Technical Notes

TECHNICAL NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes cannot exceed 6 manuscript pages and 3 figures; a page of text may be substituted for a figure and vice versa. After informal review by the editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Leading-Edge Singularity in Transonic Small-Disturbance Theory: Numerical Resolution

Barbara L. Keyfitz*
Princeton University, Princeton, N.J.
and
Robert E. Melnik† and Bernard Grossman‡
Grumman Aerospace Corporation, Bethpage, N.Y.

I. Introduction

WE apply an analytical solution to the two-dimensional transonic small-disturbance (TSD) equation developed by the authors[1] to evaluate some numerical methods for integrating the TSD equation. The analytic solution—actually a perturbation series in a coordinate similarity parameter near the leading edge—enables one to distinguish the truncation error inherent in calculating the TSD flow near its known singularity from the discrepancy between even a fully converged TSD solution and the solution to the exact isentropic potential equation (FPE). This discrepancy, in approximately the first 10% chord, results from the fact that the TSD approximation, though widely used in practice, is not uniformly valid for blunt-nosed bodies near the stagnation point. What has in the past caused some confusion is that the two sources of error can have opposite effects on the surface pressure distribution, so that incompletely converged TSD solutions display an agreement with FPE solutions which we find to be spurious and which disappears when the mesh is refined.

II. Equations of TSD Theory

The TSD approximation is derived from FPE by perturbing about uniform flow.[2] Normalize velocities by freestream U_∞^* and lengths by chord c^*, and write the potential as $\Phi^* = U_\infty^* c^* (x + \epsilon\{\phi(x,y) - Kx\} + O(\epsilon^2))$, where the x axis lies along an airfoil with symmetric thickness distribution $\tau F(x)$ and the origin is at the leading edge. Scale y by τ/ϵ where $\tau = \sqrt{2\rho^*/c^*}$ with ρ^* the nose radius. The upstream flow is at an angle $\alpha^* = O(\tau)$ to the x axis. In TSD theory, $M_\infty \sim 1$, $M_\infty^2 - 1 = O(\tau^{2/3})$, and $\epsilon = [\tau^2/G(M_\infty)]^{1/3}$ where $G(M_\infty) \to 1+\gamma$ as $M_\infty \to 1$ and γ is the ratio of specific heats. The *transonic similarity parameter* $K = [(M_\infty - 1)/\tau G(M_\infty)]^{2/3}$. The TSD equation is:

$$\phi_x \phi_{xx} - \phi_{yy} = 0 \tag{1}$$

with boundary conditions:

$$\phi_y(x,0^\pm) = \pm F'(x) \qquad 0 \le x \le 1 \tag{2a}$$

$$\phi(x,0^+) = \phi(x,0^-),\ \phi_y(x,0^+) = \phi_y(x,0^-) \quad x<0 \tag{2b}$$

$$\phi(x,0^+) - \phi(x,0^-) = \Gamma,\ \phi_y(x,0^+) = \phi_y(x,0^-) \quad x>1 \tag{2c}$$

$$\phi(x,y) \to Kx + \alpha y \qquad x^2+y^2 \to \infty \tag{2d}$$

Here Γ is the circulation and $\alpha = \alpha^*/\tau$. The pressure coefficient is:

$$C_p(x,y) = -2\epsilon\{\phi_x(x,y) - K\} \tag{3}$$

Choices of $G(M_\infty)$ include the classical Spreiter scaling,[2] $(\gamma+1)M_\infty^2$; Krupp[3] uses $(\gamma+1)M_\infty^{3/2}$, and Sirovich and Huo[4] propose $[(2\gamma+1)M_\infty^2 + 1]/2$. None of these appreciably affects the leading-edge behavior.

III. Similarity Solutions Near the Leading Edge

Motivated by the work of Germain[5] and Nonweiler,[6] we have developed a coordinate expansion for $\phi(x,y)$ near the leading edge. The effects of airfoil geometry, departure of freestream speed from sonic, and angle of attack are treated as regular perturbations of a symmetric leading term which is equivalent to Nonweiler's solution for sonic flow about a parabola.[6]

Using the similarity parameter $\zeta = xy^{-n}$, with $n = 6/7$,

$$\phi(x,y) = \sum_{i=0}^{\infty} |y|^{n_i} f_i(\zeta) + \operatorname{sgn} y \sum_{j=0}^{\infty} |y|^{m_j} g_j(\zeta) \tag{4}$$

where $n_0 < n_1 < n_2 < \ldots$ and $n_0 < m_0 < m_1 < \ldots$. The functions f_i and $(\operatorname{sgn} y) g_j$ are, respectively, symmetric and antisymmetric about the x axis. The second series is absent at zero incidence.

For the leading term, $n_0 = 4/7$, and f_0 satisfies

$$N(f_0) = (n^2\zeta^2 - f_0')f_0'' - 5n(n-1)\zeta f_0' + 3(n-1)(n-2)f_0 = 0 \tag{5}$$

The higher order terms satisfy linear equations

$$L(n_i)f_i = F_i \qquad \text{or} \qquad L(m_j)g_j = G_j \tag{6}$$

where

$$L(m) = (n^2\zeta^2 - f_0')\frac{d^2}{d\zeta^2} + [n(n-2m+1) - f_0'']\frac{d}{d\zeta} + m(m-1) \tag{7}$$

and $F_i = \Sigma f_\ell' f_k'' + \Sigma g_\ell' g_k''$, summed over ℓ and k between 0 and i such that $n_\ell + n_k - n_0 = n_i$ or $m_\ell + m_k - n_0 = n_i$. Similarly, $G_j = \Sigma f_\ell' g_k''$, where $n_\ell + m_k - n_0 = m_j$. The exponents and the boundary conditions on Eq. (6) are determined by Eqs. (2a) and (2b). In term of $p_i(\zeta) = |\zeta|^{(1-n_i)/n}(n_i f_i - n\zeta f_i')$ and $q_i(\zeta) = |\zeta|^{-n_i/n}[(1-n_i)f_i + n\zeta f_i']$ (and the same for g_j), and an assumed profile shape

$$F(x) = \sqrt{x} + \sum_{1}^{\infty} b_k x^{\ell_k}$$

Presented as Paper 77-676 at the AIAA 10th Fluid and Plasmadynamics Conference, Albuquerque, N. Mex., June 27-29, 1977; submitted Aug. 5, 1977; revision received Oct. 13, 1978.

Index categories: Transonic Flow; Computational Methods.
*Lecturer, Mechanical and Aerospace Engineering Dept. Member AIAA.
†Director of Aerophysics, Research Dept. Member AIAA.
‡Theoretical Aerodynamics Laboratory Head, Research Dept. Member AIAA.

Table 1 Series solution for a Joukowski airfoil at incidence

Term	Exponent	Coefficient	$q(\infty)$
f_0	4/7	1	−0.37846
g_0	0.60313	$\bar{E}_0$	2.2682
f_1	1.07160	E_1	−1.6694
g_1	1.10330	$E_1\bar{E}_0$	−2.8185
f_2	10/7	$3/2b_1$	0.22428
g_2	1.46027	$3/2b_1\bar{E}_0$	0.85993
g_3	1.54393	$\bar{E}_3$	−1.1925
f_3	1.57177	E_1^2	1.7341
f_4	1.57563	$\bar{E}_0\bar{E}_3$	−2.8019
g_4	1.60347	$E_1^2\bar{E}_0$	20.102
f_5	1.92874	$3/2b_1E_1$	−0.90432
g_5	1.96044	$3/2b_1E_1\bar{E}_0$	−8.4126
f_6	2.01844	E_6	2.2679

we find for f_i:

$$p_i(-\infty)=0,\ p_i(+\infty)=\begin{cases}\ell_k b_k & \text{if } n_i-1=n(\ell_k-1) \text{ for some } k\\ 0 & \text{otherwise}\end{cases} \tag{8}$$

and for g_j:

$$q_j(-\infty)=0, \qquad p_j(+\infty)=0 \tag{9}$$

Thus, Eq. (4) is found term by term. The differential equations were integrated numerically and tabulated in Ref. 1. From Eq. (3),

$$C_p(x,0^\pm)=2\epsilon[K-\Sigma(n_i/n)q_i(\infty)x^{n_i/n-1} \pm\Sigma(m_j/n)q_j(\infty)x^{m_j/n-1}] \tag{10}$$

The quantities needed to evaluate Eq. (10) for a thin Joukowski airfoil, $F(x)=\sqrt{x}(1-3/2\,x+\ldots)$ appear in Table 1.

Now, terms of Eq. (4) can arise as eigenfunctions of $L(m)f=0$ with homogeneous boundary conditions (8) or (9). In fact, m_0 itself is an eigenvalue, found numerically to be 0.60313. The second antisymmetric eigenvalue is $m_3=1.5439$; the first two symmetric eigenvalues are $n_1=1.0716$ and $n_6=2.0184$. These terms enter Eqs. (4) and (10) with undetermined multiplicative constants (E_i, $\bar{E}_j$ in Table 1), which, in practice, were evaluated by curve-fitting Eq. (10) to a numerical solution for C_p. We use the fact that $\bar{E}_0=0(\alpha)$ to drop terms of order $\bar{E}_0^2$ in Eq. (10). For the range $\alpha\leq 0.11$ which we used, this gave satisfactory results.

IV. Evaluation of Numerical Solutions

We studied Joukowski airfoils with thickness-to-chord ratios from 0.05 to 0.15, for M_∞ between 0.7 and 0.9, and α^* between 0 and 1.5 deg. We tested the Spreiter, Krupp, and Sirovich-Huo scalings. The numerical scheme was a horizontal-line mixed-flow relaxation scheme of Jameson, based on Murman and Cole,[7] and incorporating the lift spreading of Klunker and Newman.[8] Boundary conditions on the profile were enforced by means of an artificial row of mesh points; the nose was placed halfway between mesh points and the mesh tripled at each refinement. For the incidence case, the airfoil was between mesh rows. Mesh concentration was controlled by a parameter Q between 0 and $1/2\pi$ which concentrated points at the leading and trailing edges and along the airfoil. Typical results for C_p run with 45 and 135 points on the airfoil and 5%, 10%, and 16% of points in first 5% chord are shown in Fig. 1. The solution is extremely sensitive to mesh concentration; the effect of increasing the number of points in the nose region is almost the same as that of refining the entire mesh. With 135 points on the airfoil and $Q=0.1496$, the solution was considered to be effectively converged. Comparison with FPE solutions[9] clearly indicates that a much more rapid pressure rise occurs in TSD solutions, with converged solutions apparently deviating most from the FPE result. This essential difference is confirmed when the series for C_p, developed in Sec. III, is introduced (Fig. 2). In all cases, least-squares fitting of two constants $\bar{E}_0$ and E_1 ($\bar{E}_3$ and E_6 had no effect) gave an excellent match with converged solutions; for decreased leading-edge mesh concentration, the fit was not as good. As the theory would suggest, E_1 was independent of α, $\bar{E}_0$ depended mainly on α.

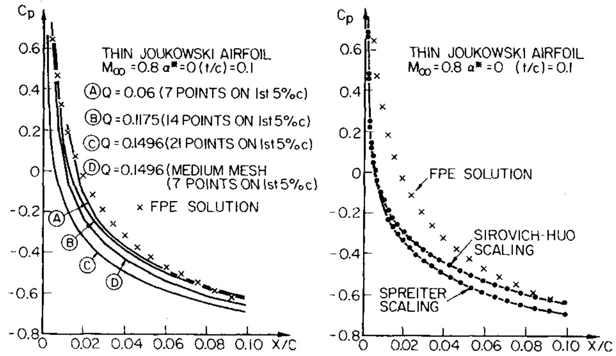

Fig. 1 Effects of mesh concentration and scaling.

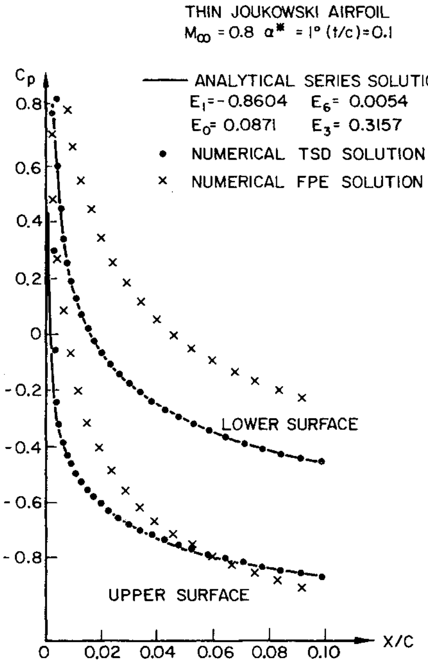

Fig. 2 Analytical and numerical solutions.

Finally, it was possible to fit series Eq. (10) to converged TSD solutions whether the Spreiter, Krupp, or Sirovich-Huo scaling was used (Fig. 1) by suitable choice of $\bar{E}_0$ and E_1. The rapid pressure rise near the leading edge is a common property of all these numerical solutions; we conclude that it is a property of the analytic solution of the TSD equations and cannot be avoided by a choice of scaling.

References

[1] Keyfitz, B.L., Melnik, R.E., and Grossman, B., "The Leading-Edge Singularity in Transonic Small-Disturbance Theory," Grumman Research Dept. Rept., RE-525, Oct. 1976; also *Quarterly Journal of Mechanics and Applied Mathematics,* Vol. XXXI, 1978, pp. 137-155.

[2] Ferrari, C. and Tricomi, F.G., *Transonic Aerodynamics,* Academic Press, New York, 1969.

[3]Krupp, J.A., "The Numerical Calculation of Plane Steady Transonic Flows Past Thin Lifting Airfoils," Boeing Scientific Research Lab. Rept. D180-12958-1, June 1971.

[4]Sirovich, L. and Huo, C., "Simple Waves and the Transonic Similarity Parameter," *AIAA Journal*, Vol. 14, Aug. 1976, pp. 1125-1127.

[5]Germain, P., "Ecoulements Transsoniques Homogenes," *Progress in Aeronautical Sciences*, Vol. 5, Pergamon Press, New York, 1964, pp. 143-273.

[6]Nonweiler, T.R.F., "The Sonic Flow About Some Symmetric Half-Bodies," *Journal of Fluid Mechanics*, Vol. 4, 1958, pp. 140-148.

[7]Murman, E.M. and Cole, J.D., "Calculation of Plane Steady Transonic Flows," *AIAA Journal*, Vol. 9, Jan. 1971, pp. 114-121.

[8]Klunker, E.B. and Newman, P.A., "Computation of Transonic Flow About Lifting Wing-Cylinder Combinations," *Journal of Aircraft*, Vol. 11, April 1974, pp. 254-256.

[9]Jameson, A., "Iterative Solutions of Transonic Flows Over Airfoils and Wings, Including Flows at Mach 1," *CPAM*, Vol. XXVII, 1974, pp. 283-309.

Use of Primitive Variables in the Solution of Incompressible Navier-Stokes Equations

K. N. Ghia*
University of Cincinnati, Cincinnati, Ohio
and
W. L. Hankey Jr.† and J. K. Hodge‡
Air Force Flight Dynamics Laboratory, Wright Patterson AFB, Ohio

I. Introduction

TWO-DIMENSIONAL laminar incompressible flows have been studied extensively by several investigators using the Navier-Stokes equations formulated in terms of vorticity ω and stream function ψ. This formulation, although remarkably useful for two-dimensional flows in simply connected regions, is not easily extendable to three-dimensional, compressible or turbulent flow applications. Two of the alternate formulations which are not as severely handicapped for complex problems are: the velocity-vorticity formulation, and the velocity-pressure formulation.

In the present study, it is planned to develop a method using the primitive variables (u,v,p) to solve the Navier-Stokes equations, with a view to later extending the method to more involved flow configurations. To carry out this task successfully, followed by a complete comparative study of this solution with the (ω,ψ) solution, it is important to choose a meaningful model flow problem. In his recent article, Roache[1] has underscored this point. An essential feature desired in the present model problem is that it be a true Navier-Stokes problem. Therefore, the driven flow in a square cavity is selected as a model problem (see Fig. 1, Ref. 2).

Presented as Paper 77-648 at the AIAA 3rd Computational Fluid Dynamics Conference, Albuquerque, N. Mex., June 27-28, 1977; submitted Oct. 13, 1977; revision received Oct. 6, 1978.

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

*Professor of Aerospace Engineering and Applied Mechanics. Member AIAA.

†Senior Scientist, Computational Fluid Dynamics Group. Associate Fellow AIAA.

‡Aerospace Engineer, Computational Fluid Dynamics Group. Member AIAA.

Burggraf[3] studied the cavity-flow problem with great care and provided results for Reynolds number, *Re*, ranging from zero to 400. Several investigators[4-8] have used the cavity problem as a model problem to test new numerical schemes, as qualitative experimental data[9,10] as well as detailed numerical results[3-5] are available for this problem. The latter is obtained by numerical schemes accurate to second order or higher.[5]

It is recognized that the corner-singularities present in the cavity-flow configuration cause some numerical difficulties. However, it is believed that the effect of these singularities is only local, and the global solution is not affected significantly. Ideally, it would be desirable to treat these singularities analytically and determine the numerical solution such that it matches smoothly with the local analytical solutions at these singular points.[11]

II. Formulation of the Problem

Governing Equations

In convective form, the differential equations governing the cavity-flow problem are:

$$u_x + v_y = 0 \tag{1}$$

$$u_t + uu_x + vu_y = -p_x + (1/Re)(u_{xx} + u_{yy}) \tag{2}$$

$$v_t + uv_x + vv_y = -p_y + (1/Re)(v_{xx} + v_{yy}) \tag{3}$$

Here, the Reynolds number *Re* is defined as $Re = \rho UL/\mu$, with U being the velocity of the moving wall. All velocities have been made dimensionless with respect to U, pressure with reference to ρU^2, and distances with respect to the width L of the square cavity.

Unfortunately, the pressure p, which is nested in this system, does not appear as a dominant variable in any of these equations. To correctly model the elliptic nature of the flow problem, the pressure p in the (u,v,p) system may be determined from a Poisson equation obtained by appropriately forming the divergence of the vector momentum equation. Thus, the following Poisson equation must replace the continuity equation.

Poisson Equation for Pressure

$$p_{xx} + p_{yy} = S_p - \frac{\partial}{\partial t}[u_x + v_y] \tag{4}$$

where

$$S_p \triangleq \frac{d}{dx}\left[-(uu_x + vu_y) + \frac{1}{Re}(u_{xx} + u_{yy})\right] + \frac{d}{dy}\left[-(uv_x - vv_y) + \frac{1}{Re}(v_{xx} + v_{yy})\right] \tag{5}$$

The derivatives of the local dilation term $D \triangleq u_x + v_y$ appearing in Eqs. (4) and (5) intentionally have not been set to zero; the reason for doing this will be explained in the next section.

Boundary Conditions

The boundary conditions for the cavity-flow problem in the (u,v,p) system are relatively straightforward (see Fig. 1, Ref. 2). For the momentum equations, the normal velocities are zero at the nonporous walls, while the tangential velocities satisfy the condition of zero slip at all walls. For the pressure equation, the boundary conditions consist of the normal gradient $\partial p/\partial n$ evaluated from the appropriate momentum equation.