[3]Tuncer, I. H., Ekaterinaris, J. A., and Platzer, M. F., "A Viscous-Inviscid Interaction Method for 2-D Unsteady, Compressible Flows," AIAA Paper 93-3019, June 1993.
[4]Tuncer, I. H., Ekaterinaris, J. A., and Platzer, M. F., "A Novel Viscous-Inviscid Interaction Method with Inviscid Wake Modeling," AIAA Paper 94-0178, Jan. 1994.
[5]Ekaterinaris, J. A., Cricelli, A., and Platzer, M. F., "A Zonal Method for Unsteady, Viscous, Compressible Airfoil Flows," *Journal of Fluids and Structures*, Vol. 8, Jan. 1994, pp. 107–123.
[6]McAlister, K. W., Pucci, S. L., McCroskey, W. J., and Carr, L. W., "An Experimental Study of Dynamic Stall on Advanced Airfoil Sections, Pressure and Force Data," Vol. 2, NASA-TM-84245, Sept. 1982.

Numerical Simulation by Cubic-Polynomial Interpolation for Unsteady, Incompressible, Viscous Flow

Hiromi Sugiyama*
Japan Automobile Research Institute, Ibaraki 305, Japan

Introduction

DURING the past three decades there has been significant progress in the numerical analysis of unsteady, incompressible flow problems. Despite the advances in algorithms and computer hardware, however, time-accurate solutions of the incompressible Navier-Stokes (N-S) equations remain a computationally intensive problem, even in two dimensions. Time-accurate solutions of the incompressible flow problems using primitive variables are particularly time consuming because of the elliptical nature of the governing equations. The existing unsteady flow solvers require excessive CPU capability; therefore, there is continuing interest in finding more efficient methods for obtaining numerical solutions to the N-S equations.

Recently, new ideas have been introduced for unsteady flow solvers. One scheme of interest is the cubic-interpolated pseudoparticle-combined unified procedure (CIP-CUP) method, originally proposed by Yabe and Wang[1] and Yabe.[2] The CIP-CUP method is an explicit time-marching scheme with a fractional-step-like approach developed to handle complex fluid flow problems, covering both compressible and incompressible flow. In the CIP-CUP formulation, the solution procedure is divided into two phases: nonadvection and advection. In the nonadvection phase, an intermediate pressure field is solved first, and then intermediate velocity and density fields are obtained from this pressure field. The pressure field is computed by solving a diffusion equation for pressure, which is obtained via manipulation of the momentum and pressure (or energy) equations. In the advection phase, the governing equations are solved for the velocity, density, and pressure at the next time step using the CIP scheme. The CIP scheme is a new class of upwind schemes using cubic polynomial interpolation and is highly accurate for solving generalized hyperbolic equations such as advection-diffusion equations and the Korteweg-de Vries (KdV) equation.[3,4]

In this Note, we present an explicit method, the CIP-artificial compressibility extension (-ACE) method for solving the incompressible N-S equations in a time-accurate and efficient manner. This method uses an artificial compressibility approach[5] to transform the governing equations into a hyperbolic system and uses CIP-CUP methodology to first decouple the pressure and the velocity fields and then to integrate them with respect to time. Although this method is applicable to three-dimensional flow problems, we confine our discussion to one- and two-dimensional flow problems.

Received April 19, 1994; revision received July 19, 1994; accepted for publication July 22, 1994.

*Research Engineer, Research and Survey Division.

Governing Equations

The incompressible N-S equations for isothermal and constant-density fluids are modified by introducing an artificial compressibility approach to the following system of equations, written in tensor notation and dimensionless form:

$$\frac{\partial p}{\partial \tau} = -\beta \frac{\partial u_i}{\partial x_i} \tag{1a}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{1b}$$

where t is time, $x_i = (x, y)$ are the Cartesian coordinates, $u_i = (u, v)$ are the corresponding velocity components, p is the pressure, τ_{ij} is the viscous stress tensor, and τ and β denote the pseudo-time and artificial compressibility parameter, respectively, which are introduced to apply the artificial compressibility formulation. The viscous stress tensor is defined as

$$\tau_{ij} = \frac{1}{Re}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

where Re is the Reynolds number. In this formulation, an incompressible flowfield (i.e., a divergence-free flowfield) is only obtained when the pressure field reaches steady state in pseudotime.

Numerical Formulation and Methodology

The CIP-ACE method presented in this Note was developed from the CIP-CUP method.[1,2] As in the CIP-CUP method, Eqs. (1a) and (1b) are divided into a nonadvection phase [Eq. (1a) and $\partial f/\partial t = G$] and an advection phase ($\partial f/\partial t + \boldsymbol{u} \cdot \nabla f = 0$), where f denotes $\boldsymbol{u} = (u, v)$ and G represents the right-hand side of Eq. (1b). The nonadvection and advection phases are then solved separately, as described in the following sections.

Nonadvection Phase

By using a fractional-step-like approach and the Euler implicit time-differencing formula for the time derivative, Eqs. (1a) and (1b) for the nonadvection phase can be written in the following vector form:

$$\frac{\partial p^{**}}{\partial \tau} = -\beta \nabla \cdot \boldsymbol{u}^{**} \tag{2}$$

$$\frac{\boldsymbol{u}^{**} - \boldsymbol{u}^{n}}{\Delta t} = -\nabla p^{**} \tag{3}$$

$$\frac{\boldsymbol{u}^{*} - \boldsymbol{u}^{**}}{\Delta t} = \frac{1}{Re}\nabla^2 \boldsymbol{u}^{n} \tag{4}$$

where the superscript n denotes the quantities at time $t = n\Delta t$, and the asterisk and double asterisks denote the first and second intermediate quantities, respectively. The values with the double asterisks are implicit values introduced by the application of the implicit time differencing.

Here, we briefly describe the essence of the algorithm for solving the system of Eqs. (2–4). First, the velocity and pressure fields are decoupled. Substituting the divergence of Eq. (3) into Eq. (2) gives the following equation:

$$\frac{\partial p^{**}}{\partial \tau} = \beta \Delta t \nabla^2 p^{**} - \beta \nabla \cdot \boldsymbol{u}^{n} \tag{5}$$

Note that this equation is a diffusion equation in pseudotime and can be easily solved by using one of the well-established algorithms (e.g., Golub and Van Loan[6]).

Next, to determine the pressure field that will force the velocity $\boldsymbol{u}^{**}$ to be divergence free, Eq. (5) is solved. When Eq. (5) is iterated in pseudotime until $\partial p^{**}/\partial \tau = 0$, Eqs. (2) and (3) are satisfied, and the divergence of the velocity is zero. Note that at steady state, Eq. (5) is very similar to but is numerically more robust than the market and cell(MAC) method.[7]

Finally, combining Eqs. (3) and (4) yields

$$\frac{\boldsymbol{u}^* - \boldsymbol{u}^n}{\Delta t} = -\nabla p^{**} + \frac{1}{Re}\nabla^2 \boldsymbol{u}^n \tag{6}$$

so that the velocity $\boldsymbol{u}^*$ is obtained without explicitly calculating $\boldsymbol{u}^{**}$, by using the steady-state solution p^{**} in pseudotime.

Advection Phase

For Eq. (1b) in the advection phase, the linearized representation obtained by evaluating $\boldsymbol{u}$ at the n time step can be written in the following vector form:

$$\frac{\partial \boldsymbol{u}}{\partial t} + \boldsymbol{u}^n \cdot \nabla \boldsymbol{u} = 0, \qquad (t^n \le t \le t^{n+1}) \tag{7}$$

where the initial value of $\boldsymbol{u}$ is the value of $\boldsymbol{u}^*$ obtained in the advection phase.

The velocity field at the new time step $\boldsymbol{u}^{n+1}$ is computed by solving the advection phase equation [Eq. (7)]. The time-integration procedure uses the CIP scheme.

We now briefly outline the CIP scheme (described in detail in Refs. 3 and 4). To illustrate the scheme, consider the following one-dimensional model equation:

$$\frac{\partial f}{\partial t} + u\frac{\partial f}{\partial x} = 0 \tag{8}$$

If u is constant, then Eq. (8) permits analytical solutions, such as $f(x,t) = f(x - ut, 0)$. Although u is a function of x and t, for sufficiently small Δt the solution is locally approximated by

$$f(x_i, t + \Delta t) = f(x_i - u\Delta t, t) \tag{9}$$

where x_i is a grid point. The numerical solution for Eq. (8) cannot be directly obtained using Eq. (9) because the value of f is given only at grid points $x_i (i = 1, 2, 3, \ldots)$ and the intergrid values corresponding to $x_i - u\Delta t$ are unknown. The CIP scheme uses the following cubic-polynomial interpolation and evaluates the intergrid profile of f:

$$F_i(x) = \{(a_i X + b_i)X + \partial_x f_i\}X + f_i \tag{10}$$

where $X = x - x_i$, and f_i and $\partial_x f_i$ are the values of f and $\partial f/\partial x$ at the grid point x_i, respectively. The values of parameters a_i and b_i depend on the known values f_i and $\partial_x f_i$. The value of $\partial_x f_i$ is obtained by solving another equation, such as $\partial(\partial_x f)/\partial t + u\partial(\partial_x f)/\partial x = -\partial_x f(\partial u/\partial x)$, which is derived from Eq. (8).

In this manner the CIP scheme is characterized by evaluating the intergrid profile of a variable on the basis of cubic-polynomial interpolation. Equation (10) also suggests that the CIP scheme is nearly third-order accurate in space.

Numerical Results

Two sample incompressible flow problems with known exact solutions were chosen to verify our CIP-ACE method. One problem is flow through a one-dimensional channel with a constant total pressure and an oscillating back pressure (a first-order solution to this problem can be found in Ref. 5). The other sample problem is flow that starts impulsively through a two-dimensional channel, also known as transient two-dimensional Poiseuille flow (the exact solution can be found in Ref. 8). For each sample problem, numerical tests were done to determine which values of β and $\Delta\tau$ result in the best pseudotime convergence. The pseudotime convergence rate was not very sensitive to the values of β and $\Delta\tau$, although it increased with larger β or $\Delta\tau$.

All calculations were done on a SPARC classic workstation. For the one-dimensional problem, the calculation took an average of 10^{-4} s of CPU time per grid point per time step, and for the two-dimensional, about 4×10^{-3} s.

One-Dimensional Channel Flow

For a channel of unit length, with a total pressure $p_0 + \frac{1}{2}$ in the channel, the static pressure at the outlet is

$$p_{\text{exit}} = p_0 + p_e \sin \omega t, \qquad (|p_e| < 1)$$

The governing equations for this problem are the same as the two-dimensional equations [Eqs. (1a) and (1b)], except that there is no gradient in the y direction, and the velocity is a function of time only. The following series solution therefore exists for $\boldsymbol{u}$:

$$u(t) = 1 + p_e u_1(t) + p_e^2 u_2(t) + \cdots \tag{11}$$

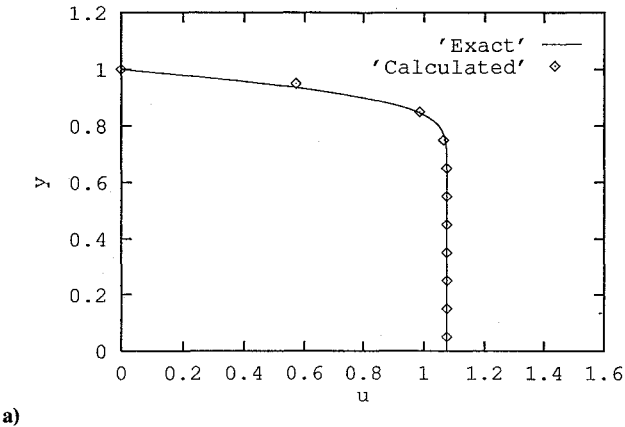
The calculation was carried out with a grid spacing of $\Delta x = 1/20$, a time step of $\Delta t = \pi/(30\omega)$, an artificial compressibility constant of $\beta = 200$, and a pseudotime step of $\Delta\tau = 100$. For an amplitude of $p_e = 1/10$ and a frequency of $\omega = 10$, at each time step two subiterations in pseudotime were required for the pressure calculation. At each time step, the maximum divergence of velocity was less than 10^{-9} and the maximum error of the velocity was less than 10^{-4}.

Transient Two-Dimensional Poiseuille Flow

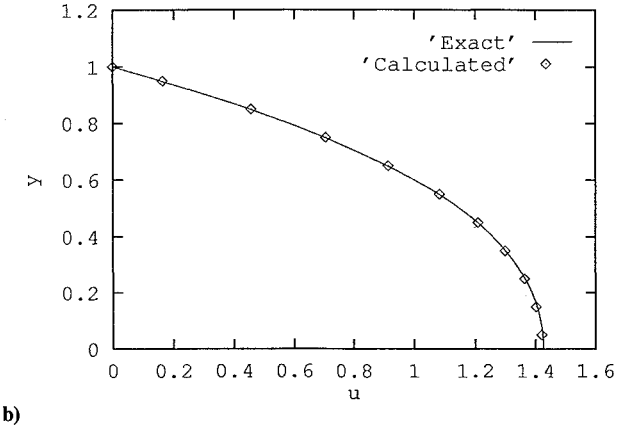
To validate our method for two-dimensional flow, we also solved for the flow through a straight channel with a characteristic length equal to half of the channel width and having a velocity profile given by

$$u(x, y, t) = 0 \qquad (t < 0) \tag{12a}$$

$$u(x, y, t) = \tfrac{3}{2}(1 - y^2) + \sum_{n=1}^{\infty} \frac{2}{\beta_n^2}\left(\frac{\beta_n \cos \beta_n y}{\sin \beta_n} - 1\right) \times \exp\left(-\frac{\beta_n^2 t}{Re}\right) \qquad (t \ge 0) \tag{12b}$$



a)



b)

Fig. 1 Calculated (diamonds) and exact (solid line) velocity profile at the channel outlet for two time points and a Reynolds number of 100: a) $t = 0.4$ and b) $t = 10.0$.

where x is the coordinate in the streamwise direction, y is the coordinate in the direction normal to the channel wall, $y = 0$ designates the centerline of the channel, β_n values are the positive roots of $\beta = \tan\beta (0 < \beta_1 < \beta_2 < \cdots)$, and Re is based on the characteristic length and the average axial velocity. This velocity field represents a flow that starts impulsively at $t = 0$, and at steady state coincides with two-dimensional Poiseuille flow.

There is no inherent upper limit to the Reynolds number that can be computed with the CIP-ACE method. We examined a representative case, namely, $Re = 100$. The calculation was carried out for a rectangular region bounded by $0 \leq x \leq 6$ and $0 \leq y \leq 1$, which was discretized into a 61×11 uniform grid. The calculation used a time step of $\Delta t = 0.01$, an artificial compressibility constant of $\beta = 2000$, and a pseudotime step of $\Delta\tau = 100$.

At each time step, Eq. (5) was solved until a steady-state solution was achieved in pseudotime. During the initial transient stage (near $t = 0$), five to six pressure subiterations were required for calculating the pressure at each time step, but decreased with time to three subiterations. The maximum divergence of velocity was less than 10^{-6} at each time step. Figure 1 shows a comparison between the calculated and the exact solution. The axial velocity profiles are shown at the channel outlet for two different time points. The calculated results are fairly close to the exact solutions except for those near the channel wall and for the initial transient stage ($t = 0.4$, for example). These differences, however, diminish with time. Moreover, agreement can be improved by using a finer grid.

Conclusions

We presented a procedure for computing time-accurate solutions to the incompressible N-S equations. The procedure, which we designated the CIP-ACE method, uses 1) an artificial compressibility approach to transform the equations into a hyperbolic system, 2) a fractional-step-like approach, and 3) cubic-polynomial interpolation to upwind difference the N-S equations. A formal procedure based on a fractional-step-like approach is used to split the momentum equation into a nonadvection equation and an advection equation. The nonadvection equation is solved using a steady-state solution of the diffusion equation for pressure in pseudotime, whereas cubic-polynomial interpolation is used to solve the advection equation. To validate this CIP-ACE method, we solved two sample incompressible flow problems with known exact solutions. For each problem, the method showed rapid convergence to a numerical solution that is reasonably compatible with the exact solution. The CIP-ACE method can be extended in a straightforward manner to solve the three-dimensional, incompressible N-S equations in generalized coordinates, making it an attractive alternative to more established numerical methods.

References

[1]Yabe, T., and Wang, P. Y., "Unified Numerical Procedure for Compressible and Incompressible Fluid," *Journal of the Physical Society of Japan*, Vol. 60, No. 7, 1991, pp. 2105–2108.

[2]Yabe, T., "A Universal Cubic Interpolation Solver for Compressible and Incompressible Fluids," *Shock Waves*, Vol. 1, No. 2, 1991, pp. 187–195.

[3]Yabe, T., and Aoki, T., "A Universal Solver for Hyperbolic Equations by Cubic-Polynomial Interpolation I. One-Dimensional Solver," *Computer Physics Communications*, Vol. 66, No. 3, 1991, pp. 219–232.

[4]Yabe, T., Ishikawa, T., and Wang, P. Y., "A Universal Solver for Hyperbolic Equations by Cubic-Polynomial Interpolation II. Two- and Three-Dimensional Solvers," *Computer Physics Communications*, Vol. 66, No. 3, 1991, pp. 233–242.

[5]Rogers, S. E., and Kwak, D., "Upwind Differencing Scheme for the Time-Accurate Incompressible Navier-Stokes Equations," *AIAA Journal*, Vol. 28, No. 2, 1990, pp. 253–262.

[6]Golub, G. H., and Van Loan, C. F., *Matrix Computations*, 2nd ed., Johns Hopkins Univ. Press, Baltimore, MD, 1990.

[7]Harlow, F. H., and Welch, J. E., "Numerical Calculation of Time-Dependent Viscous Incompressible Flow with Free Surface," *Physics of Fluids*, Vol. 8, No. 12, 1965, pp. 2182–2189.

[8]Takemitsu, N., "Finite Difference Method to Solve Incompressible Fluid Flow," *Journal of Computational Physics*, Vol. 61, No. 3, 1985, 499–518.

Assessment of Pressure-Strain Models in Predicting Compressible, Turbulent Ramp Flows

R. Abid*
High Technology Corporation, Hampton, Virginia 23666
Thomas B. Gatski†
NASA Langley Research Center, Hampton, Virginia 23681
and
Joseph H. Morrison‡
Analytical Services & Materials, Inc., Hampton, Virginia 23666

Introduction

THE purpose of this Note is to evaluate the performance of three pressure-strain correlation models (linear, quadratic, and cubic in the Reynolds stress anisotropy tensor) in the prediction of a compression ramp flow of 16 deg. Although compression ramps represent only one class of nonequilibrium flow, they have important practical application in the design of high-speed vehicles and propulsion systems. Variations in the ramp angle can lead to flows that range from an attached two-dimensional flow to a fully separated, three-dimensional unsteady flow. Clearly, the latter limit is outside the scope of the present study; however, for a ramp angle of 16 deg only incipient separation is observed experimentally at the ramp and flat-plate juncture, and experimental results[1] show that the flow is two dimensional. Thus, the flow turbulence along the ramp is initially induced by the shock-boundary-layer interaction at the flat plate and ramp juncture, and the flow can serve as a test of pressure-strain correlation models to predict an important type of compressible flowfield.

In any closure validation study, the first step is clearly the isolation of each particular correlation closure; in the present study, the focus will be on the pressure-strain correlation. In the solution of compressible flowfields, variable-density extensions to the incompressible forms of these pressure-strain models have been generally used. Thus, a variety of incompressible pressure-strain models can be tested to assess their applicability to compressible flow predictions.

The present study utilizes wall functions so that the effects of the pressure-strain models can be isolated. Care must be exercised in the implementation of the wall functions to insure a smooth functional behavior of the dependent variables throughout the inner part of the boundary layer. Equilibrium conditions consistent with the existence of a logarithmic region are assumed; these must also be consistent with the compressible models introduced for the various correlations. Unfortunately, not all turbulence models satisfy the Van Driest law of the wall when applied to compressible flows because they yield values for the von Kármán constant κ that differ from the accepted value of 0.41. In this regard, Wilcox[2] and Huang et al.[3] have examined the effect on the value of κ for both the K-ω and K-ε models with the compressibility corrections of Zeman[4] and Sarkar et al.[5] and found that these models cause a deviation of the log-law slope. For this reason, these explicit compressibility corrections for the dissipation rate and pressure dilatation are omitted, and a single-layer wall-function formulation is used in conjunction with variable-density extensions of the incompressible pressure-strain correlations.

Received June 17, 1993; revision received Oct. 5, 1994; accepted for publication Oct. 11, 1994.

*Senior Scientist, 28 Research Drive. Member AIAA.
†Senior Research Scientist, MS 128.
‡Senior Research Scientist. Member AIAA.