Zonal Approach to V/STOL Aerodynamics

Sungyul Yoo* and Daniel J. Strash*
Analytical Methods, Inc., Redmond, Washington

A zonal method based on the iterative coupling between an inviscid panel method and a Navier-Stokes (NS) method has been developed to analyze the complex flowfield associated with the V/STOL flow environment in order to improve the prediction of aerodynamic characteristics. The low-order panel method, VSAERO, developed by Analytical Methods, Inc. is used to solve for the entire flow region, which includes the V/STOL configuration and jet boundary model. In two separate investigations presented here, ARC3D, a well-known compressible thin layer NS code developed at NASA Ames Research Center, and APPL, a parabolized NS code developed at Amtec Engineering, are used to solve for the details of the jet region subject to the boundary condition provided by VSAERO. Overlapping boundaries are used for the purpose of coupling. Comparisons with the available experimental data for a jet-in-crossflow and a V/STOL configuration with lift-jet in a subsonic freestream show qualitative agreement with experiment and improved correlations over those by inviscid computations alone.

Nomenclature

C = section chord length
C_p = pressure coefficient
D_j = jet exit diameter
M = freestream Mach number
Re = Reynolds number based on freestream velocity and jet exit diameter
V_{cl} = jet centerline velocity
V_j = jet velocity/freestream velocity
V_n = normalized jet centerline velocity, $(V_{cl} - V_\infty)/[V_j(V_j - V_\infty)]^{1/2}$
V_∞ = freestream velocity
X = chordwise coordinate along freestream direction, in.
Y = spanwise coordinate normal to freestream direction, in.
Z = coordinate normal to X-Y plane, in.
α = angle of attack, deg

Introduction

SINCE the current trend in the design process for STOL and V/STOL aircraft requires the prediction of the effects of jet/airframe interaction, a more reliable method describing the flowfield around STOL and V/STOL bodies has been sought. The rising cost of wind-tunnel testing, the growing capacity of high-speed computers, and the development of new analytical tools has influenced this trend. Even though experiments using wind-tunnel testing provide reliable results, these involve procedures that are expensive and difficult to set up, especially when a variety of design conditions are tested. Thus, at an early stage of the design process, modern designers often turn to CFD (computational fluid dynamics) to obtain aerodynamic data. The modern trend towards a vectored thrust configuration designed to produce powered lift force during the takeoff and landing phases as well as provide a pilot with vectored force during combat maneuver has created a more complex environment that adds jet/surface interaction, jet-induced swirl, and jet entrainment to existing problems. The greatest impact on the airframe design is due to jet entrainment, which needs to be estimated at an early stage in the design process (Fig. 1).

The flowfield around a powered lift aircraft reveals complex phenomena, including vortical flows and separated flows from conventional high-lift devices caused by the high-energy jet flows that may be at large deflection angles relative to the direction of vehicle motion. High nozzle pressure often causes supersonic flow to exist in the region of the initial jet plane. The aerodynamic environment associated with a jet emitting perpendicular to a crossflow shows that viscous forces dominate entrainment of mainstream fluid and flow separation on the downstream side of the jet initial zone.

It is well understood that the NS equations [or Reynolds-averaged Navier-Stokes (RANS) equations for turbulent flows] are required to simulate this complex flowfield. Although, in recent years, the introduction of supercomputers has made it possible to solve the RANS equations in three-dimensional problems with some success, such solutions are restricted to local flow domains because of computing cost and large memory requirements. Therefore, solutions of the three-dimensional RANS equations are still impractical for a complete powered lift vehicle. For computational efficiency, a zonal approach dividing the flow domain into multiple zones where each set of simplified RANS equations is solved was pursued to attack this class of problem.

In the viscous equations, as a subset of the NS equations, the thin-layer NS equations (TLNS) and parabolized NS equations (PNS) are considered here. The TLNS approximation that drops the diffusion terms parallel to the body surface is quite successful in a class of flows in high-speed aerodynamics.[1] The PNS approximation, neglecting only the streamwise diffusion terms, with space marching, and subject to

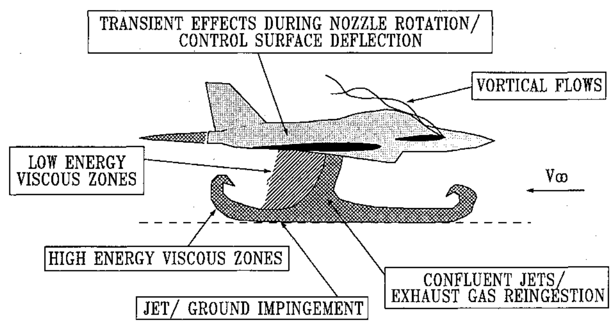

Fig. 1 Schematic of the flow environment around a V/STOL configuration.

Presented as Paper 89-2168 at the AIAA 7th Applied Aerodynamics Conference, Seattle, WA, July 31-Aug. 2, 1989; received Aug. 12, 1989; revision received March 16, 1990.

*Research Scientist. Member AIAA.

proper treatment of streamwise pressure, has proven successful when no massive separation exists.[2]

Meanwhile, in the class of inviscid equations, the integral method using the potential flow equation for subsonic or transonic flow with compressibility correction is currently favored by industry over the finite difference methods due to its computational efficiency. Flow conditions dominated by large areas of attached flow are treated with much success by this intergral method, commonly called a "panel method." Several variations of this method that utilize singularity distributions such as doublets, sources, or vortices in the form of panels to model the desired configuration, currently exist in industry. An integral equation of the second kind can be formulated using Green's identities to solve for the singularity distribution on the body. Typical examples of such methods applicable to subsonic flows are the Hess code,[3] VSAERO,[4] and PANAIR,[5] which are all three-dimensional panel methods. A comprehensive review among subsonic panel methods available as production codes, done by Margason et al.,[6] indicated that there is a tradeoff between the increased accuracy attainable with the higher order methods for a given number of panels and the significantly greater cost and execution time required for this implementation. VSAERO, a low-order method, provides adequate accuracy at a lower cost compared to the higher order methods.[4]

The general configuration modeling program, VSAERO,[4,7] was first extended for powered lift applications with very encouraging results.[8,9] The basic formulation of the method assumes that regions dominated by viscous effects are confined to thin boundary layers on the vehicle surface and to thin free shear layers representing jet "boundaries" and wakes. The problem is thereby reduced to one of free vortex sheets embedded in potential flow about the vehicle. The method includes an inner iterative "wake relaxation" loop to compute the strength and location of free vortex sheets representing jet and wake boundaries.[10,11] The computing effort for these calculations is quite reasonable, allowing for the analysis of complex configurations with the method.[7]

Roberts[12] coupled a PNS solution for the jet in a coflowing stream with a surface singularity panel method. In current STOL or V/STOL configurations, the vertical jet in strong crossflow creates a situation that may not be appropriately modeled by a PNS marching method alone, so that it becomes necessary to restore the RANS equations or to provide a means (such as the elliptic pressure equation) to restore ellipticity. A strong interaction between the jet and airframe may require multiple iterations to complete the interaction, unless the RANS-alone solutions are pursued for a flow domain large enough to impose a freestream condition at the boundary. In this regard, the panel method used for the inviscid calculation can substantially reduce the computing time involved in terms of number of iterations through an efficient coupling technique.

The concept using VSAERO/APPL was developed in previous work,[13] performed jointly by Analytical Methods, Inc. and Amtec Engineering. Due to some difficulties with the initial start of the APPL calculations at high deflection angles, a public code, ARC3D, was used as an alternative under a separate and subsequent investigation. This study, therefore, presents a zonal technique using the panel method VSAERO in combination with an NS method such as ARC3D or APPL.

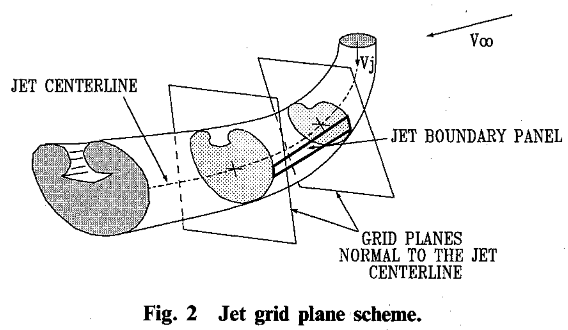

Fig. 2 Jet grid plane scheme.

Inviscid Flow Method

VSAERO[4] is a surface singularity panel method employing quadrilateral panels of uniform source and doublet. The code calculates the linearized potential given the normal velocity on surfaces bounding the flow. The method includes a coupled iterative calculation for viscous effects using integral boundary-layer methods and an inner iterative "wake relaxation" loop to compute the strength and location of free vortex sheets representing the jet and wake boundaries.[10,11]

In earlier work, the basic capabilities of VSAERO, i.e., its capacity to deal with arbitrary shapes, its coupled iterative schemes for wake relaxation and boundary-layer effects, and its simple vortex tube modeling of jet boundaries, had been evaluated on a number of V/STOL configurations.[8,9] During these evaluations, certain deficiencies in the jet model were identified relating to the modeling of entrainment and to the jet boundary relaxation when treating large jet deflection angles. A new jet model was installed to alleviate these difficulties.[14] This new jet model uses a set of grid planes constructed normal to the local jet axis shown in Fig. 2. In this scheme, relaxation calculations proceed along the jet axis in a series of steps as before, but now a normal grid plane is constructed at each step. The work[15] was partly concerned with evaluating the new model for the jets issuing at 90 deg to the freestream flow. The scope of this evaluation covered the relaxation of the jet boundaries, simple modeling of entrainment, and the effects of the boundary layer on the surface through which the jet issues. The jet model is basically similar to that of Shollenberger,[16] except that here panels representing the jet boundaries have a linear doublet distribution (i.e., constant vorticity) in the local streamwise direction.

The details of the method can be found in previous work.[13]

Viscous Flow Method

ARC3D

ARC3D, which was originally developed by Pulliam and Steger[1] at NASA Ames Research Center, is a thin layer NS method for unsteady compressible flow based on the implicit approximate factorization algorithm of Beam and Warming.[17] For turbulence closure of the equations of motion, the well-known Baldwin-Lomax turbulence model[18] is included. With these approximations in mind, the code has proven applicable to subsonic or transonic flow regimes without massive streamwise separation and adequate for flows in which the separation phenomenon is convection-dominated.

The numerical method used is the finite difference scheme with first- or second-order accuracy in time. The equations are factored (spatially split), which reduces the solution process to three one-dimensional problems at a given time level. The algorithm produces block tridiagonal systems for each space coordinate to be solved. Boundary conditions at the outer boundary are provided by the VSAERO solution, which is updated during the coupling procedure. On the jet centerline, values are linearly extrapolated in the radial direction, and treated with an averaging technique based on the previous time step. The details of the numerical algorithm are described in Ref. 1.

APPL

APPL, which was developed by Amtec Engineering, uses a second-order accurate iterative alternating-direction-implicit (ADI) algorithm to integrate the PNS equations in a single marching pass through the domain, and solves for the jet region with a three-dimensional pressure equation to recover the elliptic effects. This code uses a two-equation turbulence model with compressibility and axisymmetric flow corrections for the purpose of closure.[12,13]

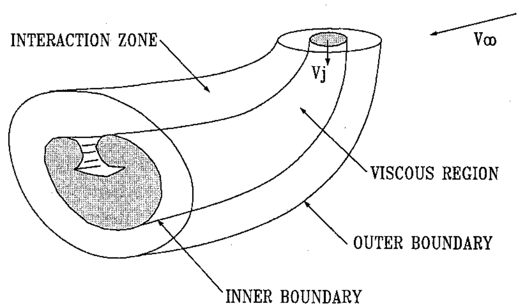

Fig. 3 Illustration of the overlap region in the coupling procedure.

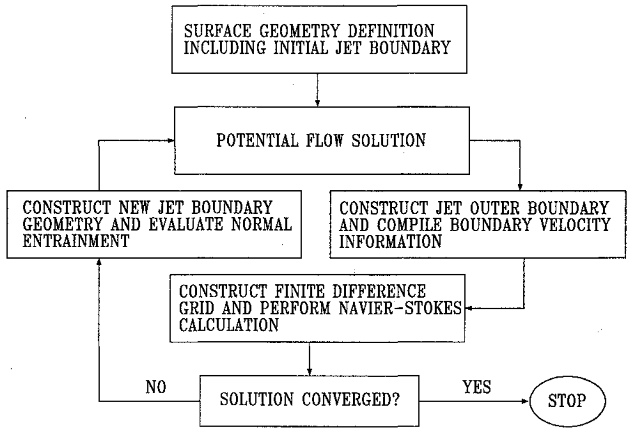

Fig. 4 Flow chart of viscous-inviscid coupling.

Table 1 Test cases

Case	Model	V_j/V_∞	α, deg	Codes used
I	60 deg jet model	8.00	0	VSAERO/APPL
II	90 deg jet model	4.00	0	VSAERO/ARC3D
		8.00	0	VSAERO/ARC3D
III	Lift-jet on a generic V/STOL model	5.00	0	VSAERO/ARC3D
		10.00	0	VSAERO/ARC3D
		5.00	4	VSAERO/ARC3D
		10.00	4	VSAERO/ARC3D

The numerical algorithm used is the finite-difference method using first-order upwind differences. A staggered mesh is used to discretize the flowfield at each plane. Pressure and density are located at the center of the cell, with velocity and temperature being defined at the cell nodes. In the PNS/elliptic pressure (EP) mode (PNS coupled to the three-dimensional elliptic pressure relation), the solution is initialized by marching through the domain in the PNS mode. The three-dimensional pressure field from this pass is saved as the initial guess for the PNS/EP mode.

The marching solution procedure in APPL requires an initial plane for which the flowfield is completely known. APPL either generates initial values on an inlet plane or obtains initial values from the VSAERO solution. The remaining boundary conditions for APPL are supplied by VSAERO. Further details are described in Ref. 13.

Coupling Procedure

The approach to coupling VSAERO with a viscous code has been based on an overlap between the viscous and inviscid regions (Fig. 3). This overlap is accomplished by locating the inner boundary approximately at the edge of the viscous domain and the outer boundary an arbitrary distance away. On the first pass, these boundaries are generated by VSAERO following the jet boundary relaxation calculation; subsequently, a viscous code provides these boundaries.

The interactions between the jet plume and the surrounding flowfield are simulated by the coupling of the component analyses. It is not sufficient to run each code once in its separate zone.[12] The codes must interact by means of overlapping boundaries so that the solution for the entire field is convergent and unique. Each code must provide information to the other code that adequately describes the physical processes being modeled in that zone. The viscous code, which calculates the jet plume development, provides VSAERO with the effects of entrainment at the plume boundary. VSAERO then uses this boundary information to compute the potential flowfield about the V/STOL aircraft on the outer boundary, and in turn to provide boundary conditions for the viscous code. The basic steps in the coupling procedure are illustrated in Fig. 4, as follows.

1) Perform the VSAERO calculation using the vortex tube model of the jet, jet efflux plane, and jet boundary relaxation. The calculation starts with a trajectory given by Margason's equation.[19]
2) Generate the outer boundary shown in Fig. 3.
3) Compute the velocities at the outer boundary.
4) Perform NS calculations subject to the velocities provided by VSAERO in step 3.
5) Assemble the information defining the inner and outer boundary geometry and the inner boundary velocities.
6) Perform VSAERO calculations subject to nonzero normal velocities on the inner boundary panels.
7) Repeat steps 3–6 until convergence is obtained.

Results and Discussion

Solutions have been obtained for several cases shown in Table 1. Calculations using VSAERO/APPL were completed under previous work[13] that contains the comprehensive case studies. Experimental data, where available, were compared with calculations. Simple turbulent jet flows were used to validate the numerical modeling incorporated in APPL and ARC3D. Results showing the centerline velocity decay for a turbulent axisymmetric jet in a coflowing stream are presented with experimental data[20] in Fig. 5 for $V_j = 2.17$.

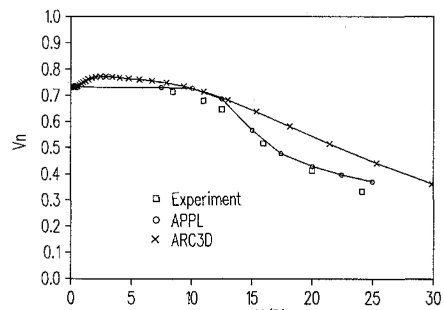

Fig. 5 Jet centerline velocity decay for turbulent jet in a coflowing stream.

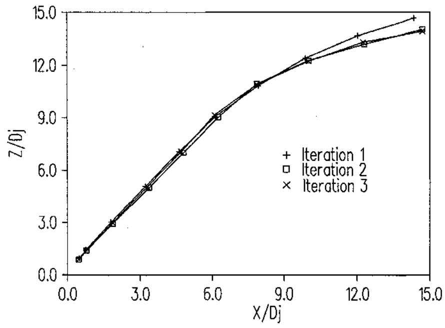

Fig. 6 Convergence of jet trajectory at 60 deg incidence.

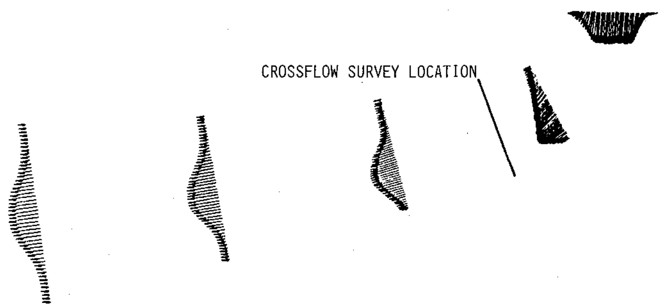

Fig. 7 Axial velocity distribution on the plane of symmetry for $V_j = 4.0$.

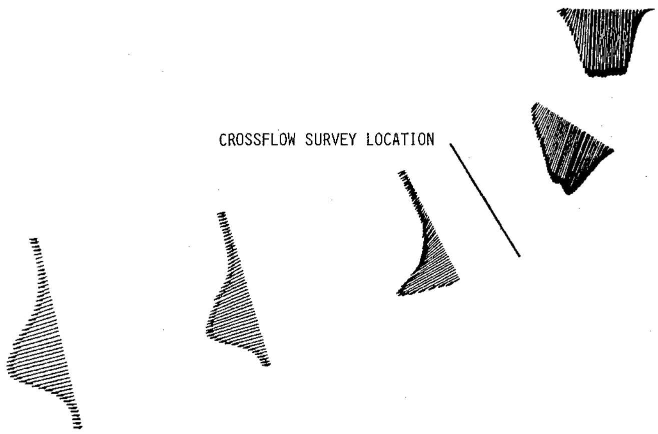

Fig. 8 Axial velocity distribution on the plane of symmetry for $V_j = 8.0$.

Case I: 60 deg Jet Model (VSAERO/APPL)

Calculations were made for a simplified nozzle-only geometry with a 60 deg injection angle at $V_j = 8.0$. In APPL, a two-dimensional pressure equation was employed in this computation. The jet centerline trajectory was calculated internally using a vorticity monitor. In VSAERO, a jet wake calculation was initiated along a trajectory by Margason.[19] From the second iteration onwards, jet wake grids are replaced by 312 (8 × 39) solid jet surface panels subject to a Neumann boundary condition of entrainment. APPL calculations here used an algebraic grid consisting of 21 radial, 40 axial, and 17 circumferential grid points. Up to five iterations were performed between VSAERO and APPL. VSAERO calculations were made on a MicroVAX computer at Analytical Methods, Inc. (AMI), and required about 1 h of CPU time per iteration, whereas APPL calculations made on a Cray X-MP 48 at NASA Ames Research Center required less than 10 min of CPU time per iteration.

Figure 6 provides a convergence history for the jet trajectory within the coupled analysis. After the second iteration, the trajectory is acceptably converged and the jet mesh could be frozen. In calculations for cases II and III, jet trajectories are frozen after the first iteration.

Case II: 90 deg Jet Model (VSAERO/ARC3D)

As part of the ARC3D validation process for use in this coupled environment, and in particular, V/STOL applications, a qualitative comparison was made with the jet-in-crossflow data of Fearn and Weston.[21] ARC3D calculations presented here used an algebraic grid consisting of 40 axial, 20 radial, and 17 circumferential (one side of the symmetry plane) for a total of 13,600 grid points, with $M = 0.1$ and $Re = 1.2$ million. About 1,100 iterations were required for convergence, and the CPU time required was 6×10^{-5} s/iteration/grid point on a Cray X-MP 24. The velocity vectors calculated by ARC3D on the plane of symmetry are shown in Figs. 7 and 8 for $V_j = 4$ and 8. These indicate a need to expand the domain in the initial region of the jet development, since ARC3D velocity profiles here are somewhat compressed. Also, the slow rate of decay of the maximum axial component of the jet velocity is consistent with the coflowing jet calculations presented in Fig. 5. The locations of crossflow velocity surveys have been indicated here for the two velocity ratios.

The crossflow velocity surveys (Figs. 9 and 10) indicated overall good qualitative agreement with the Fearn and Weston data. These surveys show the relative locations of the jet maximum axial velocity and the vortex core for both

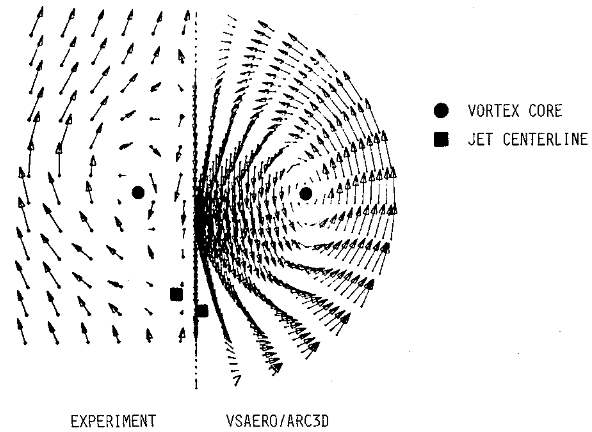

Fig. 9 Measured and computed crossflow velocity vectors for $V_j = 4.0$.

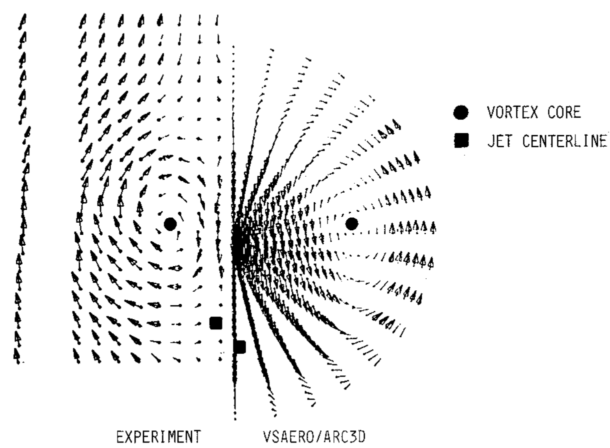

Fig. 10 Measured and computed crossflow velocity vectors for $V_j = 8.0$.

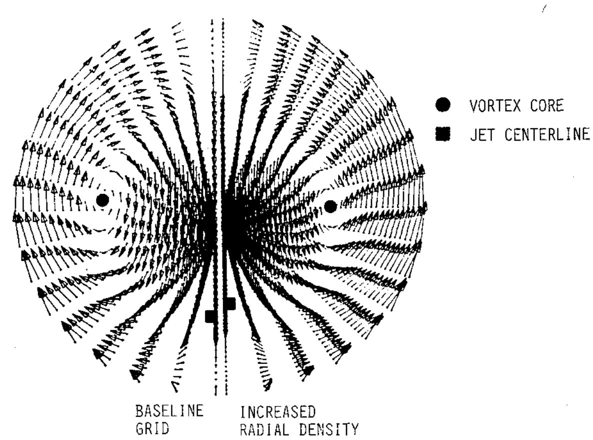

Fig. 11 Crossflow velocity vectors with baseline and increased density for $V_j = 4.0$.

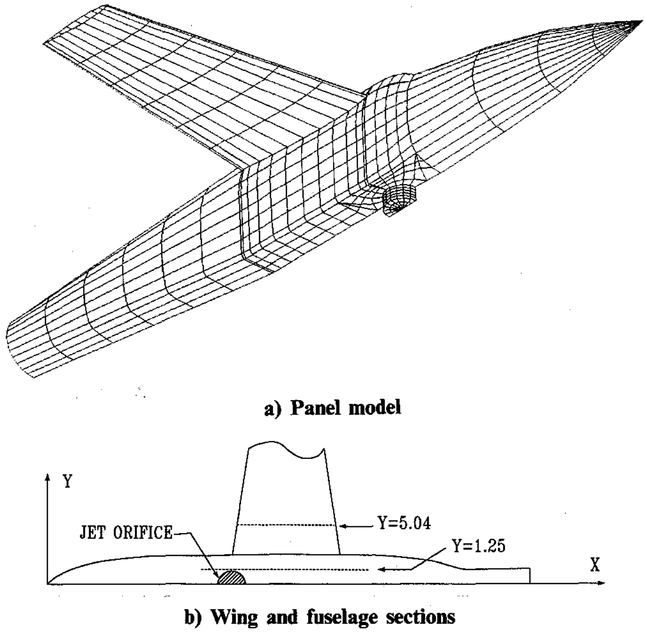

Fig. 12 V/STOL configuration.

theory and experiment. Here, it is shown that the jet centerline, as defined by the location of the maximum axial velocity, is slightly lower (in relation to the vortex core location) for the ARC3D calculation, which is consistent with the observations reached on the axial velocity surveys (Figs. 7 and 8).

A grid sensitivity study indicates that the relative distance between the vortex core and jet centerline is somewhat compressed when the number of grid points is doubled in the radial direction (Fig. 11), thus improving correlations with data. Also, calculations with ARC3D were performed on half as many mesh points as shown here in the circumferential direction, which resulted in very poor convergence behavior.

Case III: Lift Jet on a Generic V/STOL Model (VSAERO/ARC3D)

This model, developed by Minek and Margason,[22] consists of a simple slab-sided fuselage with a NACA 63A010 wing section (Fig. 12a). Figure 12b shows pressure tap locations on the wing and fuselage. It this case, the symmetry of the geometry is used for both the VSAERO and ARC3D calculations. Calculations are compared with experimental data,[22] which were not corrected for the blockage effects of the wind tunnel.

VSAERO used 762 panels for the first iteration and 1,041 panels (690 for the body, 351 for the jet surface) for the subsequent iterations. As previously mentioned, ARC3D used about 13,600 grid points (40 points along the jet axis, 17 points in the circumferential direction, and 20 points in the radial direction), with $M = 0.1$ and $Re = 1.2$ million. Approximately 1,500–2,000 iterations were made in the ARC3D calculation in order to reduce the RMS residual level by $0(10^{-3})$ from the original value. For the turbulent flow, the outer formulation in the Baldwin-Lomax model was used in the jet wake region. Both the VSAERO and ARC3D calculations were performed on a Cray X-MP 24. The CPU time required for the VSAERO and ARC3D calculations was about 2 min and 6×10^{-5} s/iteration/grid point, respectively.

The C_p distributions on the underside of the fuselage are compared with experimental data in Figs. 13a–d. It is noted that at the second iteration only slight improvement is made and no further iteration seems helpful. Results on the fore fuselage agree with experiment better than those on the rear, where both jet entrainment and separated flow on the leeward side of the jet exit play an important role. Incomplete modeling of the latter phenomena results in the consistent discrepancy between experiment and calculation. By adding the separated wake model at the junction between the fuselage and the jet surface, it was shown in previous work[9] that this correlation may be greatly improved. In other earlier work,[14] VSAERO/APPL calculations showed the same level of agreement with the experimental data. It is known from the experiment that jet effects are strongly felt at low jet velocity ratios and at the location nearest the jet.[22] As expected,

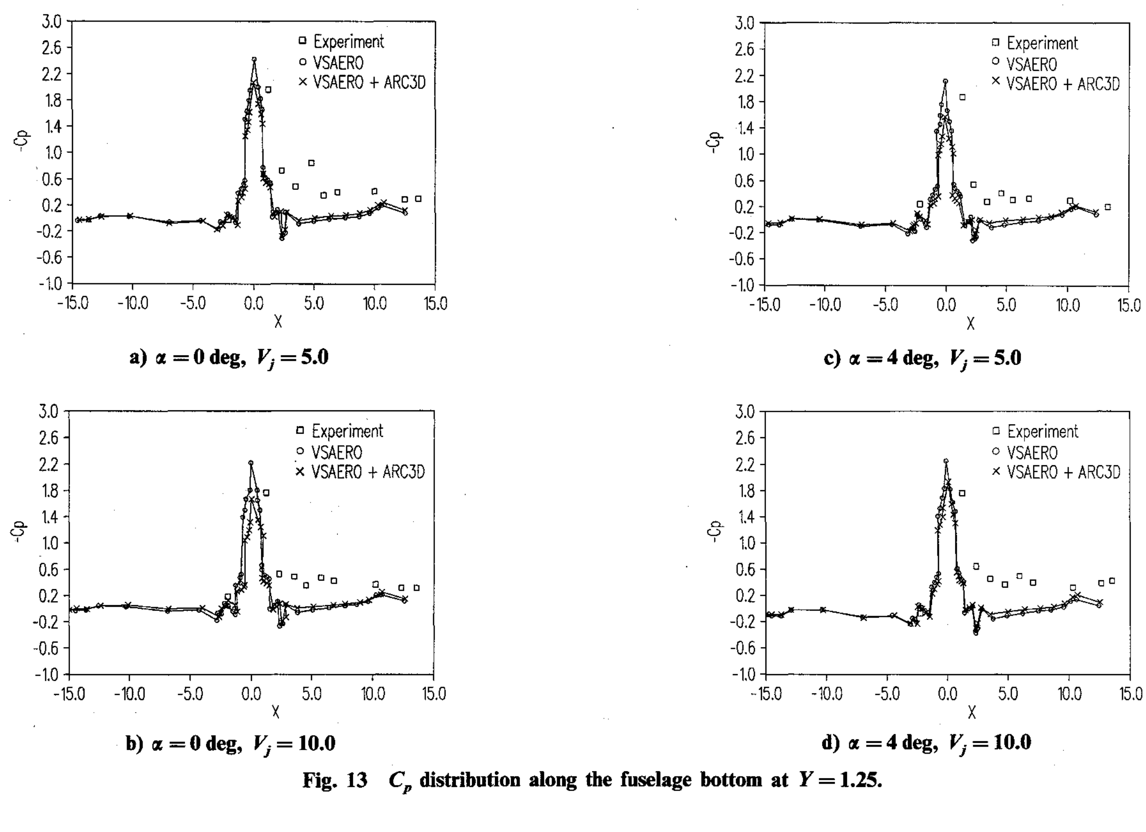

Fig. 13 C_p distribution along the fuselage bottom at $Y = 1.25$.

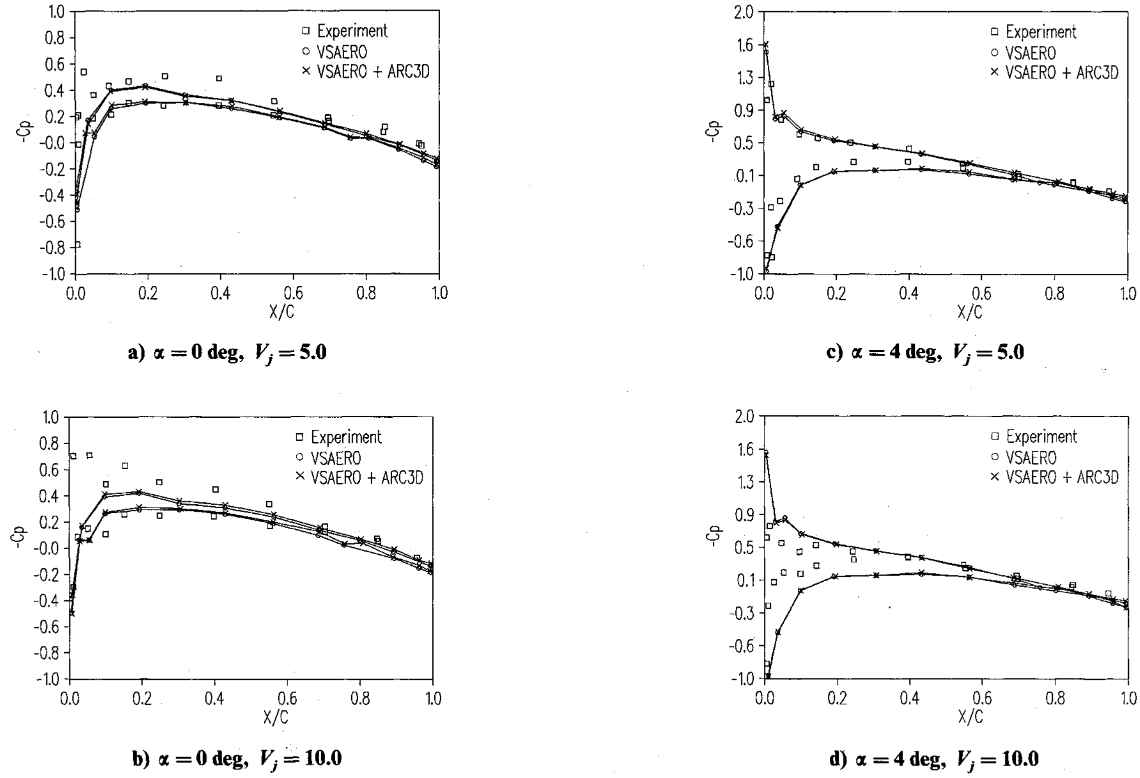

Fig. 14 Wing section C_p distribution at $Y = 5.04$.

current calculations do not indicate any noticeable change in the C_p distribution on the bottom of the fuselage for the cuts farthest from the jet plume within $V_j = 5–10$.

Figures 14a–d show the C_p distribution on a wing section. Generally, the C_p distribution on a wing section matches with the experimental data, except on the first 10% of the chord length from the leading edge. There appears to be no difference between VSAERO-alone solutions and coupled solutions. A tendency toward negative loading is noticed at a given angle of attack, due to downwash effects as V_j increases from 5 to 10. Only a slight change in the calculated C_p distribution is observed with changes in the jet velocity. This indicates that the jet effect on the lower surface near the leading edge is not sufficiently reflected in this modeling.

Conclusions

For computational efficiency, a zonal technique was developed to analyze the flowfield about a complete V/STOL aircraft and the jet issuing from the aircraft. A particular emphasis is put on the jet-induced effect on the aerodynamic characteristics. The entire aircraft, including jet simulation, is modeled by the potential flow method (VSAERO), whereas the jet region is modeled by an NS method (ARC3D or APPL). The two methods are made interactive through the coupling procedure described in this paper.

The following conclusions are derived from this study.

1) The new technique was shown to give qualitative agreement with experimental data for a jet-in-crossflow and to improve the correlation with experiment over inviscid computations alone for the V/STOL flow environment.

2) This technique allows for quantitative and qualitative analysis of the jet in the presence of the aircraft. This is an advantage over previous inviscid techniques that were unable to compute the complex jet flow and also superior to viscous methods that would routinely ignore the influence of the aircraft on the jet.

3) Some discrepancy between the calculation and experiment may be attributed to the lack of treatment of the separated wake at the juncture between the fuselage and the jet surface. This may be improved either by utilizing the separated wake model available in VSAERO or by incorporating the wake region into the NS domain.

4) Further improvement in the technique would involve automation of the coupling procedure to allow more practical use of this tool.

Acknowledgments

The development of the original concept and the basic work were carried out by Amtec Engineering, Bellevue, WA, and by Analytical Methods, Inc., Redmond, WA, under Contract NAS2-12166 funded by NASA Ames Research Center. The contract was monitored by Mike Dudley at NASA Ames. This article reports work completed under this contract as well as further work regarding ARC3D carried out by the second author under the internal research program at Analytical Methods, Inc. The authors wish to thank Donald W. Roberts for Figs. 5 and 6, made available here through the above contract. Also, the first author extends thanks to Brian Maskew for his advice throughout the duration of this contract.

References

[1]Pulliam, T. H., and Steger, J. L., "Implicit Finite Difference Simulations of Three-Dimensional Compressible Flow," *AIAA Journal*, Vol. 18, No. 2, 1980, pp. 159–167.

[2]Anderson, D. A., Tannehill, J. C., and Pletcher, R. H., *Computational Fluid Mechanics and Heat Transfer*, Hemisphere, McGraw-Hill, New York, 1984.

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

[4]Maskew, B., "Prediction of Subsonic Aerodynamic Characteristics: A Case for Low-Order Panel Methods," *Journal of Aircraft*, Vol. 19, No. 2, 1982, pp. 157–163.

[5]Magnus, A. E., and Epton, M. E., "PAN AIR—A Computer Program for Predicting Subsonic or Supersonic Linear Potential Flows about Arbitrary Configurations using a Higher-Order Panel Method," NASA CR-3251, April 1980.

[6]Margason, R. J. et al., "Subsonic Panel Methods—A Comparison of Several Prediction Codes," AIAA Paper 85-0280, Jan. 1985.

[7]Clark, D. R., Maskew, B., and Dvorak, F. A., "The Application of a Second Generation Low-Order Panel Method, Program VSAERO, to Powerplant Installation Studies," AIAA Paper 84-0122, Jan. 1984.

[8]Maskew, B., Strash, D., Nathman, J., and Dvorak, F. A., "Investigation to Advance Prediction Techniques of the Low-Speed Aerodynamics of V/STOL Aircraft," NASA CR-166479, Feb. 1983.

[9]Strash, D., Nathman, J., Maskew, B., and Dvorak, F., "The Application of a Low-Order Panel Method—Program VSAERO—to Powerplant and Airframe Flow Studies," AIAA Paper 84-2178, Aug. 1984.

[10]Maskew, B., Rao, B. M., and Dvorak, F. A., "Prediction of Aerodynamic Characteristics for Wings with Extensive Separations," AGARD CP-291, Sept. 1980.

[11]Maskew, B., "Predicting the Aerodynamic Characteristics of Vortical Flows on Three-Dimensional Configurations using a Surface Singularity Panel Method," AGARD CP-342, April 1983.

[12]Roberts, D. W., "Prediction of Subsonic Aircraft Flows with Jet Exhaust Interactions," AGARD CP-301, May 1981.

[13]Yoo, S. Y., Maskew, B., and Roberts, D. W., "Predicting Methods for Powered Lift Vehicle Aerodynamics," Analytical Methods, Inc, Redmond, WA, Rept. 8804, Aug. 1988.

[14]Maskew, B., "An Improved Prediction Method for Powered Lift Vehicle Aerodyamics," Analytical Methods, Inc., Redmond, WA, Report 8506, April 1985.

[15]Maskew, B., Strash, D., and Nathman, J., "An Investigation to Advanced Prediction Techniques of the Low-Speed Aerodynamics of V/STOL Aircraft," Analytical Methods, Inc., Redmond, WA, Rept. 8603, May 1986.

[16]Shollenberger, C. A., "Analysis of the Interaction of Jets and Airfoils in Two Dimensions," *Journal of Aircraft*, Vol. 10, May 1973, pp. 267–273.

[17]Beam, R., and Warming, R. F., "An Implicit Finite Difference Algorithm for Hyperbolic Systems in Conservation-Law Form," *Journal of Computational Physics*, Vol. 22, Sept. 1976, pp. 87–110.

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

[19]Margason, R. J., "The Path of a Jet Directed at Large Angles to a Subsonic Free Stream," NASA TN-D-4919, Nov. 1968.

[20]Landis, F., and Shapiro, A. H., "The Turbulent Mixing of Coaxial Jets," *Proceedings of Heat Transfer and Fluid Mechanics Institute*, Stanford Univ., Stanford, CA, 1951, pp. 133–146.

[21]Fearn, R. L., and Weston, R. P., "Induced Velocity Field of a Jet in a Cross Flow," NASA TP-1087, 1978.

[22]Mineck, R. E., and Margason, R. J., "Pressure Distribution on a Vectored-Thrust V/STOL Fighter in the Transition Speed Range," NASA TM-X-2867, March 1974.