Pressure-Based Navier-Stokes Solver Using the Multigrid Method

Chae M. Rhie*
Pratt and Whitney Aircraft, United Technologies Corporation, East Hartford, Connecticut

Abstract

A PRESSURE-BASED implicit procedure to solve steady-state Navier-Stokes equations is developed. A multistep pressure correction procedure with an implicit density treatment is used to establish the pressure and velocity fields. A multigrid relaxation scheme is used to solve the scalar matrices resulting from the finite-volume formulation. The algorithm is valid for all Mach number flows ranging from incompressible to supersonic flow regimes.

Contents

Computational methods for solving the Navier-Stokes equations are largely classified into two categories: "density-based methods" and "pressure-based methods."

Many existing methods (see, for example, Ref. 1) solve the continuity equation for the density (as a primary variable) and then specify the pressure (as a secondary variable) using the equation of state. Thus, they are called density-based methods. However, the density-based method has a disadvantage when the Mach number approaches zero. In the incompressible flow limit, there are no density variations, and the pressure cannot be calculated. To remedy this situation, the artificial compressibility method, originally proposed by Chorin, has been used for incompressible flows (see, for example, Ref. 3). A fictitious density term is added to the time derivative term in the continuity equation. A time-marching method then is used to calculate the pressure through a fictitious equation of state. The solution is valid when the steady state is reached. However, a problem arises when both compressible and incompressible regimes are encountered in the same computational domain.

A pressure-based method, where the pressure is used as a primary variable, has no restrictions associated with specific flow regimes (see, for example, Ref. 4). The present pressure-based method is an extension of the procedure of Rhie and Chow,[5] originally developed to solve the incompressible Navier-Stokes equations in general curvilinear coordinates. The procedure of Rhie and Chow was a generalization of the Semi-Implicit Pressure Linked Equation (SIMPLE) procedure[6] to include nonstaggered grids. In the present work, this method is extended to solve compressible flows including transonic and supersonic flow regimes. The density term is implicitly introduced in the multistep pressure correction procedure. The incompressible flow then becomes an asymptotic case of this unified solution procedure.

In the solution procedure, each momentum equation is linearized and solved individually by relaxing scalar matrices. The continuity and momentum equations are then coupled through the pressure correction equations that involve a multistep correction procedure. The concept of multistep correction was developed by Issa.[7] The present multistep correction procedure solves compressible flows using general curvilinear coordinates in which a nonstaggered grid arrangement is used as opposed to the staggered grid. All of the dependent variables are defined at the same grid location. The fully staggered grid method introduces major disadvantages when the general curvilinear coordinate system is adopted (see Ref. 5). If a fully staggered grid arrangement is used with a numerical transformation, the solution accuracy will be dependent on the grid orientation relative to the Cartesian reference frame. Best accuracy is obtained when the Cartesian velocity components defined at cell faces coincide with the contravariant velocity components that describe normal fluxes. Otherwise, interpolations to calculate the normal fluxes degrade solution accuracy.

For the solution of compressible flows, the basic idea is to update the preliminary mass flux, $(\rho u)^*$ obtained from the momentum equations by a perturbation relation $\rho u = (\rho^* + \rho')(u^* + u')$. The superscript prime represents an increment, and ρu satisfies the continuity equation. The major task is to express ρ' and u' in terms of a single variable p' and to derive a manageable linear system. The formal solution procedure is as follows.

1) Predictor step: The conservation form of the momentum equation is integrated over the control volume using finite-volume approximations. The mass fluxes then are established from the momentum equations using the preliminary pressure field p^* as

$$\rho^* G_i^* = \rho^* \Big[-B_i \Delta_i p^* - C_i \Delta_j p^* - D_i \Delta_k p^* + F_{i,\ell}\Big(\sum_m \mathrm{A}_m \mathrm{u}^*_{\ell,m}\Big) + S^*_\ell \Big] \tag{1}$$

where ρ is the density, and G_i is the linearized contravariant, velocity components in the i coordinate direction. The coefficients A, B, C, D, and F involve the density, convection, diffusion, area, and metric coefficients, etc. The second and third terms on the right-hand side are the pressure gradient terms from the nonorthogonality of the grid, and S is a source term.

2) Corrector step: A new pressure field is established to enforce the local continuity through the corresponding velocity and pressure field. Introducing the density and pressure relationship of

$$\rho' = \beta \frac{p'}{RT^*} \tag{2}$$

where T^* is the frozen static temperature and β the constant, corrections are done in an incremental manner as follows.

First correction:

$$\Delta_i \frac{\beta G_i^*}{RT^*}(p^{**} - p^*) + \Delta_i(\rho^* B_i \Delta_i)(p^{**} - p^*) = \Delta_i \rho^* G_i^* + \mathrm{DISS} \tag{3a}$$

where DISS is the explicit fourth-order dissipation term explained in the original paper with stability consideration. The superscript asterisk denotes the newly updated values.

Presented as Paper 86-0207 at the AIAA 24th Aerospace Sciences Meeting, Reno, NV, Jan. 6–9, 1986; received Feb. 20, 1986; revision received Nov. 6, 1987. Full paper available at AIAA Library, 555 W. 57th St., New York, NY 10019. Price: microfiche, $4.00; hard copy, $9.00. **Remittance must accompany order.**

*Staff Scientist.

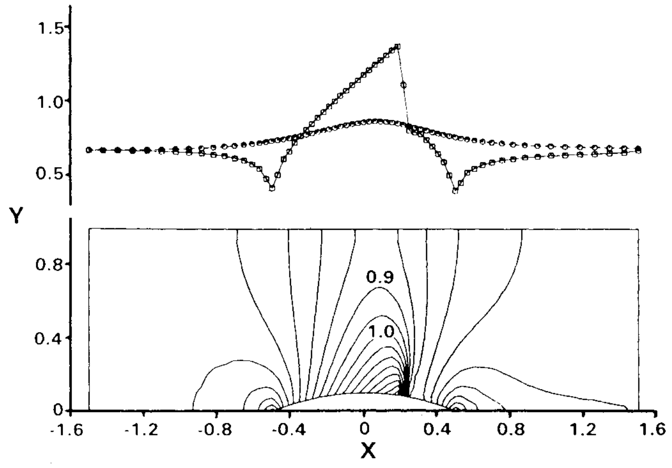

Fig. 1 Isomach lines ($\Delta M_\infty = 0.05$) for a transonic inviscid flow over a bump.

Second correction:

$$\Delta_i \frac{\beta G_i^*}{RT^*}(p^{***} - p^{**}) + \Delta_i(\rho^* B_i \Delta_i)(p^{***} - p^*)$$

$$= \Delta_i\,[-\rho^* C_i \Delta_j (p^{**} - p^*) - \rho^* D_i \Delta_k (p^{**} - p^*)] + R \quad (3b)$$

where R is the residual collection from the first correction.

Third correction:

$$\Delta_i \frac{\beta G_i^*}{RT^*}(p^{****} - p^{***}) + \Delta_i(\rho^* B_i \Delta_i)(p^{****} - p^{***})$$

$$= \Delta_i\,[\rho^* F_{i,\ell m} \textstyle\sum_m A_m (u^{***}_{\ell,m} - u^{*}_{\ell,m})] \quad (3c)$$

The significance of the present pressure correction equations, Eqs. (3a-3c), is their convective nature, which represents the transport characteristic of the density terms. In the incompressible flow limit, these terms vanish, and the correction equations become pure diffusive equations.

Multigrid Method

The present multigrid method is correction scheme according to Brandt.[8] The present procedure solves only scalar matrices in a sequential manner. The multigrid method accelerates the relaxation solution of these scalar matrices. A fixed stategy is employed in the present method. Each cycle begins on the fine grid. The alternating direction iteration (ADI) is performed once on each grid until the coarsest grid is reached. Then it is repeated going back up to the second finest grid. The multigrid method accelerates the global convergence rate. Numerical experiments for basic test cases are discussed in the original paper.

Results

Inviscid flows over a circular arc bump are computed using 65×17 grids for incompressible as well as compressible flows in the original paper. However, the real challenge of the present method is the calculation of compressible flows. The transonic flow over a 10%-thick bump is calculated and plotted in Fig. 1. In the supercritical transonic case ($M_\infty = 0.675$), the supersonic region is terminated by the shock captured over three grid points. The accurate shock capturing is achieved using a second-order-accurate central differencing. The details of the discretization can be found in the original paper.

The computer code was applied to the three-dimensional turbine cascade tested by Langston et al.[9] A total of $24 \times 20 \times 52$ grid points are used. The air inlet angle is set at 44.7 deg. The upstream inlet Mach number is 0.1, and the Reynolds number based on a blade axial chord is 5.9×10^5. The boundary layer at the end-wall is an equilibrium turbulent boundary layer with a thickness of 0.117 axial chord lengths. Wall functions are used.

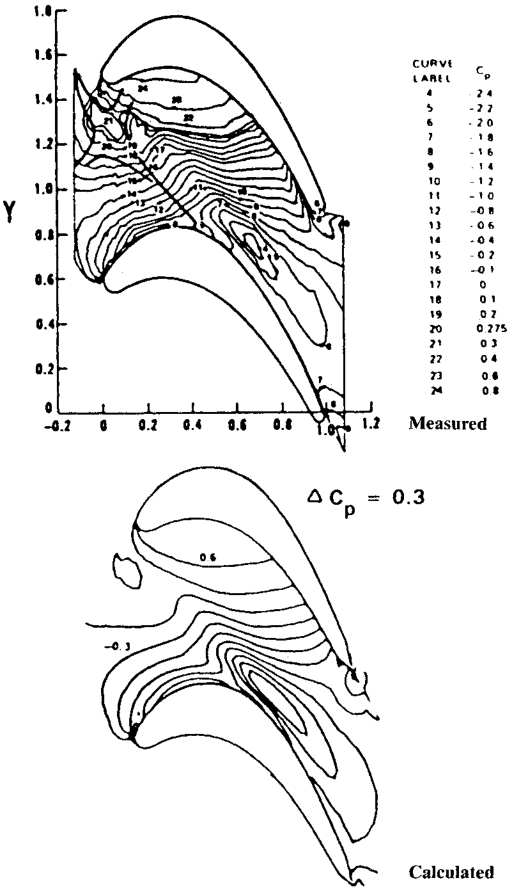

Fig. 2 Static pressure distribution on the end wall.

The end-wall static pressure distribution is plotted in Fig. 2. The wiggles in the contours, both for the present prediction and the measurements, indicate the passage vortex rollup on the end wall.

These results demonstrate that the present unified solution procedure is valid for all Mach number flow regimes. Accurate solutions can be achieved within 200 iteration cycles with the multigrid method.

References

[1]Pulliam, T. H. and Steger, J. L., "Implicit Finite-Difference Simulations of Three-Dimensional Compressible Flow," *AIAA Journal,* Vol. 18, Feb. 1980, pp. 159–167.

[2]Chorin, A. J., "A Numerical Method for Solving Incompressible Viscous Flow Problems," *Journal of Computational Physics,* Vol. 2, 1967, pp. 12-26.

[3]Chang, J. L. C., Kwak, D., Dao, S. C., and Rosen, R., "A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Main Engine, Part I – Laminar Flow," AIAA Paper 85-0175, Jan. 1985.

[4]Hirt, C. W., Amsden, A. A., and Cook, J. L., "An Arbitrary Lagrangian-Eulerian Computing Method for all Flow Speeds," *Journal of Computational Physics,* Vol. 14, 1974, p. 227.

[5]Rhie, C. M. and Chow, W. L., "Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation," *AIAA Journal,* Vol. 21, Nov. 1983, pp. 1525-1532.

[6]Caretto, L. S., Gosman, A. D., Patankar, S. V., and Spalding, D. B., "Two Calculation Procedures for Steady Three-Dimensional Flows with Recirculation," *Proceedings of the Third International Conference on Numerical Methods in Fluid Dynamics,* Paris, 1972, p. 60.

[7]Issa, R. I., "Solution of Implicitly Discretized Fluid Flow Equations by Operator-Splitting," Imperial College, Dept. Min. Resources Engrg., London, Internal Rept., 1982.

[8]Brandt, A., "Multi-Level Adaptive Solution to Boundary Value Problems," *Mathematics of Computation,* Vol. 31, No. 138, April 1977, pp. 333-390.

[9]Langston, L. S., Nice, M. L., and Hooper, R. M., "Three-Dimensional Flow Within a Turbine Cascade Passage," *ASME Journal of Engineering for Power,* Vol. 99, No. 1, 1977, pp. 21-28.