VOL. 16, NO. 6 J. AIRCRAFT JUNE 1979

ARTICLE NO. 78-1203

Three-Dimensional Viscous-Inviscid Coupling Using Surface Transpiration

L.A. Lemmerman* and V.R. Sonnad†

Lockheed-Georgia Company, Marietta, Ga.

The development of a method to obtain the subsonic flowfield about complex, three-dimensional geometries, using the viscous-inviscid iteration technique to correct the potential flow for viscous effects is described. A panel method for the calculation of three-dimensional inviscid flow is used iteratively with a two-dimensional, integral boundary-layer method. The important feature is the use of surface transpiration to simulate boundary layer effects. The cross flow is neglected in the boundary layer calculations, which are carried out along chordwise strips on the wing. A doubly underrelaxed iteration scheme has resulted in an extremely stable method, which usually gives satisfactory results after 4-5 iterations. The method has been tested on a wide variety of configurations, and the results have been uniformly satisfactory. It has been demonstrated that the method of surface transpiration is completely equivalent to the displacement thickness approach as a means of representing the boundary layer.

Nomenclature

C = chord
C_l = local lift coefficient
C_L = wing lift coefficient
C_p = pressure coefficient
M = Mach number
R = underrelaxation factor
R_E = Reynolds number
U, V, W = components of velocity along mutually orthogonal axes (depending on subscripts, they may also signify total velocity)
α = angle of attack
δ^* = displacement thickness
η = fraction of semispan
Λ = sweep angle
ρ = density
σ = source strength
ϕ = disturbance potential

Subscripts

a = analogous wing
e = inviscid flow values
L.S. = lower surface
n = normal to the body surface
surf = body surface
tot = total
T.E. = trailing edge
U.S. = upper surface
∞ = freestream values

Introduction

In an aircraft development program, there are several three-dimensional aerodynamic design problems that need to be considered. The most important of these is the design of the wing. The three-dimensional subsonic and transonic inviscid flowfields about a wing can now be efficiently calculated, and in some cases, this alone is adequate for design purposes. In some cases, however, it is necessary to account for viscous effects.

A classical, well-tested method is to correct the inviscid flow with a boundary layer effect in an iterative fashion. It has proved reliable and accurate in two-dimensional applications (within certain limitations) and is very much less expensive than solving the Navier-Stokes equations. In the viscous-inviscid iterative technique, it has been conventional to model boundary layer effects by the addition of the displacement thickness to the body surface, creating an equivalent body. Lighthill[1] suggested that the boundary layer could be equally well simulated by an outward velocity normal to the surface of the body—the surface transpiration concept. The method of transpiration has significant computational advantages over the former, since the geometry remains invariant during the iteration. This is a particularly important feature for the calculation of the flowfield about complex three-dimensional geometries. Although several attempts have been made to use this approach, the results have led to conflicting opinions regarding the validity of this approach.

In subsonic flow, the feasibility of three-dimensional coupling was demonstrated at NLR,[2] using a semi-automatic computational procedure. Only one iteration was carried out between fully three-dimensional potential flow and boundary layer codes, using surface transpiration to simulate boundary layer effects. The conclusions from Ref. 2 indicate that the surface transpiration method is satisfactory, and further iterations, although necessary, are not feasible because the computational costs using a three-dimensional boundary layer method are unacceptably high.

In July 1977, Dvorak[3] et al. presented results obtained by using the surface transpiration approach in a viscous-inviscid iteration scheme. Their results did not agree with experimental data for a low-aspect-ratio, swept wing with a symmetric section.[4] This was, however, attributed to deficiencies in the subsonic potential flow method.

Hess,[5] using the surface transpiration approach, published results in June 1977, using his panel method for the same test case as Dvorak. His calculations were nearly identical to those of Dvorak and also in poor agreement with experimental data. However, on repeating the computations with the displacement thickness approach, Hess obtained excellent correlation with experimental data. This would appear to refute Dvorak's arguments regarding deficiencies in the potential flow program. Hess further concludes that the

Presented as Paper 78-1203 at the AIAA 11th Fluid and Plasma Dynamics Conference, Seattle, Wash., July 10-12, 1978; submitted July 10, 1978; revision received Dec. 11, 1978.

Index categories: Aerodynamics; Computational Methods; Subsonic Flow.

*Associate Scientist.

†Technical Consultant. Member AIAA.

surface transpiration method does not satisfactorily represent boundary layer effects and has recommended the displacement thickness approach. Since the two methods of boundary layer simulation are mathematically equivalent, the results obtained by Hess using surface transpiration are difficult to understand.

The results obtained by NLR and Dvorak and Hess using surface transpiration appear to be contradictory and are inconclusive about the merit of surface transpiration in representing boundary layer effects, especially in three-dimensional flow. Since the transpiration approach is most attractive from an operational standpoint, additional research needs to be done to remove any doubts concerning this method.

The objective of the present work is to develop a reliable, relatively fast, design-oriented method to calculate the viscous flow over wing-body configurations in subsonic flow. This method has been developed by combining a subsonic panel method for three-dimensional potential flow calculations with a two-dimensional boundary layer method in an iterative scheme, using the surface transpiration approach to simulate boundary layer effects.

The overall approach is described in the next section. This will be followed by a brief description of the individual programs, along with the coupling process. Finally, the results obtained are presented for comparison with experimental data.

Approach

The viscous-inviscid iteration technique is based on the assumption that at practical Reynolds numbers, the viscous effects are appreciable only in a thin region near the body and can be neglected outside this region. It is therefore possible to obtain, independently, the flowfield in these two regions by solving equations that have totally different characteristics and then matching these two solutions at the interface.

The presence of a shear layer about a body affects the flow by displacing the streamlines outward from the body. In the surface transpiration approach, this effect is modeled by an outward velocity normal to the surface of the body. The sequence of iteration to obtain the viscosity-corrected flowfield is as follows:

1) Calculate the inviscid flow around the given body.
2) Using the inviscid pressure distribution, calculate the displacement thickness and, from it, the surface normal velocity required to simulate the boundary layer on lifting portions of the body.
3) Recalculate the potential flow using this normal velocity as the boundary condition, instead of the no-flow condition at the surface of the body.

The process of iteration is carried on until the selected criterion for convergence (usually imposed on the pressure distribution) is satisfied to within a given tolerance.

A panel method is used to calculate the potential flow about three-dimensional configurations. This is coupled with a two-dimensional integral boundary layer method in an iterative scheme. The cross flow is neglected in the boundary-layer calculations, which are carried out along chordwise strips on the wing. The process of iteration is performed with suitable interpolation and smoothing between the two methods of calculation.

No attempt was made to include fuselage boundary layer calculations, since the contribution of the fuselage to lift is generally small. This is illustrated later in the section on results.

Description of Programs

In the present work, there was no intent to develop methods for three-dimensional inviscid flow or for two-dimensional boundary-layer flow. Rather, the intention was to use reliable, well-tested methods and develop a suitable interfacing process for sequential iteration between the two methods. It is, however, necessary to be aware of the limitations of the respective methods and the basic assumptions made in the calculation procedures. The salient features of each program are briefly discussed.

Potential Flow Program

The Lockheed Aircraft Interference Program for the calculation of inviscid flow around three-dimensional lifting bodies is based on the method developed and documented by Hess.[6] The geometry of the body can be described to any level of detail simply by increasing the number of panels, and difficult contours do not pose any problems. The calculation procedure is different for the nonlifting and lifting portions of the body, the lifting portions being defined as those portions on which the Kutta condition is enforced.

Nonlifting Calculations

In inviscid, irrotational, incompressible flow, the continuity equation can be written as

$$\nabla^2\phi = 0$$

where ϕ is the perturbation potential. The boundary conditions are given by

$$\nabla\phi \to 0 \text{ at } \infty \tag{1}$$

$$\nabla\phi\cdot\boldsymbol{n}|_{\text{surf}} = V_\infty\cdot\boldsymbol{n}|_{\text{surf}} \tag{2}$$

where $\boldsymbol{n}$ = surface normal vector.

Disturbance source singularities (σ) are distributed on the body surface. It is possible to represent these disturbances on the basis of known geometric quantities. There results a system of linear algebraic equations that can be solved to obtain the value of σ on each panel. The velocities, and subsequently the pressure, can then be calculated on each panel.

Lifting Portion Calculations

The calculations here are different, since the Kutta condition is additionally imposed on the flow. In addition to the source distribution, a dipole distribution is used to represent the surface of the lifting section. The source distribution is used to satisfy the boundary condition of no flow through the surface, as in the nonlifting calculations, and the dipole distribution is used to satisfy the Kutta condition, which expresses the fact that the flow must leave the body smoothly at the trailing edge. As a practical matter, the Kutta condition can be applied in several theoretically equivalent ways, and the condition of equal pressures on the centroids of the trailing-edge panels is the one that is actually used.

The lifting section is described by means of "N lines" that lie on the contour of the wing and are roughly in the direction of the freestream, as shown in Fig. 1. The dipole strength μ is fixed as zero along the length of the trailing edge. Along each N line, the dipole strength varies linearly with arc length; the gradient is unknown and is determined from the Kutta condition. The spanwise variation of μ between N lines is assumed piecewise constant. The Kutta condition is satisfied using the concept of multiple-onset flows. Influence coefficients are calculated corresponding to a unit dipole gradient on a single strip and zero gradient on all other strips. By sequencing through all lifting strips, a set of equations is generated to solve for the proper distribution of dipole gradient across the lifting strips. The final values are determined iteratively, simultaneously satisfying the condition $(U_{\text{U.S.}} - U_{\text{L.S.}})_{\text{T.E.}} = 0$ on all strips. the solution for the onset flows are linearly combined using these constants to obtain the final solution for the velocities from which the pressures are calculated.

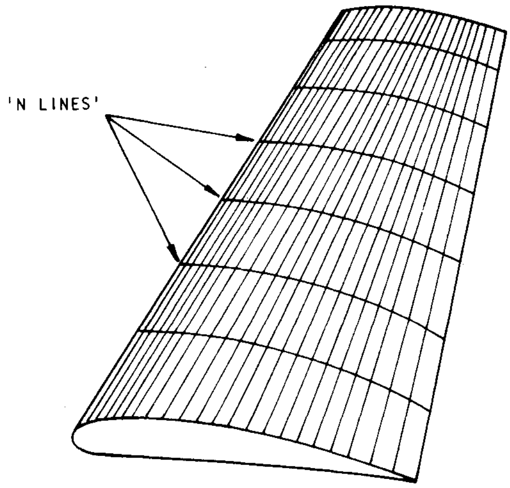

Fig. 1 Sketch of paneling for a typical wing.

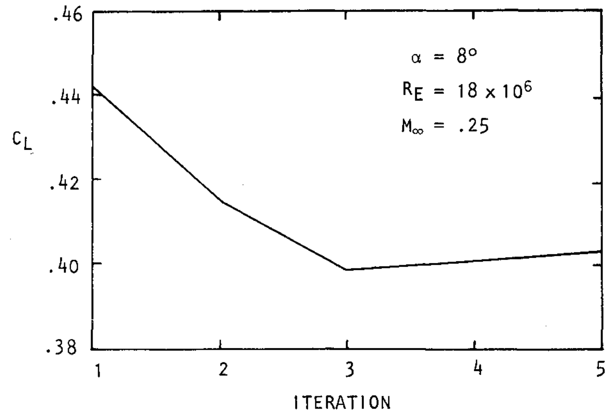

Fig. 2 Typical iteration history for a wing.

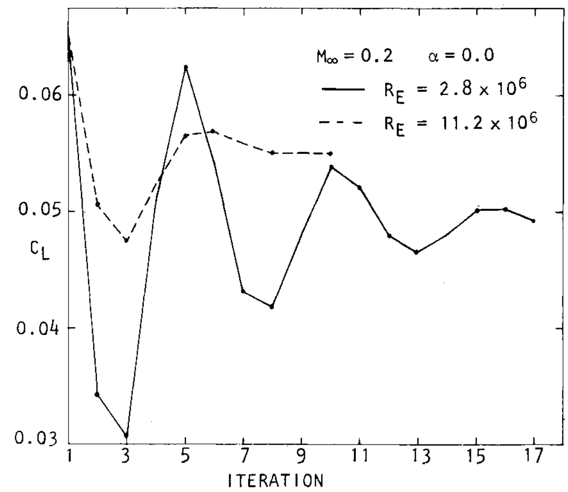

Fig. 3 Iteration history for a thick wing.

The potential flow solution method currently contains a Gothert transformation to account for the effects of compressibility. This transformation, however, has been found to be unsatisfactory at the higher subsonic Mach numbers, and the method in its present form is, therefore, most useful at lower Mach numbers.

Boundary Layer Program

A two-dimensional integral boundary layer method developed by McNally[7] has been used in this scheme. This method has been chosen for some important reasons:

1) Only the integral parameters are required in the coupling process.
2) It is very inexpensive and well behaved as compared to existing three-dimensional boundary layer methods.
3) It was required to function satisfactorily only for moderately swept wings at relatively low angles of attack, where the cross-flow effects would not be critical. (It has, however, worked exceptionally well even at 45-deg sweep angles.)

This boundary layer method has proved to be very reliable. Because of the integral method of solution, it is considerably faster than finite-difference methods and is admirably suited to the strip method, where two-dimensional boundary layer calculations have to be performed repeatedly.

The Coupling Process

The pressure distribution and the surface normal velocity are the means of interaction between the viscous and inviscid flow programs. The potential flow program calculates the pressure distributions at the centroids of the panels used to represent a given configuration. On the lifting portion of the body, the centroids lie roughly along lines that are in the direction of the freestream. This feature makes it very convenient to use with a two-dimensional boundary-layer program. The arc lengths along the wing surface and the inviscid pressure distribution, along with a few parameters, are the only quantities supplied to the boundary layer program. These pressures, after suitable interpolation and smoothing, are used by the boundary layer program to calculate the surface normal velocities. These are then smoothed and re-interpolated to calculate values of the velocities at the collation points used in the panel method. After suitable normalization, these velocities are used to modify the boundary conditions used in the potential flow program. The inviscid pressure distributions are then recalculated, and the process is repeated until satisfactory results are obtained. It is felt that the poor correlation with experimental data obtained by Hess in using the surface transpiration approach is the result of incorrect interaction procedures rather than due to any fundamental deficiencies in the method of surface transpiration as a means of representing boundary layer effects.

The important advantage of the surface velocity approach is that the body geometry remains invariant during the iteration process. For a given geometry, therefore, the matrix of influence coefficients generated by the potential flow routine is set up only once. This matrix is inverted once. Subsequent solutions of the system of equations can then be obtained through simple multiplications of the inverse matrix with new boundary conditions. The obvious applications include not only multiple viscous-inviscid iteration solutions, but also efficient multiple angle-of-attack solutions.

An underrelaxation factor is imposed on the surface normal velocities at the end of the first potential solution and on the inviscid pressure distribution at the end of the third. This double underrelaxation has proved very useful when dealing with extreme cases, such as thick wings at low Reynolds numbers. There is at present no condition for convergence imposed on the iteration process; it is left to the user to choose the number of iterations. The program has been provided with a restart option, which can be used to continue iteration from a given point onward by using data stored from previous iterations. This option can be exercised repeatedly until the results obtained are satisfactory.

There has been no attempt made at any stage to account for viscous effects in the wake. It was felt that the error due to the neglect of viscous effects on the wake was of a lower order than those associated with other approximations in this method.

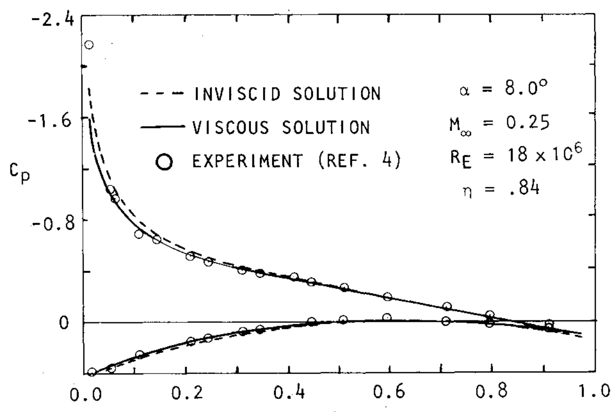

Fig. 4 Pressure distribution at a single spanwise location.

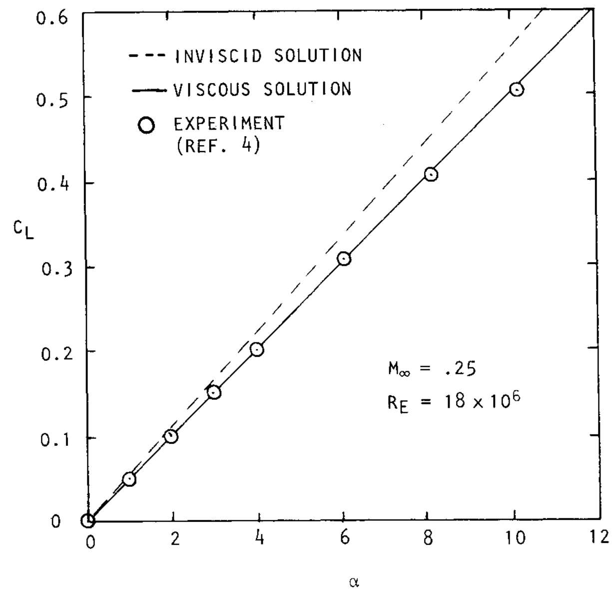

Fig. 5 Wing lift variation with angle of attack.

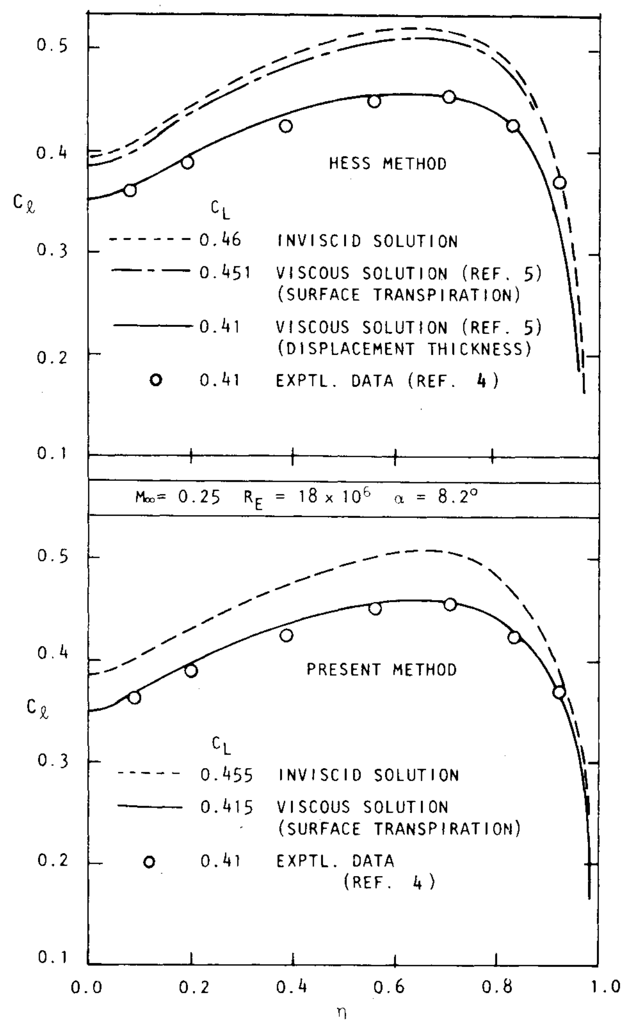

Fig. 6 Comparison of spanwise load distribution.

The method is valid only if the flow remains attached over the entire surface of the wing. Although provision has been made in the program to allow calculations to continue in the presence of separation, the basic assumptions used in the method do not hold at, or even near, separation. So, whenever separated flow is present over a portion of the wing, the results must be interpreted carefully.

Surface Sources to Simulate Boundary Layer Effects

It can be shown that in two-dimensional compressible flow, the expression for the surface normal velocity required to simulate the boundary layer effects is given by

$$V_{\text{surf}} = \frac{1}{\rho_{\text{surf}}} \frac{\partial(\rho_e U_e \delta^*)}{\partial s}$$

where δ^* is the compressible displacement thickness, s is the arc length along the surface, ρ_e, U_e, are the density and velocity at the edge of the boundary layer, and ρ_{surf} is the density at the surface of the body. Since the potential flow solution uses a Gothert transform to account for the effects of compressibility, the boundary layer solution must be related through a similar transform to an analogous body. The necessary expressions are derived as follows:

If we consider a three-dimensional body in the Cartesian coordinate system, with $\cos\alpha$, $\cos\beta$, and $\cos\gamma$ the direction cosines of the surface normal at any point, and V_n the surface normal velocity at that point in the real (physical) body system, the components of the normal velocity can be obtained by forming dot products:

$$V_{nx} = V_n \cos\alpha, \qquad V_{ny} = V_n \cos\beta, \qquad V_{nz} \cos\gamma$$

where $V_n = |V_n|$.

For flow about the analogous body, the transformed normal velocity components are given by

$$V_{xa} = (1 - M_\infty^2)\, V_{nx}, \qquad V_{ya} = (\sqrt{1 - M_\infty^2})\, V_{ny},$$

$$V_{za} = (\sqrt{1 - M_\infty^2})\, V_{nz}$$

Hence, the magnitude of the transformed velocity normal to the surface is

$$V_{na} = \sqrt{V_{xa}^2 + V_{ya}^2 + V_{za}^2}$$

$$= V_n \sqrt{(1 - M_\infty^2)\, [\, (1 - M_\infty^2) \cos^2\alpha + \cos^2\beta + \cos^2\gamma \,]}$$

In the iteration between the potential flow and boundary layer calculations, a certain degree of stability can be achieved at the expense of convergence speed by a process of underrelaxation, i.e., by allowing the conditons to change only slightly with each iteration. The method of underrelaxation used here is as follows:

1) After every boundary layer calculation, the surface normal velocities are computed and stored. These surface

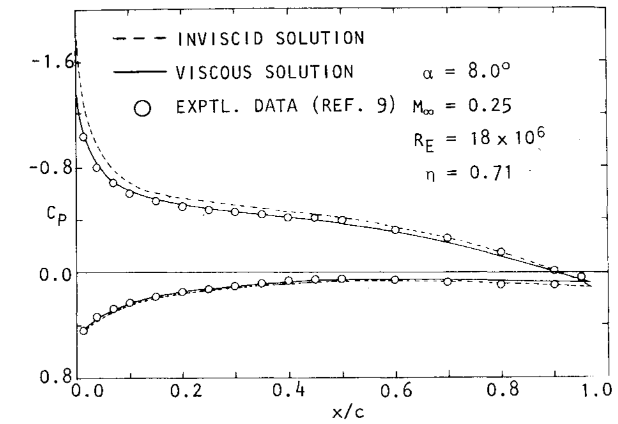

Fig. 7 Pressure distribution at a single spanwise location.

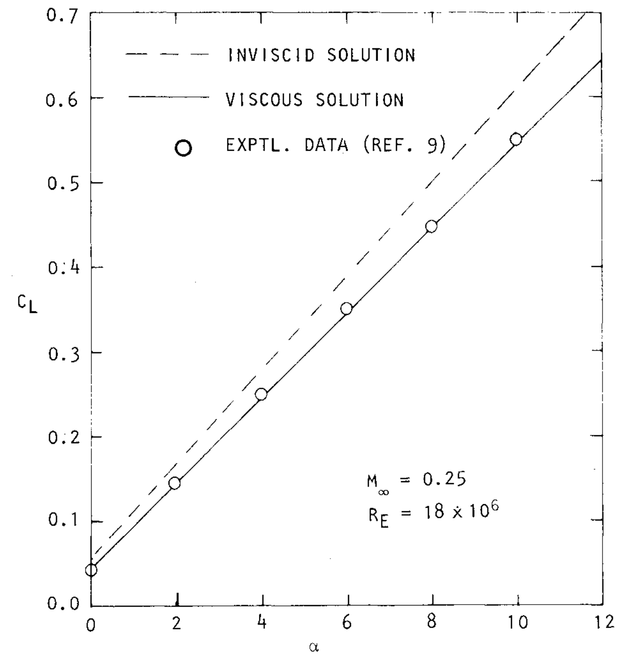

Fig. 8 Variation of wing lift coefficient with angle of attack.

velocities are modified by combining them with the surface velocities calculated at the previous iteration according to the formula

$$V_{n_{\text{used}}} = V_{n_{\text{old}}} + R(V_{n_{\text{new}}} - V_{n_{\text{old}}})$$

where R is the underrelaxation factor. The value of R determines the percentage of the old and new that are used to form the value of the surface normal velocity finally used in the calculation. A value of ⅔ was found to give satisfactory results for a wide range of wing configurations. It must be noted that at the very first iteration, $V_{n_{\text{old}}} = 0.0$ and hence

$$V_{n_{\text{used}}} = 2/3(V_{n_{\text{new}}})$$

2) After the third potential flow solution, the pressure distribution is underrelaxed in a similar manner before being used by boundary layer program:

$$C_{p_{\text{used}}} = C_{p_{\text{old}}} + R(C_{p_{\text{new}}} - C_{p_{\text{old}}})$$

Here again a value of ⅔ has been found to be most satisfactory for the underrelaxation factor R.

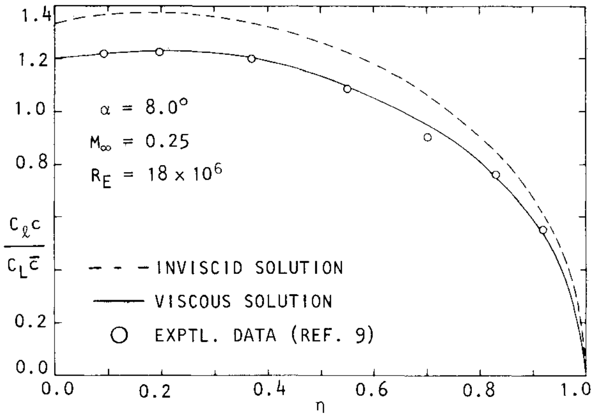

Fig. 9 Spanwise distribution of loading.

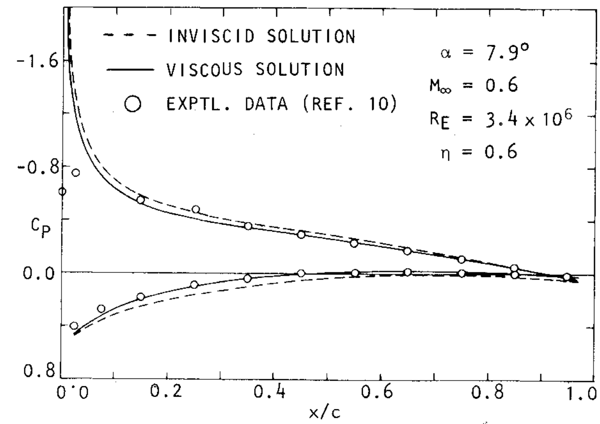

Fig. 10 Pressure distribution at a single spanwise station.

In the potential flow solution, the flow approaches a stagnation point near the wing trailing edge. Thus, on the first iteration, adjustments of the aft pressures are necessary before boundary layer calculations can be done. To prevent prediction of separation, the pressure distributions input to the boundary layer program are modified by linear extrapolation from 90% of the chord. Similarly on the first iteration only, the surface normal velocities are modified by linear extrapolation from 70% of the chord. This helps to prevent an overly strong response from the potential flow solution and avoids fishtail behavior during subsequent iterations. These measures are not usually necessary but have been incorporated to deal with extreme cases.

The use of this double underrelaxation has resulted in a stable scheme, and satisfactory results are obtained usually after 4-5 iterations. The process is illustrated for a relatively simple wing[4] in Fig. 2 and for a more difficult case,[8] which is a thick wing at low Reynolds number, in Fig. 3. The latter is intended more to demonstrate the stability of the method in an extreme case, rather than to obtain accurate results, since more iterations would be required before convergence.

Comparison of Results

It is particularly important to obtain experimental verification for results obtained from this method in order to determine the validity of the approximations used in the program. The method has been tried on several different configurations, and the results are described separately for each case.

Swept, Tapered Wing with Symmetric Cross Section

The geometric configuration, and experimental data for this wing have been obtained from Ref. 4. Typical pressure

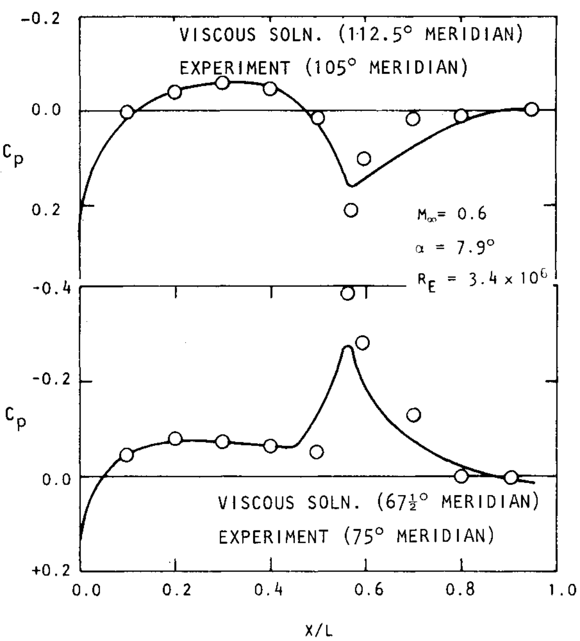

Fig. 11 Comparison of pressure distributions at two meridian locations on the fuselage.

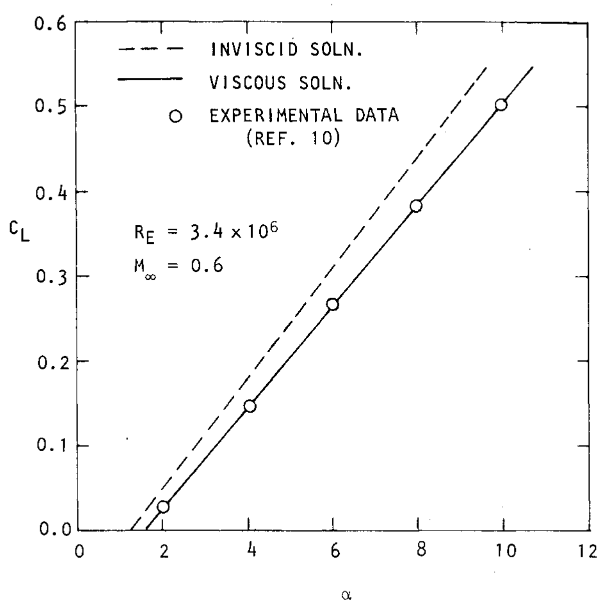

Fig. 12 Variation of wing lift coefficient with angle of attack.

distributions at a single spanwise station on this wing are compared with experimental values in Fig. 4. The wing lift coefficient shows excellent agreement with experimental data, as shown in Fig. 5. The spanwise load distribution obtained by using the present method is compared with results obtained by Hess in Fig. 6. While the results obtained by Hess using the displacement thickness approach show excellent agreement with experimental data, he indicates that the surface transpiration method does not correct adequately for viscous effects. The present results show, however, that the surface transpiration approach is adequate and is completely equivalent to the displacement thickness approach as a means of simulating the boundary layer.

Swept, Tapered, Twisted Wing with Cambered Cross Section

The geometry and experimental data have been obtained from Ref. 9. This wing has essentially the same planform shape as the wing of the previous example. In addition to sweep, the wing has a 5-deg washout between root and tip, making this a particularly severe test case for the strip method. The good agreement between predicted and experimental values for pressure and lift (Figs. 7-9) indicates that the strip method approximation of zero cross flow is valid for moderate angles of twist.

Wing-Body Combination

This configuration consists of a thin swept wing mounted in the middle of an axisymmetric fuselage. The details of the geometry are given in Ref. 10. The wing has a 4½-deg washout between root and tip. The results have been obtained at a Mach number of 0.6. It is seen that the pressure distribution on the wing and fuselage compare very well with experimental data (Figs. 10,11), as does the lift coefficient (Fig. 12). Since the fuselage pressure distributions were obtained without any viscous correction, this is an indication that a viscous correction is generally not necessary on the fuselage.

Conclusions

The use of the surface transpiration approach, in combination with the strip method, has resulted in a relatively speedy, design-oriented method to obtain the viscous correction in inviscid flow around three-dimensional lifting bodies. The surface transpiration approach has been found to be completely equivalent to the displacement thickness method in simulating boundary layer effects. The neglect of the cross flow in the boundary layer calculations appears to be a valid approximation even over swept, twisted wings.

References

[1]Lighthill, J.J., "On Displacement Thickness," *Journal of Fluid Mechanics*, Vol. 4., Pt. 4, Aug. 1958.

[2]Piers, W.J., Schipolt, G.J., and Van den Berg, G., "Calculation of the Flow around a Swept Wing, Taking into Account the Effect of the Three-Dimensional Boundary Layer," NLR TR 75076 U, May 1976.

[3]Dvorak, F.A., Woodward, F.A., and Maskew, B., "A Three-Dimensional Viscous/Potential Flow Interaction Analysis Method for Multi-Element Wings," NASA CR 152012, July 1977.

[4]Boltz, F.W. and Kolbe, C.D., "The Forces and Pressure Distributions at Subsonic Speeds on a Plane Wing Having 45° Sweepback, an Aspect Ratio of 3, and a Taper Ratio of 0.5," NACA RMA51G31, Oct. 1951.

[5]Hess, J.L., "A Fully Automatic Combined Potential-Flow Boundary Layer Procedure for Calculating Viscous Effects on the Lifts and Pressure Distributions of Arbitrary Three-Dimensional Configurations," Douglas Aircraft Co., Long Beach, Calif. MDC J7491, June 1977.

[6]Hess, J.L., "Calculation of the Potential Flow about Arbitrary Three-Dimensional Lifting Bodies," Douglas Aircraft Co., Long Beach, Calif., MDC J5679-01, Oct. 1972.

[7]McNally, W.D., "Fortran Program for Calculating Laminar and Turbulent Bondary Layers in Arbitrary Pressure Gradients," NASA TN D-5681, May 1970.

[8]Cooper, M. and Koryinski, P.F., "The Effects of Compressibility on the Lift, Pressure and Load Characteristics of a Tapered Wing of NACA 66-Series Airfoil Sections," NACA TN 1697, Oct. 1948.

[9]Boltz, W.F. and Kolbe, C.D., "The Forces and Pressure Distribution at Subsonic Speeds on a Cambered and Twisted Wing Having 45° of Sweepback, an Aspect Ratio of 3, and a Taper Ratio of 0.5," NACA RMA52D22, July 1952.

[10]Robinson, H.L., "The Effects of Wing Incidence on the Aerodynamic Loading Characteristics of a Sweptback Wing-Body Combination at Transonic Speeds," NACA RML54G23b, Oct. 1954.