Mecánica Computacional, Volumen XIX, pp. 73-78
Compilado por: F. Quintana y S. Felicelli
San Carlos de Bariloche, Noviembre de 2000

SOME ASPECTS OF THE NUMERICAL RESOLUTION OF THE UNSTEADY INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Hugo D. Pasinato
Universidad Nacional del Comahue, DMA
Calle Buenos Aires 1400, 8300-Neuquén, Argentina.
E-mail: hpasinat@uncoma.edu.ar

ABSTRACT

In this paper an iterative method to solve the unsteady incompressible Navier-Stokes equations in primitive variables are presented. The pressure problem is solved with the normal velocity as boundary condition, while the boundary normal pressure gradient is used to obtain the velocity field. A finite volume method with a second-order approximation in space and a second-order Crank-Nicolson in time are used to express the discrete equations. Numerical results for 2D steady and unsteady flow have shown a good performance of the proposed technique to resolve the gross features of the flow.

1 INTRODUCTION

The equations of motion of viscous unsteady incompressible flow are the Navier-Stokes equations

$$\frac{\partial}{\partial t}(\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u}\mathbf{u} - \mu \nabla \mathbf{u}) = -\nabla p \qquad (1-a)$$

$$\nabla \cdot (\rho \mathbf{u}) = 0 \qquad (1-b)$$

where $\mathbf{u}(\mathbf{x}, t)$ is the velocity, $p(\mathbf{x}, t)$ the pressure, μ the viscosity and ρ the specific mass. Equations (1) are subject to the specification of boundary condition for $t > 0$ of $\mathbf{u}$ at rigid no-slip boundaries or specification of u_n and $\partial u_\tau / \partial n$ at no stress boundaries, where n and τ denote the normal and tangential directions.

The numerical implementation can be done in so many different way, that is very hard to write at the present time a complete review of all used methods. When the primitive variables formulation is used, the most common used method to solve equations (1) deal with a Poisson equation for pressure, which is obtained operating the divergence on equation (1-a), and the momentum equations for velocity. The solution of these coupled equations means a pressure field that is in equilibrium with a solenoidal velocity field, which should satisfy the momentum equation.

A general technique that has been used to solve these coupled equations is the splitting or fractional step method [1], [2], [3]. Nonetheless, when a split formulation is used the pressure problem poses a major difficulty due to the lack of boundary conditions for the pressure variable. However, it seems that an alternative to the splitting method is to use directly an iterative method. In other words, a procedure that at each time step solves alternatively the equation for pressure and for velocity. One of the first work on the use of an iterative method to solve the unsteady Navier-Stokes equations belong to Fortin [4].

The author have successfully implemented an iterative method for solving the unsteady incompressible Navier-Stokes equations in 2D and 3D. The overall procedure will be reported in [5]. Here, for space reason, only the essential of the iterative method and some 2D unsteady numerical results are reported.Then, in this paper, in §2. the essential of the commented iterative method, like as some aspects of the pressure problem implementation, are reported. In §3. the numerical solutions of some two dimensional unsteady incompressible flows are reported using the finite volume method, with central second order approximation. Then in §4. it is finished with some conclusions.

2 ITERATIVE METHOD FOR INCOMPRESSIBLE UNSTEADY FLOW

The method presented here is inspired in previous reported methods [4], [6]. The following is a résumé of it: For the space discretization a finite volume formulation with a staggered mesh is used. A centered second-order approximation is used to evaluate the flux at the sub volume surfaces. The second-order Crank-Nicolson scheme is used for time advancing. The pressure into the domain Ω is evaluated using only the normal velocity at the boundaries, while the pressure normal gradient at the boundary Γ is used to evaluate pressure at Γ.

Aiming at to present the essential of the method, the equations (1) are discretized according to the finite volume method. Hence, for an arbitrary subdivision of the physical domain into sub volumes fix in time, the conservation of momentum, for instance for the sub volume Ω_u belong to velocity u, and mass for the sub volume Ω_m, are

$$\frac{\partial}{\partial t}\int_{\Omega_u}(\rho\mathbf{u})\,d\Omega + \oint_{\Gamma_u}(\rho\mathbf{u}\mathbf{u} - \mu\nabla\mathbf{u})\cdot\mathbf{n}\,d\Gamma = -\int_{\Omega_u}\nabla p\,d\Omega \tag{2}$$

$$\int_{\Omega_m}(\rho\mathbf{u})\cdot\mathbf{n}\,d\partial\Omega = 0 \tag{3}$$

where u, v and w are the velocity components with direction coincident with axis x, y and z respectively, the sub volume Ω_m for the continuity equation is centered at the node ijk coincident with the point where pressure is considered, and the sub volumes of velocity u Ω_u are centered at a point staggered in x direction half sub volume from ijk.

Taking the θ scheme, which for $\theta = 0.5$ is the second-order Crank-Nicolson scheme, for the time discretization of the momentum equation and the continuity equation evaluated at time $(n+1)\Delta t$, the integral equations (2-3) at the discrete level are

$$(\forall\rho\mathbf{u})^{n+1}/\Delta t - (\forall\rho\mathbf{u})^{n}/\Delta t + \theta\mathbf{ADF}^{n+1} + (1-\theta)\mathbf{ADF}^{n} = S^{n+1} \tag{4}$$

$$\mathbf{MF}^{n+1} = 0 \tag{5}$$

where $\forall$ is the volume of the discrete sub volume Ω_u for velocity, the terms **ADF** is the net advection - diffusion flux through the surfaces of the sub volume Ω_u, **MF** is the net flux of mass through the surfaces of the sub volume Ω_m, and S is the source term represented by pressure.

Using a centered second-order approximation to express the flux terms through the sub volumes surfaces, the discrete momentum equation for the component u of velocity, which is centered at $i-1/2jk$ is

$$a_u\,u_{ijk}^{n+1} = b_u\,u_{i-1jk}^{n+1} + \ldots + g_u\,u_{ijk+1}^{n+1} + h_u\,(p_{i-1jk} - p_{ijk}) + i_u \tag{6}$$

and the discrete continuity equation centered at node ijk is

$$a_m\,u_{ijk}^{n+1} + b_m\,u_{i+1jk}^{n+1} + c_m\,v_{ijk}^{n+1} + d_m\,v_{ij+1k}^{n+1} + e_m\,w_{ijk}^{n+1} + f_m\,w_{ijk+1}^{n+1} = 0 \tag{7}$$

Aiming at to obtain a Poisson equation, the momentum equation (6) for u is arranged as

$$u_{ijk}^{n+1,m+1} = \frac{1}{a_u}\{b_u\,u_{i-1jk}^{n+1,m} + \ldots + g_u\,u_{ijk+1}^{n+1,m} + i_u\} + \frac{h_u}{a_u}(p_{i-1jk} - p_{ijk}) \tag{8}$$

and the continuity equation is considered as

$$a_m\,u_{ijk}^{n+1,m+1} + b_m\,u_{i+1jk}^{n+1,m+1} + \ldots + e_m\,w_{ijk}^{n+1,m+1} + f_m\,w_{ijk+1}^{n+1,m+1} = 0 \tag{9}$$

Then, substitution of the discrete momentum equation (8) into equation (9) results in the following discrete Poisson equation for pressure

$$a_p\,p_{ijk}^{n+1,m+1} = b_p\,p_{i-1jk}^{n+1,m+1} + \ldots + f_p\,p_{ijk-1}^{n+1,m+1} + g_p\,p_{ijk+1}^{n+1,m+1} + h_p \tag{10}$$

where $a_u, b_u, c_u, \ldots$ and $a_p, b_p, c_p, \ldots$ are the coefficients of the discrete equations.

Equation (10) allows to obtain the pressure in the domain Ω. If the velocity field is known at $(n+1)\Delta t$, through equation (10) the real pressure in the domain is obtained at $(n+1)\Delta t$. The numerical algorithms of the iterative method to advance the solution from time $(n)\Delta t$ to time $(n+1)\Delta t$ is:

(i) Normal pressure gradient at Γ is evaluated for time $(n+1)$ and iteration $(m+1)$.

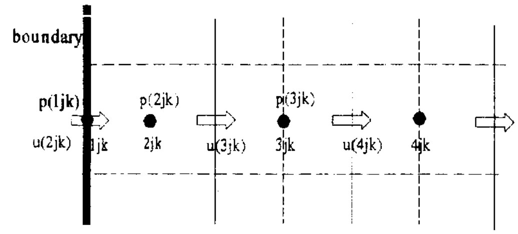


Figure 1: Sub volumes for the momentum equation for velocity u next to the left boundary

(ii) The coefficients $a_u; b_u; c_u; \ldots; a_v; b_v; c_v; \ldots$; for velocity u; v and w are evaluated at $(n+1), (m+1)$.
(iii) Pressure into Ω is evaluated at $(n+1), (m+1)$ from equation (10).
(iv) Velocity u, v and w at $(n+1)(m+2)$ are obtained using the previous coefficients and the evaluated pressure field at the Ω.
(v) Step (i) - (iv) are repeated until a convergence norm is achieved.
(vi) Time is advanced one step.

The pressure field into the Ω is solved using the normal velocity at the boundaries as boundary condition. Using the conservative integral form of the momentum and mass equations to formulate the pressure problem with a staggered mesh, it can be shown [5] that giving the normal velocity at the boundaries for the pressure problem is equivalent to give the proper boundary condition. At this point it is important to remark that only pressure into the domain Ω is obtained from the Poisson equation (10). The pressure at the boundary should be obtained in a different way. It is important to remark also that the previous comment on the pressure problem implementation doesn't mean that boundary condition for pressure, for instance the normal pressure gradient, is not necessary to solve the whole problem. Figure 1 shows a discretized region of the physical space for the component u of velocity, next to the left boundary. The discrete momentum equation for $u(3jk)$ is

$$a_u\, u_{3jk}^{n-1,m-1} = b_u\, u_{2jk}^{n+1,m} + \ldots + g_u\, u_{3jk+1}^{n+1,m} + h_u(p_{2jk} - p_{3jk}) + i_u(p_{1jk} - p_{2jk}) + j_u \qquad (11)$$

where $p(1jk)$ is the pressure at the boundary. Because $(p_{1jk} - p_{2jk})$ represent a source term in the momentum equation, the pressure at boundaries should be evaluated based on an extra equation, based on the pressure boundary condition. In this work the normal pressure gradient at the boundary is used to evaluate $p(1jk)$, as it is shown in [5].

Hence, if the velocity field is known at a time $(n+1)\Delta t$, the pressure gradient at Γ can be evaluated and therefore the coefficients of the momentum equations for u, v and w can also be evaluated. Then, based on these coefficients the pressure field into Ω can be resolved at this time. Therefore, as it was commented above, even though the pressure is evaluated using only the normal velocity at the boundary, in order to enforce momentum conservation in the physical domain, the pressure at the boundaries should be evaluated using the proper boundary condition for pressure, for instance, the normal pressure gradient.

3 NUMERICAL RESULTS

The propose of this section is to perform comparison with the help of some numerical tests. The iterative method reported in §2. has been implemented for both 2D and 3D unsteady flow in Cartesian coordinates. Here some comparison of steady and unsteady 2D flow are reported. Numerical results for steady and unsteady 3D flow will be reported in [5]. The discussion on the numerical results are centered on the comparison between the iterative method presented here and other resolution reported in the literature and with a high-order resolution performed by the author in a previous work [7]. For space reason, only a comparison on the ability of the method to resolve the gross features of the flow is presented. Some 2D

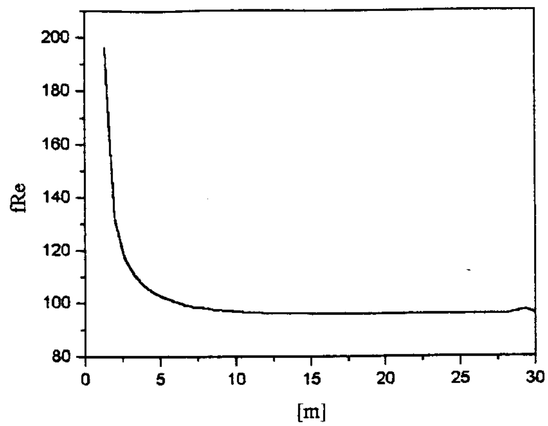


Figure 2: Skin friction coefficient for a developing flow between plates for Re=400

steady and unsteady driven cavity flow and channel flow are reported. The reason of this choice results from the simplicity of the geometry and from the number of numerical results available.

Figure 2 shows the skin-friction coefficient for a developing flow between plates. The only problem in this solution is a little bump presented by the coefficient next to the outflow boundary. It is thought that this oscillation of the pressure field next to the outflow boundary could be yield by numerical errors in the evaluation of the normal pressure gradient at the boundary. However, more numerical test with outflow boundary conditions is necessary to check this problem, and also to check the ability of the method to resolve boundary layers next to rigid boundaries.

For the driven cavity flow the domain is a square of dimensions $[1, 1]$ in both direction, x and y, with the no-slip condition imposed on three sides and with velocity tangent to the fourth ones. In most of the works the fluid velocity have a constant value equal to 1 on this fourth side. The solution of this problem is singular at the corner making more difficult the comparisons. Thus, here the Boucier and Francois driven cavity flow is considered [4]. In this flow the upper side of the square moves with velocity $u_b(x)$. Here, two tests with the Boucier and Francois driven cavity flow were performed. First, for the final steady state case with Reynolds equal to 400 and for the flow at time $t = 2$ seconds for Reynolds equal to 10^4. In the first case, $u_b(x) = -16x^2(1-x)^2$ and the initial date are $u = -(3y^2 - 2y)16x^2(1-x)^2$ and $v = (y^3 - y^2)(1-2x)(1-x)32x$. This test has been extensively solved in [4] with a coharse 20x20 mesh for different schemes, thus aiming at to perform comparison here the same mesh was chosen. In the second case, $Re = 10^4$, $u_b(x) = 16x^2(1-x)^2$ and initial date are $u = (3y^2 - 2y)16x^2(1-x)^2$ and $v = -(y^3 - y^2)(1-2x)(1-x)32x$. Because this case has been solved by E and Liu [8] with a 257×257 mesh, the same mesh was also chosen here for comparison reason.

Figure 3 shows the comparison of velocity u on the line $x = 0.5$ for the first case, for the proposed method and for a fourth-order resolution with a compact scheme [7]. Figure 4 shows also for this case the final steady state for the vorticity field for the iterative method, which uses a second-order volume finite approximation and for the fourth-order scheme. Figure 5 shows the state of the flow for the second case with the driven cavity flow with Re=10^4, for time $t = 2$ seconds. This solution has been also reported by [8]. As it is shown in Figure 5-(a), the characteristic bump of this flow in the upper side of the right-low vortice of this Figure is well resolved.

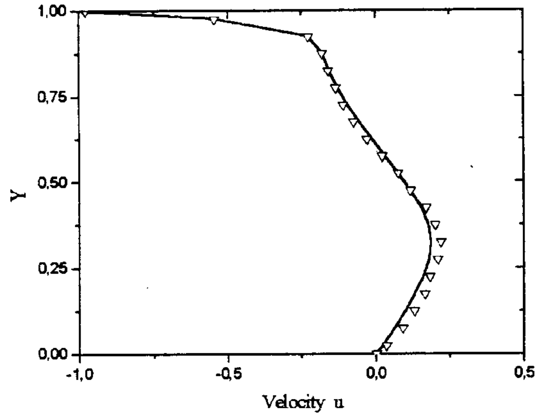


Figure 3: Velocity u on the line $x = 0.5$, for the final steady state for the driven cavity flow for Re=400: (a) —— iterative method, 20 × 20 uniform mesh; (b) ∇ ∇ ∇ ∇ fourth-order compact scheme, 64 × 64 uniform mesh [7]

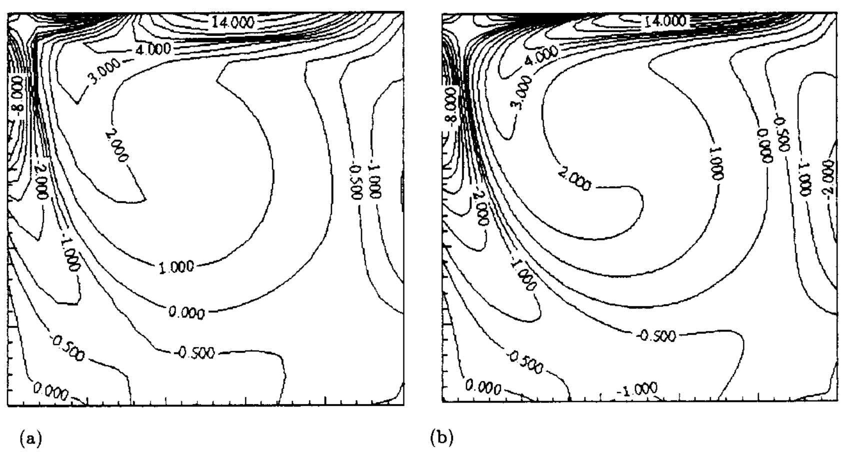
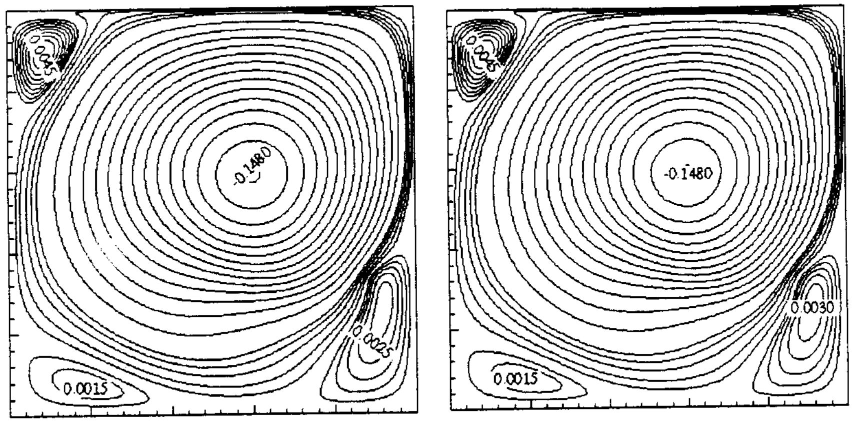


Figure 4: Final state of vorticity for the driven cavity flow for Re=400: (a) Iterative method with a 20 × 20 uniform mesh: (b) Fourth-order compact scheme with a 64 × 64 uniform mesh [7]



(a) (b)

Figure 5: Stream function for the driven cavity flow for $Re = 10^4$ at t=2 seconds. (a) Iterative method, 257×257 uniform mesh; (b) Fourth-order resolution, 128×128 uniform mesh [7]

4 CONCLUSIONS

An iterative method, that resolve alternatively a pressure equation and the momentum equation, to solve the incompressible unsteady viscous flow has been presented. The procedure takes some features of other previous numerical implementation reported in the literature. The pressure equations is solved using the normal velocity at the boundaries and the proper pressure boundary condition is used in the momentum equation. Some steady and unsteady 2D and 3D incompressible flow were tested. Here, for space reason, only some 2D steady and unsteady flow for a channel and for the driven cavity problem were reported. The comparisons with other numerical solution on the ability to resolve the gross features of the flow have shown in general a good performance of the method.

REFERENCES

[1] **Harlow, F.H. and J.E. Welsh** *Numerical Calculation of Time-Dependent Viscous Incompressible Flow of Fluid with Free Surface, Physics of Fluids*, vol. 8, 1965, pp. 2182-2189.

[2] **Chorin, A.J.** *Numerical Solution of the Navier-Stokes Equations, J. Math. Computation*, vol. 22, 1971, pp. 745-762.

[3] **Yanenko, N.** *The Method of Fractional Steps*, Springer-Verlag, New York, 1971.

[4] **Peyret, R. and T. D. Taylor** *Computational Method for Fluid Flow*, Springer-Verlag, New York, 1983.

[5] **Pasinato, H.D.** *Iterative Method for Solving the Unsteady Incompressible Navier-Stokes Equations*, 2000, preprint.

[6] **Patankar, S.V.** *Numerical Heat Transfer and Fluid Flow*, McGraw-Hill, New York, 1980.

[7] **Pasinato, H.D.** *A High-Order Resolution of the Incompressible Navier-Stokes Equations*, MECOM'99, 1999, Mendoza.

[8] **E Weinan and J.G. Liu** *Essentially Compact Schemes for Unsteady Viscous Incompressible Flows, J. Comp. Physics*, vol 126, 1996, pp. 122-138.