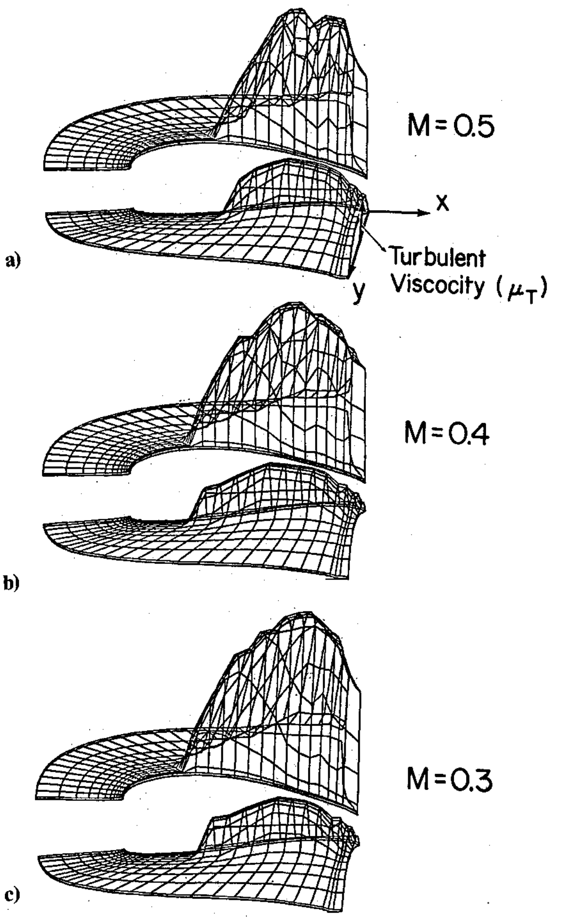

Fig. 2 Three-dimensional plot of turbulent viscosity; $Re = 100{,}000$, angle of attack = 7 deg.

the regions of turbulence in these low Reynolds number flows and illustrate no unexpected trends.

Conclusion

In summary, the present Note describes the first work to assess, albeit in an elementary sense, how low Reynolds numbers may affect the standard compressibility corrections for airfoil properties. Our purpose here is to indicate that such low Reynolds number effects may be important. Moreover, it appears that the highly viscous effects result in a relatively constant C_L, at least up to $M_\infty = 0.5$.

Acknowledgments

Acknowledgment is given to the Office of Naval Research, which supported this work under Grant N00014-85-K-0246, with Robert Whitehead, T.C. Tai, and Spiro Lekoudis as monitors.

References

[1]Mueller, T.J. and Batill, S.M., "Experimental Studies of the Laminar Separation Bubble on a Two-Dimensional Airfoil at Low Reynolds Numbers," AIAA Paper 80-1440, July 1980.

[2]Arena, A.V. and Mueller, T.J., "Laminar Separation, Transition, and Turbulent Reattachment Near the Leading Edge of Airfoils," *AIAA Journal,* Vol. 18, July 1980, pp. 747–753.

[3]Pohlen, L.J. and Mueller, T.J., "Boundary Layer Characteristics of the Miley Airfoil at Low Reynolds Numbers," *Journal of Aircraft,* Vol. 21, Sept. 1984, pp. 658–664.

[4]Marchman, J.F. III and Abtahi, A.A., "Aerodynamics of an Aspect Ratio 8 Wing at Low Reynolds Numbers," *Journal of Aircraft,* Vol. 22, July 1985, pp. 628–634.

[5]*Proceedings of the Conference on Low Reynolds Number Airfoil Aerodynamics,* edited by Thomas Mueller, Notre Dame University, Notre Dame, IN, June 17–18, 1985.

[6]Kothari, P. and Anderson, J.D. Jr., "Flows Over Low Reynolds Number Airfoils—Compressible Navier-Stokes Numerical Solutions," AIAA Paper 85-0107, Jan. 1985.

[7]MacCormack, R.W., "A Numerical Method for Solving the Equations of Compressible Flow," AIAA Paper 81-0110, 1981.

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

[9]Anderson, J.D. Jr., *Fundamentals of Aerodynamics,* McGraw-Hill, New York, 1984.

[10]Anderson, J.D. Jr., *Modern Compressible Flow: With Historical Perspective,* McGraw-Hill, New York, 1982.

[11]Bastedo, W.G. and Mueller, T.J., "Spanwise Variation of Laminar Separation Bubbles on Wings at Low Reynolds Numbers," *Journal of Aircraft,* Vol. 23, Sept. 1986, pp. 687–694.

Application of a Panel Code to Unsteady Wing-Propeller Interference

A.A. Rangwalla* and L.N. Wilson†
Iowa State University, Ames, Iowa

Introduction

THE present work deals with the prediction of unsteady, incompressible, potential flow using a low-order paneling method. The method used is an extension of VSAERO, developed by Maskew.[1]

Paneling methods have been used by others to predict unsteady lifting flows. A short representative review of the methods and problems solved are given in Refs. 2–10. One of the first attempts at predicting three-dimensional, unsteady lifting flows is reported in Ref. 3. Here, the complete nonlinear problem was solved; that is, the small perturbation approximation was not made. However, no examples of unsteady, three-dimensional wake structures were presented. In Ref. 9, a free wake analysis of helicopter rotors in incompressible unsteady potential flows was presented. The methodology presented in Ref. 9 is very similar to the present one. More references by the same author can be found there.

The present approach is a nonlinear one in that the small perturbation approximation is not made. This approach has the advantage of being able to solve quite general problems such as the one that was the primary objective of this development, the unsteady interference between a two-bladed propeller and a wing. To the authors' knowledge, this problem has not been solved before.

Formulation of the General Problem

For the formulation of the problem and the calculation of the subsequent wake development, the reader is referred to Refs. 1 and 11, where greater detail is provided. It is sufficient to state that the numerical procedure consisted of solving the Laplace equation at every time step. The wing and propeller were impulsively started. The position of the propeller with respect to the wing was changed between time steps. The wake emanated from the trailing edges of the wing and propeller. This wake was convected downstream with the local velocity calculated at the corner points of the wake panels.

Received Oct. 14, 1986; revision received Jan. 31, 1987.

*Graduate Assistant; presently, Research Associate.

†Professor and Chairman, Aerospace Engineering Department. Associate Fellow AIAA.

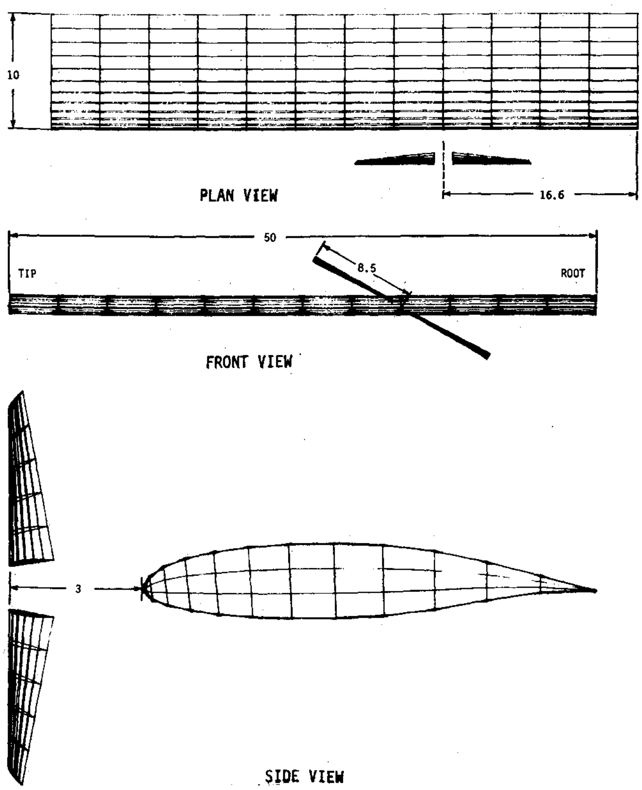

Fig. 1 Three views of the paneled wing and propeller configuration.

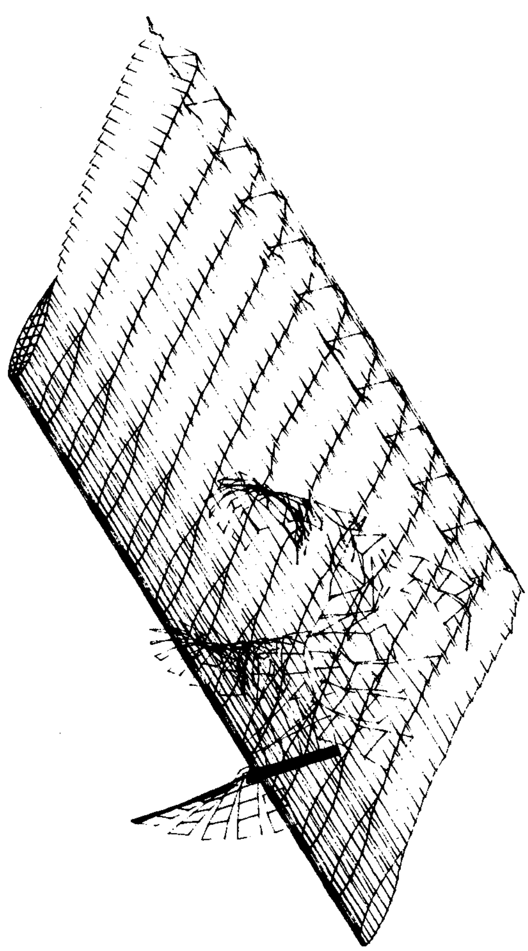

Fig. 2 Configuration after time step 22.

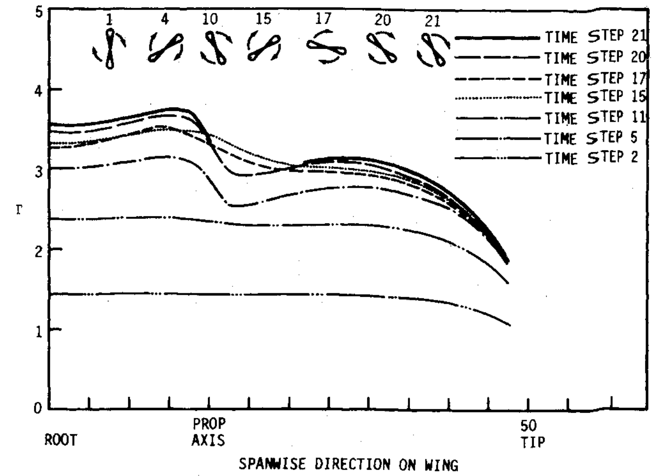

Fig. 3 Development of spanwise circulation on the wing.

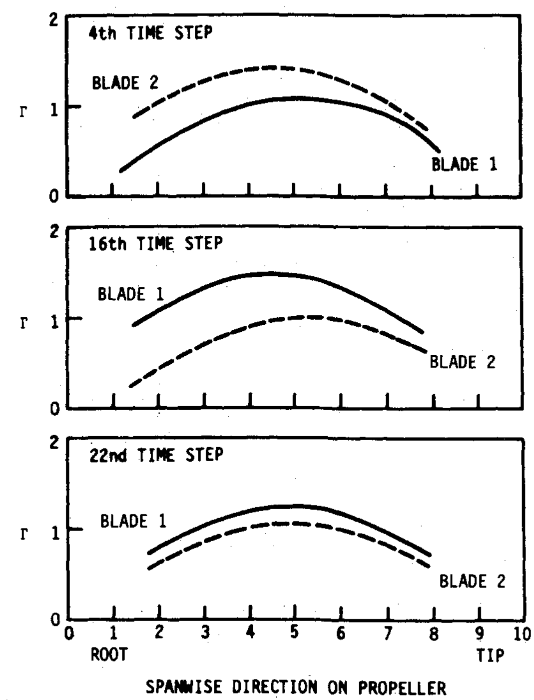

Fig. 4 Development of spanwise circulation on the propeller blades.

Stability of the Wake

A finite vortex wake is inherently unstable and rolls up at its ends. Hence, when attempting to track the wake numerically, care must be taken not to introduce any spurious instabilities. This was achieved by using the method given by Chorin and Bernard.[12] Here, a cutoff radius was chosen for each vortex line such that the velocity predicted by the vortex outside this radius was the correct value but was bounded inside. More details are given in Ref. 11.

Wake Snipping

It was mentioned that the corner points of all wake panels are convected at their calculated velocities. However, in multibody cases like a propeller and wing or a wing and tail combination, it is likely that this would result in wake panels intersecting a downstream solid body. Since the wake panels are actually ring vortices, in a real flow viscous effects would diffuse the vorticity in the vortex filament as it approached close to a solid body. This results in a dissipation or loss of

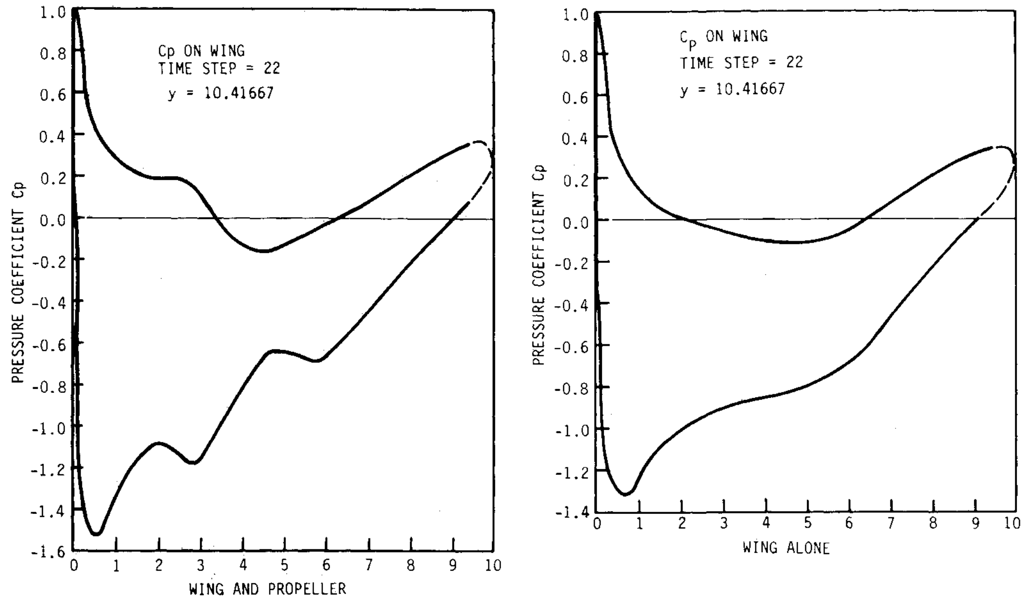

Fig. 5 Development of pressure distribution on the wing.

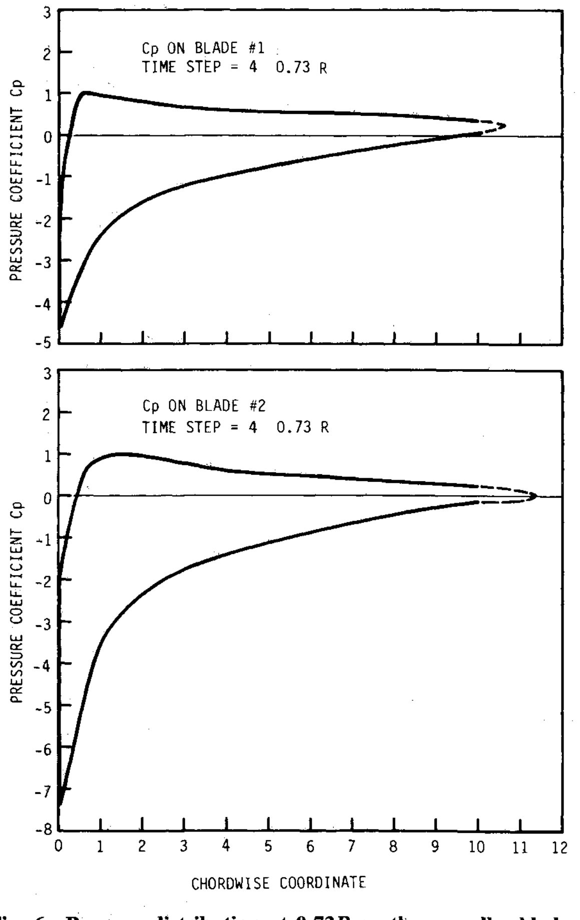

Fig. 6 Pressure distribution at 0.73*R* on the propeller blades.

the filament. Since one of the objects of a numerical code is to simulate as closely as possible the physical phenomena it is trying to model, a "wake snipping" procedure was introduced to allow for the loss of such filaments. This was achieved in a somewhat crude manner by dropping from the calculation any wake panel that intersected a solid body. More refined techniques could have been used.

Results

The method was first used to generate flow over a high aspect ratio rectangular wing starting impulsively from rest. This was done to validate the code, and the results are provided in Ref. 11.

Figure 1 shows three different views of the configuration. The wing has a GAW-1 section. There is a plane of symmetry at the root of the wing. The airfoil section of the propeller is the Clark-Y section of 11% thickness with its chord varying linearly from 1.0 near the hub to 0.6 at the tip. The propeller blade is twisted to give a sectional angle of attack of about 15 deg at 73% radius of the blade, with respect to the relative velocity for the downward-moving blade.

The whole configuration was impulsively started. The angular velocity of the propeller was 0.2353 rad/s. This results in an advance ratio equal to approximately 1.57.

This problem has some interesting aerodynamic features. Unlike some other studies (see Refs. 13–15) where the steady-state problem was solved, we have an unsteady interference between the propeller and the wing. The whole configuration is at an angle of attack; hence, the rotating propeller blades will experience a periodic loading. Also, whether the propeller is heavily or lightly loaded makes no difference in the computational approach. Recently, Kroo[16] carried out a quasisteady analysis of propeller-wing combinations in incompressible inviscid flow. The optimal, spanwise, time-averaged loading on the wing was obtained and gave results quite consistent with those obtained in this work.

Figure 2 shows the configuration after many time steps. Here the wing and propeller wake structure are shown. The spanwise distribution of circulation on the wing (Fig. 3) shows two effects: that of the impulsive start and that of the propeller. The unsteadiness due to the impulsive start results in a monotonic increase in the circulation at every spanwise station. The interference due to the propeller shows two characteristics. One is the interference due to the swirl, which creates an antisymmetric distribution centered about the spanwise station directly behind the propeller. The other effect is due to the displacement velocity, which causes a symmetric distribution. For this particular case, the displacement effect dominated when the propeller was nearly horizontal whereas the swirl effect dominated when the propeller was nearly vertical.

Figure 4 shows the spanwise circulation on the propeller blades. It is found, as expected, that the downward-moving blade generates more circulation because the local sectional angle of attack is larger than that for the upward-moving blade. The pressure distribution on the wing at one spanwise location with and without the propeller (Fig. 5) clearly shows the interference effects on the pressure.

The pressure coefficient on a particular spanwise location on the propeller blades is shown in Fig. 6. This has been normalized with respect to the local relative velocity of that section. The angle of attack of this section with respect to the relative velocity is about 15 deg.

Conclusions

A piecewise constant singularity panel method has been extended to solve for a large class of unsteady, incompressible potential flows in three dimensions. The method was used to solve the unsteady interfering flowfield between a propeller and a wing. Due to the geometry and flowfield, a wake snipping procedure had to be introduced.

The present approach provides an efficient method for calculating the unsteady potential flow about quite complicated configurations. The efficient low-order method used makes an iterative solution practical.

Acknowledgments

This work was carried out under the support of the Engineering Research Institute of Iowa State University and NASA Grant 422-20-12-1432 from the NASA Ames Research Center. The authors wish to acknowledge Prof. J.M. Vogel for his helpful discussions.

References

[1]Maskew, B., "Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods," *Journal of Aircraft,* Vol. 19, Feb. 1982, pp. 157-163.

[2]Giesing, J.P., "Non-Linear Two Dimensional Unsteady Potential Flow with Lift," *Journal of Aircraft,* Vol. 5, March-April 1968, pp. 135-143.

[3]Djojodihardjo, R.H. and Widnall, S.E., "A Numerical Method for the Calculation of Nonlinear, Unsteady Lifting Potential Flow Problems," *AIAA Journal,* Vol. 7, Oct. 1969, pp. 2001-2009.

[4]Kalman, T.P., Rodden, W.P., and Giesing, J.P., "Application of the Doublet-Lattice Method to Nonplanar Configurations in Subsonic Flow," *Journal of Aircraft,* Vol. 8, June 1971, pp. 406-413.

[5]Morino, L. and Kuo, C.-C., "Subsonic Potential Aerodynamics for Complex Configurations: A General Theory," *AIAA Journal,* Vol. 12, Feb. 1974, pp. 191-197.

[6]Belotserkovski, S.M., "Study of the Unsteady Aerodynamics of Lifting Surfaces Using the Computer," *Annual Review of Fluid Mechanics,* Vol. 9, 1977, pp. 469-494.

[7]Basu, B.C. and Hancock, G.J., "The Unsteady Motion of a Two Dimensional Aerofoil in Incompressible Inviscid Flow," *Journal of Fluid Mechanics,* Vol. 87, July 1978, pp. 159-178.

[8]Katz, J., "A Discrete Vortex Method for the Nonsteady Separation Flow Over an Airfoil," *Journal of Fluid Mechanics,* Vol. 102, Jan. 1981, pp. 315-328.

[9]Morino, L., Kaprielian, Z. Jr., and Sipcic, S., "Free Wake Analysis of Helicopter Rotors," *Vertica,* Vol. 9, Feb. 1985, pp. 127-140.

[10]Kim, M.J. and Mook, D.T., "Application of Continuous Vorticity Panels to General Unsteady Two Dimensional Lifting Flows," AIAA Paper 85-0282, Jan. 1985.

[11]Rangwalla, A.A., "Application of a Potential Code to General Unsteady Problems in Three Dimensions," Ph.D. Thesis, Iowa State University, Ames, IA, 1986.

[12]Chorin, A.J. and Bernard, P.S., "Discretization of a Vortex Sheet, with an Example of Roll-up," *Journal of Computational Physics,* Vol. 13, Nov. 1973, pp. 423-429.

[13]Hess, J.L. and Valarezo, W.O., "Calculation of Steady Flow About Propellers by Means of a Surface Panel Method," AIAA Paper 85-0283, Jan. 1985.

[14]Nishida, M., "The Effect of Displacement Velocity on Propeller Performance," *Journal of Aircraft,* Vol. 20, Aug. 1983, pp. 734-736.

[15]Kobayakawa, M. and Onuma, H., "Propeller Aerodynamic Performance by Vortex-Lattice Method," *Journal of Aircraft,* Vol. 22, Aug. 1985, pp. 649-654.

[16]Kroo, I., "Propeller-Wing Intersection for Minimum Induced Loss," *Journal of Aircraft,* Vol. 23, July 1986, pp. 561-565.

Lift of an Airfoil with a Jet Issuing from Its Surface

D. A. Tavella*
Stanford University, Stanford, California,
and
K. Karamcheti†
Florida State University, Tallahassee, Florida

THE lift induced on a two-dimensional wing due to a thin jet emerging from the airfoil's lower surface is estimated by a simple inviscid model. When a thin jet sheet issues from the surface of a wing exposed to a freestream, the jet becomes part of the boundary of a separated flow region. In this analysis, such a separated flow domain is idealized as a dead-water region with constant pressure, bounded by the jet, part of the airfoil surface, and a separation streamline originating at the trailing edge of the wing, as illustrated in Fig. 1. In this model, the effect of the jet is to cause the appearance of a separation cavity under the wing. The cavity is assumed to have a constant pressure equal to the pressure of the freestream. In this case, the cavity must be open and semi-infinite.[1] A similar concept with ground plane interaction was studied by Agarwal and Deese[2] by numerical solution of the Euler equations. Measurements for weak blowing intensities were reported by Krothapali et al.[3] A three-dimensional version of this problem was studied experimentally by Tavella et al.[4]

The mathematical task is a boundary-value problem where the location of part of the boundary is not known a priori. It can be stated as

$$\nabla^2\Phi=0 \text{ in } \Omega \tag{1}$$

$$\nabla\Phi\cdot n=-U_\infty \text{ on } c_1,\ c_2,\ c_3 \tag{2}$$

$$\nabla\Phi=0 \text{ on } c_1 \tag{3}$$

$$\frac{(\Delta\Phi+U_\infty)^2}{U_\infty^2}=1-\kappa C_\mu \text{ on } c_2 \tag{4}$$

where Φ is the perturbation potential, U_∞ the freestream velocity, κ the jet curvature, and the jet momentum coefficient is defined as $C_\mu=m_j/q_\infty$, with m_j the jet momentum flux per unit length and q_∞ the freestream dynamic pressure. The problem is solved by approximating the wake shape through the vertical coordinates q: y_1, y_2, y_3..., of selected points on the free streamline and jet trajectories. The trajectories are defined by spline interpolation through those points. Equation (1) with the boundary conditions given by Eq. (2) is solved with a panel method. With Eq. (2) and (4), an objective function is

Presented as Paper 82-0220 at the AIAA 20th Aerospace Sciences Meeting, Orlando, FL, Jan. 11-14, 1982; received Feb. 8, 1987.

*Research Associate, Department of Aeronautics and Astronautics. Member AIAA.

†Department Head, Department of Mechanical Engineering. Fellow AIAA.