Acknowledgment

The authors gratefully acknowledge the financial support of the National Science Council of the Republic of China under Grant NSC-82-0401-E-014-009.

References

[1]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solutions of the Euler Equations by a Finite Volume Method Using Runge–Kutta Time-stepping Schemes," AIAA Paper 81-1259, June 1981.

[2]Jameson, A., "Numerical Solution of the Euler Equations for Compressible Inviscid Fluids," *Numerical Methods for the Euler Equations of Fluid Dynamics*, edited by F. Angrand, A. Dervieux, J. A. Desideri, and R. Glowinski, Society for Industrial and Applied Mathematics, Philadelphia, PA, 1985, pp. 199–245.

[3]van Leer, B., Tai, C.-H., and Powell, K. H., "Design of Optimally Smoothing Multi-stage Scheme for the Euler Equations," AIAA Paper 89-1923, June 1989.

[4]Lynn, J. F., and van Leer, B., "Multi-stage Schemes for the Euler and Navier–Stokes Equations with Optimal Smoothing," AIAA Paper 93-3355, June 1993.

[5]van Leer, B., Lee, W. T., Roe, P. L., Powell, K. G., and Tai, C.-H., "Design of Optimally Smoothing Multistage Scheme for the Euler Equations," *Communications in Applied Numerical Method*, Vol. 8, 1992, pp. 761–769.

[6]Tai, C.-H., Sheu, J.-H., and van Leer, B., "Optimal Multistage Schemes for Euler Equations with Residual Smoothing," *AIAA Journal*, Vol. 33, No. 6, 1995, pp. 1008–1016.

Use of a Wake-Integral Method for Computational Drag Analysis

J. Mark Janus* and Animesh Chatterjee†
Mississippi State University/National Science Foundation Engineering Research Center for Computational Field Simulation, Mississippi State, Mississippi 39762-6176

Introduction

OVER the past few decades the operational requirements of aircraft have placed a premium on high values of aerodynamic efficiency with aircraft having relatively restricted span lengths. Lift-induced drag, for subsonic aircraft, accounts for almost 50% of the total drag of such aircraft under cruise conditions.[1] As a result, much interest exists in the aerospace community to study the causative mechanism involved and to devise new designs to minimize induced drag while maintaining relatively large values for lift coefficient.

Experimentally, the separation of form drag and induced drag involves detailed flowfield measurements, a task both time consuming and costly. Rapid testing of prototypes via numerical flowfield simulation overcomes some of the time and cost constraints linked with experimental techniques, at the same time providing complete flowfield data at no extra cost. The primary issue of interest in this study is the accurate and reliable calculation of induced drag. This interest developed as a consequence of an effort to numerically simulate an experimental investigation designed to study the effect of some innovative configurational arrangements on induced drag.[2] To predict the induced drag for the configurations studied, a wake-integral method was used. Earlier works using a similar approach[3–5] indicate this to be a relatively accurate technique to calculate the lift and induced drag for a finite wing at moderate angles of attack. In this Note, some procedures that were found to be necessary to produce satisfactory induced drag calculations based on this wake-integral approach are discussed.

Presented as Paper 95-0535 at the AIAA 33rd Aerospace Sciences Meeting, Reno, NV, Jan. 9–12, 1995; received March 16, 1995; revision received Sept. 5, 1995; accepted for publication Sept. 11, 1995.

*Assistant Professor, Department of Aerospace Engineering, P.O. Box 6176. Senior Member AIAA.

†Graduate Student, Department of Aerospace Engineering; currently Graduate Student, Syracuse University, 111 College Place, Syracuse, NY 13244-4100.

Analysis

The flow solver used to numerically simulate the flowfields studied herein is based on a finite volume method to solve the three-dimensional Euler equations. Interface fluxes are determined using a high-resolution Riemann scheme.[2,6] The flow solver incorporates characteristic-based concepts, hence characteristic-variable boundary conditions are employed where applicable.

Traditional approaches to lift and drag evaluation for an aerodynamic configuration use either surface force integration or a measurement of the total forces on the system in the wind tunnel. In a numerical simulation using an Euler method, the simplest method toward calculating lift and drag is based on surface pressure integration. Numerical dissipation present in all numerical flow solvers poses quite a problem when considering this approach. The detrimental effects of numerical dissipation is most pronounced at the leading and trailing edges (regions of high gradients) of the body. As a result significant errors can occur, especially in the drag calculations since these regions of tainted pressures are largely oriented in the streamwise direction. The references cited indicate that a wake-integral method appears to be the most accurate method for evaluating induced drag utilizing present computational fluid dynamics (CFD) capabilities. Therefore the approach used here for induced drag and lift determination is the generalized wake-integral method of Wu et al.[3]

Let $\mathcal{V}(t)$ represent a material volume within a fluid mass at time t that completely encloses a body; see Fig. 1. The volume is bounded by planes normal to the freestream velocity both upstream and downstream of the body and by surfaces parallel to $\boldsymbol{V}$ on the sides. Conservation of linear momentum states that the time rate of change of linear momentum of the material volume $\mathcal{V}(t)$ is equal to the resultant force on the volume. Therefore, for the material volume $\mathcal{V}(t)$ bounded by surface $\mathcal{S}(t)$

$$\boldsymbol{F} = \frac{\mathrm{d}}{\mathrm{d}t}\int_{\mathcal{V}(t)} \varrho \boldsymbol{V}\,\mathrm{d}\mathcal{V} \tag{1}$$

For steady flows with no body or viscous forces, the near-field (integral over $\mathcal{S}_9$) and far-field (integral over $\mathcal{S}_{1,2,3,4,5,6}$) expressions for lift are written as

$$L = \int_{\mathcal{S}_9} p n_y\,\mathrm{d}\mathcal{S} = -\int_{\mathcal{S}_{1,2,3,4,5,6}} [p n_y + \varrho v(\boldsymbol{V}\cdot\boldsymbol{n})]\,\mathrm{d}\mathcal{S} \tag{2}$$

and similarly for drag,

$$D = \int_{\mathcal{S}_9} p n_x\,\mathrm{d}\mathcal{S} = -\int_{\mathcal{S}_{1,2,3,4,5,6}} [p n_x + \varrho u(\boldsymbol{V}\cdot\boldsymbol{n})]\,\mathrm{d}\mathcal{S} \tag{3}$$

The near-field expressions are simply surface pressure integration. It is well established that the near-field method of determining the aerodynamic forces on a body works quite well for lift calculations but is tainted by numerical dissipation for drag calculations.

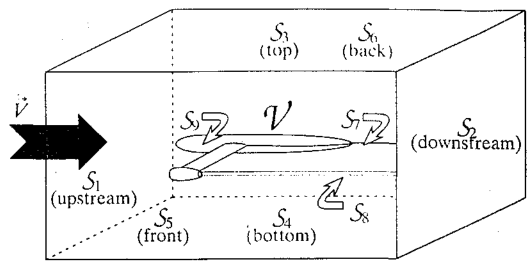

Fig. 1 Control volume for development of the wake-integral theory.

Following the assumptions for wake-integral development put forth by van Dam et al.,[5] the far-field expression for lift simplifies to the wake integral

$$L = -\varrho_\infty U_\infty \int_{S_2} z\bar{\xi}\, \mathrm{d}S \tag{4}$$

where $\bar{\xi}$ is the streamwise vorticity component given by

$$\bar{\xi} = \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \tag{5}$$

A formulation for induced drag as a wake integral can be developed along similar lines. The following continuity equation is introduced,[3]

$$\frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = -\frac{\partial u}{\partial x} = f \tag{6}$$

along with two scalar functions $\psi(y, z)$ and $\phi(y, z)$ that satisfy the equations, respectively,

$$\frac{\partial^2 \psi}{\partial y^2} + \frac{\partial^2 \psi}{\partial z^2} = -\bar{\xi} \tag{7}$$

and

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = f \tag{8}$$

Again following the assumptions for wake-integral development put forth by van Dam et al.,[5] one can obtain the following simplified wake-integral expression for induced drag:

$$D = \frac{\varrho_\infty}{2} \int_{S_2} (\psi \bar{\xi})\, \mathrm{d}S \tag{9}$$

It is interesting to note that after the simplifying assumptions have been taken, whether from the free-flight analysis of van Dam et al.[5] or the wind-tunnel analysis of Wu et al.,[3] the wake-integrals obtained are the same. Wu et al. give enlightening physical meaning to the terms involved in the wake-integral development.

All computations for the wake-integral expressions for both lift and induced drag are carried out on the plane S_2 oriented normal to the freestream flow. This plane is generated from the computational grid at selected stations downstream of the wing using linear interpolation. The vorticity component normal to the plane is calculated within the computational domain and then mapped from the computational domain onto S_2 using a search algorithm.[6]

The stream function ψ can be obtained as a solution to the Poisson's equation, Eq. (7). A method for evaluating ψ that was found to work quite well is given by the following equation[7]:

$$\psi(y, z) = -\frac{1}{2\pi} \int_S [\bar{\xi}(y', z') \ell_n r]\, \mathrm{d}S + \psi_0(y, z) \tag{10}$$

where $r^2 = (y - y')^2 + (z - z')^2$, and boundary conditions are applied through ψ_0, where ψ_0 is the solution to the Laplace equation

$$\frac{\partial^2 \psi_0}{\partial y^2} + \frac{\partial^2 \psi_0}{\partial z^2} = 0 \tag{11}$$

Transforming the terms in Eq. (11) for use on the curvilinear transverse grid (η, ζ) and discretizing using second-order accurate difference approximations yields a block-tridiagonal system. The resulting linear system was solved using either a direct method or an iterative successive over-relaxation (SOR) method. The boundary condition on ψ is the homogeneous Dirichlet boundary condition, $\psi_b = 0$, where the subscript b denotes the boundaries of S_2.

Results and Discussion

The test case geometry here is that studied by van Dam et al.[5] The characteristic features of this wing are an elliptical leading edge and an unswept, straight trailing edge. The spanwise chord distribution is given by the equation $c(\eta) = c_r\sqrt{(1 - \eta^2)}$, where c_r represents the reference (root) chord that was taken as 1.0, and $\eta = z/(b/2)$.

Table 1 Lift and induced drag data for $X_t = 1$ wing at $M_\infty = 0.2$

α, deg	$\Delta x/c_r$	C_{Lsp}	C_{Lw}	C_{Dsp}	C_{Dw}	C_{Dll}
0	0.5	0.0001	−0.0001	0.0022	0.0002	0.0000
4	0.1	0.3470	0.3470	0.0100	0.0057	0.0054
4	0.5	0.3470	0.3470	0.0100	0.0054	0.0054
4	1.0	0.3470	0.3467	0.0100	0.0053	0.0054

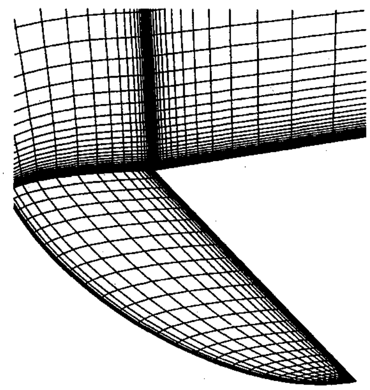

Fig. 2 Grid for the $X_t = 1$ wing.

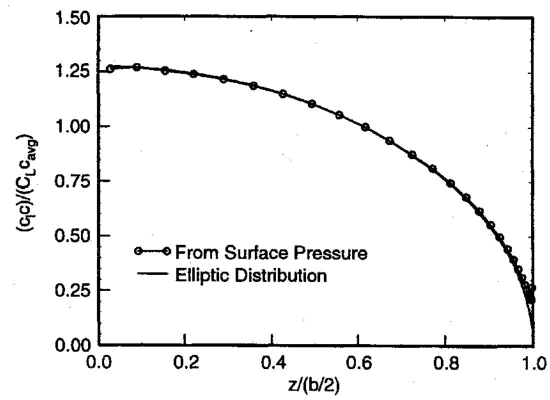

Fig. 3 Spanwise load distribution for $X_t = 1$ wing at $M_\infty = 0.2$ and $\alpha = 4$ deg.

The semispan was selected as $b/2 = 2.749$ to yield an aspect ratio of 7. The sectional shape for this wing is given by the NACA 0012 airfoil.

This geometry was modeled using a chordwise C grid and an H grid in the normal and spanwise directions. All far-field boundaries were placed six reference chords from the body. The flowfield was discretized using a point distribution of $121 \times 40 \times 51$ $(c \times n \times s)$ with a surface grid of 62×30. Figure 2 shows the surface grid and symmetry plane for this problem. Note the fine grid spacing used both normal to the body surface and in the axial direction at the trailing edge to accurately capture the trailing vortical sheet. The flowfield about this body was simulated at $M_\infty = 0.2$ for angles of attack of 0 and 4 deg. Comparison of the spanwise loading curve for this wing with a theoretical elliptic distribution is shown in Fig. 3. Here c_l denotes the sectional lift coefficient, c is the local chord, C_L is the lift coefficient for the complete wing, and c_{avg} is the average chord.

The wake-integral routines to compute the lift and drag from Eqs. (4) and (9), respectively, were applied to this test case for pitch angles of 0 and 4 deg. Values for lift and drag coefficients as obtained from surface pressure integration (denoted with the subscript Sp), wake-integral theory (denoted with the subscript w) and Prandtl's lifting-line theory (denoted with the subscript ll) for various downstream locations of the transverse plane are shown in Table 1. Here $\Delta x/c_r$ is the normalized distance from the transverse plane S_2 to the wing trailing edge.

A known problem with current numerical simulations of fluid flow is the generation of spurious vorticity in the flowfield. For an incoming irrotational, inviscid flow, the only source of vorticity is that shed in the trailing vortical sheet. However, it was observed that very low levels of background vorticity appeared in the entire flowfield. This poses a serious problem for the application of the wake-integral theory since, when implemented computationally, the force integrals in Eqs. (4) and (9) involve summation over the entire S_2 plane. In theory the only contributions to these integrals should come from the vorticity present in the infinitesimally thin vortical sheet shed behind the wing. However, because of numerical dissipation, this sheet obtains a finite thickness forming a vortical wake, making it difficult if not impossible to isolate it from the surrounding low-level vortical field. Though the background vorticity level is several orders in magnitude smaller than that present in the vortical sheet (wake), it is spread over an area several orders larger than that constituting the vortical sheet. Consequently, when this low-level background vorticity is included, the errors in the force integrals were observed to be significant. This dilemma is somewhat analogous to determining the edge of a boundary layer, which is arbitrarily defined. It cannot be overemphasized how critical a role the definition of the vortical wake edge plays in determining the prediction performance of this wake-integral method.

The approach adopted to circumvent this problem was to employ a vorticity filter. This filter was constructed by setting a threshold value for vorticity, and any vorticity values smaller than this threshold were then dropped out. The correct filter was obtained iteratively by selecting a threshold value and computing the lift coefficient using the wake-integral method. Since the lift coefficient using surface pressure integration is a proven good estimate of the true lifting characteristics of a body, the lift coefficient obtained from wake integration was compared with that obtained from surface pressures. The filter level that yielded a match (within a given tolerance) was selected as the correct filter for that case. The stream function and the drag integral were then computed using that filter. Since finding the edge of the vortical wake was found to be a function of the particular problem being simulated, a new filter threshold value had to be set for each case.

The results presented in the table indicate that the wake-integral method is accurate in predicting the lift-induced drag for a finite wing. It should be noted, however, that an accurate resolution of the vortical sheet is absolutely indispensable in this approach, since the more the vorticity in the sheet is dissipated (again forming what amounts to a wake), the greater the ambiguity in determining its extent into the surrounding field.

Conclusions

The capacity to predict drag due to lift using numerical simulations of the flowfield and a wake-integral approach has been studied. An important finding during the course of this research was the presence of low-level spurious vorticity in the flow. Future work must address the problem of requiring arbitrary filters to determine the extent of the vortical wake that ultimately governs the integral limits. Since the postprocessor uses a wake-based approach to compute the lift and drag about a lifting body, it was found that to correctly predict these force components it is necessary to very accurately resolve the flowfield in this region. An accurate resolution of the vortical field is critical when using the present approach for induced drag calculations. To maintain the performance of this approach, a sufficiently dense mesh is required in the wake region. This yields somewhat of a tradeoff situation since the errors associated with surface pressure integration can also be reduced with a sufficient increase in mesh density local to the body. Perhaps the use of a computational method specifically designed for the unmolested transport of vorticity would be the ultimate complement to the wake-integral method.

Acknowledgments

Funding for this project was provided by NASA Grant NAG-1-1271, with Dennis Bartlett as technical monitor. The authors would also like to thank the Mississippi Center for Supercomputing Research for providing several hours of Cray time for project software debugging and a portion of the runs presented here.

References

[1]Henderson, W. P., and Holmes, B. J., "Induced Drag—Historical Perspective," Society of Automotive Engineers, SAE Technical Paper Series, SAE Aerotech 1989 Conf., Sept. 1989.

[2]Janus, J. M., Chatterjee, A., and Cave, C., "Computational Analysis of Methods for Reduction of Induced Drag," AIAA Paper 93-0524, Jan. 1993.

[3]Wu, J. C., Hackett, J. E., and Lilley, D. E., "A Generalized Wake-Integral Approach for Drag Determination in Three-Dimensional Flows," AIAA Paper 79-0279, Jan. 1979.

[4]van Dam, C. P., Nikfetrat, K., Vijgen, P. M. H. W., and Fremaux, C. M., "Calculation and Measurement of Induced Drag at Low Speeds," Society of Automotive Engineers, SAE Paper 901935, Oct. 1990.

[5]van Dam, C. P., Chang, I. C., Vijgen, P. M. H. W., and Nikfetrat, K, "Drag Calculations of Wings Using Euler Methods," AIAA Paper 91-0338, Jan. 1991.

[6]Chatterjee, A., "Numerical Prediction of Induced Drag for Wings and Wing–Body Combinations," M.S. Thesis, Dept. of Aerospace Engineering, Mississippi State Univ., Mississippi State, MS, Dec. 1993.

[7]Lamb, H., *Hydrodynamics*, 6th ed., Dover, New York, pp. 219, 220.

New Mixed Van Leer Flux Splitting for Transonic Viscous Flow

Pascal Ferrand* and Stéphane Aubert†
École Centrale de Lyon, Ecully 69131, France

Introduction

THE Van Leer[1] (VL) scheme is a very popular flux vector splitting method used for inviscid transonic flow. It presents an excellent shock capturing capability with few oscillations in these areas. It has been demonstrated that the VL method has the ability and robustness to capture intense nonlinear waves, but it is well known that it generates an excess of diffusion at low Mach number. Indeed, this method is efficient for Euler equations discretization in transonic applications but not for Navier–Stokes equations discretization. On the other hand, the central schemes (CS) give very good accuracy in the boundary layer, but may allow large oscillations and losses across shocks. Different alternatives have been developed. For example, flux difference splitting[2] presents better accuracy through the contact discontinuities. Another way consists of modifying the original VL scheme to minimize the numerical error, with more[3,4] or less[5] success. Various hybridizing schemes have been also developed by several authors; see, for example, Coquel and Liou.[6] Basically, these approaches combine two complementary schemes by choosing the best scheme for each point of the grid.[4]

In this Note, we propose an alternative to the hybrid scheme that is very simple and resolves the same difficulties at the interface of the different schemes. Additionally, the introduction of this formulation is quasi-immediate in all VL MUSCL methods. This new approach is based on VL flux vector splitting and called the mixed VL method (MVL). The proposed MVL method has been constructed to conserve the advantages of the CS method at low Mach number and the advantages of the VL method elsewhere. This construction is based on the same VL principles to define the fluxes, and keeps the monoticity condition by defining an adapted weighting function between the CS and the VL scheme.

New MVL Method

In this Note, our objectives are the following: 1) to obtain an efficient modification at low Mach number, 2) to conserve the VL properties through the shocks, 3) to define a formulation easy to incorporate in existing VL methods, 4) to eliminate supplementary tests of local flow conditions, and 5) to limit the increase of memory and CPU time.

Received May 11, 1994; revision received July 21, 1995; accepted for publication July 25, 1995.

*Research Scientist, Laboratoire de Mécanique des fluides et d'acoustique, Centre National de la Recherche Scientifique, URA 263, Université Lyon I.

†Associate Professor, Laboratoire de Mécanique des fluides et d'acoustique, Éducation Nationale, Université Lyon I.