Navier-Stokes Solution of the Turbulent Flowfield about an Isolated Airfoil

S. J. Shamroth* and H. J. Gibeling†
Scientific Research Associates, Inc., Glastonbury, Conn.

Abstract

A COMPRESSIBLE time-dependent solution of the Navier-Stokes equations, including a transition-turbulence model, is obtained for the isolated airfoil flowfield problem. The equations are solved by a consistently split linearized block implicit scheme due to Briley and McDonald. A nonorthogonal body-fitted coordinate system is used, which has maximum resolution near the airfoil surface and in the region of the airfoil leading edge. The transition-turbulence model is based upon the turbulence kinetic energy equation and predicts regions of laminar, transitional, and turbulent flow. Mean flowfield and turbulence field results are presented for an NACA 0012 airfoil at zero and nonzero incidence angles at Reynolds number up to one million and low subsonic Mach numbers.

Contents

Flow about an isolated airfoil is an important problem which arises in a variety of practical applications, including the aircraft wing, control surfaces, and the helicopter rotor blade problems. To date, several investigators have developed Navier-Stokes analyses to predict airfoil-type flows; a number of these approaches are discussed in Ref. 1. However, most of these analyses are restricted by assumptions of incompressible laminar flow. The procedure of Ref. 1 concentrates on solving airfoil flowfields that are both compressible and turbulent.

When solving the Navier-Stokes equations for flow about airfoils, it is highly advantageous to solve the equations in a coordinate system having the airfoil surface as a coordinate line. Such a procedure is followed in the present approach, as the equations are solved in a constructive coordinate system having spatial variables ξ and η, and a temporal variable τ. The new independent spatial variables ξ and η are functions of the Cartesian coordinates x and y, and time t. Two possible approaches for solving the equations in the resulting nonorthogonal coordinates are the strong conservation approach and the quasilinear approach. The strong conservation form of the equations is:

$$\frac{\partial W/D}{\partial \tau}+\frac{\partial}{\partial \xi}\left[\frac{W\xi_t}{D}+\frac{F\xi_x}{D}+\frac{G\xi_y}{D}\right]+\frac{\partial}{\partial \eta}\left[\frac{W\eta_t}{D}+\frac{F\eta_x}{D}+\frac{G\eta_y}{D}\right]$$
$$=\frac{1}{Re}\left[\frac{\partial}{\partial \xi}\left(\frac{F_I\xi_x}{D}+\frac{G_I\xi_y}{D}\right)+\frac{\partial}{\partial \eta}\left(\frac{F_I\eta_x}{D}+\frac{G_I\eta_y}{D}\right)\right] \tag{1}$$

where

$$D=\xi_x\eta_y-\xi_y\eta_x$$

$$W=\begin{bmatrix}\rho\\ \rho u\\ \rho v\end{bmatrix},\quad F=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\end{bmatrix},\quad G=\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\end{bmatrix}$$

$$F_I=\begin{bmatrix}0\\ \tau_{xx}\\ \tau_{xy}\end{bmatrix},\quad G_I=\begin{bmatrix}0\\ \tau_{xy}\\ \tau_{yy}\end{bmatrix} \tag{2}$$

ρ is density, p is pressure, u and v are Cartesian velocity components, and τ_{ij} is a shear stress component. The quasilinear form of the equations is:

$$\frac{\partial W}{\partial \tau}+\xi_t\frac{\partial W}{\partial \xi}+\xi_x\frac{\partial F}{\partial \xi}+\xi_y\frac{\partial G}{\partial \xi}+\eta_t\frac{\partial W}{\partial \eta}+\eta_x\frac{\partial F}{\partial \eta}+\eta_y\frac{\partial G}{\partial \eta}$$
$$=\frac{1}{Re}\left[\xi_x\frac{\partial F_I}{\partial \xi}+\eta_x\frac{\partial F_I}{\partial \eta}+\xi_y\frac{\partial G_I}{\partial \xi}+\eta_y\frac{\partial G_I}{\partial \eta}\right] \tag{3}$$

It should be noted that the solution of Eq. (1) or (3) requires evaluation of the metric coefficients $\xi_x,\xi_y,\eta_x,\eta_y$; these coefficients may be evaluated either analytically or numerically. The present approach utilizes an analytic evaluation of the metric data. When the analytic evaluation was used in conjunction with the strong conservation form, Eq. (1), physically unrealistic results dominated by geometric truncation error were obtained for the flow about a circular cylinder. However, good results were obtained when the quasilinear approach was used. It has since been noted in Ref. 2 that use of the strong conservation approach requires numerical differentiation of the metric data for accurate solutions. Therefore, as suggested in Ref. 3, the discrepancy between the two approaches for the cylinder flowfield noted in Ref. 1 is likely to be a result of using the strong conservation form in conjunction with analytic determination of the metric data. It appears that with appropriate evaluation of the metric coefficients, either approach can be used for coordinate systems having time-independent Jacobians. The quasilinear approach was used in the present study.

Since the present effort addresses the problem of turbulent flow, it is necessary to specify a turbulence model suitable for this problem. However, since the flowfield of interest has regions of laminar, transitional, and turbulent flow, any proposed model must be capable of dealing with all three regimes. The present approach calculates the turbulent viscosity from a model utilizing the turbulence energy equation and an algebraic length-scale equation. These equations are solved in conjunction with the mean flow equations to predict the mean flowfield, the turbulence energy field, and the turbulent viscosity field. The turbulent structural coefficients required for the solution of the turbulence

Presented as Paper 79-1543 at the AIAA 12th Fluid and Plasma Dynamics Conference, Williamsburg, Va., July 23-25, 1979; submitted Jan. 18, 1980; synoptic received May 16, 1980. Full paper available from AIAA Library, 555 W. 57th Street, New York, N.Y. 10019. Price: Microfiche, $3.00; hard copy, $7.00. **Remittance must accompany order.**

Index categories: Computational Methods; Viscous Nonboundary-Layer Flows.

*Vice-President. Member AIAA.
†Research Scientist.

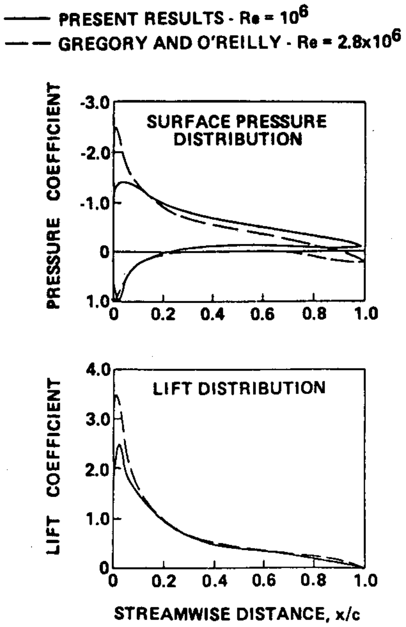

Fig. 1 Surface pressure distribution for NACA airfoil at 6 deg.

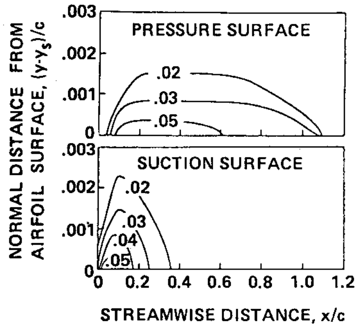

Fig. 2 Turbulence energy contours—NACA airfoil at 6 deg.

energy equation are taken to be a function of turbulence Reynolds number, thus making the model applicable to laminar, transitional, and turbulent flow regimes. The structural model used follows that of Ref. 4 and is given in detail in Ref. 1.

Results were obtained for an NACA 0012 airfoil immersed in a stream of Mach number 0.147 and a chord Reynolds number of 10^6 for both 0- and 6-deg incidence.[1] The runs were made by assuming a profile at $t=0$, and then marching the equations in time until a steady solution was reached. In the low-incidence cases considered, a steady flow was obtained in the vicinity of the airfoil within 150 time steps. The calculations were made in a highly stretched coordinate system which gave high resolution both near the airfoil surface and in the vicinity of the airfoil leading edge region. A preliminary study of mesh resolution indicated that inadequate resolution of the boundary layer may give an inaccurate prediction of the boundary-layer development, and this in turn may lead to a deterioration of the predicted pressure distribution. Details of the initial conditions, boundary conditions, coordinate system, and marching technique are given in Ref. 1. The pressure distribution predicted for the 6-deg case is compared with the data of Gregory and O'Reilly[5] in Fig. 1. As shown in Fig. 1, the major discrepancy between data and analytic prediction occurs in the leading edge region where the analysis fails to predict the correct suction peak. This discrepancy is at least partially a result of grid resolution.[1] However, the predicted lift distribution agrees well with data. In regard to other aspects of the flowfield, the predicted suction surface transition location occurs at $x/c \approx 0.08$. The data of Gregory and O'Reilly give $x_T/c \approx 0.04$ for a Reynolds number of 2.8×10^6 and $x_T/c \approx 0.08$ for a Reynolds number of 1.48×10^6, where x_T is the transition location. Thus, the predicted transition location is in excellent agreement with the data. The turbulence energy field on both the suction and pressure surfaces is shown in Fig. 2. Since the regions of high-turbulence energy are concentrated very near the airfoil surface, they are presented as lines of constant k on a plot of distance along the airfoil surface vs distance from the airfoil surface. On the suction surface, large turbulence energies are generated in the region of the high adverse pressure gradient. However, as the boundary layer proceeds in the high adverse pressure gradient region and approaches separation, $x/c > 0.4$, the turbulence energy drops as a result of the low transverse velocity gradients associated with nearly separated boundary layers which can no longer produce turbulent energy. The turbulence energy field on the pressure surface exhibits a different character. Although the analysis does not predict the maximum turbulence kinetic energy on the pressure surface to be as large as that on the suction surface, relatively high-turbulence levels are present over the entire pressure surface flowfield. Again, this is to be expected since the boundary layer in this region does not approach separation. Further results for both the 6- and 0-deg incidence cases are given in Ref. 1.

Acknowledgment

This work was supported under a joint NASA/Army program between the Structures Laboratory, U.S. Army Research and Technology Laboratory (AVRADCOM), and NASA Langley Research Center under Contract NAS1-15214.

References

[1] Shamroth, S. J. and Gilbeling, H. J., "A Time-Dependent Compressible Solution of the Navier-Stokes Equations for Turbulent Flow About an Isolated Airfoil," NASA CR-3183, Oct. 1979; also, "The Prediction of the Turbulent Flow Field about an Isolated Airfoil," AIAA Paper 79-1543, 1979.

[2] Thomas, P. D. and Lombard, C. K., "The Geometric Conservation Law—A Link Between Finite-Difference and Finite Volume Methods of Flow Computation on Moving Grids," *AIAA Journal*, Vol. 17, Oct. 1979, pp. 1030-1037.

[3] Thompson, J., Mississippi State University, private communication, 1979.

[4] McDonald, H. and Fish, R. W., "Practical Calculation of Transitional Boundary Layers," *International Journal of Heat and Mass Transfer*, Vol. 16, No. 9, Sept. 1973, pp. 1729-1744.

[5] Gregory, N. and O'Reilly, C. L., "Low Speed Aerodynamic Characteristics of NACA 0012 Airfoil Section, Including the Effects of Upper Surface Roughness Simulating Hoarfrost," Aero Rept. 1308, National Physics Lab., 1970.