit@gmail.com
[2]*Professor, Department of Mechanical Engineering, JNTU College of Engineering, Hyderabad, Andhra Pradesh, India*
[3]*Department of Mathematics, Smt. V.G. College for Women, Gulberga, Karnataka, India*
[4]*Professor and Dean, PG(T.P.E.) PDA College of Engineering, Gulbarga Karnataka, India*
[*]*Corresponding author: E-mail address: nsreddysrsit@gmail.com*

Abstract

A pressure based solver is developed for an incompressible laminar Newtonian fluid using finite volume method. In the present work the coupling between density and pressure is removed, as well as the coupling between the energy equations for compressible flows. And also the effect of increased mesh resolution and mesh grading towards the walls is investigated. The solver is validated with the existing solver in the literature. The results are analyzed for standard test case driven cavity flow for different mesh sizes.

Keywords: Finite volume method, Discretisation, pressure based solver

1. Introduction

Since the inception of the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm in 1972 [1], the continuity and momentum equations have been typically solved using segregated approach. Over the last quarter century significant advances have been made in pressure based methods including improvements in convergence rate [2], flow and heat transfer [3], unstructured grid arrangement, ability to handle complex geometry [4], simulating high speed flows[5], handling deforming geometries and moving grids [6] and several aspects. A detailed description of pressure-based methods can be found in a monograph by Ferzieger and peric [7].Solution of the Navier-Stokes equations is complicated by the lack of an independent equation for the pressure, whose gradient contributes to each of the three momentum equations. In compressible flows the continuity equation can be used to determine the density and the pressure is calculated from equation of state. This approach is not appropriate for incompressible flows. The current paper presents pressure based coupled solution approach for flow simulations in a finite volume frame work using Rhie and Chow interpolation [8].

2. The pressure equation and its solution

The incompressible continuity and momentum equations are given by

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) - \nabla \cdot (\nu \nabla \mathbf{u}) = -\nabla p \tag{2.1}$$

The non linearity in the convection term is handled using an iterative solution technique. There is no pressure equation, but the continuity equation imposes a scalar constraint on the momentum equation (since $\nabla.\mathrm{u}$ is a scalar). There is no pressure equation for incompressible flow, so we use the continuity and momentum equations to derive a pressure equation. Start by discretizing the momentum equation, keeping the pressure gradient in its original form:

$$a_P^{\mathbf{u}}\mathbf{u}_P + \sum_N a_N^{\mathbf{u}}\mathbf{u}_N = \mathbf{r} - \nabla p \tag{2.2}$$

Introduce the H (u) operator:

So that

$$\mathbf{H}(\mathbf{u}) = \mathbf{r} - \sum_N a_N^{\mathbf{u}}\mathbf{u}_N \tag{2.3}$$

$$\begin{aligned} a_P^{\mathbf{u}}\mathbf{u}_P &= \mathbf{H}(\mathbf{u}) - \nabla p \\ \mathbf{u}_P &= (a_P^{\mathbf{u}})^{-1}(\mathbf{H}(\mathbf{u}) - \nabla p) \end{aligned} \tag{2.4}$$

Substituting this in the incompressible continuity equation ($\nabla.\mathrm{u} = 0$) to get pressure equation for incompressible flow

$$\nabla \cdot \left[(a_P^{\mathbf{u}})^{-1}\nabla p\right] = \nabla \cdot \left[(a_P^{\mathbf{u}})^{-1}\mathbf{H}(\mathbf{u})\right] \tag{2.5}$$

3. Finite Volume Method

The computational domain is subdivided into a finite number of continuous control volumes, where the resulting statements express the exact conservation of relevant properties for each of the control volumes. At the centroid of the control volumes, the variable values are calculated. Interpolation is used to express variable values at the control volumes surface in terms of the center values and suitable quadrature. Mathematical formulae are applied to approximate the surface and volume integrals.

$$\left(\frac{\partial \phi}{\partial x}\right) = \frac{1}{\Delta V}\int_V \frac{\partial \phi}{\partial x} dV = \frac{1}{\Delta V}\int_A \phi \, dA^{x} \approx \frac{1}{\Delta V}\sum_{i=1}^{N} \phi_i A_i^{x} \quad (3.1)$$

4. Discretization

Discretisation of the solution domain is shown in figure 4.1. The space domain is discretized into computational mesh on which the Partial differential equations are subsequently discretized. Discretisation of time, if required, is simple: it is broken into a set of time steps Δt that may change during a numerical simulation, perhaps depending on some condition calculated during the simulation. On a more detailed level, discretisation of space requires the subdivision of the domain into a number of cells, or control volumes. The cells are contiguous, i.e. they do not overlap one another and completely fill the domain. A typical cell is shown in figure4.1

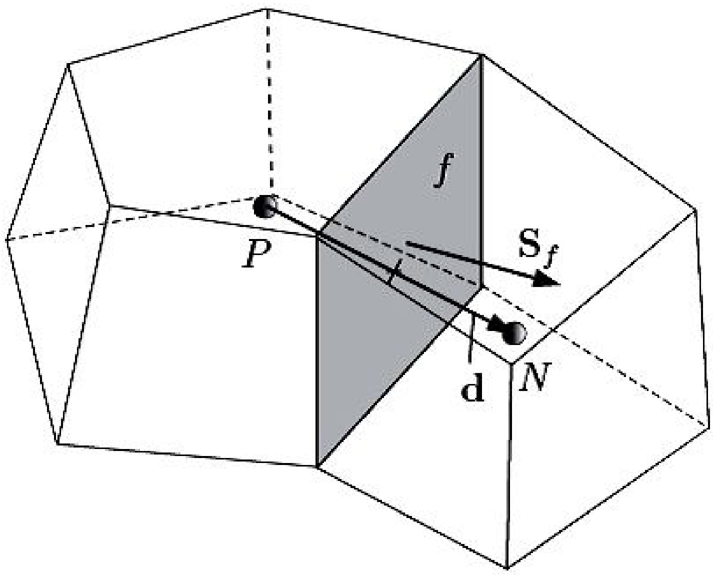


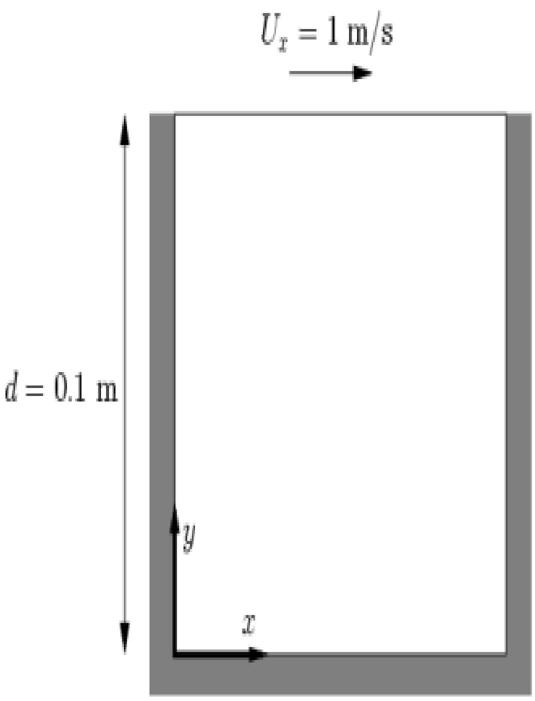
Figure 4.1 Finite volume method for typical cell

5. Numerical schemes

Here we use the Euler implicit temporal discretization, and the linear (centraldifference) scheme for convection. The p linear equation system is solved using the Conjugate Gradient solver PCG. The U linear equation system is solved using the Conjugate Gradient solver PBICG. The solution is considered converged when the residual has reached the tolerance 1e-05 for each time step.

6. Numerical Experiments

The geometry of the lid driven cavity as shown in figure 6.1 in which all the boundaries of the square are walls. The cavity with fine mesh of 20 x 20 as shown in figure 6.2 .The top wall moves in the x direction at a speed of 1 m/s while the other three are stationary. Initially the flow will be assumed laminar and will be solved on a uniform mesh using a solver for laminar, isothermal and incompressible flow. This case will be run with a Reynolds number of 10, $\nu = 0.01\,\mathrm{m^2\,s^{-1}}$.



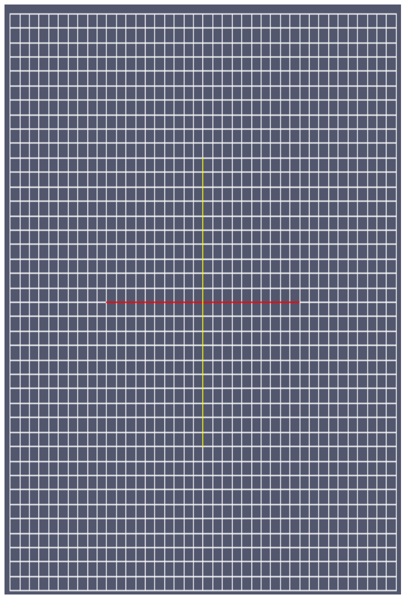
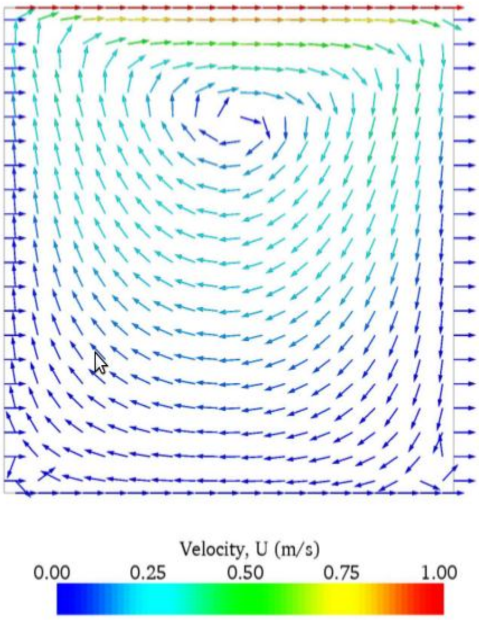


Figure 6.1 Geometry of the lid driven cavityFigure 6.2 Fine Mesh with 20 x 20

7. Results



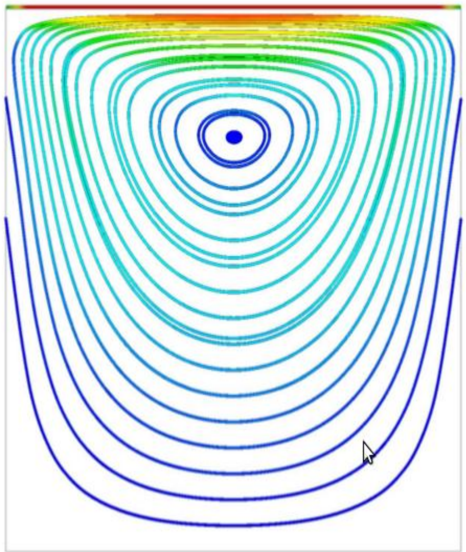
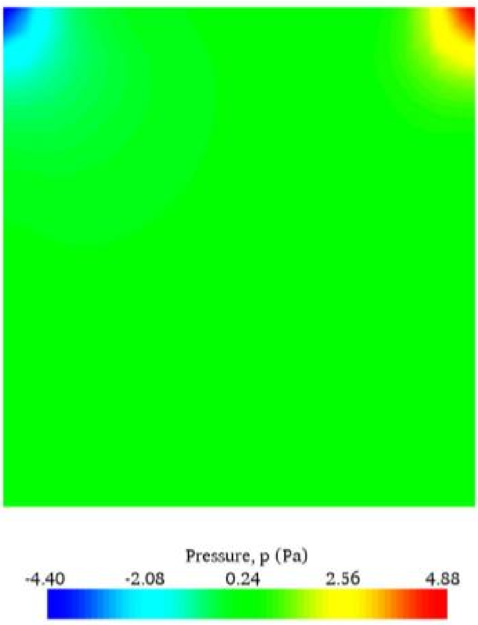


Figure 7.1. Velocity vectors for coarse mesh of 10 X 10 Figure 7.2. Stream lines for coarse mesh of 10 X 10



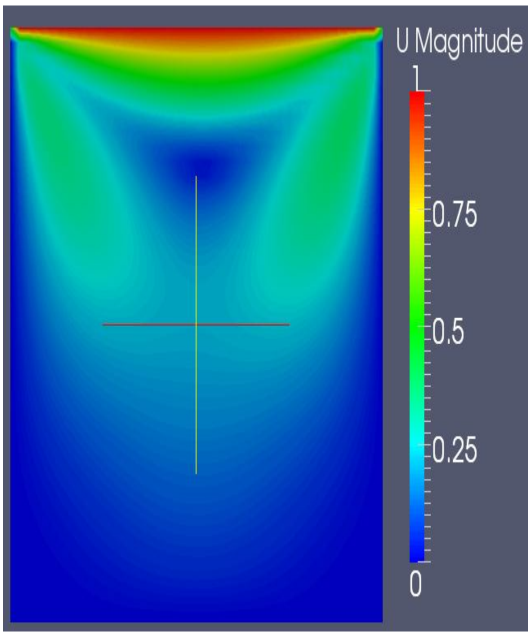
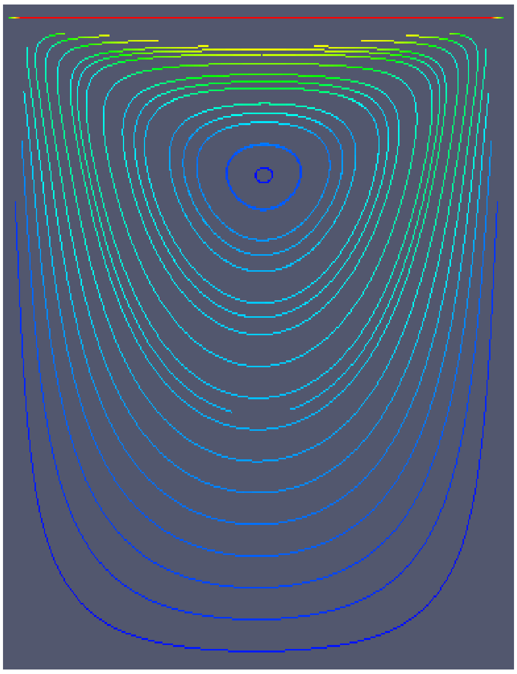


Figure 7.3. Pressure contours for the coarse mesh of 10 X 10 Figure 7.4. Velocity U for the fine mesh 20 x 20



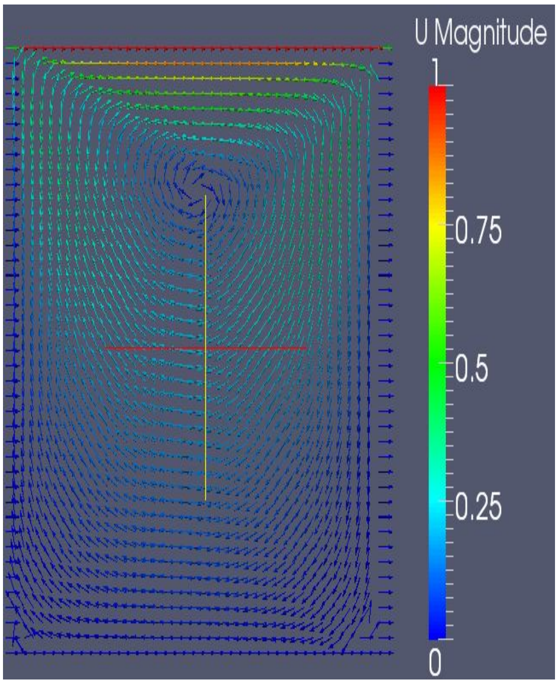
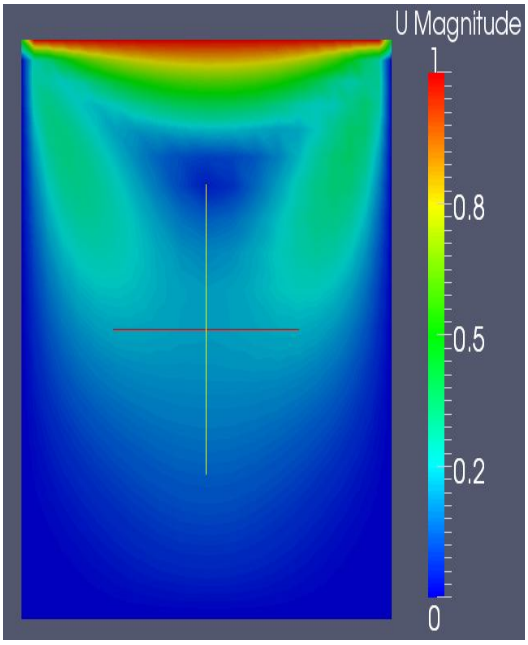


Figure 7.5. Stream lines for fine mesh of 20 x 20 Figure 7.6. Velocity vectors for fine mesh of 20 x 20



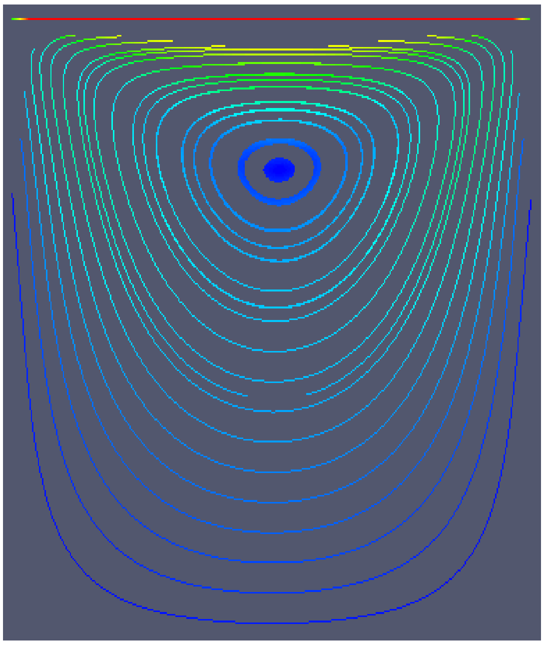


Figure 7.7. Velocity U for graded mesh Figure 7.8. Stream lines for graded mesh

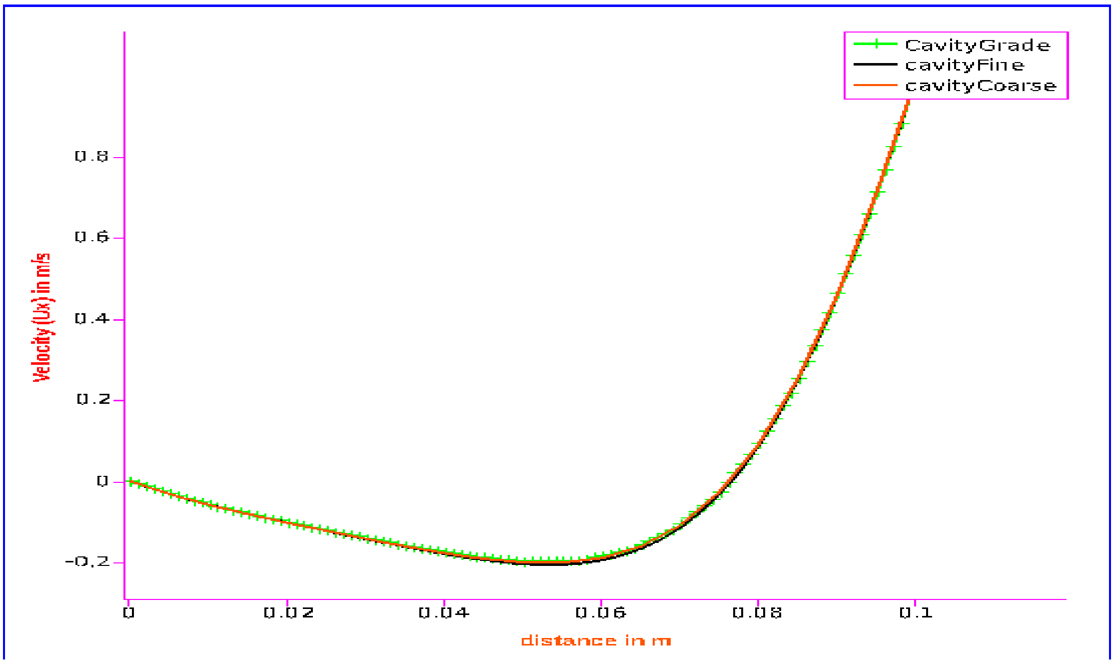


Figure 7.9 Velocity in x direction at mid plane

8. Conclusions

A pressure based solver has been presented for solving fluid dynamics equations for incompressible flows. The method uses pressure based formulation and solves primitive variables (u, v and p). The method has been tested on CFD bench mark case lid driven cavity. The results are good agreement with existing results in the literature [9].In lid driven cavity case excellent convergence rate has been observed.

References

[1] S. V. Patankar, D.B. Spalding, A calculation procedure for heat, mass and momentum transfer in three dimension parabolic flows, Int. j. Heat Mass transfer 15 (1972) 1787-1806.

[2] J.P. Van Doormaal, G.D. Raith, Enhancement of the SIMPLE method for predicting incompressible fluid flow, Numer. Heat Transfer 7 (1984) 147-163.

[3] P.F. Galpin, G.D. Raithby, Numerical solution of problems in incompressible fluid flow: treatment of temperature velocity coupling, Numer. Heat Transfer 10 (1986) 105-129.

[4] Y.Jiang, A.J. Przekwas, Implicit pressure-based incompressible Navier-Stokes equations solver for unstructured meshes,AIAA 94-0305,1994.

[5] Y.Jiang, A.J. Przekwas, 3D simulation of complex flows at all speeds with an implicit multi-domain approach, AIAA 93-3124,1993

[6] H.Q. Yang, A.J. Przekwas, Pressure-based high-order TVD methodology for dynamic stall simulation, AIAA,93-0980,1993.

[7] J.H Ferziger, M. Peric, Computational Methods in Fluid Dynamics, Springer-Verlag, Berlin, Heidelberg, New York,1996.

[8] C.M. Rhie, W.L. Chow, A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation, AIAA J, 21 (1983) 1525-1532.

[9] U. Ghia, K.N. Ghia and C.T. Shin, High Re solutions for incompressible flow using the Navier-Stokes equations and a multy grid method, Journel of Computational Physics,(1982) 48,387-411