Numerical Solution of Navier-Stokes Equations for Two-Dimensional Viscous Compressible Flows

Sunil Kumar Chakrabartty*
National Aeronautical Laboratory, Bangalore, India

Abstract

A NODAL-POINT, finite-volume space discretization of viscous fluxes in compressible Navier-Stokes equations is presented. To advance the solution in time, an explicit five-stage Runge-Kutta scheme has been used. To accelerate the rate of convergence to steady state, local time stepping, residual averaging, and enthalpy damping have been employed. The scheme has been evaluated by solving laminar flow over a semi-infinite flat plate and an NACA 0012 air foil using thin-layer approximation. It has been observed here that fourth-order artificial dissipation is sufficient for numerical stability. The results have been compared with available theoretical and numerical solutions.

Contents

According to the idea of the present nodal-point scheme, the flow quantities are ascribed at the vertices of the computational cells. It gives at least first-order accurate derivatives even for stretched and skewed grids. Another advantage of this scheme is that the pressure can be calculated directly on the boundary.

In the finite-volume formulation, the generation of body-fitted grids using curvilinear coordinates and the solution process are separated because no global transformation is used. The governing equations considered here are two dimensional, unsteady, Navier-Stokes equations with following boundary conditions.[1] At the body surface, the velocity components are zero (no slip), and the wall temperature is either prescribed or the normal derivative of it is zero (adiabatic wall in the present case). A periodicity condition has been applied along the cut boundary in a C-type mesh. For the flat-plate case, a symmetry condition is necessary in front of the flat plate on the line of the symmetry. At the downstream boundary of the flat plate computation, either all the unknown variables are extrapolated from the interior or pressure is prescribed with other variables extrapolated. The far-field boundary conditions are based on Rieman invariants for one-dimensional flow normal to the boundary.[1] For circulatory flows, freestream variables appearing in the Riemann invariants have been modified by considering the effect of a single vortex situated on the airfoil in a compressible medium.[2] This reduces the extent of the far-field boundary and hence the computational domain. Computation of viscous fluxes in the present scheme (Fig. 1) can be summarized as follows:

1) Find the first derivatives of all the flow variables at $(i+\frac{1}{2},j)$ for all (i,j) using the Green's theorem and calculate the stress tensors at the same points.

2) Find the difference in the fluxes across two surfaces AB and HG to get only the viscous fluxes over the cell HGBA. Note that the streamwise-like differences are neglected (thin-layer assumption) at this stage.

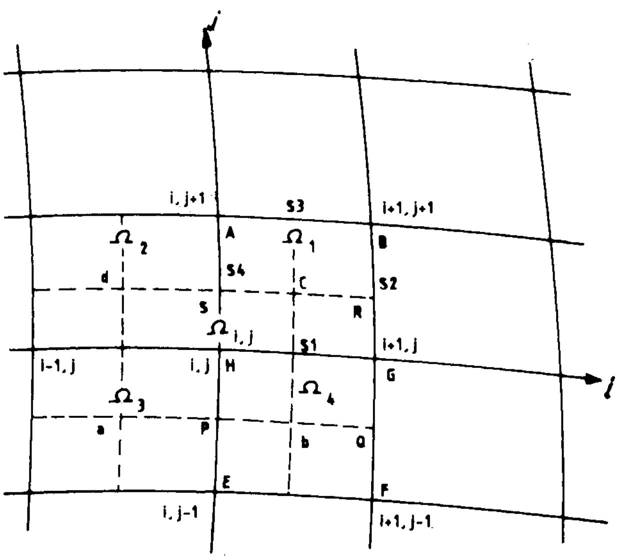
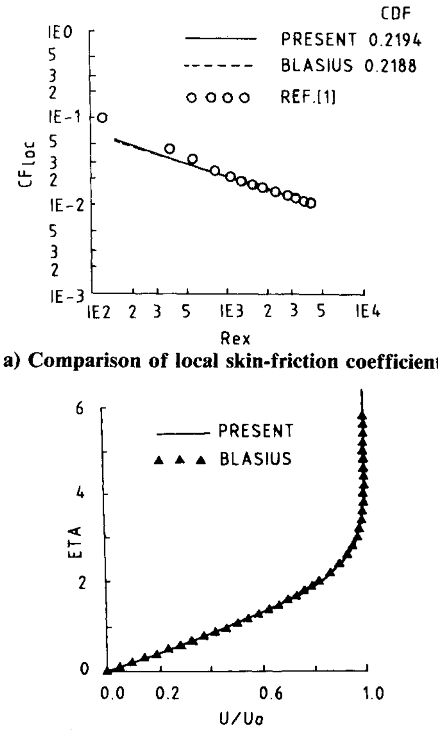


Fig. 1 Finite-volume mesh for the nodal-point scheme.



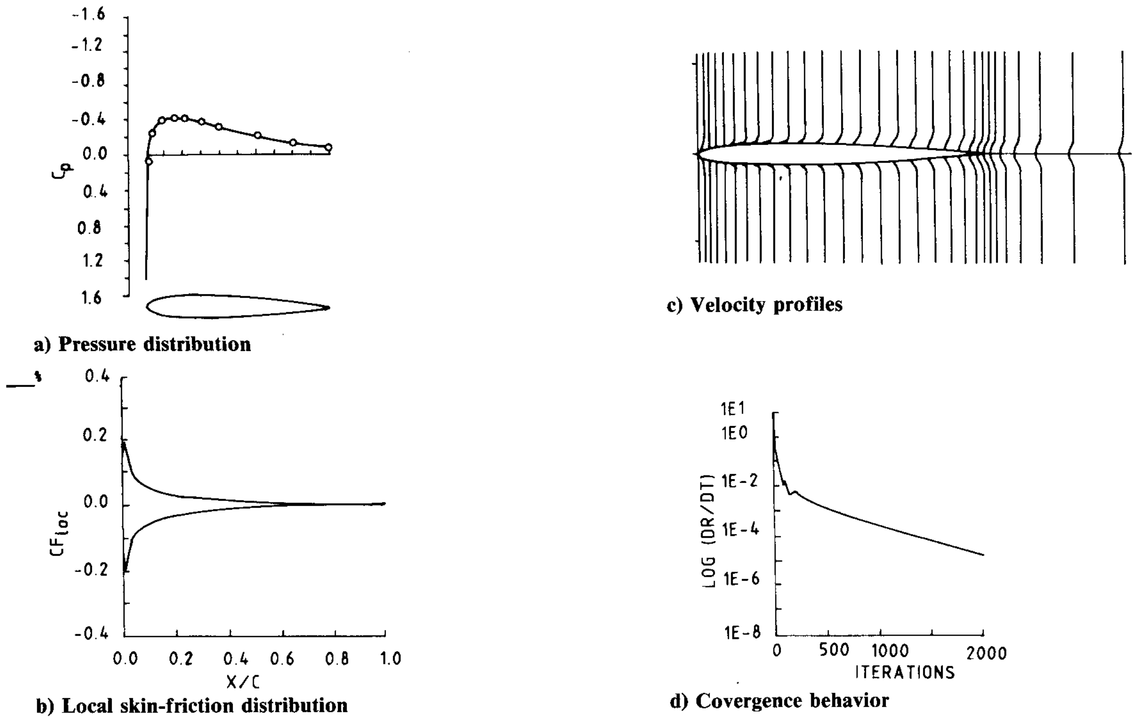
a) Comparison of local skin-friction coefficient

b) Comparison of velocity profiles

Fig. 2 Viscous flow over a flat plate at $M_\infty = 0.50$, $\alpha = 0$, $Re = 5000$.

Received March 2, 1988; revision received July 11, 1988.

*Scientist, Fluid Mechanics Division.



a) Pressure distribution

b) Local skin-friction distribution

c) Velocity profiles

d) Covergence behavior

Fig. 3 Laminar flow over a NACA 0012 airfoil ($M_\infty = 0.50$, $\alpha = 0$, $Re = 5000$, —present investigation, o Ref. 1).

3) Add these with the corresponding Euler flux for the same cell.

4) From the four neighboring cells of the point (i,j), take the average to get the flux across $\Omega i,j$ surround point (i,j).

It is to be noted that in the present scheme the first derivatives were calculated using all the four sides of the control volume. The thin-layer theory has been applied only for the second derivatives. In Ref. 1, the streamwise-like differences were neglected even at the first stage.

The present method has been applied to the following test problems: flow over a flat plate and flow over an NACA 0012 airfoil. Only the laminar cases have been considered. In both cases, the freestream Mach number $M_\infty = 0.25$–0.85 and freestream Reynolds number (with respect to length of the flat plate and chord of the airfoil) $Re = 500$–5000 have been considered. The airfoil case also have been computed for different angles of attack α. The full results have been reported in Ref. 2. The effect of grid size, effect of artificial viscosity, effect of far-field boundary location/flow condition, etc., have been studied in detail for the flat-plate problem. The variation of local skin-friction coefficient CF_{loc} with local Reynold's number Re_x has been compared with the classical Blasius solution, and the numerical solution of Swanson and Turkel [1] and is shown in Fig. 2a. The present solution compares very well with Blasius solution throughout the plate. Figure 2b shows the exact reproduction of the Blasius velocity profile ($\text{ETA} = \sqrt{0.5 Re_x}\, y/x$ vs u/u_∞).

The computation time on a CRAY-1 computer needs about 0.8 times 10^{-5} s/grid point/iteration. A comparison of pressure distribution on NACA 0012 airfoil is shown in Fig. 3. The agreement between the two computations is very good. The local skin-friction distribution, velocity profiles on the airfoil and wake, and convergence history also are shown. Separated behavior of the velocity profiles as well as the formation of the boundary layer and the wake can be seen clearly in the figure.

The flow past the same airfoil at $M_\infty = 0.80$, $\alpha = 10$ deg. and $Re = 500$ also has been considered. The comparison of aerodynamic coefficients among various versions of the present method, including a different way of implementing the thin-layer theory and other numerical solutions [3,4] are summarized in Table 1 (shown in full paper). The present methods predict more C_L and CD_F than those of the reference values, where the artificial viscosity was used for stability and the combined effect of two viscosities give low C_L and CD_F. The effect of single vortex on the far-field boundary predicts more C_L, as it should. The pressure drag and moment coefficients about the leading edge agree well with each other (opposite convention of sign used here).

Acknowledgment

This is a part of work done from November 1986 to August 1987 at the Institute of Design Aerodynamics, DFVLR Braunschweig under a joint collaboration program between Council of Scientific and Industrial Research (India) and Deutsche Forschungs und Versuchsanstalt für Luft und Raumfahrt (West Germany).

References

[1]Swanson, R. C. and Turkel, E., "A Multistage Time-Stepping Scheme for the Navier-Stokes Equations," AIAA Paper 85-0035, Jan. 1985.

[2]Chakrabartty, S. K., "Numerical Solution of Two-Dimensional Navier-Stokes Equations by Finite Volume Method," DFVLR Braunschweig, FRG, Rept. IB-129-87/23, 1987.

[3]Muller, B., "Navier-Stokes Solution for Laminar Transonic Flow over a NACA-0012 Airfoil," FFA Rept. 140, 1986.

[4]Muller, B. and Rizzi, A., "Runge-Kutta Finite Volume Simulation of Laminar Transonic Flow over at NACA 0012 Airfoil Using the Navier-Stokes Equations, Försöksanstalten, Stockholm, Sweden, FFA TN 1986-60, 1986.