Numerical Analysis of an NACA 0012 Airfoil with Leading-Edge Ice Accretions

Mark G. Potapczuk*
University of Akron, Akron, Ohio

Abstract

THIS synoptic presents current progress in analysis of a NACA 0012 airfoil with leading-edge ice. Calculations have been performed using a Navier-Stokes code coupled with a grid generation code. The computed results were compared to experimental information obtained for an airfoil with a well-defined artificial ice shape. Computational results agree well with experimental information at angles of attack below stall. Further work is needed for evaluation of the stalled airfoil.

Contents

Aircraft icing occurs when an aircraft enters a cloud containing super-cooled water droplets. These droplets can settle on aircraft surfaces, particularly leading-edge surfaces, and form ice. The ice accretion can result in a decrease in lift and an increase in drag. These two effects, however, are not easily determined before an actual icing encounter. In an effort to provide more reliable information for the design of ice protection systems, it is desirable to be able to predict these effects as part of any testing program associated with aircraft design. In addition, detailed flowfield information for an airfoil with leading-edge ice accretions is essential for the development of a program to analyze general aircraft icing. The approach used in this study is based on the solution of the thin-layer Navier-Stokes equations in a transformed coordinate system.

The codes used were developed at NASA-Ames Research Center, and further information is available in Refs. 1–4. The grid generation code developed by Sorenson[1] is called GRAPE. The Navier-Stokes code, which was developed by Steger[2] and later modified by Pulliam,[3] is called ARC2D. The turbulence model employed in ARC2D is the Baldwin-Lomax model.[4]

Two distinct types of ice accretion can develop on an airfoil, depending on the temperature. These are called rime and glaze ice. The glaze ice shape is less aerodynamic; consequently, it is more detrimental to airfoil performance and constitutes the more difficult modeling problem. In an associated experimental program conducted by Bragg,[5] an artificial glaze ice shape was constructed in order to obtain detailed flowfield information. His experimental results serve as a basis for evaluating computations.

The computational grid for this airfoil and ice shape consisted of 253 points in the direction around the surface and 64 points in the direction "normal" to the surface. The spacing of the first grid point normal to the surface was 0.2×10^{-4} chord lengths. There were 46 points in the wake, thus leaving 207 points on the body for definition of the airfoil and ice shape geometry. This amount of detail is necessary for accurate representation of the horns and concave surfaces of the ice shape geometry. All flowfield calculations were for $M_\infty = 0.12$ and $Re = 1.41 \times 10^6$.

The corners, or horns, of an ice shape cause separation of the boundary layer. If the angle of attack is low, the flow reattaches aft of the ice, and the separation process is analogous to that of a backward facing step. At the point where the pressure becomes approximately equal to that of a turbulent boundary layer with no separation, the shear layer reattaches to the surface of the airfoil, forming a steady separation bubble. On the other hand, at higher angles the boundary layer will not reattach and remains separated. The pressure on the surface does not recover to that of an unseparated turbulent boundary layer. In this unsteady flow, the vorticity is able to move along with the freestream until it is shed off the end of the airfoil. This loss of circulation on the airfoil's upper surface results in stall. Flow visualization experiments indicate that, for the geometry studied, the change from an attached flow with separation bubble to a completely separated flow occurs at approximately 7 deg angle of attack (AOA).

The lift, drag, and moment coefficients are shown compared to Bragg's experimental results in Fig. 1. These results indicate good agreement between experiment and computation, especially in the case of angles of attack less than 7 deg. This range, 0 deg $\leq AOA \leq$ 6 deg, corresponds to the steady flow discussed previously.

The calculated results for angles of attack below stall show the small separation bubbles aft of the horns. As the AOA is increased, the upper surface bubble increases in size, and the lower surface bubble decreases in size. Comparison of

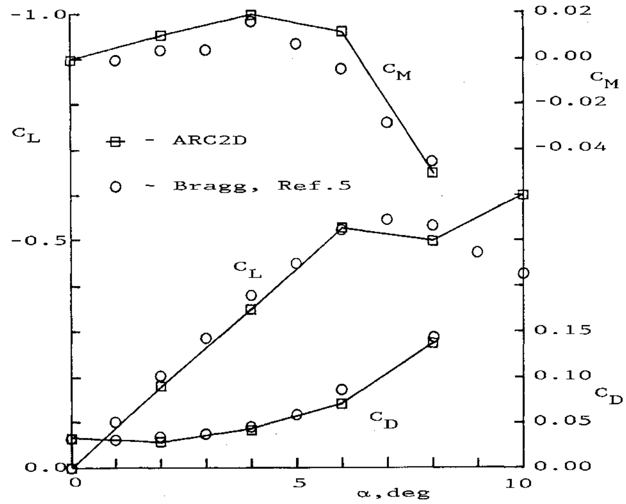

Fig. 1 Lift, drag, and moment coefficients.

Presented as Paper 87-0101 at the AIAA 25th Aerospace Sciences Meeting, Reno, NV, Jan. 12-15, 1987; received April 20, 1987; revision received July 10, 1987. Full paper available from AIAA Library, 555 W. 57th St., New York, NY 10019. Price microfiche, $4.00; hard copy, $9.00. **Remittance must accompany order.**

*Instructor in Mechanical Engineering.

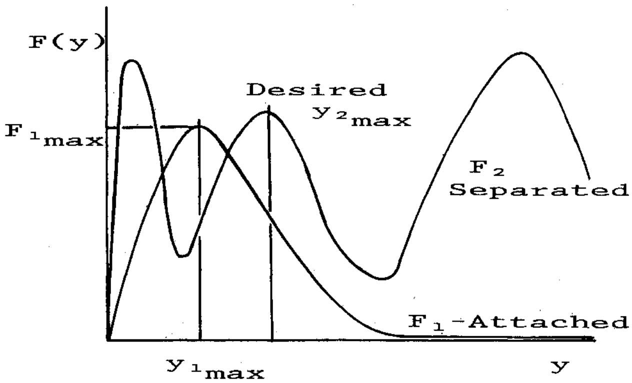

Fig. 2 Behavior of $F(y)$ for attached and separated flows.

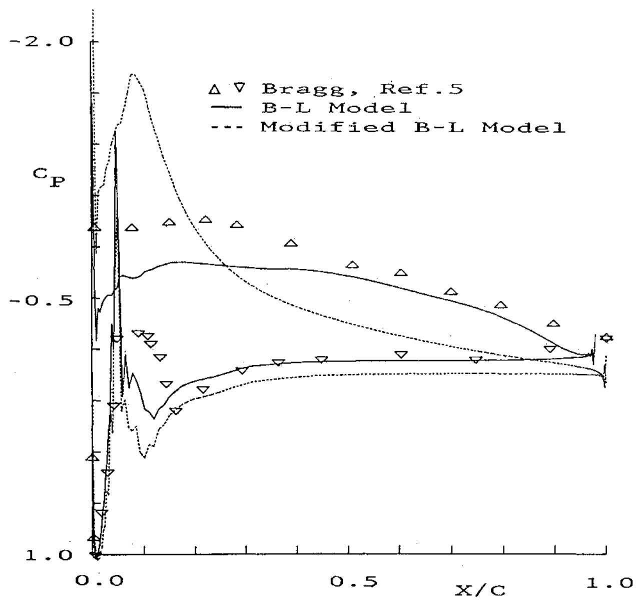

Fig. 3 Pressure coefficient distribution for 8 deg AOA.

computed velocity profiles in the bubble with Bragg's data indicate that the general features of the bubble are modeled quite well. Further discussion of the separation bubble calculation can be found in the full paper.

At approximately 7 deg AOA, the separation bubble detaches from the surface, and an unsteady separated flow develops on the upper surface. This result, indicated by flow visualization experiments, is also captured by the calculations. The calculated results indicate the development of a separation bubble aft of the horns, as in the lower AOA cases, which then moves along the airfoil surface and is eventually shed into the wake. The large suction peak associated with this vortex structure also moves along the surface until, as the vortex is shed, the peak collapses, and the lift of the airfoil decreases.

As calculated by the Baldwin-Lomax turbulence model, the eddy viscosity levels in the vortex structure and at points before it along the airfoil are much lower than in the region aft of the vortex. This is seen consistently as the vortex moves along the airfoil. This behavior of the eddy viscosity is not consistent with expectations and requires further examination.

In the Baldwin-Lomax turbulence model, the length scale in the outer region of the boundary layer is based on the function

$$F(y) = y\,|\omega|\,[1 - \exp(-y^+/A^+)]$$

The maximum value of $F(y)$ and the y-value at which $F(y)$ reaches this maximum are then selected to determine the value of the eddy viscosity at the given location. In the separated flowfield, some difficulties occur in the selection of these F_{MAX} and y_{MAX} values.

Figure 2 indicates what the difficulty is. The first profile is that of F vs y for an equilibrium boundary layer. The second profile indicates what can occur in the massively separated flowfield of the iced airfoil. If the peak at the largest y-location is picked, the eddy viscosity is too large, and the flowfield has too much dissipation. Selection of the inner peak leads to smaller than expected values of eddy viscosity. Thus, a minimum acceptable value of F is selected, and then the first maximum F value above this level is used as F_{MAX}. This approach was used to re-evaluate an 8 deg AOA case.

Results show that the airfoil has a large recirculation bubble over the entire upper surface. The calculation also converged to this flow pattern as a steady-state solution. This seems to contradict the unsteady behavior indicated from the flow visualization study. Evaluation of the pressure coefficient also indicates the discrepancy with experimental results, as seen in Fig. 3. Thus, the use of this "appropriate" F_{MAX} value did not help in evaluation of the separated flowfield.

Further examination of this case, without prior selection of the F_{MAX} value, indicates that the vortex shedding mode may be the correct representation of the actual flowfield. By averaging the pressure coefficient values at each location over several shedding periods, values similar to the experimental results were obtained. This is shown in Fig. 3. The lift coefficient obtained from this calculation is used in the global results of Fig. 1. If a better method of determining the dissipation level were used, it is expected that the results would be closer to the experimental values. With this purpose in mind, an effort is currently underway to implement both a $k-\epsilon$ turbulence model and an ordinary differential equation model of Johnson and King[6] as alternatives to the Baldwin-Lomax model.

References

[1]Sorenson, R.L., "A Computer Program To Generate Two-Dimensional Grids about Airfoils and Other Shapes by the Use of Poisson's Equation," NASA TM-81198, May 1980.

[2]Steger, J.L., "Implicit Finite-Difference Simulation of Flow about Arbitrary Two-Dimensional Geometries," *AIAA Journal,* Vol. 16, July 1978, pp. 679–686.

[3]Pulliam, T.H., "Euler and Thin-Layer Navier-Stokes Codes: ARC2D, ARC3D," *Notes for Computational Fluid Dynamics User's Workshop,* UTSI Publication No. E02-4005-023-84, March 12-16, 1984.

[4]Baldwin, B.S. and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Jan. 1978.

[5]Bragg, M.B., "An Experimental Study of the Aerodynamics of a NACA 0012 Aifoil with Simulated Glaze Ice," AIAA Paper 85-0484, 1986.

[6]Johnson, D.A. and King, L.S., "A New Turbulence Model for Boundary Layer Flows with Strong Adverse Pressure Gradients and Separation," AIAA Paper 84-0175, Jan. 1984.