THE 5$^{\text{TH}}$ ASIAN COMPUTATIONAL FLUID DYNAMICS
BUSAN, KOREA, OCTOBER 27 ~ 30, 2003

An Implicit Pressure Correction Method for Incompressible Navier-Stokes Equations on Unstructured Cartesian Grids

Dartzi Pan
Institute of Aeronautics and Astronautics, National Cheng Kung University
Tainan, Taiwan ROC
Email: dpan@mail.ncku.edu.tw

ABSTRACT

An implicit pressure correction method on unstructured Cartesian grid is developed for the incompressible Navier-Stokes equations. An immersed boundary method is also incorporated to treat the body geometry. Tests show that with an appropriate amount of dissipation, the method is second order accurate both in time and space. The driven cavity flows with and without immersed bodies are computed to demonstrate the capability of the present scheme.

Keyword: *Implicit Pressure Correction Method, Unstructured Cartesian Grids, Immersed Boundary Method*

1. Introductions

The numerical method for the incompressible Navier-Stokes equations has been highly developed in recent years. One popular method is the pressure correction method (or fractional step method) [1-5] in which the divergence-free condition is obtained via an appropriate pressure field. In these methods the velocity is updated by a 2-step predictor-corrector time integration. In the predictor step the intermediate velocity field computed from the momentum equations is usually not divergence free. Then in the corrector step a pressure field is applied to correct the non-divergence-free part of the intermediate velocity. This pressure field obeys a Poisson equation. Normally in these methods staggered grids are used to achieve pressure-velocity coupling [1-3]. If a non-staggered grid is used, then a cell-face velocity is defined in addition to the cell-center variables [4,5] to compute the volume flux. The time integration method used for velocity is usually semi-implicit. That is, the predictor step of the time integration usually employs an explicit scheme (Adams-Bashforth or RK3) for the inviscid terms and an implicit scheme (Trapezoidal rule) for the viscous terms. It is well known that the RK3 scheme has a CFL limit of $\sqrt{3}$ and the Adams-Bashforth scheme is weakly unstable for the model convection equation [1]. These stability limitations may become sever when the grid is locally refined or stretched.

Traditionally a body-fitted [1,5] structured or unstructured grid system is employed to handle the immersed bodies in the flow field. More recently the immersed boundary method [2,3,4] on fixed Eulerian grids has gained popularity for their ease in treating the complex geometries. In Fadlun et al. [2] and Kim et al. [3], a body in the flow field is modeled by adding momentum forcing to the appropriate grid cells to simulate the appropriate boundary condition. In Mittal et al. [4], the grid cells cut by the body are treated with special differencing schemes. The usefulness of these methods lies on the appropriateness and simplicity of the procedure to model the true body geometry and its effects on the flow field. In this aspect, the capability of local refinement around body boundaries is desirable.

In this work an implicit pressure correction method is developed to solve the incompressible Navier-Stokes equations. The fully implicit time integration is achieved by the technique of sub-iteration [6]. The unstructured Cartesian grid system is employed to allow local refinement when necessary. The pressure Poisson equation is solved by an implicit multi-grid method [7]. A fourth order artificial dissipation is added to the pressure field to suppress the even-odd decoupling of pressure. An immersed boundary method is also developed to handle the immersed bodies. The no-slip boundary condition is enforced by the method of direct forcing [2]. The driven cavity flows with immersed bodies are computed to demonstrate the capability of the present scheme.

2. Some Results

The analytical solutions of decaying vortices [8] are used for order of accuracy test. Tests show that with an appropriate amount of dissipation the present scheme is second order accurate both in time and space. Also, the published data [9] on driven cavity flows are used for comparison. Figure 1 shows the computed velocity profiles *u(y)* along the vertical centerline and *v(x)* along the horizontal centerline of a driven cavity for *Re=3200.* The comparison with the published data [9] is good. Figure 2 shows the computed particle paths in a driven cavity with a centered circular cylinder, which is modeled by the immersed boundary method. The smoothness of the streamline over the cylinder indicates the effectiveness of present method to model a curved surface on a Cartesian grid.

Acknowledgement

This work is funded by National Science Council under the grant NSC91-2212-E006-104. The support from NSC is highly appreciated.

References

[1] Rai, M.M., "Direct Simulations of Turbulent Flow Using Finite-Difference Schemes", *J. Comput. Phys.* Vol. 96, (1991), pp.15-53.

[2] Fadlun, E.A., Verzicco, R., Orlandi, P. and Mohd-Yusof, J., "Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations", *J. Comput. Phys*, Vol. 161, (2000), pp. 35-60.

[3] Kim, J., Kim, D., and Choi, H., "An Immersed-Boundary Finite-Volume Method for Simulations of Flow in Complex Geometries", *J. Comput. Phys.*, Vol. 171, (2001), pp. 132-150.

[4] Ye, T., Mittal, R., Udaykumar, H.S. and Shyy, W., "An Accurate Cartesian Grid Method for Viscous Incompressible Flows with Complex Immersed Boundaries", *J. Comput. Phys.*, Vol. 156, (1999), pp. 209-240.

[5] Zang, Y., Street, R. L., and Koseff, J. R., "A Non-staggered Grid, Fractional Step Method for Time-Dependent Incompressible Navier-Stokes Equations in Curvilinear Coordinates," *J. Comput. Phys.*, Vol. 114, (1994), pp. 18-33.

[6] Pan, D. and Chang, C.H., "The Capturing of Free Surfaces in Incompressible Multi-Fluid Flows", *International Journal For Numerical Methods In Fluids*, Vol. 33, (2000), pp. 203-222.

[7] Pan, D., "An Unstructured Cartesian Poisson Solver," Proceedings of The 9th National Computational Fluid Dynamics Conference, Aug. 2002, Tainan, Taiwan ROC

[8] Ethier, C. R. and Steinman, D.A., "Exact Fully 3D Navier-Stokes Solutions for Benchmarking", *Int. J. Num. Met. Flu.*, Vol. 19, (1994), pp. 368-375.

[9] Ghia, U., Ghia, K.N. and Shin, C.T., "High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method", *J. Comput. Phys.*, Vol. 48, (1982), pp. 387-411.

[10] Xiao, F., "A Computational Model for Suspended Large Rigig Bodies in 3D Unsteady Viscous Flows," *J. Comput. Phys.*, Vol. 155, (1999), pp. 348-379.

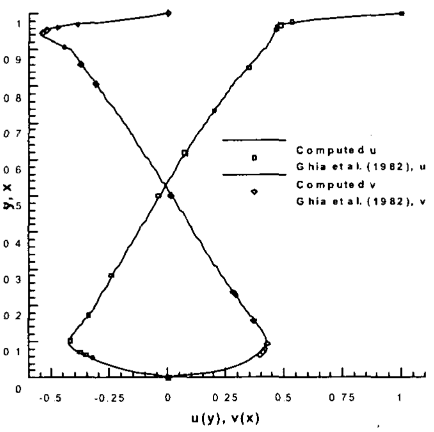

Fig. 1 Driven Cavity, *u(y)* along *x=0.5* and *v(x)* along *y=0.5, Re=3200, cdisp=0.01*

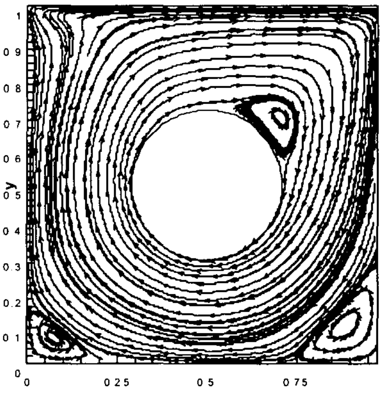

Fig. 2 Particle Paths for driven cavity with centered cylinder, *Re=1000.*