다중격자와 인공점성항을 이용한 3차원 비압축성 흐름에 관한 수치모형 해석

Numerical Simulation of Three Dimensional Incompressible Flows Using the Navier-Stokes Equations with the Artificial Dissipation Terms and a Multigrid Method

박기두*, 이길성**

Kidoo Park, Kil Seong Lee

Abstract

The governing equations in generalized curvilinear coordinates for 3D laminar flow are the Incompressible Navier-Stokes (INS) equations with the artificial dissipative terms. and continuity equation discretized using a second-order accurate, finite volume method on the nonstaggered computational grid. This method adopts a dual or pseudo time-stepping Artificial Compressibility (AC) method integrated in pseudo-time. Multigrid methods are also applied because solving the equations on the coarse grids requires much less computational effort per iteration than on the fine grid. The algorithm yields practically identical velocity profiles and secondary flows that are in excellent overall agreement with an experimental measurement (Humphrey et al., 1977).

Keywords: Incompressible Navier Stokes Equations, Generalized Curvilinear Coordinates, Artificial Compressibility Algorithms, Artificial Dissipative Terms, Multigrid Method

1. Introduction

The past decade has witnessed a great deal of progress in the area of computational fluid dynamics (CFD). Developments in computer technology as well as in advanced numerical algorithms have enabled attempts to be made towards analysis and numerical solution of highly complex flow problems. However, we can investigate some difference between numerical solution and experimental data as to use the application of the numerical solution. Almost numerical errors that appear in the literatures (Ferziger et al., 1994; Figitas et al., 1993) are associated with the discretized convective terms in motion even if turbulence modeling is not included. The numerical method needs to introduce the artificial dissipation terms (Rogers et al., 1990; Rogers et al., 1991) in order to eliminate the numerical errors. Multigrid method applies an iterative scheme in this study since the total amount of computational effort does not significantly increase even though the method converges faster (Tang et al., 2003). This numerical study with the artificial dissipation terms for the convective terms is employed and the multigrid method is also implemented.

* 서울대학교 공과대학 건설환경공학부 박사과정 · E-mail : hydrol88@snu.ac.kr

** 정회원·서울대학교 공과대학 건설환경공학부 교수·E-mail : kilselee@snu.ac.kr

2. Governing Equations

The governing equations in generalized curvilinear coordinates, using dual or pseudo time-stepping Artificial Compressibility (AC) method to couple pressure and velocities, are the three-dimensional, incompressible Navier-Stokes (NS) equations and continuity equation, nondimensionlized by ρ, U_o, and L_o. The governing equations follow

$$\Gamma\frac{\partial Q}{\partial t}+J\frac{\partial}{\partial t}(F^l-F_\nu^l)=0, \tag{1}$$

$$\Gamma=diag(0,1,1,1),$$

$$Q=(p,u_1,u_2,u_3)^T,$$

$$F^l=\frac{1}{J}(U^l,\ u_1U^l+p\xi_{x_1}^l,\ u_2U^l+p\xi_{x_2}^l,\ u_3U^l+p\xi_{x_3}^l),$$

$$F_\nu^l=\frac{1}{JRe}\left(0,\ g^{ml}\frac{\partial u_1}{\partial\xi^m},\ g^{ml}\frac{\partial u_2}{\partial\xi^m},\ g^{ml}\frac{\partial u_3}{\partial\xi^m}\right)^T,$$

where x_l are the Cartesian coordinates x, y, and z. ξ^l are curvilinear coordinates ξ, η, and ζ, respectively, $\xi_{x_m}^l$ are the metrics of the geometric transformation, g^{ml} are the components of the contravariant metric tensor, and J is the Jacobian of the geometric transformation. U^l are the contravariant velocity components, $U^l=u_m\xi_{x_l}^m$, u_m are the Cartesian velocity components, and u, v, and w, and p is the static pressure divided by the density. Finally, Re is the Reynolds number of the flow, which is based on characteristic length and velocity scales and the kinematic viscosity of the fluid.

The governing equations are discretized in strong conservation form using a three-point backward, second-order accurate Euler implicit scheme for the temporal derivative and three-point, second order accurate central differencing for the spatial derivatives,

$$\Gamma\frac{1}{J}\left(\frac{dQ}{dt}\right)_{i,j,k}+\left(\delta_{\xi^l}\tilde{F}^l+\delta_{\xi^l}\tilde{F}_\nu^l\right)_{i,j,k}^{n+1}=0 \tag{2}$$

where

$$\delta_{\xi^1}(\ )_{i,j,k}=\frac{(\ )_{i+1/2,j,k}-(\ )_{i-1/2,j,k}}{\triangle\xi^1}.$$

The flux $\tilde{F}$ is the flux F at the cell interfaces and D is the artificial dissipation flux, especially, the matrix-valued scheme (Lin and Sotiropoulos, 1997),

$$\tilde{F}_{i+1/2,j,k}^1=\frac{1}{2}\left(F_{i,j,k}^1+F_{i+1,j,k}^1\right)+D_{i+1/2,j,k}^1 \tag{3}$$

$$D_{i+1/2,j,k}^1=\epsilon\delta_{\xi^1}\left(|A^1|\delta_{\xi^1}\delta_{\xi^1}\right)Q_{i+1/2,j,k} \tag{4}$$

where ϵ is a constant, $|A^l|$ is the absolute value of the Jacobian matrix $A^l=\partial F^l/\partial Q$.

3. Numerical Methods

The efficiency and robustness of the algorithm can be enhanced by implementing the local dual-time-stepping and multigrid methods are also applied because solving the equations on the coarse grids requires much less computational effort per iteration than on the fine grid.

It is marched in time to advance the solution to the next time step, and couples the pressure and velocity fields at the same level by adopting compressible flow techniques. A pseudo-time derivative term is added to the discrete governing equations until the pseudo-time derivative is reduced to a small residual and the time accurate NS equations are satisfied at the next physical time step. However, the unsteady problem is excluded in this study since we just focus on the multigrid method and the effect of artificial dissipation.

$$Q_{i,j,k}^{n+1,m} = Q_{i,j,k}^{n+1,0} - \alpha_m \Delta\tau \left[\Gamma \frac{3Q_{i,j,k}^{n+1,m-1} - 4Q_{i,j,k}^{n} + Q_{i,j,k}^{n-1}}{2\Delta t} + J\left(\delta_{\xi^l}\tilde{F}^l + \delta_{\xi^l}\tilde{F}_\nu^l\right)_{i,j,k}^{n+1,m-1} \right] \quad (5)$$

where $\Delta\tau$ is the pesudo-time step increment, α_m are the Runge-Kutta coefficients $(=1/m, \text{for } m=1,2,3,4)$. Thus, Q terms at the pesudo-time level is treated as $Q^{n+1,0} = [Q^{n+1}]^\tau$, $Q^{n+1,4} = [Q^{n+1}]^{\tau+\Delta\tau}$.

The local time step is computed

$$\Delta\tau = \frac{CFL}{\max\left(\lambda_{\xi^1}, \lambda_{\xi^2}, \lambda_{\xi^3}\right)} \quad (6)$$

where λ_ξ is the spectral radii of the Jacobian matrices, CFL is the Courant-Friedrich-Lewis number.

As shown in Fig 1, the multigrid cycling strategy to implement the multigrid method is the V-cycle using the Runge-Kutta scheme. The equation is computed on h grid level and the results are injected to $2h$ grid level. The equation is also calculated on $2h$ grid level, and then the procedure is continued to $4h$ grid level until the coarsest grid is reached. The data is prolonged from $4h$ grid level, through $2h$ grid level, to h grid level.

Boundary conditions are specified at the inlet, outlet, and solid wall boundaries. Inflow boundary conditions are implemented by Dirichlet conditions for the fully developed velocity distribution (White, 1991). No slip and no flux boundary conditions are used on the solid walls. Zero gradient boundary conditions at the outflow boundary and on the symmetry plane (at $z=0$) are used (Neumann condition). The pressures at all boundaries are obtained by linear extrapolation from interior nodes.

4. Results and Conclusion

The phenomenon of steady fully developed laminar flow of viscous fluid in curved circular tube is now well understood and the accurate friction factor correlation equations are available in the literatures (Cheng et al., 1976; Taylor et al., 1981; Humphery et al., 1997). The numerical solution is to validate the accuracy of this solver for steady laminar flow through a 90° curved rectangular tube. This test case was experimentally studied by Humphery et al.

(1977) who carried out velocity measurements at Reynolds number Re = 790, based on the bulk velocity and the hydraulic diameter. To evaluate the performance of the multigrid method in a 3D flow of a curved 90 degree square bend, we choose these test cases: The finest meshes are $49 \times 41 \times 21$ grid nodes, the coarser meshes are $49 \times 21 \times 11$ nodes, and finally, $49 \times 11 \times 6$ nodes are selected as the coarsest mesh. The CFL number used in this numerical study is CFL = 3.0.

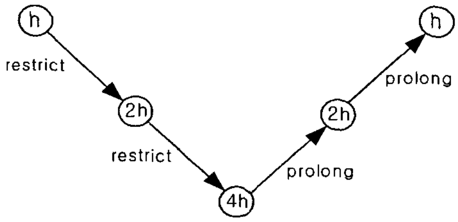

Fig 1. Multigrid V-cycle strategy

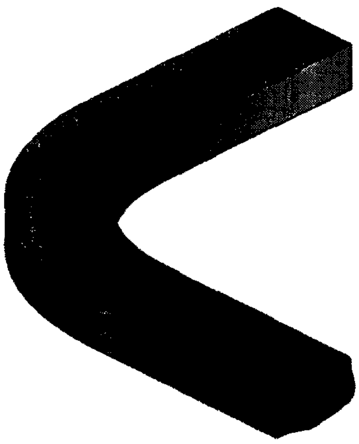

Fig 2. Pressure and velocity distributions

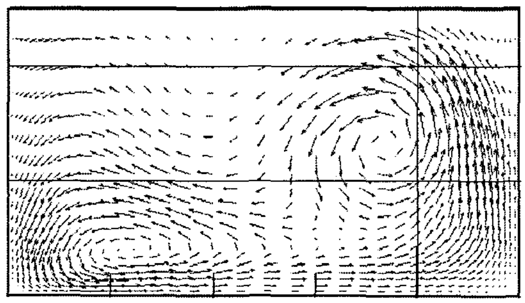

Fig 3. Secondary flow at $\theta = 90^\circ$

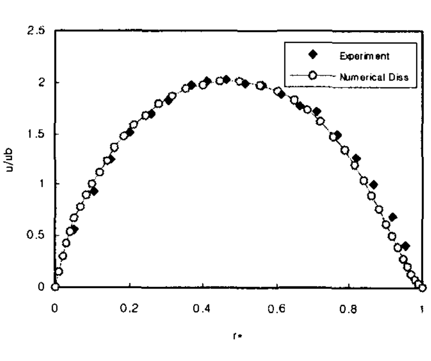

Fig 4. Streamwise velocity profiles along radial lines at $z = 0$ and $\theta = 0^\circ$

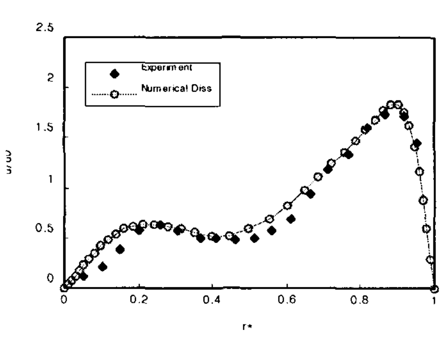

Fig 5. Streamwise velocity profiles along radial lines at $z = 0$ and $\theta = 90^\circ$

As shown in Fig 2 to Fig 5, this algorithm yields practically identical velocity profiles and secondary flows that are in excellent overall agreement with an experimental measurement (Humphrey et al., 1977).

Acknowledgement

The frist author is grateful to Dr. Fotis Sotiropoulos and Dr. Liang Ge at Georgia Institute of Technology for their help conducting the numerical analysis in this study. This research was conducted as a part of the academically promoted frontier research programs of Safe and Sustainable Infrastructure Research and Sustainable Water Resource Research Center (# 1-7-3). Their partial financial supports are greatly acknowledged.

References

Cheng, K. C., Lin, R., and Ou, J. (1976). "Fully developed laminar flow in curved rectangular channels." *ASME J. Fluids Eng.*, pp. 41-48.

Ferziger, J. H. (1994). "Comments on the policy statement on numerical accuracy." *ASME J. of Fluids Eng.*, Vol. 116, p. 396.

Freitas, C. J. (1993). "Editorial policy statement on the control of numerical accuracy." *ASME J. of Fluids Eng.*, Vol. 115, p. 339.

Humphrey, J. A. C., Taylor, A. M. K., and Whitelaw, J. H. (1997). "Laminar flow in a square duct of strong curvature." *Journal of Fluid Mechanics*, Vol. 83, pp. 509-527.

Lin, F. B., and Sotiropoulos, F. (1997). "Assessment of artificial dissipation models for three-dimensional incompressible flow solutions." *ASME J. Fluids Eng.*, 119(2), pp. 331-340.

Rogers, S. E., and Kwak, D. (1990). "Upwind differencing schemes for the time-accurate incompressible Navier-Stokes equations." *AIAA Journal*, Vol. 28, No. 2, pp. 253-262.

Rogers, S. E., Kwak, D., and Kiris, C. (1991). "Steady and unsteady solutions of the incompressible Navier-Stokes equations." *AIAA Journal*, Vol. 29, No. 4, pp. 603-610.

Tang, H. S., Jones, S. C., Sotiropoulos, F. (2003). "An overset-grid method for 3D unsteady incompressible flows." *J. Comp. Phys.* Vol. 191, pp. 567-600.

Taylor, A. M. K. P., Whitelaw, J. H., and Yianneskis, M. (1982). "Curved ducts with strong secondary motion: Velocity measurements of developing laminar and turbulent flow." *AIAA Journal*, Vol. 104, pp. 350-359.

Yeo, R. M., Wood, P. E., and Hrymak, A. N. (1991). "A numerical study of laminar 90-degree bend duct flow with different discretization schemes." *ASME J. Fluids Eng.*, Vol. 113, pp. 563-569.

White, F. M. (1991). *Viscous Flow*, McGraw-Hill, New York.