DIRECT SIMULATION OF SPATIALLY EVOLVING COMPRESSIBLE TURBULENT BOUNDARY LAYERS

Man Mohan Rai

NASA Ames Research Center, Moffett Field, California

Thomas B. Gatski

NASA Langley Research Center, Hampton, Virginia

and

Gordon Erlebacher

ICASE, NASA Langley Research Center, Hampton, Virginia

INTRODUCTION

Data from direct numerical simulations of turbulent shear flows are becoming increasingly valuable in identifying the essential energy balances occurring within such flows. This information is useful in verifying the various turbulent closures developed for higher-order turbulent models. In the incompressible regime, such simulations of turbulent channel (Ref. 1) and boundary-layer (Ref. 2) flows have been able to identify the relative distribution of the various terms in the Reynolds stress and dissipation rate budgets. In addition, analyses of the turbulent structures (Refs. 3-4) in turbulent channel flow have also been made. These studies have significantly contributed to the understanding of turbulence as well as the development of better turbulence models.

Recently, there has been renewed interest in the computation of turbulent compressible flows because of the interest in designing high speed vehicles and the associated propulsion systems. Unfortunately, databases of turbulent flows, that are more commonly available in the incompressible regime, have not been generated for high-speed compressible flows. The effort described in this paper is the first direct numerical simulation of a compressible, turbulent, supersonic, spatially evolving boundary-layer flow. The motivation is to simulate a turbulent flow (of relevance to engineering purposes) at moderate Reynolds numbers which can be used as a guide in understanding the fundamental turbulent physics of such flows as well as providing the relevant Reynolds stress and dissipation budgets which have proved useful in the incompressible studies.

The flow chosen is the zero-pressure-gradient, flat-plate boundary layer flow of Shutts et al. (Ref. 5) (Case 55010501) at $M_\infty = 2.25$ with freestream $Re_\theta \approx 6,000$. Such a large free stream Re_θ can be simulated because of the dependence of viscosity on temperature. Under the adiabatic conditions of the experiment, the temperature increases as the wall is approached and the Re_θ based on local conditions is correspondingly reduced. Thus in regions where resolution requirements are most stringent the increased viscosity effectively reduces the grid density required to compute the flow. Related work for supersonic flow through a channel performed at Mach numbers of 1.5 and 3.0 with Reynolds number of about of $300-400$ is reported in Ref. 6. A direct comparison of the Reynolds numbers used in Ref. 6 and in the present study is difficult to make because of the cold wall condition used in Ref. 6, and, other differences.

The computational method used in this study is a high-order-accurate, upwind-biased, implicit, finite-difference method that was developed in Ref. 7 and used to compute transition to turbulence on a flat plate at the low Mach number of $M_\infty = 0.1$ and subject to high freestream disturbance levels. The technique involves using upwind-biased differences for the convective terms, central differences for the viscous terms and an iterative-implicit time integration scheme. The computational results from this study indicated that the essential features of the transition process had been captured. The computational method as developed in Ref. 7 uses the non-conservative form of the governing equations to obtain the solution. Therefore, this approach is confined to flows which do not contain discontinuities. In the present flat plate simulation, the freestream Mach number is assumed to be sufficiently low to preclude eddy shocklets (the computed results confirm this assumption).

Hence, the method of Ref. 7 is used with minor modifications in the current investigation.

NUMERICAL METHOD

The integration method used in this study for the compressible form of the Navier-Stokes equations is a high-order-accurate, upwind-biased, finite-difference technique that is used in conjunction with an iterative-implicit time-advancement scheme. The convective terms are evaluated as in Ref. 8 (but with high-order-accurate differences instead of second-order-accurate differences), and the iterative implicit technique of Ref. 9 is used to integrate the equations of motion in time.

To describe the method, we consider the unsteady, compressible, nonconservative formulation of the Navier-Stokes equations in three spatial dimensions:

$$Q_t + AQ_x + BQ_y + CQ_z = \frac{1}{\rho}(R_x + S_y + T_z) \tag{1}$$

where Q is the vector of dependent variables

$$Q = \begin{pmatrix} \rho \\ u \\ v \\ w \\ p \end{pmatrix} \tag{2}$$

In Eq. (2), ρ is the density; u, v, and w are the velocities in the x, y, and z directions, respectively; and p is the pressure. The matrices A, B, and C are obtained from the matrix D,

$$D = \begin{pmatrix} q & k_1\rho & k_2\rho & k_3\rho & 0 \\ 0 & q & 0 & 0 & k_1/\rho \\ 0 & 0 & q & 0 & k_2/\rho \\ 0 & 0 & 0 & q & k_3/\rho \\ 0 & k_1\gamma p & k_2\gamma p & k_3\gamma p & q \end{pmatrix} \tag{3}$$

where q is defined as $q = uk_1 + vk_2 + wk_3$. The matrix A is obtained from D with $k_1 = 1, k_2 = 0$, and $k_3 = 0$; the matrix B is obtained with $k_1 = 0, k_2 = 1$, and $k_3 = 0$; and the matrix C is obtained using $k_1 = 0, k_2 = 0$, and $k_3 = 1$. The viscous term R_x on the right-hand side of Eq. (1) is given by

$$R_x = \begin{pmatrix} 0 \\ (2\mu u_x + \lambda(u_x + v_y + w_z))_x \\ (\mu(u_y + v_x))_x \\ (\mu(u_z + w_x))_x \\ \rho(\mu Pr^{-1}(c^2)_x)_x + \phi \end{pmatrix} \tag{4}$$

where c is the local speed of sound and

$$\begin{aligned} &\lambda = -2\mu/3 \\ &\phi = \rho(\gamma - 1)(u_x(2\mu u_x + \lambda(u_x + v_y + w_z)) + v_x(\mu(u_y + v_x)) + w_x(\mu(u_z + w_x))) \end{aligned} \tag{5}$$

The viscous terms S_y and T_z are obtained from similar expressions.

Before describing the time-integration method, it is helpful to consider the techniques used to compute the convective and viscous terms. The convective terms AQ_x, BQ_y, and CQ_z [Eq. (1)] are evaluated as in Ref. 8. To illustrate the technique, we consider the term AQ_x. The matrix A can be written as

$$A = P\Lambda P^{-1} \tag{6}$$

where P is the matrix of the left eigenvectors of A, and Λ is a diagonal matrix containing the eigenvalues of A. The term AQ_x is evaluated as

$$AQ_x = A^+ Q_x^- + A^- Q_x^+ \tag{7}$$

where $A^\pm = P\Lambda^\pm P^{-1}$, and Λ^+ and Λ^- are diagonal matrices containing the positive and negative eigenvalues of A, respectively. The terms Q_x^+ and Q_x^- are forward and backward differences of the vector Q, respectively. Additional details regarding this method of evaluating the convective terms can be found in Ref. 8. In the present study, Q_x^+ and Q_x^- are computed using fifth-order-accurate forward- and backward-biased finite differences using a seven-point stencil as

$$Q_x^- = \frac{-6Q_{i+2} + 60Q_{i+1} + 40Q_i - 120Q_{i-1} + 30Q_{i-2} - 4Q_{i-3}}{120\Delta x}$$

$$Q_x^+ = \frac{4Q_{i+3} - 30Q_{i+2} + 120Q_{i+1} - 40Q_i - 60Q_{i-1} + 6Q_{i-2}}{120\Delta x}$$

on a grid that is equispaced in the x-direction. The remaining convective terms are evaluated in a similar manner. For stretched meshes (for example, the type used in the wall-normal direction in this study), the coefficients in the difference formulas are evaluated numerically using Lagrange polynomials so as to retain high-order accuracy even on grids where the rate of change of grid spacing is not sufficiently smooth (obtaining explicit formulas for these coefficients is a laborious task for differences with higher than second-order accuracy). Additional details regarding this procedure can be found in Ref. 10.

The viscous terms are computed using central differences and a five-point stencil (fourth-order accuracy). To illustrate, we consider the term $(\mu(u_y + v_x))_x$ in the vector R_x. This term is first expanded as

$$(\mu(u_y + v_x))_x = \mu_x(u_y + v_x) + \mu(u_{xy} + v_{xx}) \tag{8}$$

Each of the derivatives on the right-hand side of Eq. (8) is then evaluated using central differences. As in the case of the convective terms, the coefficients in the difference formulas are evaluated numerically using Lagrange polynomials.

The fully implicit finite-difference representation of Eq. (1) can now be written as

$$\begin{aligned} &\frac{3Q^{n+1} - 4Q^n + Q^{n-1}}{2\Delta t} + \\ &\qquad (A^+Q_x^- + A^-Q_x^+ + B^+Q_y^- + B^-Q_y^+ + C^+Q_z^- + C^-Q_z^+)^{n+1} \\ &\qquad\qquad = (\frac{1}{\rho}(R_x + S_y + T_z))^{n+1} \end{aligned} \tag{9}$$

Equation 9 is second-order accurate in time and represents a system of nonlinear equations. This system of nonlinear equations can be solved using a Newton-Raphson-type iterative technique at each time step. The linearization of Eq. (9) results in

$$
\begin{aligned}
&(1.5I + \Delta t(\frac{A^+\nabla_x}{\nabla x_i} + \frac{A^-\Delta_x}{\Delta x_i}) + \Delta t(\frac{B^+\nabla_y}{\nabla y_j} + \frac{B^-\Delta_y}{\Delta y_j} - M(\frac{\Delta_y}{\Delta y_j} + \frac{\nabla_y}{\nabla y_j})- \\
&N(\frac{\Delta_y}{\Delta y_j} - \frac{\nabla_y}{\nabla y_j})) + \Delta t(\frac{C^+\nabla_z}{\nabla z_k} + \frac{C^-\Delta_z}{\Delta z_k}))^p(Q^{p+1} - Q^p) = -\Delta t(\frac{3Q^p - 4Q^n + Q^{n-1}}{2\Delta t}+ \\
&(A^+Q_x^- + A^-Q_x^+ + B^+Q_y^- + B^-Q_y^+ + C^+Q_z^- + C^-Q_z^+)^p - (\frac{1}{\rho}(R_x + S_y + T_z))^p)
\end{aligned}
\tag{10}
$$

where ∇ and Δ are backward and forward difference operators, respectively; the matrices M and N represent the linearization of the first and second derivatives in the viscous terms, and p is an iteration index. For the viscous terms, only the derivatives in the y or wall-normal direction have been retained on the left-hand side (LHS). On the LHS of Eq. (10), the convective terms are represented with first-order-accurate formulations and the viscous terms with second-order formulations as in Ref. 9. The reason this does not affect the accuracy of the scheme is as follows. In Eq. (10), Q^p is an approximation to Q^{n+1}. When p = 0, $Q^p = Q^n$, and when Eq. (10) is iterated to convergence at a given time-step, $Q^p \rightarrow Q^{n+1}$. Since the LHS of Eq. (10) can be driven to zero at each time-step, linearization errors (such as those mentioned above) can be driven to zero during the iteration process. It should be noted that the discretization presented in Eq. (10) is valid only for rectangular grids (however, the grid spacing in each direction may vary).

The LHS of Eq. (10) represents a sparse matrix of large bandwidth and, therefore, is computationally expensive to solve. Reference 9 uses an approximate factorization technique, as was done in Ref. 11, to overcome this problem. The factored form of Eq. (10) is given by

$$
\begin{aligned}
&(\alpha I + \beta\Delta t(\frac{A^+\nabla_x}{\nabla x_i} + \frac{A^-\Delta_x}{\Delta x_i}))^p \times \\
&\quad(\alpha I + \beta\Delta t(\frac{B^+\nabla_y}{\nabla y_j} + \frac{B^-\Delta_y}{\Delta y_j} - M(\frac{\Delta_y}{\Delta y_j} + \frac{\nabla_y}{\nabla y_j}) - N(\frac{\Delta_y}{\Delta y_j} - \frac{\nabla_y}{\nabla y_j})))^p \times \\
&\quad(\alpha I + \beta\Delta t(\frac{C^+\nabla_z}{\nabla z_k} + \frac{C^-\Delta_z}{\Delta z_k}))^p(Q^{p+1} - Q^p) = -\Delta t(\frac{3Q^p - 4Q^n + Q^{n-1}}{2\Delta t}+ \\
&(A^+Q_x^- + A^-Q_x^+ + B^+Q_y^- + B^-Q_y^+ + C^+Q_z^- + C^-Q_z^+)^p - (\frac{1}{\rho}(R_x + S_y + T_z))^p)
\end{aligned}
\tag{11}
$$

where $\alpha = 1.5^{1/3}$ and $\beta = 1.5^{-2/3}$. The approximation made earlier of using a three-point stencil in formulating the LHS of Eq. (10) has resulted in systems of block-tridiagonal matrices in Eq. (11) instead of block-heptadiagonal matrices; this reduces the computing time required at each iteration at the possible expense of the number of iterations to converge. The factorization error, just like the linearization error, can be driven to zero during the iteration process. Additional details regarding the computational method can be found in Ref. 7.

An important limitation imposed by using the nonconservative formulation of the Navier-Stokes equations is that the method can only be used to compute flows that are free of flow discontinuities (such as purely subsonic flows). The nonconservative formulation of the governing equations has been chosen for this investigation because of the efficiency of this approach, the availability of a validated computer program that uses this approach, and, because the flow does not exhibit discontinuities. Methods that use the conservative form of the equations, that are conservative in a discrete sense, and, that are high-order accurate on generalized grids are developed in Ref. 12. In summary, the present method is second-order accurate in time and fourth-order accurate in space.

BOUNDARY CONDITIONS

The computational region of interest is shown in Fig. 1. In the streamwise (x) direction it extends from x = 4.0 inches to x = 15.0 inches for the turbulence simulation. The grid spacing in the streamwise direction is sufficiently fine to resolve the flow features of interest only up to x = 9.0 inches. Thereafter the grid gradually becomes extremely coarse so that the disturbances in the boundary layer rapidly attenuate and a simple boundary condition can be used at the exit boundary. The dimensions of the computational region in the wall normal (y) and spanwise (z) directions are 3.0 inches and 0.35 inches, respectively. The boundaries of the computational region consist of the inlet boundary, the exit boundary, the lower plate boundary, the upper symmetry boundary and the two boundaries in the spanwise direction. The boundary conditions used at each of these boundaries is discussed below.

Inlet Boundary

The inlet boundary is for the most part a supersonic inlet boundary except for that portion of the boundary layer that is subsonic. The values of all the dependent variables are fixed at this boundary (the data for this boundary is obtained from a boundary layer analysis).

Exit Boundary

The exit boundary is a supersonic exit boundary except for that portion of the boundary layer that is subsonic. All the variables are extrapolated from the interior to the exit boundary for those exit boundary points at which the flow Mach number is greater than unity. For boundary points at which the streamwise Mach number is subsonic the extrapolation process described above is implemented for all the variables except the pressure. The pressure is specified as equal to the pressure at the grid point at which the Mach number first becomes supersonic. In order to be accurate this approximation requires that the disturbances in the boundary layer be attenuated before the exit boundary is reached. As described before this attenuation is accomplished by using a grid that gradually becomes very coarse in the streamwise direction.

Upper and Spanwise Boundaries

A symmetry boundary condition is used at the upper boundary. This boundary condition in reality makes the present simulation a channel simulation instead of a flat plate simulation. However, the ratio of the width of the channel to the momentum thickness at x = 9.0 inches is about 300. Therefore the computed solution will be very close to a flat plate solution. Additionally, the grid coarsens in the y direction so that disturbances are attenuated before they reach the upper boundary.

A periodic boundary condition is used in the spanwise direction. The width of the computational region in the spanwise direction was determined with the help of two point correlations of the velocity components in this direction obtained using coarse grids and wider domains. The width was chosen such that the two point correlation values were sufficiently small at the tail end of the correlation curve.

Lower Surface Boundary

The lower boundary of the computational domain corresponds to the plate surface. Except for the portion of the plate where blowing and suction is implemented a no-slip boundary condition together with a temperature condition (adiabatic wall temperature) is specified. The dependent variables are obtained from

$$T_1 = T_{aw}$$

$$u_1 = 0$$

$$v_1 = 0$$

$$w_1 = 0$$

$$p_1 = p_2$$

where the subscripts 1 and 2 correspond to the wall and the first point away from the wall, respectively. The density at the wall is computed using the pressure and the temperature. In the region of wall blowing and suction (in this computation, $4.5 \leq x \leq 5.0$) the normal component of the velocity is computed as

$$v_1 = v_{bs}$$

where v_{bs} is velocity of suction and blowing and the rest of the dependent variables are computed as before. The wall-normal velocity at the wall is prescribed as

$$v_{bs} = Au_{\infty}v_w(x)sin(2\pi\beta t)$$

$$v_w(x) = 4sin(\theta)(1 - cos(\theta))/(27)^{1/2}$$

$$\theta = 2\pi(x - x_a)/(x_b - x_a)$$

where x_a and x_b are the streamwise locations of the beginning and the end of the blowing/suction strip, respectively, A is the amplitude of the imposed disturbance and β is the temporal frequency of the disturbance.

EVOLUTION OF SMALL AMPLITUDE DISTURBANCES

The numerical method and boundary conditions dicussed in the previous sections and the corresponding computer program were tested by studying the evolution of two-dimensional, small amplitude disturbances in flat plate flow. This investigation closely resembles that reported in Ref. 15 (CASE 2). The Mach number for this computation is 4.5, the freestream temperature is $110°R$ and the Reynolds number based on freestream conditions is 83333/inch. The boundary layer is perturbed with small amplitude eigenfunctions that were obtained using the Parabolized Stability Equation (PSE) method. These eigenfunctions are superimposed on the boundary layer solution that is implemented at the inlet boundary. The dimensionless frequency of this eigensolution is $F = 2.2 \times 10^{-4}$ (80960 Hz). This frequency excites a second-mode disturbance. The spatial evolution of this disturbance is computed using the method described earlier.

The computational domain extends from x = 5.88 inches to 20.88 inches with only the first 9.0 inches being well resolved, the last 6.0 inches are discretized using a grid that gradually becomes very coarse to attenuate the disturbances. The grid resolution in the x direction is 50 points/inch (16 points per wavelength as in Ref. 15). The first point away from the wall is at y = 0.0005 inches. The total number of grid points used in the computation is 120,731 (481×251 grid). Three hundred time steps were used per time period with two iterations per time step.

Figure 2 shows the maximum amplitude of fluctuation of the various quantities at each streamwise location as a function of the distance along the plate. The figure includes both the computed results using the present methodology as well as the PSE results of Ref. 15. The agreement between the two sets of results is excellent.

RESULTS

Results obtained for the flat plate flow are presented in this section. These results were obtained in several stages. The initial solution was obtained on an extremely coarse grid and then interpolated onto successively finer grids. The computation on each of the intermediate grids was carried to a point where the skin-friction distribution (after spanwise averaging) showed minor changes in time and exhibited a time-average that was essentially in equilibrium.

The computation was modeled after the experimental configuration of Shutts et al. (Ref. 5). The Reynolds number based on inlet conditions is 635,000/in., and the inlet Mach number is 2.25. The freestream temperature is $305°R$ and the adiabatic wall temperature is $580°R$. The computational domain consists of three regions. The first region is fairly well resolved and contains the region of blowing and suction and some of the end stage transition phenomena ($2.540 \times 10^6 \leq Re_x \leq 4.445 \times 10^6$). The second region also contains some of the end stage transition phenomena as well as the turbulent portion of the flow field ($4.445 \times 10^6 \leq Re_x \leq 5.715 \times 10^6$) and has the densest grid in the computation. The third region is discretized using a grid that gradually becomes very coarse in the streamwise direction ($5.715 \times 10^6 \leq Re_x \leq 9.525 \times 10^6$). The total number of grid points used in the computation is about 17 million ($971 \times 55 \times 321$ grid). Based on the wall shear velocity at x = 8.8 inches, or $Re_x = 5.588 \times 10^6$, the resolution in the second region is $\Delta x^+ = 27.0$, $\Delta z^+ = 10.4$ and the first grid point of the wall is at $y^+ = 0.95$.

Figure 3 shows the computed skin-friction, the experimental data of Ref. 5, laminar skin-friction data obtained from a boundary layer analysis and the turbulent correlation of White and Christoff (Ref. 13) given by

$$C_f = \frac{0.455}{S^2(ln(\frac{0.06}{S} Re_x \frac{\mu_\infty}{\mu_{wall}} \sqrt{\frac{T_\infty}{T_{wall}}}))^2}$$

where T is the temperature, μ is the viscosity of the fluid, Re_x is the Reynolds number based on freestream conditions, and S is given by

$$S = \frac{\sqrt{T_{wall}/T_\infty - 1}}{sin^{-1}(A)}$$

and

$$A = (r\frac{\gamma - 1}{2}M_\infty^2 \frac{T_\infty}{T_{wall}})^{1/2}$$

where r is the recovery factor and M is the Mach number. The region of blowing and suction can be discerned in the region $4.5 \leq x \leq 5.0$ as small variations in the computed skin-friction. Further downstream, in the end stage transition region, the skin-friction increases dramatically, reaches a maximum and then begins to decrease in a more gradual fashion. This behavior is consistent with the known data. The experimental data of Shutts et al. (Ref. 5) is about 19% lower than the computed value at $x = 8.8$ inches. However, the computed results are closer to the turbulent correlation. It is reported in Ref. 13 that the RMS variation of numerous sets of experimental data with respect to the correlation is 7.8%. The corresponding band of variation is also shown in Fig. 3. The presence of small oscillations in the computed skin-friction is a reflection of a small statistical sample.

Figure 4 shows the computed Van Driest effective velocity normalized by the wall shear velocity plotted as a function of the wall normal distance (in wall units) at the streamwise location $x = 8.8$ inches. This location is about 15 times the boundary layer thickness at x = 8.8 from the end of transition. The effective velocity is obtained from

$$u_{VD} = \frac{u_\infty}{A} sin^{-1}(\frac{Au}{u_\infty})$$

Also plotted in Fig. 4 are the curves $u_{VD}^+ = y_w^+$ and the log law $u_{VD}^+ = 2.5ln(y_w^+) + 5.5$. Clearly, there is excellent agreement between the computed velocity profile and the log law. A similar agreement is seen in the near wall region between the computed results and the curve $u_{VD}^+ = y_w^+$. Another feature of interest is the presence of the wake profile for $y_w^+ \geq 500$.

Figure 5 shows turbulence intensity profiles normalized by the freestream velocity. The computed results agree qualitatively with existing experimental data. Detailed comparisons between the computed results and experimental data will be presented in the final paper. The turbulence intensities normalized by the local mean streamwise velocity at the streamwise location $x = 8.8$ inches are presented in Fig. 6. The symbols represent experimental data for incompressible flow from Ref. 14. Since the flow Mach number near the wall is subsonic, incompressible flow data is appropriate for the purpose of comparison. The streamwise and spanwise components are seen to agree well with the experimental data. The limiting value at the wall of the streamwise component is very nearly the same as the experimental value of 0.39. The normal component is lower than that of the experimental data. It should be noted that the experimental $v_{rms}/\overline{u}$ data do not exhibit the expected linear behavior in the near-wall region; instead, they increase as the wall is approached thus indicating experimental errors.

Figure 7 shows the computed Reynolds shear-stress profile normalized by the square of the wall shear velocity and plotted in wall units at the streamwise location x = 8.8 inches. The incompressible flow experimental data of Ref. 14 is also included to provide a frame of reference for looking at the computed results. Compressible flow data from Ref. 5 will be included in the final paper. The computed peak value is lower than the incompressible data (as expected).

The distributions of the turbulent intensies for the pressure, density, and temperature, and velocity (turbulent Mach number) are shown in Fig. 8. The scaling for all variables is based on the local mean value including that for the turbulent Mach number. The pressure fluctuations are shown to be relatively constant across the layer, but with a slight decrease towards the outer part of the layer. The other variables, however, show a rapid rise in the vicinity of the wall at $y \approx 0.0025$ ($y_w^+ \approx 25$), and then a gradual decline towards the outer part of the boundary layer. The peak levels of the density and temperature fluctuations are about 11%, and for the turbulent Mach number the intensity level peaks at about

0.15. These levels suggest that the turbulence is weakly compressible and that Morkovin's hypothesis would be applicable in describing such a flow. Nevertheless, these results are useful in validating and calibrating various models for correlations such as the mass and heat flux, as well as pressure dilatation and dilatation dissipation.

Figures 9-11 show instantaneous contours of spanwise vorticity in the x-z plane just above the wall ($y^+ = 0.95$). Figure 9 shows contours in the region $5.0 \leq x \leq 6.5$. Isolated patches of vorticity are seen in immersed in relatively quiescent regions. The appearance of the clustered vortical structures, signifying the end of the transition process and the begining of the early turbulent flow, is very abrupt. Figure 10 shows contours in the region $6.5 \leq x \leq 8.0$. The flow in this region is mostly turbulent except in the far upstream region. Streaky structures of the kind found in incompressible flows are clearly evident. These structures are more clearly observed in Fig. 11 which shows contours in the region $8.0 \leq x \leq 8.5$. The high speed streaks (core of blue) are on the average 125 wall units in width and 500 wall units in length. They correspond approximately in size and shape with high speed streaks observed in incompressible flow. In retrospect this should have been expected since the Mach number at this y location is less than 0.1. The low speed streaks are generally much longer than the high speed streaks.

SUMMARY

This investigation involves the use of high-order-accurate, finite-difference methods for direct simulations of compressible turbulent flow. The technique involves using upwind-biased differences for the convective terms, central differences for the viscous terms and an iterative-implicit time integration scheme. Validation of the numerical technique is achieved by computing the growth of small amplitude disturbances and then comparing these results with PSE results.

Using blowing and suction to trip the incoming flow, a spatially evolving compressible turbulent flat-plate boundary layer at M = 2.25 and freestream $Re_\theta = 6000$ has been simulated. Preliminary results from this simulation including turbulent statistics and flow visualizations are presented. The effort described in this paper is the first direct numerical simulation of a compressible, turbulent, supersonic, spatially evolving boundary-layer flow. The final paper will include more detailed investaigations of the turbulent statistics and flow visualizations and the results of a grid refinement study.

REFERENCES

1. Mansour, N. N., Kim. J.,and Moin P., "Reynolds-Stress and Dissipation Rate Budgets in a Turbulent Channel Flow," *J. Fluid Mech.*, Vol. 194, pp. 15–44, 1988.

2. Moin, P., Shih, T.-H., Driver, D., and Mansour, N.N., "Direct Numerical Simulation of a Three-Dimensional Turbulent Boundary Layer," *Phys. Fluids A*, Vol. 2, pp. 1846-1853, 1990.

3. Moser, R.D., "Statistical Analysis of Near-Wall Structures in Turbulent Channel Flow," *Proc. Zaric Memorial Intl. Seminar on Wall Turbulence*, may 16–20, Dubrovnik, Yugoslavia, 1988.

4. Moin, P., and Moser, R.D., "Characteristic- Eddy Decomposition of Turbulence in a Channel," *J. Fluid Mech.*, Vol. 200, pp. 471-509, 1989.

5. Fernholz, H.H., and Finley, P.J., "A Critical Compilation of Compressible Turbulent Boundary Layer Data," *AGARDograph No. 223*, Case 55010501, 1977.

6. Coleman G.N., "Direct Numerical Simulation of Isothermal-Wall Supersonic Channel Flow," *Center for Turbulence Research: Annual Research Briefs*, pp. 313–328, 1993.

7. Rai, M. M., and Moin, P., "Direct Numerical Simulation of Transition and Turbulence in a Spatially Evolving Boundary Layer," *Journal of Computational Physics*, Vol. 109, No. 2, December 1993, pp. 169-192.

8. Chakravarthy, S. R., Anderson, D. A., and Salas, M. D., "The Split-Coefficient Matrix Method for Hyperbolic Systems of Gas Dynamic Equations," AIAA Paper 80-0268, Pasadena, Calif., 1980.

9. Rai, M. M., and Chakravarthy S. R., "An Implicit Form for the Osher Upwind Scheme," *AIAA Journal*, Vol. 24, No. 5, May 1986, pp. 735-743.

10. Rai, M. M., and Moin, P., "Direct Simulations of Turbulent Flow Using Finite-Difference Schemes," *Journal of Computational Physics*, Vol. 96, No. 1, September 1991, pp. 15-53.

11. Beam, R. M., and Warming, R. F., "An Implicit Factored Scheme for the Compressible Navier-Stokes Equations," *AIAA Journal*, Vol. 16, No. 4, April 1978, pp. 393-402.

12. Rai, M. M. and Chakravarthy, S. R., "Conservative High-Order-Accurate Finite-Difference Methods for Curvilinear Grids," AIAA Paper No. 93-3380, AIAA 11th Computational Fluid Dynamics Conference, Orlando, Florida, July 6-9, 1993.

13. White, F. M., *Viscous Fluid Flow* , McGraw-Hill Book Company, 1974.

14. Karlson, R. I. and Johansson, T. G., "LDV Measurements of Higher-Order Moments of Velocity Fluctuations in a Turbulent Boundary Layer," *Laser Anemometry in Fluid Mechanics* , Ladoan - Instituto Superior Tecnico, Lisbon Portugal, 1988.

15. Pruett, C. D., Zang, T. A., Chang, C.-L. and Carpenter, M. H., "Spatial Direct Numerical Simulation of High-Speed Boundary-Layer Flows - Part I: Algorithmic Considerations and Validation," to appear in *Theoretical and Computational Fluid Mechanics*
.

FIGURE CAPTIONS

1. Schematic of computational region (not to scale).

2. Variation of disturbance amplitude maxima in the streamwise direction.

3. Computed skin friction along the length of the flat plate.

4. Mean velocity profile normalized by wall-shear velocity at x = 8.8 inches.

5. Profiles of the turbulence intensities normalized by the freestream velocity at x = 8.8 inches.

6. Turbulence intensities at x = 8.8, normalized by local mean velocity and plotted in wall coordinates.

7. Reynolds shear-stress distribution at the streamwise location x = 8.8, normalized by the square of the wall-shear velocity and plotted in wall coordinates.

8. Profiles of the turbulent Mach number M_t and the rms values of the density, pressure and temperature.

9. Spanwise vorticity contours in an (x,z) plane in the region $4.0 \leq x \leq 6.5$ at $y^+ = 0.95$.

10. Spanwise vorticity contours in an (x,z) plane in the region $6.5 \leq x \leq 8.0$ at $y^+ = 0.95$.

11. Spanwise vorticity contours in an (x,z) plane in the region $8.0 \leq x \leq 8.5$ at $y^+ = 0.95$.