# PRESSURE BOUNDARY CONDITIONS FOR A SEGREGATED APPROACH TO SOLVING INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

*Tony W. H. Sheu, Morten M. T. Wang, and S. F. Tsai*
*Department of Naval Architecture and Ocean Engineering, National Taiwan University, 73 Chou-Shan Road, Taipei, Taiwan, Republic of China*

***It has been well accepted that Dirichlet and Neumann boundary conditions for the pressure Poisson equation give the same solution. The purpose of this article is to reveal that the above statement is computationally acceptable but is not theoretically correct. Analytic proof as well as computational evidences are presented through examples in support of our observation. In this work we address that the mixed finite-element formulation for solving incompressible Navier-Stokes equations in primitive variables is equivalent to the formulation that involves solving the pressure Poisson equation, subject to Neumann boundary conditions, iteratively with the momentum equations provided the velocity field is classified as having divergence-free and conservative properties.***

## 1. INTRODUCTION

Navier-Stokes equations, together with the divergence-free constraint condition, represent a classical formulation for solving the incompressible Navier-Stokes fluid flows. The pressure in the compressible Navier-Stokes equations serves as a thermodynamic property and is related to other thermodynamic properties through the kinetic equation of state. As to the pressure in their incompressible counterparts, it serves as a Lagrangian multiplier [1]. The equation needed to account directly for the pressure unknown is lacking. It is the distinct nature of the pressure in the flow equations that suggests it is impractical to compute an incompressible Navier-Stokes flow problem by virtue of the compressible code. In the discussion that follows, we will restrict our attention to incompressible Navier-Stokes equations cast in primitive variables.

Major hurdle in the numerical simulation of incompressible fluid flow analysis lies in ensuring that the velocity vector accommodates the divergence-free (solenoidal) property. The necessity of incorporating this kinematic constraint

Received 26 September 1997; accepted 21 May 1998.

This research was supported by the National Science Council under contract NSC82-0209-E-002-422. The authors wish to thank the Computer Center of National Taiwan University and the National Center for High-Performance Computing (NCHC) for making a CRAY XMP EA/116se and an IBM ES/9000-860, respectively, available, which made this work possible.

Address correspondence to Prof. Tony W. H. Sheu, National Taiwan University, Institute of Naval Architecture, 73 Chow-Shan Road, Taipei, Taiwan, Republic of China. E-mail: sheu@indy.na.ntu.edu.tw

**Numerical Heat Transfer, Part B, 34:457–467, 1998**

**1040–7790/98 $12.00 + .00**

## NOMENCLATURE

| | | | |
|---|---|---|---|
| $a$ | vector defined in Eq. (19) | $u_0$ | reference velocity |
| $f$ | boundary force per unit volume | $\gamma_\xi, \gamma_\eta$ | Peclet numbers defined in Eq. (6) |
| $g$ | boundary velocity vector | $\nu$ | kinematic viscosity |
| $L$ | characteristic length | $\Gamma$ | boundary of $\Omega$ |
| $n$ | unit outward normal vector of $\Gamma$ | $\Omega$ | physical domain |
| $N^i$ | shape function | | |
| $p$ | modified pressure | **Subscript** | |
| Re | Reynolds number ($\equiv u_0 \mathrm{L}/\nu$) | | |
| s | unit tangent vector of $\Gamma$ | ref | reference quantities |

condition presents a challenge for the development of an incompressible solution algorithm because this flow analysis becomes a mathematical constraint problem. Circumvention of this difficulty can be achieved by an exact enforcement of the continuity constraint. In what follows, this class of methods is referred to as the mixed formulation. By definition, primitive variables are solved from a coupled differential system consisting of the divergence-free constraint equation and equations of motion (or momentum equations).

While use of mixed formulation is very widespread, it suffers from the lack of pressure unknowns in the continuity equation. This leads to an ill-conditioning matrix with as many diagonal zeros as continuity equations in the incompressible flow simulation. The resulting indefinitive and unsymmetric matrix equations are extremely detrimental to iterative solvers [2]. This potential difficulty limits the algorithmic designer's choices for applying existing iterative solvers and motivated the development of uncoupled methods.

Common to uncoupled approaches is that they all involve using the pressure Poisson equation (PPE) as a replacement for the continuity equation [3]. The pressure Poisson equation is derived by differentiating momentum equations with respect to $x$ and $y$ and adding them together. In the uncoupled formulation, the pressure equation is solved iteratively with the momentum equations to render primitive variable solutions as the convergence criterion set by the user is reached. Uncoupled formulations present an important advantage over their coupled counterparts in that the discrete approximations of the unknown variables are not subject to $\mathscr{LBB}$ (Levy-Brezzi Babŭska) div-stability condition [4, 5]. This inf-sup div-stability condition must be satisfied instead in the choice of finite elements for primitive variables in the mixed formulation [6]. However, the pressure equation has the vector momentum equations as its implicit pressure boundary conditions. One can adopt the normal component of the momentum equations to yield a Neumann boundary condition. The tangential component of the momentum equations is also permissible as a boundary condition for the pressure Poisson equation. The question is raised as to which boundary condition is permissible as the boundary condition for the PPE. This presents a challenging problem in computational fluid dynamics (CFD) to prescribe numerically the proper boundary conditions. Traditionally, the PPE is solved with the Neumann boundary condition obtained from applying the normal component of the momentum equations at the boundary. The rationale behind this approach is due to Gresho and Sani, who showed that Dirichlet and Neumann boundary conditions for the PPE give the same solution [3]. This finding is consistent with the study of Orszag and Israeli [7]. In contrast, Moin and Kim [8] pointed out that the Neumann and Dirichlet boundary conditions may not provide the same solution. The analysis formulated on staggered grids can be used to circumvent the difficulty regarding the ambiguity of applying boundary condition used together with the PPE [9]. On staggered grids, the boundary can be arranged to pass through the velocity nodes. With this method, no pressure boundary condition is required for the solution of incompressible flows. Instead, boundary values needed are inferred from information on the boundary velocities [10, 11]. The reader is referred to a thorough review of the pressure boundary conditions for the incompressible Navier-Stokes equations [3].

Within the finite-element framework, we are aiming at clarifying proper use of boundary conditions for the PPE equation. To answer this question, we considered two sets of divergence-free velocity vector. Given the first velocity vector, the pressure can be derived analytically from the incompressible Navier-Stokes equations. This is not the case for the second velocity vector. The pressure can only be derived analytically from the pressure Poisson equation, subject to the Neumann boundary condition. In this study, an incompressible Navier-Stokes code is developed with the aim of clarifying the importance of applying the legitimitive boundary condition in the segregated approach.

The remainder of this article is organized as follows. An overview of the working equations is given in Section 2. Two finite-element models used for predicting the incompressible fluid flow are described in Section 3. We present in Section 4 a discussion of boundary conditions for closure of a truly incompressible Navier-Stokes flow. This is followed by the presentation of computed results in Section 5 to confirm our finding. In Section 6, we offer conclusions.

## 2. WORKING EQUATIONS

We consider in this article the steady-state equations for incompressible and viscous fluid flows. Of several existing variable settings for numerical simulation of this class of flows, the primitive-variable formulation has been very widespread because of its accommodation of the closure boundary condition [1]. Recognizing this, working equations are cast in the following dimensionless form:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\mathbf{u} \cdot \Delta \mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}} \Delta^2 \mathbf{u} + \mathbf{f} \tag{2}$$

In Eq. (2), the Reynolds number is defined as $\mathrm{Re} = u_0 L / \nu$, where $L$ is a characteristic length, $u_0$ is a reference velocity, $\nu$ is the kinematic viscosity, and $\mathbf{f}$ is the body force per unit volume. The pressure $p$ and $\mathbf{f}$ in Eq. (2) have been normalized by $\rho u_{\mathrm{ref}}^2$ and $\rho u_{\mathrm{ref}}^2 / L$, respectively, where $\rho$ is the fluid density. In order to close the above elliptic-type differential system, it is imperative that the boundary velocity, $\mathbf{u} = \mathbf{g}$, satisfy the following constraint condition:

$$\int_{\partial\Omega} \mathbf{n} \cdot \mathbf{g} \, ds = 0 \tag{3}$$

In Eq. (3), $\mathbf{n}$ denotes the unit outward normal to the boundary $\partial\Omega$. According to Ladyzhenskaya [12], specification of pressure at $\Gamma$ is not permitted on the no-slip wall; otherwise, the differential system (1)–(3) will be overdetermined. As is now clear from (1)–(2), there is no explicit pressure equation to solve for the pressure field. Since equations of motion account for the vector quantities $\mathbf{u}$ and $\mathbf{v}$, the remaining equation demands that all admissible solutions to the momentum equations must satisfy the solenoidal kinematic constraint, $\nabla \cdot \mathbf{u} = 0$. This shows that pressure plays two distinct roles in the differential system given by (1)–(2). The pressure serves not only as a force in the mechanical balance law for conservation, but also as a continuity constraint.

## 3. NUMERICAL MODELS

### Mixed Finite-Element Formulation

As noted earlier, our strategy to solve for finite-element solutions of equations (1)–(2) is to apply a mixed finite-element model. To obtain the discrete problem, we make expansions $\mathbf{u}^h = \Sigma N^i u_i^h$ and $p^h = \Sigma M^i p_i^h$. Here, the basis for all primitive variables is given by bilinear pressure and biquadratic velocity approximation. It has long been known that this element setting meets the div-stability condition, thus preventing a possible occurrence of even–odd spurious pressure modes [4, 5]. In the numerical prediction of advection-dominated transport equations, prediction deteriorates as a result of false diffusion errors [10] and oscillations in the solution. False diffusion errors are extremely detrimental to multidimensional flow simulations, particularly the flow at an angle to the grid line. These cross-wind errors may distort the transport profile in the direction normal to the flow direction. Reduction of this type of error can be made by resorting to the streamline-upwind finite-element formulation implemented in a multidimensional context.

To avoid oscillatory velocities in the case of problems with higher Reynolds numbers, we adopted the Petrov-Galerkin finite-element model. Our refinement to the Galerkin formulation is to add $B^i$, defined below, to the shape function $N^i$ to render upwinding. That is,

$$B^i = \tau\left(N^j \tilde{V}_k^j\right) \frac{\partial N^i}{\partial X_k} \tag{4}$$

As seen in Eq. (4), the free parameter $\tau$ determines the degree of upwinding needed to suppress oscillatory velocities. In the present study, $\tau$ is derived through an operator-splitting approximation. The resulting expression for $\tau$ takes the following form:

$$\tau = \frac{\delta(\gamma_\xi)\tilde{V}_\xi h_\xi + \delta(\gamma_\eta)\tilde{V}_\eta h_\eta}{2|\tilde{V}|^2} \tag{5}$$

In quadratic elements, use of $\tau$ in Eq. (5) provides nodally exact solutions in one dimension provided that the coefficient $\delta$ is given as

$$\delta(\gamma) = \frac{1}{2}\coth\left(\frac{\gamma}{2}\right) - \frac{1}{\gamma} \tag{6}$$

In the above equations, $\hat{e}_\varepsilon$ and $\hat{e}_\eta$ denote the local coordinate basis unit vectors. The Peclet numbers $\gamma_\xi$ and $\gamma_\eta$ are denoted by $\gamma_\xi = \tilde{V}_\xi h_\xi \,\mathrm{Re}/2$, $\gamma_\eta = \tilde{V}_\eta h_\eta \,\mathrm{Re}/2$.

## Uncoupled PPE Formulation

Uncoupled methods represent another class of incompressible algorithms for solving the primitive-variable form of Navier-Stokes equations [9, 10, 13]. The main feature of this class of methods is that one avoids solving the divergence-free equation directly. These methods are favorably applied to three-dimensional analyses because the demand on computer memory can be largely reduced. Typical of uncoupled approaches is that the intermediate velocity field has been mapped onto a zero-divergence velocity subspace. Use of an operator-splitting or fractional-step method helps reduce the computational time as well as disk storage. These methods, however, suffer from uncertainties in specifying intermediate boundary conditions and, thus, introduce splitting errors. Compared with the SIMPLE-type segregated solution algorithm [10], the projection method has the advantage of getting rid of the perplexing pressure-correction calculations from the momentum equations. Unfortunately, one cannot dispense with computation of the pressure from the Poisson equation, subject to the appropriated boundary condition. Recognizing this, a decision is made to choose a legitimative boundary condition to close the Poisson differential equation. For the problem to be well-posed, pressure boundary conditions should guarantee the satisfaction of the continuity equation for the fluid even close to the boundaries. This presents a significant research topic to retain the divergence-free velocity condition, which is an important requirement in the simulation of incompressible fluid flow.

Application of the divergence operator to Eq. (1), together with use of Eq. (2), yields the so-called pressure Poisson equation in $\Omega$:

$$\nabla^2 p = \nabla\left(\frac{1}{\mathrm{Re}}\nabla^2\mathbf{u} - \mathbf{u}\cdot\nabla\mathbf{u} + \mathbf{f}\right) \tag{7}$$

The above equation is used to replace the continuity equation in the uncoupled formulation for solving incompressible Navier-Stokes equations. In Eq. (7), the elliptic nature demands that the boundary condition for pressure be specified.

According to Gresho and Sani [3], a judicious boundary equation equipped with the PPE formulation is as follows:

$$\frac{\partial p}{\partial n} = \left( \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + f \right) \cdot \mathbf{n} \tag{8}$$

Equations (7) and (8), along with Eq. (2), constitute a decoupled formulation for the incompressible viscous flow problem. In what follows, we will refer to this problem as a Poisson/Neumann problem.

## 4. BOUNDARY CONDITION FOR PRESSURE POISSON EQUATION

We consider in this article two sets of divergence-free velocity vectors to discuss pressure boundary conditions.
*Problem 1*:

$$\mathbf{u}_1(x, y) = \left( -\frac{2(1+y)}{(1+x)^2 + (1+y)^2}, \frac{2(1+x)}{(1+x)^2 + (1+y)^2} \right) \tag{9}$$

$$p_1(x, y) = c_1 - \frac{2}{(1+x)^2 + (1+y)^2} \tag{10}$$

*Problem 2*:

$$u_2(x, y) = (\cos(\pi x)\cos(\pi y), \sin(\pi x)\sin(\pi y)) \tag{11}$$

$$\begin{aligned} p_2(x, y) = c_2 &- \frac{1}{4}\cos(2\pi x) + \frac{1}{4}\cos(2\pi y) \\ &+ \frac{2\pi}{\mathrm{Re}} \operatorname{csch}(\pi)\{\cosh(\pi x) + \cosh[\pi(x-1)]\}\cos(\pi y) \end{aligned} \tag{12}$$

In Eqs. (10) and (12), $c_1$ and $c_2$ are constant values. Equations (9)–(10) and (11)–(12) are all solutions to Eqs. (7)–(8). Through mathematical manipulation, one can show that equations (9)–(10) also satisfy exactly the working equations (1)–(2). This is not the case for the Problem 2. It is important to note that the velocity–pressure pair $(\mathbf{u}_1, p_1)$ also satisfies the following condition along the tangential direction s:

$$\frac{\partial p}{\partial s} = \left( \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{f} \right) \cdot \mathbf{s} \tag{13}$$

By contrast, the pair $(\mathbf{u}_2, p_2)$ satisfies the PPE equation/Neumann condition but violates Eq. (13). It is this striking difference that motivates us to divide the divergence-free velocity vectors into two exclusive subsets: $\mathbf{u}_1 \in \mathrm{DIV}_1$ and $\mathbf{u}_2 \in \mathrm{DIV}_2$. In the following, we try to answer "under what conditions can we solve for **u** and $p$ from the uncoupled formulation given by Eqs. (2), (7), and (8), subject to the Dirichlet-type velocity boundary condition." To answer this question, we present the following two theorems.

**Theorem 1.** The pressure computed from Eqs. (2), (7), and (8) is analytic only if the following two conditions hold for a velocity vector $\mathbf{u}$:

$$\nabla p = \mathbf{a}(\mathbf{u}) \tag{14}$$

$$\nabla \times \mathbf{a} = 0 \tag{15}$$

where

$$\mathbf{a}(\mathbf{u}) = \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{f} \tag{16}$$

**Proof.** According to Eqs. (1) and (2), the solutions sought are those satisfying the following two relations:

$$\nabla \cdot \mathbf{u} = 0 \tag{17}$$

$$\nabla p = \mathbf{a}(\mathbf{u}) \tag{18}$$

where

$$\mathbf{a}(\mathbf{u}) = \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{f} \tag{19}$$

With Eq. (18), the following equation holds analytically:

$$\nabla \times \mathbf{a}(\mathbf{u}) = 0 \tag{20}$$

This implies that for incompressible Navier-Stokes fluid flows, it is impossible to find a scalar $p$ that coexists with a velocity field $\mathbf{u}$ under the circumstances:

$$\nabla \times \mathbf{a}(\mathbf{u}) \neq 0 \tag{21}$$

**Theorem 2.** In the numerical simulation of steady-state incompressible Navier-Stokes equations, the mixed formulation given in Eqs. (1)–(2) is equivalent to the uncoupled formulation comprising a PPE equation, subject to the Neumann condition given by Eq. (8), and momentum equations, subject to the Dirichlet type of velocity boundary condition, on condition that the prescribed boundary velocity satisfies Eq. (20). Under the circumstances, use of boundary conditions (8) and (13) for the pressure Poisson equation gives the same solution.

**Proof.** According to [14], the coupled PPE and momentum equations are equivalent to the continuity equation and momentum equations in the interior domain. This completes the first part of the proof. By virtue of Theorem 1, if the boundary velocity $\mathbf{u}$ takes a divergence-free form and can yield a conservative field $\mathbf{a}$, the momentum equations are satisfied at the boundary on a pointwise basis. In other words, along the boundary, the following equation holds:

$$\nabla p = \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{f} \tag{22}$$

By virtue of the above equation, Eqs. (8) and (13) are obviously attainable. This corresponds to saying that if there exists a divergence-free velocity that can result in a conservative field **a**, the coupled equations (2), (7), (8) are equivalent to Eqs. (1)–(2), subject to velocity boundary conditions.

## 5. COMPUTED RESULTS

We will now numerically justify whether or not Eq. (8) is permissible as a boundary condition for the pressure Poisson equation (7) in the numerical simulation of incompressible viscous fluid flow. In the first place, we have to validate the proposed streamline upwind Petrov-Galerkin finite-element (SUPG) model developed to simulate incompressible Navier-Stokes flows. In a unit square, calculations were carried out on continuously refined grids, thus facilitating the convergence test for the finite-element model. For this study (Re = 1,000), we plot in Table 1 the computed $L_2$ error norms for the cases with $\Delta x = \Delta y = 1/5, 1/10, 1/20$, and $1/40$. It is clear from Table 1 that the solutions improve with mesh refinement. Also given in Table 1 are rates of convergence computed from the errors. These computed rates of convergence agree with the theoretical prediction of the quadratic convergence.

We then applied the analytically verified quadratic SUPG finite-element code to solve the incompressible viscous problem, subject to the boundary condition given by Eq. (11). It is surprising to find from Figure 1, which plots computed contours for $u_2, v_2$, and from Figure 2, which plots the computed pressure $p_2$, that these solutions agree well with velocities given by Eq. (11) and the pressure given by Eq. (12). Theoretically, Eqs. (11) and (20) are not solutions to Eqs. (1)–(2). Instead, Eqs. (11)–(12) are solutions of Eqs. (7)–(8) but not for the equations given by (7) and (13). Inferring from the computed results, we can conclude that while conditions given by Eqs. (16) and (20), together with Eq. (17), are required to yield an exact equivalance between the mixed and PPE formulations, it is computationally impossible to implement them in real simulations to ascertain whether or not the specified boundary has the properties given by Eqs. (16), (17), and (20). Example calculations support that it is computationally accepted to employ the PPE, subject to the Neumann-type boundary condition, and equations of motion to solve the incompressible Navier-Stokes equations.

**Table 1.** Computed results and convergent order based on a finite-element model, where pressure gradients are upstream-weighted

| Mesh size | $\Vert \mathbf{u} - \mathbf{u}_{\text{exact}} \Vert$ | Convergent order | $\Vert p - p_{\text{exact}} \Vert$ | Convergent order |
|---|---|---|---|---|
| $5 \times 5$ | $1.374 \times 10^{-3}$ | | $2.404 \times 10^{-3}$ | |
| $10 \times 10$ | $3.757 \times 10^{-4}$ | 1.871 | $6.085 \times 10^{-4}$ | 1.982 |
| $20 \times 20$ | $8.817 \times 10^{-5}$ | 2.091 | $1.421 \times 10^{-4}$ | 2.098 |
| $40 \times 40$ | $1.863 \times 10^{-5}$ | 2.242 | $3.446 \times 10^{-5}$ | 2.044 |

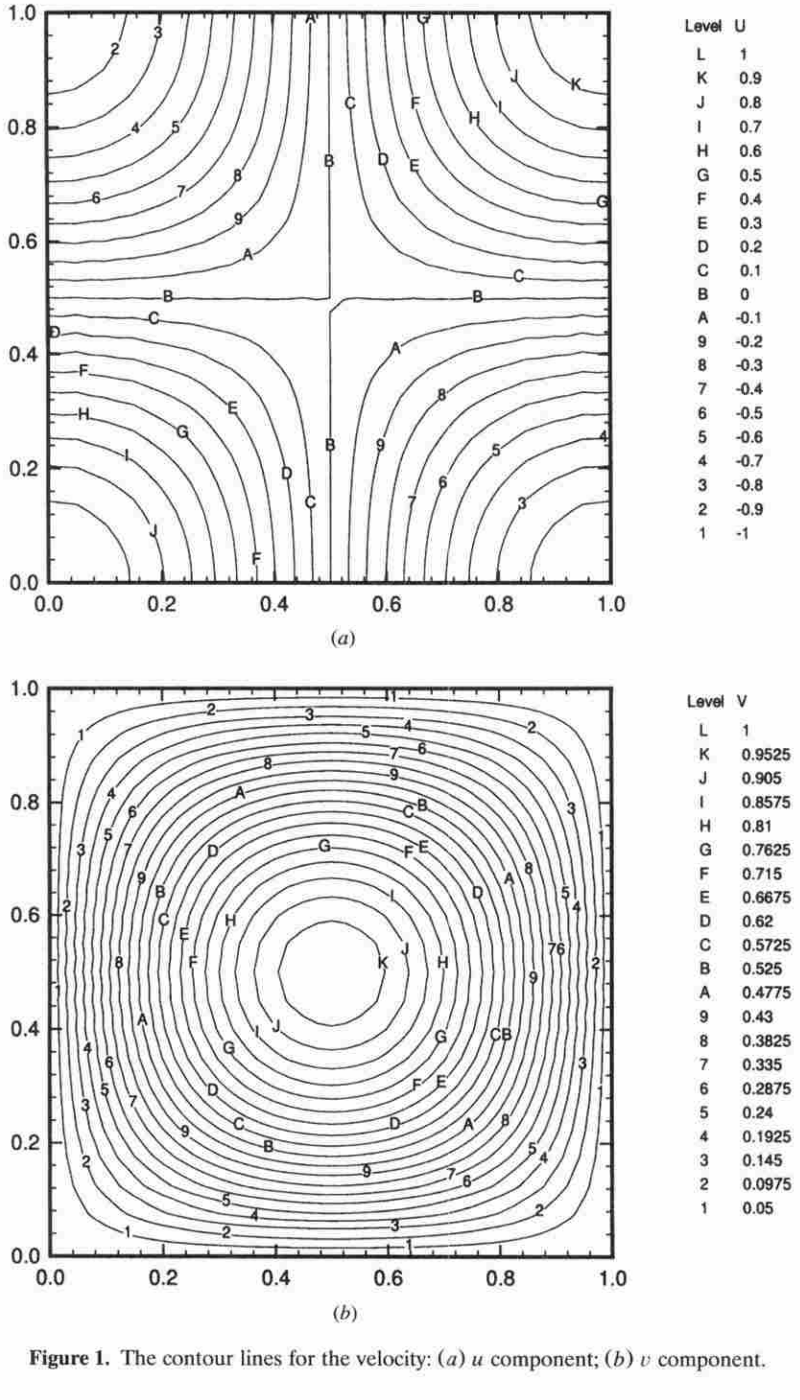


**Figure 1.** The contour lines for the velocity: (*a*) *u* component; (*b*) *v* component.

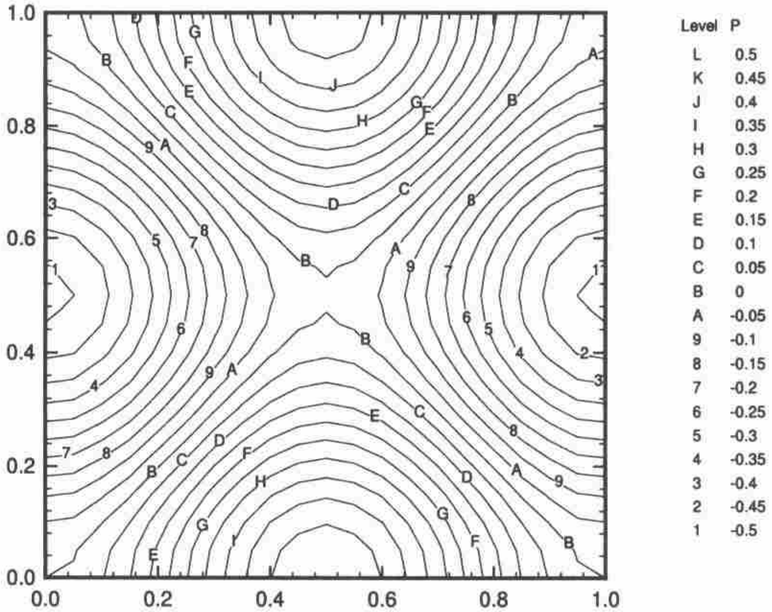


**Figure 2.** The contour lines for the pressure.

## 6. CONCLUSIONS

In the numerical simulation of incompressible Navier-Stokes equations, we showed that the mixed formulation is equivalent to the PPE formulation provided the divergence-free boundary velocity field satisfies the condition

$$\nabla \times \mathbf{a} = 0$$

where

$$\mathbf{a}(\mathbf{u}) = \frac{1}{\mathrm{Re}} \nabla^2 \mathbf{u} - \mathbf{u} \cdot \nabla \mathbf{u} + \mathbf{f}$$

The present study showed that even when the acceleration-like vector field $\mathbf{a}(\mathbf{u})$ is inaccessible, the finite-element solutions computed iteratively from the momentum equations and the pressure Poisson equation, subject to the Neumann boundary condition, are computationally acceptable.

## REFERENCES


1. L. Quartapelle, *Numerical Solution of the Incompressible Navier-Stokes Equations*, Birkhäuser Verlag, Berlin, 1993.
2. Morten M. T. Wang and Tony W. H. Sheu, An Element-by-Element BiCGSTAB Iterative Method for Three-Dimensional Steady Navier-Stokes Equations, *J. Comput. Appl. Math.*, vol. 79, pp. 147–165, 1997.
3. P. M. Gresho and R. L. Sani, On Pressure Boundary Conditions for the Incompressible Navier-Stokes Equations, *Int. J. Numer. Meth. Fluids*, vol. 7, pp. 1111–1145, 1987.

4. F. Brezzi, On the Existence, Uniqueness and Approximation of Saddle Point Problems Arising from Lagrangian Multipliers, *RAIRO, Anal. Num.*, vol. 8 (R2), pp. 129–151, 1974.
5. I. Babuška, Error Bounds for Finite Element Methods, *Numer. Math.*, vol. 16, pp. 322–333, 1971.
6. M. D. Gunzburger, *Finite Element Methods for Viscous Incompressible Flows, A Guide to Theory, Practice, and Algorithms*, Academic Press, San Diego, 1989.
7. S. Orszag and M. Israeli, Numerical Simulation of Viscous Incompressible Flows, *J. Atmos. Sci.*, vol. 24, pp. 337–355, 1972.
8. P. Moin and J. Kim, On the Numerical Solution of Time-Dependent Viscous Incompressible Fluid Flows Involving Solid Boundaries, *J. Comput. Phys.*, vol. 35, pp. 381–392, 1980.
9. F. H. Harlow and J. E. Welch, Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface, *Phys. Fluids*, vol. 8, pp. 2182–2189, 1965.
10. S. V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington, DC, 1980.
11. J. P. van Doormaal and G. D. Raithby, Enhancements of the SIMPLE Method for Predicting Incompressible Flows, *Numer. Heat Transfer*, vol. 7, no. 5, pp. 147–163, 1984.
12. O. Ladyzhenskaya, *The Mathematical Theory of Viscous Incompressible Flow*, Gordon & Breach, New York, 1969.
13. A. J. Chorin, A Numerical Method for Solving Incompressible Viscous Flow Problems, *J. Comput. Phys.*, vol. 2, pp. 12–26, 1967.
14. P. M. Gresho, Some Current CFD Issues Relevant to the Incompressible Navier-Stokes Equations, *Comput. Meth. Appl. Mech. Eng.*, vol. 87, pp. 201–252, 1991.