Analysis Method for Viscous Flow over Circulation-Controlled Airfoils

F.A. Dvorak*
Analytical Methods, Inc., Bellevue, Wash.
and
R.J. Kind†
Carleton University, Ottawa, Ontario, Canada

A method developed for the analysis of the incompressible viscous flow over circulation-controlled airfoils is described. A surface vorticity method is used to solve the inviscid portion of the flow, and a combination of integral and finite-difference methods is used to calculate the development of the viscous layers. An iterative process is used to arrive at final solutions which satisfy an appropriate trailing-edge condition and incorporate the interaction between the viscous and potential regions of the flow. Comparisons between calculated and experimental results show good agreement for surface pressure distributions and lift coefficients over a range of blowing momentum coefficients from 0 to 0.12.

Nomenclature

c = airfoil chord length
C_L = lift coefficient, based on airfoil chord length
C_p = static pressure coefficient
C_μ = blowing momentum coefficient $= J/\frac{1}{2}\rho U_\infty^2 c$
H = boundary-layer shape factor $= \delta^*/\theta$
J = blowing momentum flux per unit span
J_{ex} = excess momentum flux in wall jet per unit span
l = length of panels used to represent airfoil
N = number of panels used to represent airfoil
p = static pressure
q = surface source strength (volume efflux per unit surface area)
R = radius of curvature
R_θ = momentum thickness Reynolds number $= U\theta/\nu$
s = arc length along airfoil surface
u, v = flow velocity components in potential flow analysis
U = local flow velocity at outer edge of viscous layer
U_∞ = freestream velocity
γ = strength per unit area of surface vorticity distribution
Γ = circulation around airfoil
δ^* = boundary-layer displacement thickness
θ = boundary-layer momentum-defect thickness
ν = fluid viscosity
ρ = fluid density

Subscripts

sepl = separation of lower surface flow
sepu = separation of upper surface flow
slot = conditions at blowing slot
trans = transition

Introduction

A CIRCULATION-CONTROLLED airfoil is somewhat unusual in that its trailing edge is bluff instead of sharp. According to potential flow theory, the circulation around such sections can have any arbitrary value; that is, potential flow solutions, with finite velocity everywhere, exist for any value of the circulation. In viscous flow, on the other hand, separation occurs near the trailing edge, and the circulation assumes a definite value which depends on the separation positions of the upper- and lower-surface flows. To realize the high values of circulation and lift which are possible in potential flow over these sections, it is only necessary to use small amounts of blowing air to delay separation of the upper-surface flow. On circulation-controlled sections, air is injected tangentially from one or more slots located on the upper surface near the trailing edge. Figure 1 illustrates the flow over a circulation-controlled airfoil.

Circulation control can achieve very high ratios of lift coefficient to flowing momentum coefficient (C_L/C_μ). Ratios in excess of 50:1 have been demonstrated, together with good lift-to-effective-drag ratios.[1,2] Furthermore, the lift developed by a circulation-controlled section can be easily and rapidly changed simply by varying the supply of blowing air. The section shape facilitates the attainment of relatively high structural stiffness. These and other advantages are particularly attractive in the design of lifting rotors, and much work has been done with this application in mind.[3,4] The scheme is also of interest for fixed-wing aircraft, where it offers STOL performance and direct lift control (see, for example, Ref. 5).

This paper describes a method for predicting the pressure distribution, lift, drag, and pitching moment on circulation-controlled airfoils of arbitrary geometry in incompressible flow. The method has been programmed; the inputs required by the self-contained program are section geometry, slot location and thickness, angle of attack, Reynolds number based on chord length, and blowing momentum coefficient

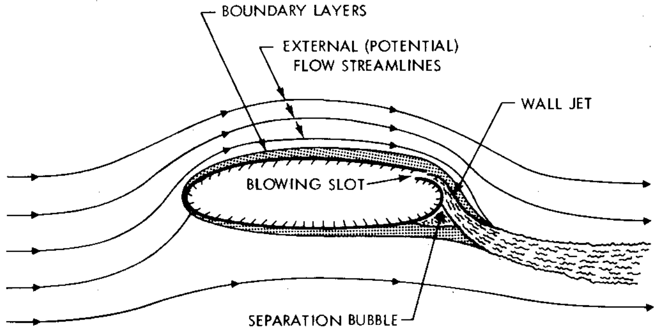

Fig. 1 Flow over a circulation-controlled airfoil.

Presented as Paper 78-157 at the AIAA 16th Aerospace Sciences Meeting, Huntsville, Ala., Jan. 16-18, 1978; submitted Feb. 13, 1978; revision received July 25, 1978.

Index categories: Aerodynamics; Jets, Wakes, and Viscid-Inviscid Flow Interactions.

*President.

†Associate Professor, Department of Mechanical and Aeronautical Engineering. Member AIAA.

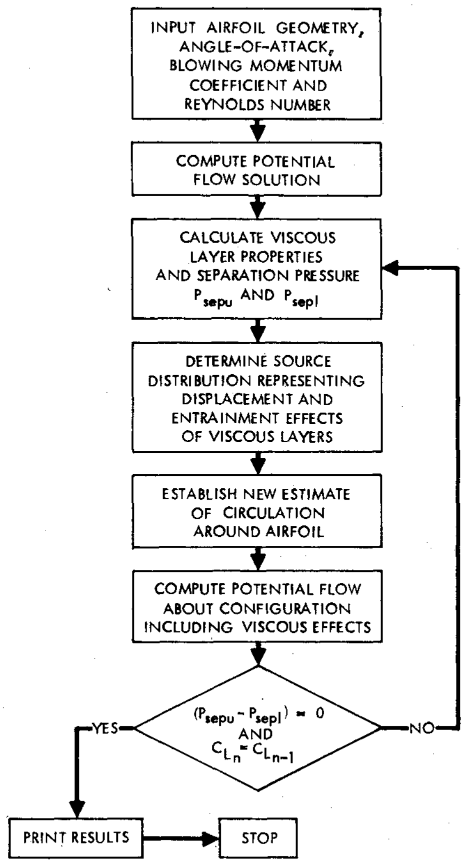

Fig. 2 Calculation procedure.

C_μ. A potential flow analysis is used to solve the flow outside the viscous layers. The development of the boundary layers on the upper and lower surfaces of the airfoil and the development of the combined boundary-layer/wall-jet flow downstream of the blowing slot are calculated. The potential flow analysis is then repeated, taking into account the displacement and entrainment effects due to the viscous layers. The entire process is repeated until a converged solution is obtained, usually after four to five iterations. Viscous effects are thus fully accounted for. The method provides a powerful tool for the optimization of circulation control sections for various applications.

Several methods for calculating the performance of circulation-controlled airfoils have been published previously.[6-9] The present method represents a considerable advance over these. None of the earlier methods takes into account the interaction between the viscous and the potential regions of the flow due to boundary-layer displacement and wall-jet entrainment effects. Allowance for these effects is particularly important to accurate prediction of drag on the airfoil. The methods of Refs. 6-8 use integral methods to calculate the flow downstream of the blowing slot; although fast, these methods cannot accurately represent the complex and rapidly changing velocity profile shapes which exist downstream of the slot where a previously developed boundary layer mixes with high-velocity blowing air. Accurate predictions of development and separation of this portion of the flow are crucial to accurate predictions of airfoil performance. Finite-difference methods are not restricted to an assumed family of velocity profile shapes and therefore have the greater flexibility required to accurately analyze the flow downsteam of the blowing slot. Gibbs and Ness[9] use a finite-difference method to analyze all of the viscous layers on the circulation-controlled airfoil. The present method, on the other hand, uses a finite-difference method only for the flow downstream of the slot on the upper surface; integral methods are used for the conventional boundary layers which develop on the lower surface and upstream of the slot on the upper surface. This choice was made because integral methods are about 100 times more economical of computer time and, for conventional boundary layers, just as accurate as finite-difference methods.[10] The present method also uses a potential flow analysis better suited to automatic computation than the Theodorsen method used by Gibbs and Ness. Computer time considerations are particularly important because of the iterative procedure required to account for viscous/potential flow interaction effects.

Description of the Analysis Method

General Description

The present method is in many respects similar to methods for the analysis of the viscous flow over conventional airfoil sections, that is, sections having a sharp trailing edge. In fact, the present computer program can treat such airfoils (with or without slot blowing) as a special case. The flow chart in Fig. 2 shows the sequence of the principal steps in the analysis. The first step is a calculation of the potential flow around the airfoil. Next, boundary-layer and wall-jet developments are computed using pressure distributions obtained from the potential flow analysis. Then the viscous interaction effects are modeled and the potential flow analysis is repeated, and so on, until a converged solution is arrived at.

For circulation-controlled airfoils, the potential flow analysis cannot make use of conventional forms of the Kutta condition because there is no sharp trailing edge of known position to delineate between upper-surface and lower-surface flows; in effect, one does not know, a priori, where the Kutta condition should be applied. To circumvent this difficulty, the Kutta condition is replaced by an equation which specifies the value of the total circulation around the airfoil; this value corresponds to an estimated value for the lift coefficient. The potential flow solution gives, among other things, the positions of the forward and aft stagnation points, and these serve as the divisions between the upper-surface flow and the lower-surface flow.

Starting at the forward stagnation point, the development of the boundary layer along the lower surface of the airfoil is calculated, and the pressure at which it separates, P_{sepl}, is determined. Starting again at the forward stagnation point, the development of the boundary layer along the upper surface is calculated up to the position of the blowing slot. At the slot, another flow development calculation is initiated using an initial velocity profile made up of the boundary-layer velocity profile and the known velocity distribution (essentially uniform) of the blowing air at the slot exit. This development calculation proceeds until flow separation pressure, P_{sepu}, is noted. All of the aforementioned flow-development calculations use static pressure distributions given by the previous potential flow analysis.

Measured static pressure distributions around circulation-controlled airfoils clearly show that the separation pressures of the upper- and lower-surface flows, P_{sepu} and P_{sepl}, are equal. This observation is physically reasonable because it merely implies that the static pressure is constant throughout the separation bubble shown in Fig. 1. Experiment[1] also indicates that there is a negligible static pressure difference across the wall jet at its separation position; the equality of P_{sepu} and P_{sepl} then implies that no net vorticity is being convected into the wake, which is consistent with existence of a fixed value of circulation and lift on the airfoil.[11] There is thus ample justification for assuming that the analysis has arrived at the correct solution for any particular set of conditions if P_{sepu} and P_{sepl} are equal for that solution. The goal of the iteration process thus is to find that solution which gives P_{sepu} equal to P_{sepl}.

Of course, the values of P_{sepu} and P_{sepl} calculated during the first pass through the analysis normally will not be equal to one another, and, even if they were, it would be necessary to repeat the analysis in order to incorporate the

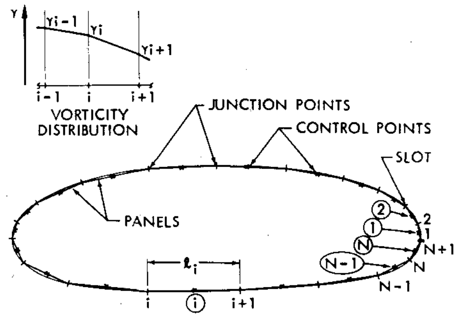

Fig. 3 Airfoil paneling scheme.

viscous/potential flow interaction effects. To proceed, then, a new value of the lift or circulation around the airfoil is estimated on the basis of the current values of lift and P_{sepu}-P_{sepl}. A surface source distribution is generated to model the displacement and entrainment effects due to the viscous layers. The source distribution is incorporated into the potential flow analysis, which is then repeated with the circulation specified at the new value. The viscous flow development calculations then are repeated using the pressure distributions from the potential flow analysis, and so on. The solution is deemed to have converged to the correct result when the lift coefficient remains essentially unchanged between iterations and at the same time P_{sepu}-P_{sepl} is essentially zero.

Description of the Major Elements of the Method

The following paragraphs describe, in outline form, the major elements in the analysis method. Full details can be found in the cited references.

Potential Flow Analysis

The airfoil is paneled as shown in Fig. 3. The potential flow over the airfoil is modeled by distributing vorticity along the panels. The strength γ of the vortex panels or sheets is made to vary linearly along each panel and is continuous at all junction points. Thus, if there are N panels on the airfoil, there are N unknown strengths, γ_I to γ_N. The boundary condition of zero net velocity normal to the panel surface is applied at the midpoint of every panel. This gives a set of N equations in N unknowns. In addition, it is necessary to introduce an equation to specify the value of the total circulation Γ around the airfoil. This equation takes the form

$$\sum_{i=I}^{N}\left[l_i\frac{(\gamma_i+\gamma_{i+I})}{2}\right]=\Gamma \qquad (1)$$

where l_i is the length of the ith panel. To avoid having a greater number of equations than unknowns, an additional unknown is provided by adding the influence of a uniform strength source distribution just inside the airfoil surface. It should be noted that the solution obtained for this unknown source strength is always very close to zero.

The equations are set up in matrix form with γ_I to γ_N and the uniform source strengths as unknowns. The influence coefficients of the unknowns are computed on the first pass through the analysis method, and their matrix is inverted. On subsequent passes through the analysis, a surface source distribution is introduced to model viscous/potential flow interaction effects. The strength variation of this source distribution is, however, determined by the results of the viscous flow analysis, so that it only introduces an additional known induced velocity on the right-hand side of each boundary-condition equation. The total circulation Γ also appears only on the right-hand side. Therefore, the matrix of influence coefficients is not altered, and only matrix multiplication is required to obtain the potential flow solution for the second and later iterations.

Pressure coefficients on the surface are calculated using

$$C_p = I - u^2 - v^2 \qquad (2)$$

where u and v are the nondimensional velocity components parallel and normal to the surface at the control points. Pressure coefficients are also calculated for a grid of off-body points in the trailing-edge region downstream of the blowing slot. The static pressure in this region varies rapidly along both tangents and normals to the airfoil surface, and the information is required for the finite-difference flow development calculation used downstream of the slot. To speed convergence of the iteration process, it has been found advantageous to use a weighted average of the newly computed and previous values of the pressure coefficients in the continuing calculations.

Lift and pitching moment coefficients are determined by integration of the static pressure distribution. The drag coefficient is determined after the viscous flow calculations by integration of the skin friction and static pressure distributions.

Before proceeding to the viscous flow calculations, the potential flow solution is searched for the forward and aft stagnation points. These are considered to be located where the vorticity γ changes sign. The static pressure data generated by the potential flow analysis are then rearranged into two arrays. Each of these begins at the junction point nearest to the forward stagnation point and runs downstream, one along the upper surface and the other along the lower surface of the airfoil. Each array is terminated two junction points past the position of the rear stagnation point. These arrays are used for the viscous flow calculations, which must, of course, begin essentially at the forward stagnation point for both the upper- and lower-surface flows.

The potential flow analysis method has been tested extensively in this and other applications, and it generally gives very good agreement with exact solutions. A detailed description of the basic method is available in Ref. 12.

Boundary-Layer Development Calculation along Lower Surface

The boundary-layer development calculation begins essentially at the forward stagnation point with the Heimenz stagnation flow solution.[13] Curle's[14] extension of Thwaites' method is used for calculation of the laminar boundary-layer development downstream of the stagnation point. The calculation proceeds until either transition or laminar separation is indicated.

The procedure of Granville[15] is used to predict transition. In some cases, the pressure gradient is sufficiently adverse to cause separation of the laminar boundary layer prior to transition. Usually the flow will then quickly become turbulent and reattach as a turbulent boundary layer. From measurements of Gaster[16] and others, a correlation is formed which can predict the occurrence of both separation with subsequent reattachment and separation without reattachment (see Ref. 12 for details). The turbulent boundary-layer calculation is begun using the final laminar value as the initial turbulent momentum thickness. The initial turbulent value of H is determined from the following empirical relation developed from data obtained by Coles[17]:

$$H=\frac{1.4754}{\log_{10}R_{trans}}+0.9698 \qquad (3)$$

The turbulent boundary-layer development is calculated using the Nash-Hicks[18] method. This integral method has been found to be as good as the best finite-difference methods in predicting the development of conventional boundary layers in a wide variety of pressure distributions. The turbulent boundary-layer calculation is continued until separation occurs, at which time its position and pressure, P_{sepl}, are noted.

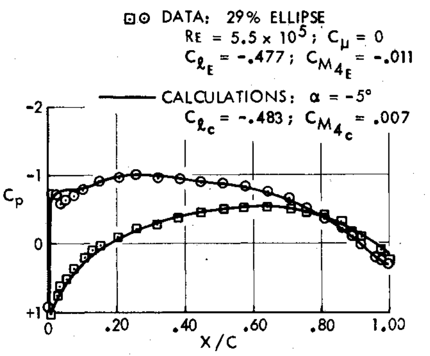

Fig. 4 Comparison between calculated and measured pressure distributions for a 29% ellipse, $C_\mu = 0$.

Viscous Flow Development Calculation along Upper Surface

This calculation begins at the forward stagnation point and proceeds along the upper surface to the blowing slot position using the same subroutines as the lower-surface boundary-layer calculations. Normally, separation without reattachment does not occur upstream of the blowing slot, but, if it does, the calculated separation position and pressure, P_{sepu}, are stored, and the iteration process is continued in the usual way. Normally, the flow at the slot is attached, and it is necessary to calculate the development of the rather complex viscous flow downstream of the slot. Here, the air from the previously developed boundary layer begins mixing with the high-velocity jet issuing from the slot. The flow proceeds around a highly curved surface in very strong pressure gradients and eventually separates at some pressure P_{sepu}.

Accurate predictions of the static pressure rise sustainable by the upper-surface flow are crucial to the overall accuracy of the analysis method. The physical function of the blowing air on a circulation-controlled airfoil is to enable the upper-surface flow to negotiate the static pressure rise between the aft suction peak and the final separation pressure. This separation pressure coefficient is approximately constant (equal to $C_{p_{sepl}}$), but the suction peak rises as the circulation around the airfoil increases. Thus stronger blowing is required to sustain the increased static pressure rise associated with increased circulation, and the circulation is directly and strongly dependent on the pressure rise that can be sustained.

A wide variety of velocity profile shapes can occur between the slot and separation; the profile shape also tends to change quite rapidly. Integral methods are not well suited to dealing with such complex and varied velocity profile shapes. The present analysis method accordingly utilizes a finite-difference method to calculate development of the viscous flow downstream of the slot. The method, described in detail in Ref. 19, uses an eddy viscosity model for the Reynolds shear stress behavior. It has the necessary attributes of being both reasonably fast and accurate for the type of flow in question.

The finite-difference calculation is started at the slot with an initial velocity profile consisting of the final profile from the Nash-Hicks boundary-layer calculation, under which is placed a nearly uniform velocity profile to represent the blowing air issuing from the slot. The slot air is assumed to have the same density as the ambient air, and its velocity is determined such that it has the correct momentum flux assuming isentropic expansion to local static pressure at the slot exit.

The effects of the strong convex curvature downstream of the blowing slot must be accounted for in the finite-difference calculations. The equations of motion are formulated with the required curvature terms and are solved for static pressure fields, which include the radial variation of static pressure due to flow curvature. The potential flow analysis gives the radial variation of static pressure, where the flow stagnation pressure is approximately equal to the freestream. When the

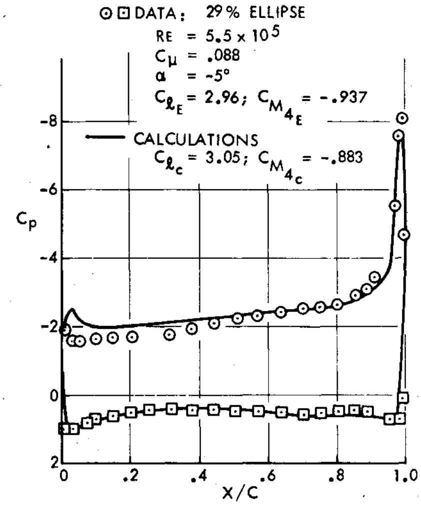

Fig. 5 Comparison between calculated and measured pressure distributions for a 29% ellipse, $C_\mu = 0.088$.

freestream value is exceeded, an additional variation is superimposed; this is assumed to vary linearly from $-(J_{ex}/R)$ at the solid surface to zero at the minimum in the velocity profile. This superimposed variation represents the extra radial pressure difference required to balance the excess momentum flux J_{ex} in the wall-jet portion of the flow; the assumption of linearity is approximately justified on the basis of Ref. 20. The convex curvature also has a pronounced effect on the turbulence structure because the flow is radially unstable where its velocity decreases with increasing radial distance from the surface. The eddy viscosity model includes factors to account for this.

Surface curvature information is supplied to the calculation by fitting a spline under tension[21] to the airfoil coordinate points. The splined curve is differentiated analytically at each coordinate position, and a second spline is fitted to the first derivative values. Differentiation of the second spline yields good values for the radius of surface curvature.

Modeling Viscous/Potential Flow Interaction Effects

As already mentioned, surface source distributions are used to model viscous/potential flow interaction effects. This approach has two major advantages over the addition of a displacement thickness to the original airfoil geometry. First, as pointed out earlier, the matrix of aerodynamic influence coefficient remains the same for all iterations, with attendant savings in computer time. Second, as pointed out in Ref. 22, the addition of displacement thicknesses tends to produce peculiar surface geometries, especially in the trailing-edge region, with attendant difficulties in obtaining correct values for lift and pressure distribution; such difficulties are completely avoided when surface source distributions are used.

The source distribution is placed on the planar panels, which are used to represent the airfoil. The source strength q varies linearly between junction points and is continuous at the junction points. At any junction point, it is given the value

$$q_i = \frac{d}{ds}(U\delta^*) \tag{4}$$

where s denotes arc length, U is the local external (inviscid) flow velocity, and δ^* is the local boundary-layer displacement thickness, determined by the viscous flow analysis. Equation (4) follows from the fact that boundary-layer displacement effects will be modeled correctly if the total mass flux emitted by all upstream surface sources equals the local mass flow deficit in the boundary layer. No special methods are needed to deal with the entrainment effects due to the wall jet downstream of the blowing slot; the product $U\delta^*$ tends to

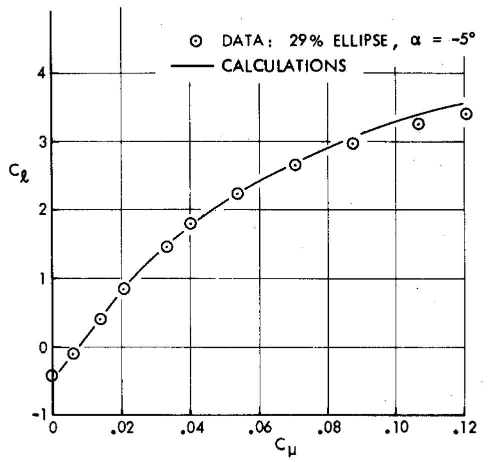

Fig. 6 Comparison between calculated and measured lift coefficients for a range of momentum coefficients.

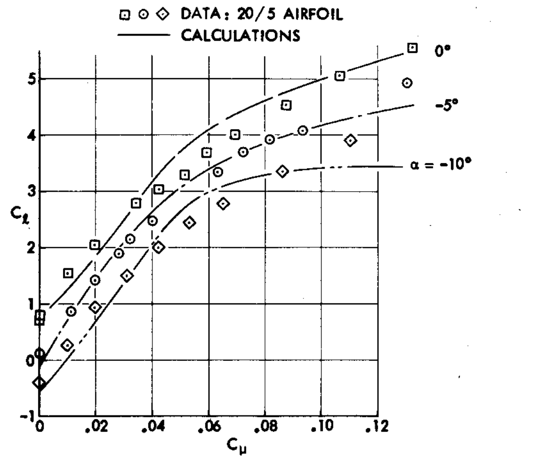

Fig. 8 Comparison between calculated and measured lift coefficients for a range of momentum coefficients.

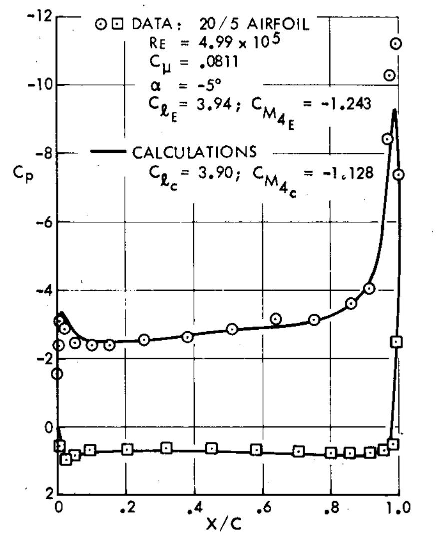

Fig. 7 Comparison between calculated and measured pressure distributions for a 20% cambered ellipse, $C_\mu = 0.0811$.

decrease here, and the source strengths then simply become negative (i.e., sinks).

Iteration Scheme

The main features of the iteration scheme have already been outlined. The analysis begins with a rough estimate of the circulation. The second and subsequent estimates of circulation are obtained using

$$\Gamma_{n+1} = \Gamma_n + k(C_{P_{\mathrm{sepu}}} - C_{P_{\mathrm{sepl}}}) \tag{5}$$

where the numerical constant k has a value of $0.1 \rightarrow 0.3$. This relation follows from two assumptions:

1) At fixed angle of attack, an increase in circulation or in C_L corresponds to an approximately uniform increase in suction on the upper surface and little pressure change on the lower surface.

2) For any particular value of C_μ, an approximately fixed pressure rise, $C_{P_{\mathrm{slot}}}$-$C_{P_{\mathrm{sepu}}}$, can be sustained by the flow downstream of the blowing slot.

If the estimated circulation is too low relative to the correct value, a problem tends to arise. That is, the static pressure rise seen by the flow development calculation downstream of the slot is too low, and separation from the surface is not necessarily predicted. The calculation then tends to predict reverse flow away from the wall in the boundary between the wall jet and the remnant of the upstream boundary layer. In such cases, the iteration process is continued using an estimate for the separation position and pressure based on an extrapolation to zero skin-friction coefficient from the point of midstream reversal. The value of $C_{P_{\mathrm{sepu}}}$ given by the extrapolation will generally be greater than $C_{P_{\mathrm{sepl}}}$, and it is only used for obtaining the next estimate for the circulation Γ. Such external flow separation is a real but uncommon possibility on circulation-controlled airfoils. When it occurs in the calculations, it is usually only because the current estimate of circulation around the airfoil is too low.

The calculations are not allowed to stop unless at least two iterations have been completed. This insures that the viscous/potential flow interaction effects are accounted for in the final results. The calculations are terminated when the lift coefficient remains essentially unchanged between iterations and at the same time $C_{P_{\mathrm{sepu}}}$-$C_{P_{\mathrm{sepl}}}$ is essentially zero.

Discussion of Results

The calculation method has been applied to the analysis of circulation-controlled airfoils of interest to the U.S. Navy. The case of zero momentum coefficient is of special interest in the event of blowing power loss. An example of this is shown in Fig. 4, where measured and calculated pressure distributions are compared for a 29% ellipse[23] having a circular trailing edge. The agreement for the zero blowing case is excellent and demonstrates a capability not heretofore published in the literature.

Further calculations were made for the 29% ellipse over a range of momentum coefficients. Calculated and measured pressure distributions for a momentum coefficient of 0.088 are compared in Fig. 5. The calculated and measured lift coefficient characteristics of this airfoil for a range of momentum coefficients at an angle of attack of -5 deg are given on Fig. 6. In general, the results for this airfoil are very encouraging.

Additional computations were made for comparison with experimental data obtained on a cambered 20% ellipse,[2] and these are shown on Figs. 7 and 8. The calculated and experimental pressure distributions for a momentum coefficient of 0.0811 are in good agreement. The lift-momentum coefficient characteristics of the cambered ellipse for an angle-of-attack range from 0 to -10 deg are reproduced quite well, as indicated on Fig. 8.

Conclusions

The initial results of comparison with experiment are very encouraging. The method was successfully applied over a wide range of momentum coefficients, including the zero blowing case. A current lack of measured shear stress distributions has precluded the correlation between theory and experiment, and the question remains, is the shear stress closure model adequate or should improved or alternative models be considered? Airfoils with trailing edges other than circular also need to be investigated.

Acknowledgments

This work was supported by the U.S. Navy under Contract N00600-76-C-1494. This support is gratefully acknowledged.

References

[1] Kind, R.J. and Maull, D.J., "An Experimental Investigation of a Low-Speed Circulation-Controlled Aerofoil," *Aeronautical Quarterly*, Vol. XIX, May 1968, pp. 170-182.

[2] Williams, R.M. and Howe, H.J., "Two-Dimensional Subsonic Wind Tunnel Tests on a 20 Percent Thick, 5 Percent Cambered Circulation-Control Airfoil," NSRDC, Washington, D.C., TN AL-176, Aug. 1970.

[3] Dunham, J., "Experiments Towards a Circulation-Controlled Lifting Rotor," *Aeronautical Journal*, Vol. 74, Jan.-Feb. 1970, Pt. I, pp. 91-103, Pt. II, pp. 175-186.

[4] Williams, R.M., "Some Research on Rotor Circulation Control," *CAL/AVLABS Symposium: Aerodynamics of Rotary Wing and V/STOL Aircraft, 3rd Proceedings*, Vol. 2, June 1969.

[5] Englar, R.J., Trobaugh, L.A., and Hemmerly, R.A., "STOL Potential of the Circulation-Control Wing for High-Performance Aircraft," *Journal of Aircraft*, Vol. 15, March 1978, pp. 175-181.

[6] Dunham, J., "A Theory of Circulation-Control by Slot-Blowing Applied to a Circular Cylinder," *Journal of Fluid Mechanics*, Vol. 33, July 1968, pp. 495-514.

[7] Kind, R.J., "A Calculation Method for Circulation Control by Blowing Around a Bluff Trailing Edge," *Aeronautical Quarterly*, Vol. XIX, Aug. 1968, pp. 205-223.

[8] Kind, R.J., "A Calculation Method for Boundary Layer Control by Tangential Blowing," *Canadian Aeronautics and Space Institute Transactions*, Vol. 4, Sept. 1971, pp. 88-98.

[9] Gibbs, E.H., and Ness, N., "Analysis of Circulation-Controlled Airfoils," *Journal of Aircraft*, Vol. 13, Feb. 1976, pp. 158-160.

[10] Kline, S.J., Morkovin, M.V., Sorvan, G., and Cockrell, D.J. (eds.), *Proceedings of Computation of Turbulent Boundary Layers, AFOSR-IFP-Stanford Conference*, 1968.

[11] Thwaites, B. (ed.), *Incompressible Aerodynamics*, Oxford University Press, Oxford, England, pp. 198-200.

[12] Dvorak, F.A. and Woodward, F.A., "A Viscous/Potential Flow Interaction Method for Multi-Element Infinite Swept Wings," NASA CR-2476, Nov. 1974.

[13] Schlichting, H., *Boundary Layer Theory*, 4th Ed., McGraw-Hill, New York, 1960, pp. 147-151.

[14] Curle, H., "A Two-Parameter Method for Calculating the Two-Dimensional Incompressible Laminar Boundary Layer," *Journal of the Aeronautical Society*, Vol. 71, Feb. 1967.

[15] Granville, P.S., "The Calculation of Viscous Drag of Bodies of Revolution," David Taylor Model Basin, Rept. 849, 1953.

[16] Gaster, M., "The Structure and Behavior of Laminar Separation Bubbles," Aeronautical Research Council, ARC 28-226, 1967.

[17] Coles, D.E., "Measurements in the Boundary Layer on a Smooth Flat Plate in Supersonic Flow," Jet Propulsion Lab., Rept. 20-69, 1953.

[18] Nash, J.F. and Hicks, J.G., "An Integrated Method Including the Effect of Upstream History on the Turbulent Shear Stress," *Proceedings of Computation of Turbulent Boundary Layers, AFOSR-IFP-Stanford Conference*, Vol. 1, 1968.

[19] Dvorak, F.A., "Calculation of Turbulent Boundary Layers and Wall Jets over Curved Surface," *AIAA Journal*, Vol. 11, April 1973, pp. 517-524.

[20] Kind, R.J., "Calculation of the Normal Stress Distribution in a Curved Wall Jet," *Aeronautical Journal*, Vol. 75, May 1971, pp. 343-348.

[21] Cline, A., "Splines Under Tension," National Center for Atmospheric Research, Boulder, Colo. 1973.

[22] Seebohm, T. and Newman, B.G., "A Numerical Method for Calculating Viscous Flow Round Multi-Section Aerofoils," *Aeronautical Quarterly*, Vol. 26, Aug. 1975, pp. 176-188.

[23] Abramson, J., "Two-Dimensional Subsonic Wind Tunnel Evaluation of a 29% Thick Circulation Control Airfoil," Aviation and Surface Effects Department Tech. Memo. 16-76-42, Dec. 1975.