Fluid Dynamics Research 3 (1988) 225–230
North-Holland

Bound vortex boundary layer control with application to V/STOL airplanes

V.J. MODI, F. MOKHTARIAN

Dept. of Mechanical Engineering, University of British Columbia, Vancouver, B.C., Canada V6T-1W5

T. YOKOMIZO, G. OHTA and T. OINUMA

Dept. of Mechanical Engineering, Kanto Gakuin University, Yokohama, Japan

Abstract. Effectiveness of the bound vortex boundary layer control is assessed with reference to airfoils modified with a leading edge rotating cylinder. Results of the test program and the numerical models suggest the following:

- The surface singularity method in conjunction with the boundary layer correction scheme is capable of predicting useful information concerning bound vortex boundary layer control. The predicted pressure distributions are in good agreement with experiment almost up to the point of complete separation from the the airfoil surface except near the trailing edge where more accurate results of the flow field would require the modelling of the separated flow region using the full Navier–Stokes equations.
- The concept of bound vortex boundary layer control appears to be quite promising. With cylinder rotation, the flow never separated completely from the upper surface. The higher rates of rotation promoted reattachment of the partially separated flow giving a significant improvement in the maximum lift and stall characteristics.

1. Introduction

Ever since the introduction of boundary layer concept by Prandtl, there has been a constant challenge faced by scientists and engineers to minimize its adverse effects and control it to advantage. Methods such as suction, blowing, vortex generators, turbulence promoters, etc., have been investigated at length and employed in practice with a varying degree of success. However, the use of moving wall for boundary layer control has received relatively little attention.

The investigation reported here studies fluid dynamics of airfoils with the rotating cylinder boundary layer control using a numerical surface singularity approach incorporating separated flow and wall confinement effects, and a finite difference boundary layer scheme to account for viscous corrections. The calculated pressure distribution and associated lift characteristics are substantiated through comparison with corresponding experimental results.

2. Numerical approach

A surface singularity potential flow model involving a distribution of vorticity on the airfoil surface and a boundary condition specified in terms of the stream function was used. The method, described in detail by Kennedy in his Ph.D. dissertation (Kennedy 1977) is thought to be the simplest available. Although the viscosity effects are ignored, it provides a fairly accurate estimate of the real flow around the section.

This method is extended to permit the analysis of airfoils with finite thickness trailing edges. The vortex sheet defining the airfoil surface is left open at the trailing edge and a source singularity of unknown strength is required within the contour of the body to model the flow inside the airfoil wake which theoretically extends to infinity. The solution of the flow field, in this case, requires the specification of two Kutta conditions at the trailing edge. The tangential velocities are contributions from the surface vorticity distribution as well as the uniform flow and the point source inside the contour, and are easily calculated. The system of equations can be easily extended to solve the case of multi-component airfoil sections.

In order to model the flow effectively over an airfoil section with boundary layer control, however, effects of viscosity must be accounted for. These are confined to the boundary layer on the surface and to the separated flow region near the trailing edge. The streamlines are shifted by a distance conventionally referred to as the "displacement thickness" of the boundary layer. The variation of the static pressure along the surface within the shear layer, therefore, depends on the shape of the surface and on the displacement effect of the boundary layer. The flow outside this boundary layer is then approximated by the potential flow over the displacement surface.

A practical method of solving this problem is, therefore, to attempt to match the outer potential flow solution with the inner boundary layer solution. The thin shear layer approximations of the Navier-Stokes equations for steady two-dimensional, incompressible flow are used. The finite difference method used to solve the boundary layer problem is the so called "box" method (Cebeci and Bradshaw 1977).

Finally, using the above approach, the effect of the cylinder can be considered:

- through the potential flow model using a simple model of a pair of counter rotating vortices below the leading edge;
- and through the boundary layer scheme by setting the appropriate boundary conditions at the location of the leading edge cylinder.

3. Wind tunnel test program

The wind tunnel tests were carried out using models spanning the tunnel test-section, $0.9 \times 0.68 \times 2.6$, to create essentially two-dimensional flow conditions. A typical model carried a central aluminum ring provided with 37 pressure taps, suitably distributed over the circumference, to yield detailed information concerning the surface loading.

The wind tunnel test results were complemented by a flow visualization study using a water tunnel in conjunction with a film of aluminum powder at the free surface. Both still and movie pictures were obtained during steady and transient conditions.

4. Results and discussion

Fig. 1a shows the pressure distributions on the surface of a Joukowsky model, without rotating cylinder replacing its nose. Thus it represents pressure profiles for a conventional Joukowsky airfoil. Due to practical difficulty in locating pressure taps in the cusp region there is a discontinuity in the pressure plots near the trailing edge. However, that region has little importance in the present discussion. It is apparent that the airfoil stalls at an angle of attack somewhere between 16°–18°.

Fig. 1b shows the adverse effect of replacing the nose by a non-rotating cylinder ($U_c/U = 0$, $U_c \equiv$ cylinder surface velocity, $U \equiv$ free stream velocity). Note only the nose geometry is slightly altered, however, now we have a two-element airfoil with a gap in the form of a step on

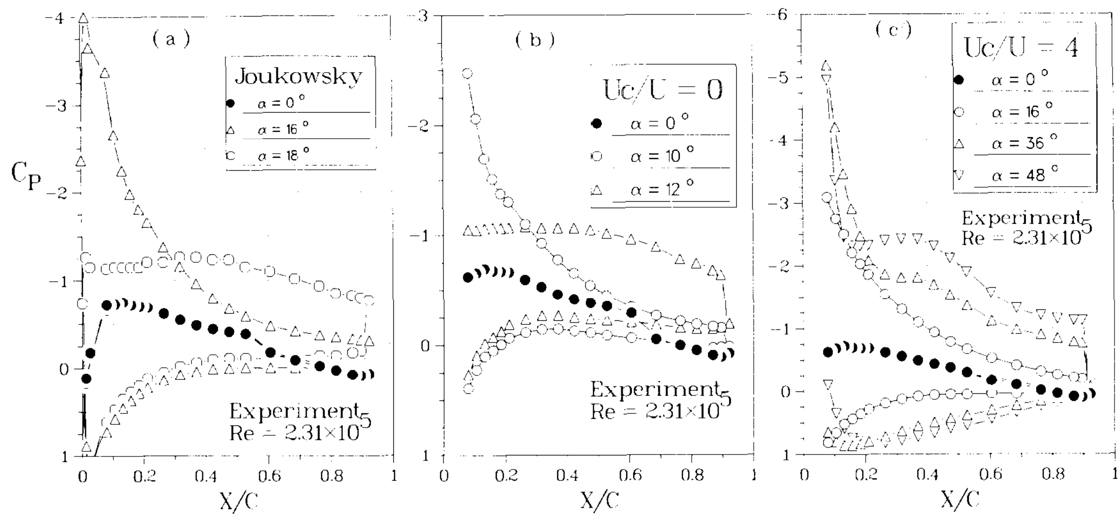

Fig. 1. Typical experimentally obtained pressure-distribution plots for Joukowsky airfoil: (a) unmodified airfoil; (b) effect of modification with the leading edge cylinder; (c) the effect of cylinder rotation.

the otherwise smooth surface. This not only reduces the negative presure peak, compared to that in fig. 1a, but also the stall angle which is now between 10°–12°. The discontinuity in the pressure plots near the leading edge is due to practical difficulty in measurement of pressure at the surface of the cylinder. However, since in the subcritical flow regime, the peak negative pressure on the surface of a circular cylinder occurs at around 70°, location of the first pressure tap (top and bottom surfaces) would come quite close to it.

Fig. 1c shows the effect of cylinder rotation on the pressure distribution and the onset of stall at very high angles of attack. In general, effect of the leading edge rotating cylinder is to increase the peak negative pressure, however, the relative increase is less at higher U_c/U. With an increase in cylinder surface velocity to free stream velocity ratio, the stall angle corresponding to complete separation (i.e. no reattachment) is delayed. Note, without rotation the separation (on the top surface) occurs at around 12° whereas with rotation a part of the surface always has an attached flow up to $x/c \approx 0.25$. Also with higher rates of rotation the onset of flow separation on the top surface occurs at a higher angle of attack and there is a tendency for the boundary layer to reattach. One would therefore expect the overall effect of the cylinder rotation would be to increase $C_{L,max}$ due to stall at much higher angles of attack and higher C_L/C_D at any given angle of attack.

A typical pressure distribution as predicted by the inviscid flow model is shown in fig. 2a. Slightly higher suction is predicted on the upper surface due to wall confinement. The correction scheme uses these results to calculate the displacement effect of the boundary layer as well as the point of separation before the final "corrected" pressure distribution can be obtained. The results are in very good agreement with experiment except near the trailing edge. This is mainly due to the nature of the boundary layer scheme and its simplifying assumptions. Particularly, the assumption that the pressure is constant across the shear layer is no longer valid after the shear layer has seaparated and is only an approximation when the size of the separation region near the trailing edge of the airfoil is small.

Fig. 2b shows a typical result at a large angle of attack with the boundary layer control scheme. The theoretical approach appears to be quite reasonable considering the very complicated nature of the flow. The location of the stagnation point on the lower surface and the separation point on the upper surface are predicted very accurately. At very high angles of

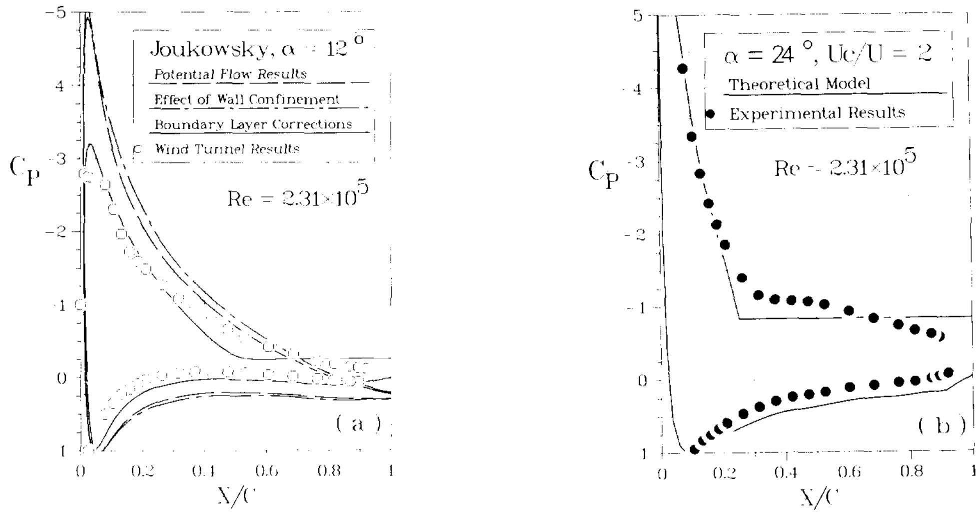

Fig. 2. Typical predicted pressure distributions on a Joukowsky airfoil and comparison with experiment: (a) unmodified airfoil; (b) airfoil modified with the leading edge rotating cylinder.

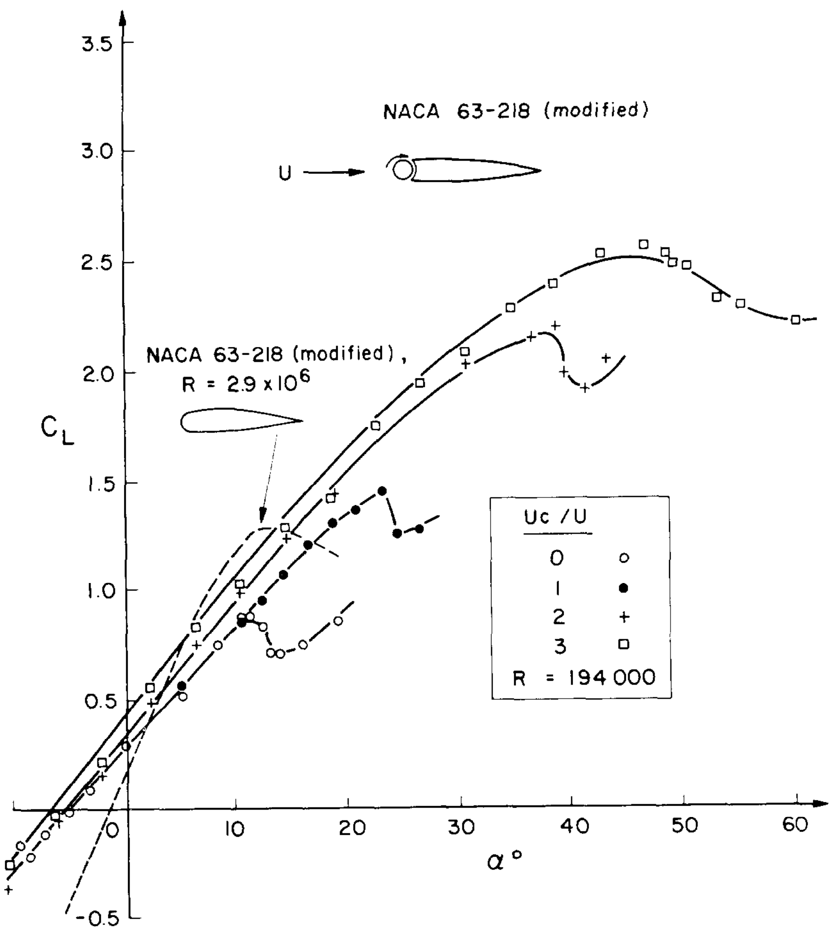

Fig. 3. Variation of the lift coefficient with angle of attack as affected by leading edge cylinder and cylinder rotation.

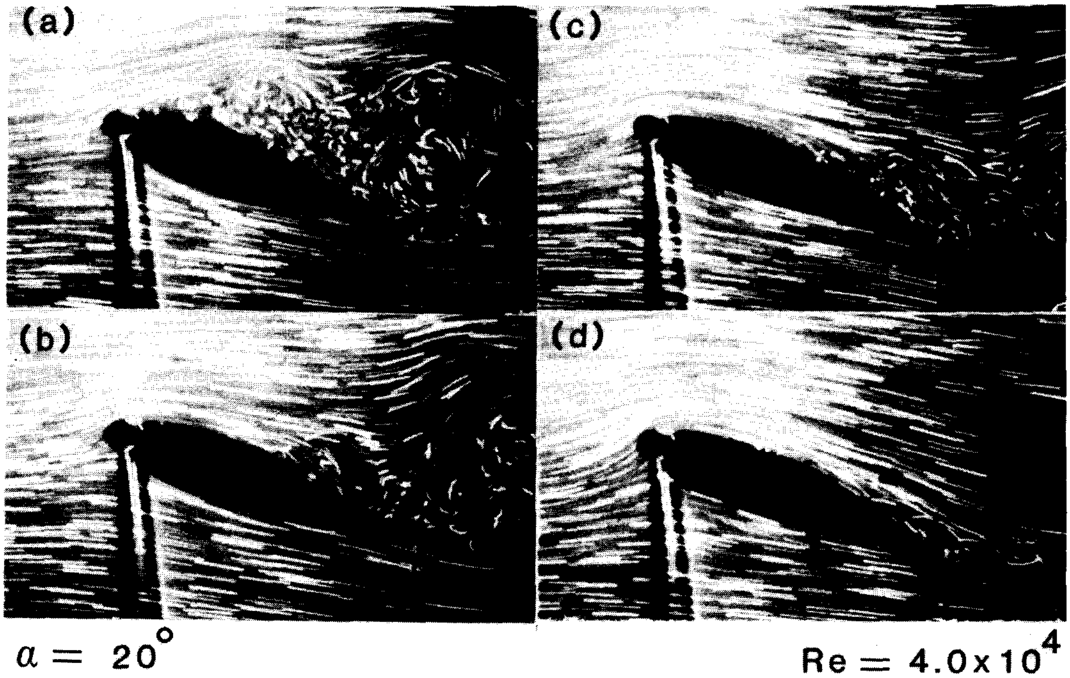

Fig. 4. Flow visualization photographs showing the transition from the highly separated flow in the absence of cylinder rotation to the completely reattached flow at $U_c/U = 6$.

attack, where the separation region near the trailing edge grows larger, more accurate results of the flow field would require the modelling of the separated flow region using the full Navier–Stokes equations.

Similar experimental data for a modified NACA 63-218 airfoil at different rates of rotation of the cylinder are summarized in the form of lift data in fig. 3. The basic airfoil (without cylinder) has a maximum lift coefficient of around 1.3. However, bluffness of the cylinder and the associated gap cause the slope of the lift curve as well as $C_{L,max}$ to diminish. With cylinder rotation, a large well developed suction peak at the leading edge of the wing suggested a delay in the stall. In fact, the data showed the stall to occur around 45° ($U_c/U = 3$) with an increase in the lift coefficient by about 200%. Note also that the effect of rotation is to extend the lift curve without affecting its slope, and flatten the stall peak.

The flow visualization study (fig. 4) dramatically shows the effectiveness of the moving surface boundary layer control. With the cylinder rotational speed increasing from $U_c/U = 0$ to $U_c/U = 6$, the highly separated flow at $\alpha = 20°$ reattaches almost completely.

Acknowledgements

The model was fabricated in the Mechanical Engineering workshop. Assistance of Mr. E. Abell, Senior Technician, in design and construction of the model is gratefully acknowledged.

Investigation reported here was supported by the Natural Sciences and Engineering Research Council of Canada, Grant No. A-2181.

References

[1] T. Cebeci and P. Bradshaw, Momentum Transfer in Boundary Layers (Hemisphere-McGraw Hill, Washington, 1977).
[2] J.L. Kennedy, The design and Analysis of Airfoil Sections, Ph.D. dissertation, The University of Alberta (1977).
[3] F. Mokhtarian and V.J. Modi, Fluid Dynamics of Airfoils with Moving Surface Boundary Layer Control, to be published: Journal of Aircraft (1988).