

Pressure Based Method for Solving Viscous Incompressible Flows using Rhie and Chow Interpolation

Anjan H¹, N.Sreenivasalu Reddy², Kanchiraya S³

^{1,3}Dept. of Mechanical Engineering, Govt. Engineering College, Hassan

²Asso. Professor, Dept. of Mechanical Engineering, Rajarajeshwari College of Engineering, Bangalore

¹anjanh7@gmail.com, ²nsreddysrit@gmail.com, ³Kanchirayas@gmail.com

Abstract - In this paper a pressure based finite volume method is formulated to solve the viscous incompressible flows using rhie chow interpolation. The method involves employing a different interpolation for advective velocities and provides a connection between the nodal velocity and pressure so there is no odd-even decoupling. The present solver is tested for one Dimensional diffusion problem with and without source term, potential flow equation, and laminar viscous incompressible flows. The results are compared with existing results in the literature. The present study has established that numerical method can provide a reasonable good solution for incompressible viscous flows.

Keywords — Finite volume method, Navier-Stokes equation, Potential flow, Lid driven cavity.

I. Introduction

Computational fluid dynamics methods for solving incompressible flow fields have followed two evolutionary paths. The first method of incompressible flow algorithms was the MAC (Marker and Cell) method due to Harlow and Welch [2], the projection method due to Chorin [3] and the popular SIMPLE (Semi Implicit Method for Pressure Linked equations) family of schemes due to Patankar and Spalding [1]. These methods collectively referred to as pressure based methods. A new formulation of the Navier-Stokes equation is introduced to solve incompressible flow problems by Ping Lin [4]. The SIMPLE algorithm and its variants have dominated the entire field of numerical simulation of incompressible flows, over the past twenty years. Based up on the SIMPLE algorithm using finite volume method employing Cartesian velocity components and non-staggered arrangements have been successfully used by Venka [6] in their three dimensional solution. In the staggered grid arrangement if the mesh is not Cartesian then the velocity nodes may cease to lie between the pressure nodes that drive them. The importance of using collocated grid arrangement for curvilinear coordinates has full-fledged since Peric [7] published an application of SIMPLE algorithm. The utilization of non-staggered grids significantly decreases the storage memory and cut down the computational time in three-dimensional calculations, especially for unstructured/curvilinear body- fitted grids. On the other hand, they are prone to produce a false pressure field checkerboard pressure. Hence, in the 1980s and before, non-staggered grids were infrequently used in the primitive variable method for incompressible flows. However, since 1983 the collocated grid has been adopted more and more widely, after Rhie and Chow [8] proposed a momentum interpolation method to remove the checkerboard pressure and subsequent modification by Peric [7].

Goldstein[9] proposed a model called virtual boundary formulation, was used to simulate turbulent flow over a ribbelt-covered surface. Ye [10] propose a method called Cartesian grid method for simulating two dimensional viscous, incompressible flows over complete geometry. Lai & Peskin [11] have proposed another method second order accurate boundary which was used for a flow over a stationary cylinder. Present solver is a useful to validate the potential flow is such that an analytical solution exists for cases whose geometries are relatively simple.

Lid driven cavity flow is the motion of a fluid inside a square cavity and it serves as the benchmark for the numerical method in terms of accuracy and efficiency. It is easy to code and apply boundary condition for the driven cavity flow due to simplicity of the geometry. Erturk & Gokcol[12], Liao & Zhu [13] have presented the solution of two dimensional incompressible flow in the driven cavity for $Re \leq 100$. Prasad & Koseff[14] have done few experiments on three dimensional driven cavity flows. These experimental studies present important information about the physics of the flows in the driven cavity. In the present study we have used two dimensional navier stokes equation and all of the solution presented based on the assumption that the flow is two dimensional.

II. Pressure based solver

A. Finite volume method for one dimensional steady state diffusion problems

Here we develop the numerical method based on integration, the finite volume (or control volume) method, by considering the simplest transport process of pure diffusion in the steady state. The governing equation of steady diffusion can easily be derived from the general transport equation. This gives

$$\text{Div}(\Gamma \text{grad}\phi) + S_{\phi} = 0 \quad (1)$$

Consider the steady state diffusion in a one-dimensional domain as shown in figure 1 . The process is governed by

$$\frac{d}{dx} \left(\Gamma \frac{d\phi}{dx} \right) + S = 0 \quad (2)$$

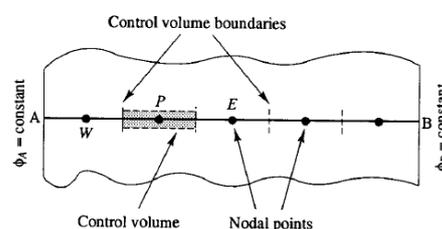


Figure 1: One-dimensional domain

Where Γ is the diffusion coefficient and S is the source term. Boundary values at points A and B are prescribed.

Step 1: Grid generation

The first step in the finite volume method is to divide the domain into discrete control volumes. The boundaries (or faces) of control volumes are positioned mid-way between adjacent nodes. Thus each node is surrounded by a control volume or cell.

Step 2: Discretisation

The key step of the finite volume method is the integration of the governing equation (or equations) over a control volume to yield a discretised equation at its nodal point. For the control volume defined above this gives.

$$\frac{d}{dx} \left(\Gamma \frac{d\phi}{dx} \right) dV + \int_{\Delta V} S dV = \left(\Gamma A \frac{d}{dx} \right)_e - \left(\Gamma A \frac{d}{dx} \right)_w + \bar{S} dV = 0 \quad (3)$$

Above equation states that the diffusive flux leaving the east face minus the diffusive flux entering the west face is equal to the generation.

Step 3: Solution of equations:

Discretised equations of the form 4 must be set up at each of the nodal points in order to solve a problem. For control volumes that are adjacent to the domain boundaries the general discretised equation 4 is modified to incorporate boundary conditions. The resulting system of linear algebraic equations is then solved to obtain the distribution of the property at nodal points. Any suitable matrix solution technique may be enlisted for this task.

$$a_P \phi_P = a_W \phi_W + a_E \phi_E + S_u \quad (4)$$

III. Pressure-Velocity Linkage

In the momentum equation, pressure forces appear as a source of momentum; e.g. in the x-momentum equation

$$\text{Net pressure force} = (P_w - P_e) A$$

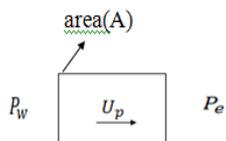


Figure 2: control volume for momentum equations

The discretised momentum equation is of the form

$$a_p u_p - \sum_F a_F u_F = A(P_w - P_e) + \text{other forces} \quad (5)$$

		N	
		.	n
W	.	w	P
	.		e
		S	
		.	S

Figure 3: Storage of variables

$$\begin{aligned} 0 &= (uA)_e - (uA)_w + \dots \\ &= (\rho Ad)_e (P_P - P_E) - (\rho Ad)_w (P_W - P_P) + \dots \\ &= -a_W P_W + a_P P_P - a_E P_E + \dots \end{aligned} \quad (6)$$

This has the same algebraic form as the scalar-transport equations.

IV. Results and Discussions

Test case 1: One dimensional diffusion problem with source term

By considering a source heat conduction equation with fixed temperature boundary condition of 100°C and 200°C at two ends with uniform heat generation and constant thermal conductivity as shown in figure 4 and it is governed by equation (7).

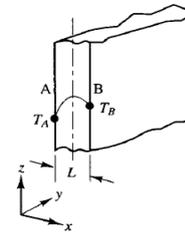


Figure 4 : Large plate

$$\frac{d}{dx} \left(k \frac{dt}{dx} \right) + q = 0 \quad (7)$$

To incorporate the boundary condition at nodes 1 and 5 we apply the linear approximation for temperature between a boundary point and the adjacent nodal point.

Nodes	1	2	3	4	5
Distance (meter)	0.002	0.006	0.01	0.04	0.018
Numerical solution	150	218	254	258	230
Exact solution	146	214	250	254	226
Percentage error	2.73	1.86	1.60	1.57	1.78

Table 1: numerical and analytical results

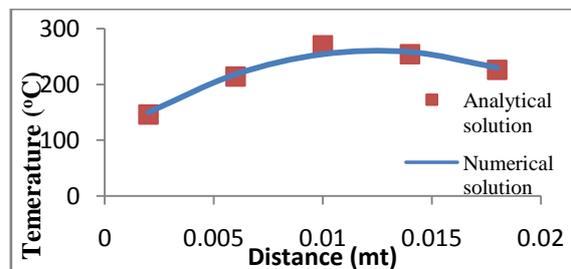


Figure 5: Comparison of numerical results with exact solution for coarse grid.

Figure 5 shows the exact solution and the numerical results coinciding. The table1 shows the exact solution of governing

equation. Maximum percentage error is 2.73 . For the same problem numerical solution can be improved by employing a finer grid, the numerical values of the coefficients and source term are different owing to the smaller grid spacing.

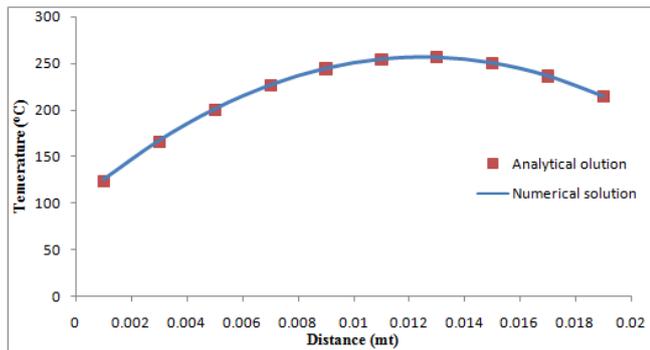


Figure 6: Comparison of numerical results with analytical solution for fine grid.

The comparison of results of the fine grid (10 nodes) is shown in figure 6. Finer grid results shows the better agreement with the analytical solution and the percentage error is 0.8.

Test case 2: Potential equation

A square domain is used to simulate the flow over a stationary cylinder in order to validate the numerical procedure. Present solver is validating with potential flow equations since the assumptions of potential flow are such that an analytical solution exists for cases whose geometries are relatively simple. The domain is two dimensional and has the length of 4mt & width of 4mt. A circular cylinder is placed inside the domain so that its centre has co-ordinate $x=2d$ & $y=2d$.

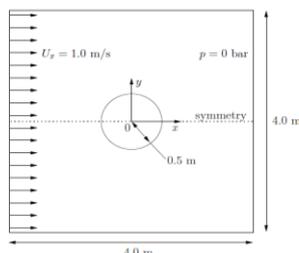


Figure 7 : Geometry of flow round the cylinder

Governing equations

- Mass continuity for an incompressible fluid

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

Pressure equation for an incompressible, irrotational fluid assuming steady-state conditions

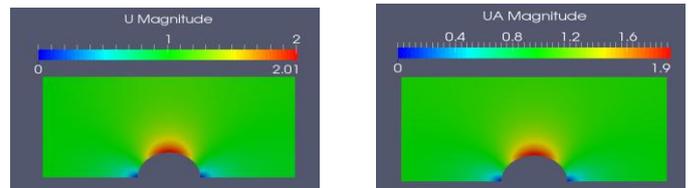
$$\nabla^2 p = 0$$

Boundary conditions

- Inlet (left) with fixed velocity $\mathbf{U} = (1, 0, 0)$ m/s.
- Outlet (right) with a fixed pressure $p = 0$ Pa.

- No-slip wall (bottom);
- Symmetry plane (top).

No fluid properties need to be specified since the flow is assumed to be incompressible and inviscid. Present method executes an iterative loop around the pressure equation which solves in order that explicit terms in the Laplacian term may be updated in successive iterations. In the first instance the pressure equation is solved once, and there is no non-orthogonal correction.



a: Numerical solution **b:** Analytical solution

Figure 8: Velocity counter plots of potential flow.

The solution is shown in Figure 8(a) the velocity counter plots passing across the domain are smooth with no significant error as in the analytical solution in Figure 8(b).

Test case 3: Lid-Driven cavity flow

The lid driven cavity flow is most probably one of the most studied fluid problems in computational fluid dynamics field. The simplicity of the geometry of the lid driven cavity flow makes the problem easy to code and apply boundary conditions and etc. Even though the problem is simple in many ways, the flow in a cavity retains all the flow physics with counter rotating vortices appear at the corners of the cavity.

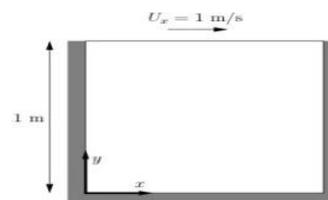


Figure 9: Velocity profile for Reynolds number 100

Governing equations

The incompressible Navier-stokes equation may be written in dimensionless form.

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

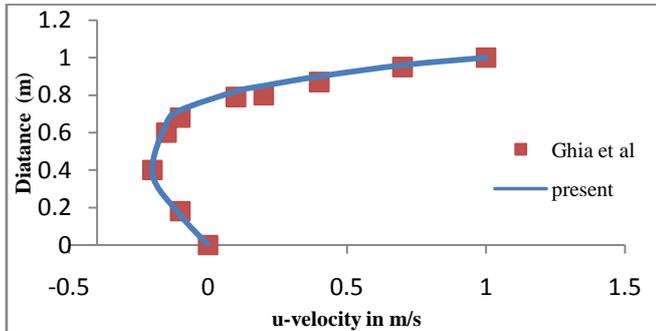
$$\frac{\partial \mathbf{u}}{\partial x} + (\mathbf{u} \cdot \nabla) \mathbf{u} = - \nabla p + \frac{1}{Re} \nabla^2 \mathbf{u}$$

Where, in the usual notation, $\mathbf{u} = (u,v)$ denotes the velocity field, p is the pressure and Re is a Reynolds number.

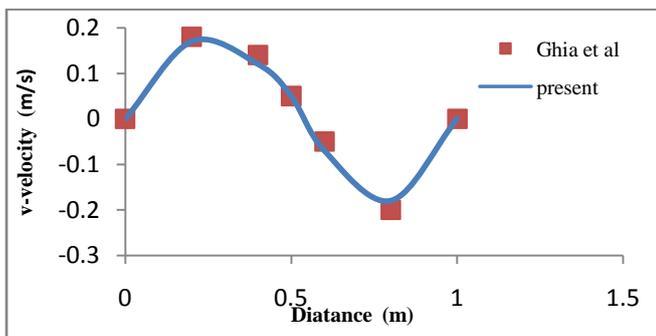
Boundary conditions

No-slip velocity boundary condition ($U=V=0$) is applied to all the walls, except the top lid. On the top lid ($U=1$ & $V=0$) is applied. The bottom boundary is modeled as wall is shown in figure 9.

For an assessment of the accuracy of the present results at Reynolds number 100, the velocity components through the vertical and horizontal centerlines of the cavity are compared with the corresponding numerical results of Ghia et al [15] in figure.



a: u-velocity profile along a vertical line



b: v-velocity profile along a horizontal line

Figure 10: Comparison of velocity profile along a vertical line and horizontal line passing through the centre of the cavity at Reynolds number 100.

Comparison shows the solutions by present solver are smooth and in excellent agreement with benchmark results in Ghia et al[15].

V. Conclusions

- The present pressure based method is tested for 1-D scalar transport Problem using finite volume method and compared the results with analytical solutions. The results are good agreement with analytical solutions for diffusion problem.

- We are validated the solver with potential flow equations for flow over cylinder, the analytical results and the numerical results are exactly matching as shown in figure
- The method is also tested for lid-driven cavity problem at ($Re=100$) and solutions are accurate and in excellent agreement with benchmark results in the literature[16]. Although we have presented only a two-dimensional application of the present method, the extension of the method to three-dimensional problems is direct.

References

- i. Patankar, S. V. and Spalding, D. B., A Calculation Procedure for Heat Mass and Momentum Transfer in Three-Dimensional Parabolic Flows, *Int. J. Heat Mass Transfer*, Vol. 15, p. 1787, 1971.
- ii. Harlow, F., Welsh, J. "Numerical Calculation of Time-Dependent Viscous Incompressible Flow with Free Surface", *Physics of Fluids*, vol.8, p. 2182-2189, 1965.
- iii. Chorin, A. J., "A Numerical Solution of the Navier-Stokes Equations", *Math. Comp.*, 22:742-768, Vol. 2, p. 12-26, 1968.
- iv. Ping Lin Qingping Guo, Xianqiao Chen, "A fully explicit method for incompressible flow computation", *Comput. Methods Appl. Mech. Engrg.*, 192, p. 2555-2564, 2003.
- v. Long Lee, "A class of high-resolution algorithms for incompressible flows", *Computers & Fluids*, 39, p. 1022-1032, 2010.
- vi. D.F. Fletcher, A. Thyagaraja, "A finite difference error arising from the use of a staggered grid", *Applied Mathematical Modelling*, p.496-498, 1991.
- vii. M. peric, R. Kessler, G. Scheuerer. Comparison of Finite-volume numerical methods with staggered and collocated grid. *Computers & Fluids*, Volume 16, Issue 4, P.389-403.
- viii. Rhie, C. M. and Chow, W. L., "A Numerical Study of the Turbulent Flow Past an Isolated Airfoil with Trailing Edge Separation", *AIAA J.*, Vol. 21, p. 1525, 1983.
- ix. D. Goldstein, R. Hadler, L. Sirovich, "No-slip flow boundary with an external force field", *J. Comp.* 105 p. 354, 1993.
- x. T. Ye. R. Mittal, H S. Udaykumar, W. Shyy, "An accurate Cartesian grid method for viscous incompressible flows with complex boundaries", *J. Comp.* P.156-209, 1999.
- xi. M.C. Lai, C.S. Peskin, "An immersed boundary method with formal second-order accuracy and reduced numerical viscosity", *J. Comp.* P.160-705, 2000.
- xii. Erturk, E. gokcol, "Fourth Order Compact Formulation of Navier-Stokes Equation and Driven Cavity Flow at High Reynolds Numbers. *International Journal for Numerical Methods in Fluids* 50, 421-436.
- xiii. Liao, S.J, Zhu, J.M, "A Short Note on Higher-Order Streamfunction- Vorticity Formulation of 2-D Steady State Navier-Stokes Equations. *International Journal for Numerical Methods in Fluids* 22, P.1-9, 1996.
- xiv. Prasad, A.K, Koseff J.R, "Reynolds Number and End-Wall Effects on a Lid Driven Cavity Flow, *Physics of Fluids*, P.208-218, 1989.
- xv. Ghia U, Ghia KN, Shin CT Rose ME. "A numerical study of two dimensional Navier-Stokes equations and multigrid method. *Journal of computational physics* 1982; 387-411.