Prediction Methods for Jet V/STOL Propulsion Aerodynamics

Max F. Platzer
Naval Postgraduate School, Monterey, Calif
and
Richard J. Margason
NASA Langley Research Center, Hampton Va.

Introduction

THE Navy's interest in vertical/short takeoff and landing (V/STOL) aircraft for deployment on smaller ships down to frigate size and on new vertical support ships to be introduced into the fleet in future years has created a need for the systematic study of all aspects of V/STOL aircraft technology. V/STOL aircraft needed to support future naval operations cover a performance spectrum comprising the following general categories: type A – general purpose subsonic aircraft for basic fleet support missions (transport and antisubmarine warfare); and type B – tactical aircraft for attack and fighter missions.

The propulsion concept for type A requirements may be the lift/cruise fan. The most suitable type B aircraft propulsion system will have to be based on a thorough evaluation of several competing concepts: lift/lift cruise engines as incorporated in the VAK-191 prototype aircraft; thrust-vectoring devices as applied in the Harrier-AV-8A operational aircraft; and ejector wings such as those currently under development for the XFV-12A technology demonstrator prototype aircraft.

V/STOL aircraft, incorporating either the lift/lift cruise engine, vectored thrust, lift fan, or ejector wing concepts, all generate high-speed jet exhausts. Thus the term *jet* V/STOL is used to characterize this class of aircraft. However, special attention and understanding are required for a number of unique problems which occur on jet V/STOL aircraft. A detailed classification and listing of those problems can be found in the U. S. Navy V/STOL Technology Assessment[1] prepared by the Naval Air Systems Command in June 1975.

The propulsion systems of jet V/STOL aircraft must be designed to generate the forward thrust for conventional flight, the lifting force for hovering flight, and in many cases, additional force components for control purposes during hover and transition flight. This dual- or triple-function character of V/STOL propulsion systems leads to design features which make them significantly different from conventional airbreathing propulsion systems. In most aircraft configurations, the lifting force for hovering flight is provided at two or more locations to permit moment trim and control about all three axes. This requirement is met either by location of propulsion units at several places on the aircraft (i.e. the lift-plus/lift-cruise engine concept), or by the addition of internal flow transfer ducts (i.e. the augmenter concept or the tip-driven lift/cruise-fan concept), or by use of shafts to transmit power (i. e. helicopters or shaft-driven lift/cruise-fan concept). Each of these concepts introduces its own set of problems. For example, the transfer ducts often exhibit complex geometric features such as asymmetric cross-section transitions, short-radius bends, branches, and so forth. In addition, they often terminate in similarly complex flow distribution devices such as slots, vanes, nozzles, wide-angle diffusers, and so forth. In the case of V/STOL aircraft using the ejector principle, complicated turbulent flow-mixing processes occur in a three-dimensional mixed internal/external flow environment.

In all cases, the lift jets issuing from the aircraft mix with the external flow to generate extremely complicated three-dimensional flow phenomena. In general, the jet-induced effects cause additional forces and moments on the aircraft both statically and dynamically. The character and magnitude of these jet-induced effects is influenced by the flight regime being encountered as well as the specific aircraft configuration. Several authors have surveyed and described these V/STOL propulsion-induced effects.[2-5]

In the low-speed flight regime, these effects are present in several areas: 1) the performance losses sustained while hovering out-of-ground effect, 2) the performance changes and hot-gas ingestion problems occuring while hovering in-ground effect, and 3) the induced aerodynamic effects in transition flight from hover to wing-borne flight out-of-ground effect and in-ground effect during a STOL flight mode. Resolution of the conflicts which arise from the design requirements imposed by these different modes of flight present a significant challenge to the aircraft designer. Satisfactory solution is needed to provide necessary lift forces and adequate control power in low-speed flight.

Even in cruise flight, V/STOL aircraft have more complicated inlet integration requirements, lower fineness ratio, more difficult propulsion system selection requirements, and

M. F. Platzer graduated from the Technical University of Vienna with Diploma Engineer and Doctor of Technical Sciences degrees. He came to the United States in 1960 where he worked at the NASA Marshall Space Flight Center and the Lockheed-Georgia Research Laboratory on aerodynamic and aeroelastic problems of space launch vehicles and transport aircraft. Since joining the Naval Postgraduate School in 1970, he devoted his research efforts to various naval aviation problems and is also serving as a consultant to the Naval Air Systems Command.

Richard J. Margason graduated from San Jose State University (B. S. ME) in 1960 and Virginia Polytechnic Institute and State University (M. S. AE). He is currently employed at NASA Langley Research Center as head of the Low-Speed Aerodynamics Branch. He is in charge of the V/STOL tunnel at Langley and is involved with aerodynamics research for high-lift systems for long-haul aircraft as well as aero-propulsion interactions for V/STOL aircraft and rotorcraft.

Presented as Paper 76-932 at the AIAA Aircraft Systems and Technology Meeting, Dallas, Texas, Sept. 27-29, 1976; submitted Oct. 12, 1976; revision received Nov. 2, 1977.
Index categories: Aerodynamics; Jets, Wakes, and Viscid-Inviscid Flow Interactions; Airbreathing Propulsion.

more complex nozzles when compared with conventional aircraft. These characteristics can lead to significant penalties in the ability of a V/STOL aircraft to perform its mission.

As a result, additional development activity is required for both low- and high-speed flight regimes to provide an understanding of the associated flowfields. For this reason, from July 28-31, 1975, the Naval Air Systems Command conducted a workshop on "Prediction Methods for Jet V/STOL Propulsion Aerodynamics."[6] The conference was organized into eight sessions: 1) internal flow problems, 2) flows in vectoring devices, 3) thrust-augmenting ejectors, 4) lift fans, 5) ground interference and thermal effects, 6) jet-induced flow effects and induced loads during both 7) hover, and 8) transition flight. An earlier paper[7] summarized the conference including the experimental results. The present paper concentrates on the prediction methods which represented less than 20% of the conference papers.

Current Status of Prediction Methods

Generally, the following hierarchy of prediction methods can be distinguished in modern aircraft aerodynamics: 1) purely empirical methods or engineering methods with large empirical content, 2) inviscid (potential) flow methods, 3) patched potential-flow/viscous-flow methods, and 4) "exact" viscous-flow methods.

Empirical Methods

The inherently limited range of validity of empirical methods makes it difficult to review them within limited space and in the absence of specific configurations. Thus, no attempt is made here to do justice to these methods. However, the general objective of a prediction method is, of course, to determine the aerodynamic characteristics of a new configuration prior to the availability of wind-tunnel and flight-test information. On V/STOL aircraft, this task is complicated by the presence of large, viscous, turbulent, and, in most cases, separated flow regions which present an enormous hurdle to the development of "rational" fluid mechanics approaches (i.e. approaches based on solutions of sufficiently general governing equations which can account for these phenomena). Thus, empirical methods are likely to retain their utility in the design of V/STOL aircraft for many years to come.

Inviscid Flow Methods

The second class of methods which is applicable to configurations dominated by attached flow regions has seen a rapid expansion in recent years with the availability of high-speed large-memory computers. These methods are generally classified as panel methods which model rather arbitrary aircraft configurations by singularity distributions (sources, doublets, vortices). Typical examples of such methods are the Boeing TEA 230 program[8] and the Hess three-dimensional flow method.[9] It should be noted that jets or jet sheets can also be included in the modeling process by proper vortex and sink distributions. Typical panel methods as well as jet-flap methods are described in Ref. 10. These methods will be discussed later in this paper where their results are compared with experimental data.

Patched Potential-Flow/Viscous-Flow Methods

Methods typical of the third hierarchy are those coupling potential flow with the boundary-layer computations for the determination of overall aircraft drag or the performance of flap systems. A recent review of hierarchy 2 and 3 methods currently used for conventional aircraft was given by Tulinius and Margason.[10] The latest status of boundary-layer theory and computation can be found in Ref. 11.

Viscous-Flow Methods

Category 4 methods, that is, solutions based on certain approximations of the Navier-Stokes equations or complete Navier-Stokes solutions applicable to practical aircraft configurations are still in the early stages of development. Practical implementation of this solution class is heavily dependent on the development of adequate turbulence modeling and numerical methods which provide sufficient accuracy in reasonable computing times. It must be stated at the outset that the accurate numerical prediction of flows such as those described above (i.e. those exhibiting large flow gradients, turbulence, unsteadiness, three-dimensionality, and separated flow zones) is still unattainable at this time. Nevertheless, significant progress has recently been made in the treatment of simpler flows.[12-14]

Various numerical schemes have been investigated in the past 10 to 15 years to obtain solutions of the Navier-Stokes equations. These can generally be classified into finite-difference, integral relation methods, spectral methods, and finite-element methods. More detailed information concerning these methods can be found in the recent reviews by Peyret and Viviand[12], Roache[15], Patankar[14], Orszag and Israeli[16], and Shen.[17] The problem of turbulence modeling is reviewed in Ref. 18 and 19.

It should be noted that no answer can presently be provided to the often-asked question about the best numerical method to solve the Navier-Stokes equations, because the best scheme strongly depends on the prescribed boundary conditions and on the desired accuracy. In particular, efforts to achieve higher accuracy may lead to physically inconsistent schemes.[20] This problem of physical/numerical inconsistency is further enhanced by the dearth of exact reference solutions or of sufficiently detailed and reliable experiments. In view of these uncertainties, as well as because of the large computer run times, it is not surprising that the major effort in computational viscous flow solutions is still being concentrated on two-dimensional flow problems, blunt-body flows, leading-edge flows, shock-wave/boundary-layer interactions, and so forth. Nevertheless, a few three-dimensional flow solutions have recently been attempted.[21-28]

Internal Flow Studies

A duct may be defined as any channel with two or more identifiable inlet or exit openings. Such ducts may include subsonic and supersonic inlets, diffusers, nozzles, bends, and secondary ducts for distributing air to various V/STOL aircraft propulsion units.

Unfortunately, the conditions at the ports are almost never known, making it necessary to simulate adjacent elements by prescribing "realistic" distributions of mean velocity, turbulence properties, pressure, and so forth at the ports. Obviously, the specification of such distributions requires considerable experience and/or experimental information. The large variety of possible distributions vastly increases the number of cases to be studied. Furthermore, the turbulence levels typically encountered in propulsion system flows are quite high, introducing the additional problem of proper turbulence modeling. Also, there is a lack of generally adopted reference quantities and coefficient definitions for a consistent assessment of duct element performance. Therefore, it is not surprising that one-dimensional flow concepts, together with semiempirical corrections and adjustments, are mostly used at the present time for duct flow analysis and design.[29, 30]

Similarly, the papers on flow vectoring devices[6] provide further evidence that the performance prediction of these devices rests mainly on empirical data. Hodder[6] (p. 461) and Hudson[6] (p. 42) emphasize the need for improved prediction methods, especially analytical techniques to predict secondary flow patterns.

At the workshop several internal flow solutions were presented which are indicative of the present status of duct flow modeling. Fagan et al.[6] (p. 125) based their approach on steady inviscid flow assumptions and obtained a three-dimensional rotational flow solution by iteratively updating the computed stream surfaces starting with irrotational flow as a first guess. The results show that streamwise vorticity components of significant magnitude can be expected at the exit of turning ducts and that the resulting vortex structure significantly alters the pressure and velocity distributions. Eiseman and McDonald[6] (p. 83), in contrast to Fagan's analysis, used the viscous equations. The essential idea in this approach is to solve an approximate set of viscous equations rather than the complete three-dimensional Navier-Stokes equations or the inviscid three-dimensional equations together with boundary-layer theory. Stepwise integration in the direction of the primary flow from a given set of upstream initial conditions produces a parabolic flow approximation of the full Navier-Stokes equations whereby the flowfield is obtained by a sequence of two-dimensional calculations, thus substantially reducing the required computer time. This approach is due to Patankar and Spalding[28] and Briley[23] and is based on the assumption that viscous diffusion and thermal conduction in the primary flow direction are negligibly small and that variations of the viscous correction computed for the pressure field can be treated separately. In essence, this procedure can be regarded as a natural extension of the boundary-layer approach.

Work is presently in progress to apply the method to subsonic flow through a turbine vane passage.[25, 31] Fully automated computer graphics displays greatly add to the utility of these computations. Such a capability is presently being implemented at the NASA Lewis Research Center by B. Anderson and collaborators. Although applicable to a fairly large class of duct cross sections, it must be remembered that the method can be applied only to ducts with slowly varying geometries. This restriction arises from the parabolic approximation of the Navier-Stokes equation which requires positive axial (primary) flows. This means that primary flow separation cannot be treated although secondary flow separation; that is, the formation of streamwise vortices, can be studied. A typical solution for laminar flow in a square duct[23] is shown in Fig. 1. Further results using parabolic or partially parabolic approximations were recently obtained by Ghia and Sokhey,[32] Rubin et al.,[33] Patankar et al.,[34] Pratap and Spalding,[35] and Dodge.[36] The most recent work[37, 38] is aimed at removing these approximations by more sophisticated iteration procedures making it possible to predict three-dimensional flows with embedded separated flow regions.

Thrust Augmenting Ejectors

Since the first application of thrust-augmenting ejectors in the XV-4A Hummingbird V/STOL aircraft, significant advances in ejector technology have been achieved which have led to a reevaluation of the ejector concept. As a result, the Navy XFV-12A V/STOL prototype aircraft is currently under development using ejectors installed in the lifting surfaces. Advantages expected from this concept include good engine matching to the cruise condition, large control forces for vertical and transition flight, and low jet exhaust velocities and temperatures. However, the attractiveness of the thrust-augmented wing concept is critically dependent on the attainment of both high augmentation ratio and low ejector volume. These two inherently conflicting requirements make it mandatory to fully understand the various flow phenomena and parameters which influence ejector performance. As discussed by Schum[6] (p. 639) the augmenter performance is strongly dependent on the product of nozzle thrust coefficient and the square root of entrainment rate. Thus, compact primary jet nozzles which provide high entrainment rates at acceptable thrust efficiencies are essential components of practical V/STOL ejectors.

This problem places great emphasis on the need to develop techniques to increase the jet entrainment rates (e.g. by hypermixing[39] or other methods[40-42]),and, more generally, to improve the understanding of the underlying turbulent entrainment mechanisms. The ejector design problem is further complicated by the use of high-energy wall jets and by the presence of strong adverse pressure gradients and of three-dimensional flow effects. The presently available analytical methods are quite inadequate to describe these complicated flow processes and therefore, lengthy test programs are required for ejector optimization. This process is further hampered by uncertainties about transferring the results of model tests to the full-scale configuration, (e.g. Hill and Jenkins[6] p. 629, Garland[6] p. 518 and Stewart[43]).

Nevertheless, a number of useful ejector analyses have been developed in recent years which provide valuable design information and guidance. These can generally be classified into one-, two-, and quasi-three-dimensional methods. The one-dimensional methods[44, 45] build upon von Karman's classical analysis[46] but incorporate various loss mechanisms, primary/secondary flow mixing modes, and compressibility effects.

Two-dimensional flow effects can be incorporated in an integral manner by using empirical laws for the entrainment rates and by assuming selfpreserving shapes for the jet velocity profiles. The resulting computing times are very short.[47]

A more rigorous two-dimensional flow solution was recently obtained using finite-difference techniques together with an eddy viscosity model.[48, 49] A different mixing length is used for each distinctly different region of the flow, based on information available in the literature. In this way, the jet velocity profiles develop as a result of the turbulent stresses, but the method requires two orders of magnitude greater computing times than the previously described entrainment

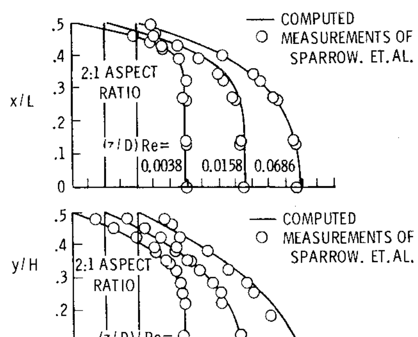

Fig. 1 Variation of the axial velocity ratio for selected axial locations in a duct with 2:1 aspect ratio.[23] The top plot gives the profiles across the wide dimension at $y/H=0$. The lower plot gives the profiles across the narrow dimension at $x/L=0$. (Reprinted with permission from the *Journal of Computational Physics*, Vol. 14, 1974, p. 24.)

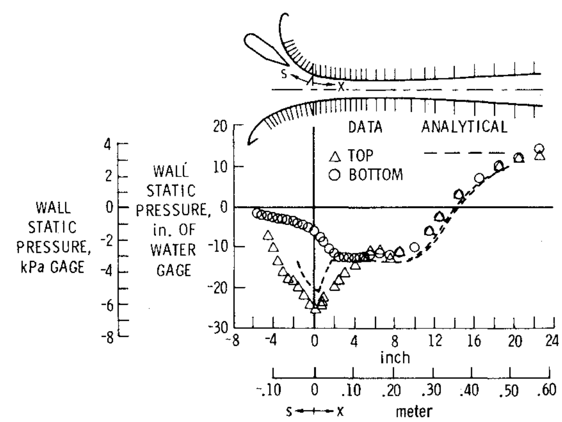

Fig. 2 Comparison of experimental and analytical wall static pressure distributions for an asymmetric ejector mixing section.[49]

method. An example of such a computation[49] is shown in Fig. 2.

Two attempts to predict three-dimensional flow effects have recently been published using the finite-element[26] and the finite-difference[27] approach. This work is based on the previously described parabolic flow approximation of the Navier-Stokes equations in combination with the two-equation turbulence model of Launder and Spalding[50] and computations are attempted for increasingly complex flow problems. The greater challenges yet to be faced in the future can be appreciated by remembering that the above approaches still assume a two-dimensional channel and only the jets are fully three dimensional (hypermixing). Thus, ejector end wall and taper effects are as yet unaccounted for.

An additonal problem area is introduced by the need to predict the augmenter wing characteristics in forward flight. Stewart's[6] (p. 664) measurements on a high-aspect-ratio augmenter wing configuration are in good agreement with Wilson's two-dimensional augmenter wing solution.[51] Also, three-dimensional jet wing computer programs[52, 53] have been developed, but little information is available on low-aspect-ratio configurations with swept and tapered ejectors typical of the XFV-12A aircraft.

Inlet Flows

The engine inlet flow problem is particularly critical on V/STOL aircraft due to the need to operate efficiently over a wide range of mass flow, flight speed, and incidence angle. Stockman[6] (p. 722) has reviewed the inlet design program currently used at the NASA Lewis Research Center. It is schematically summarized in Fig. 3.

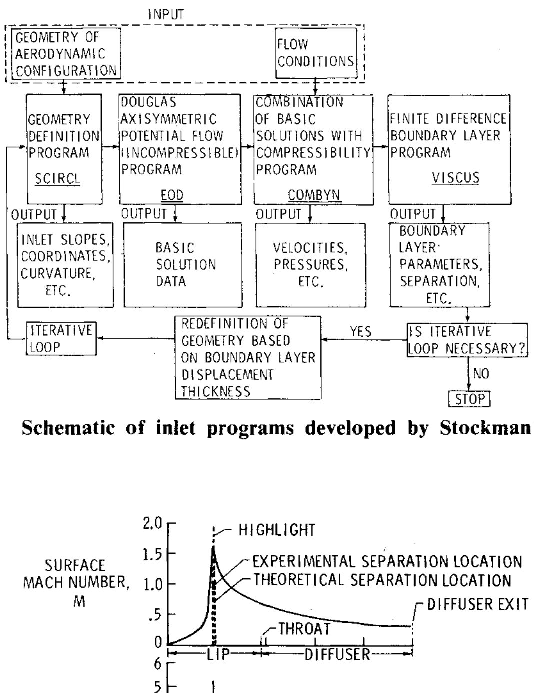

Fig. 3 Schematic of inlet programs developed by Stockman[6] (p. 722).

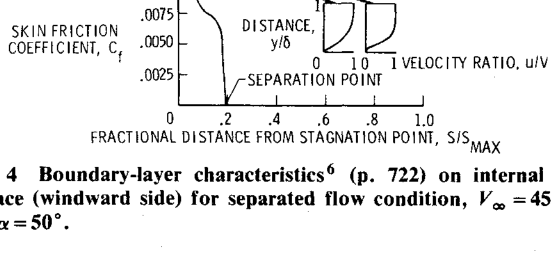

Fig. 4 Boundary-layer characteristics[6] (p. 722) on internal inlet surface (windward side) for separated flow condition, $V_\infty = 45$ m/s and $\alpha = 50°$.

A sample calculation[54] is shown in Fig. 4 illustrating the boundary-layer characteristics for a separated flow condition for an inlet at a 50 deg incidence angle. As can be seen, good agreement is obtained between the experimental and theoretical separation locations. However, it should be noted that the boundary-layer calculations are two-dimensional, and therefore, neglect three-dimensional, that is circumferential, flow effects. Also, at present, shock boundary-layer interactions are neglected which are of interest in many inlets with locally supersonic flows. The most recent developments in inlet flow computations can be found in Refs. 55 and 56 and in an informal report summarizing the proceedings of an inlet workshop held at NASA Lewis, January 11-13, 1977.

Lift Fan Modeling

For an understanding of lift-fan operation, it is essential to determine the combined inlet/through-flow/exit flow solution. Schaub[6] (p. 777) presented a solution to this problem by modeling the crossflow-induced inflow distortions by means of a polar lattice vortex, potential flow model. The crossflow-induced distortions at the fan's exit plane are simulated by a three-dimensional network of vortex lines attached to the surface of the jet plume whose centerline trajectory is described by an experimentally determined functional relationship. The fan itself is modeled by a collection of constant-flow-area fan segments whose average through-flows are determined from standard cascade loss and deflection data. These segment through-flows are matched with the external flow by adjustment of the back-pressure distribution to account for the interaction between the crossflow and the efflux jet. The complete fan-in-crossflow solution then is obtained by coupling together the three analytical models and iterating toward an operating point.

This model made it possible to determine the aerodynamic performance of lifting fans subjected to two simultaneous distortions: A potential flow distortion in the inlet, and a distortion of the back pressure due to efflux jet and crossflow interactions. The parallel compressor concept was used to divide the fan into eleven circumferentially spaced, equal-area segments. Thus it allowed each segment to find its own operation point consistent with the imposed upstream and downstream boundary conditions. Comparisons with experiment are shown in Fig. 5 indicating reasonable agreement.

Ground Effect

A unique flight condition of V/STOL aircraft occurs near the ground where the propulsion flows impinge on the ground, flow outward from the aircraft, and in some cases, form fountains which flow up to the aircraft (Fig. 6). These so-called recirculation flowfields can cause several performance losses on V/STOL aircraft. At the present time, the

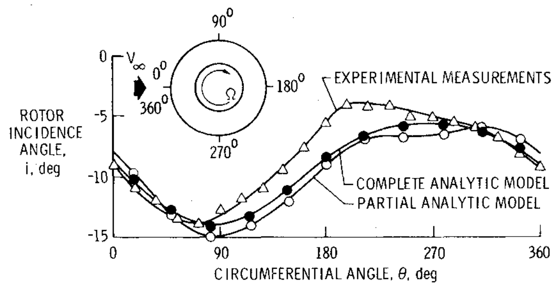

Fig. 5 Comparison of the rotor incidence angle in the inlet a distance of 0.625 diam above the fan plane which was obtained from experimental measurements and from two analytic models[6] (p. 777). The partial analytical model uses a segmented fan model while the complete analytic model adds inlet and jet plume models, $V_\infty = 32.5$m/s, fan speed = 10,000 rpm, and tip Mach number = 0.468.

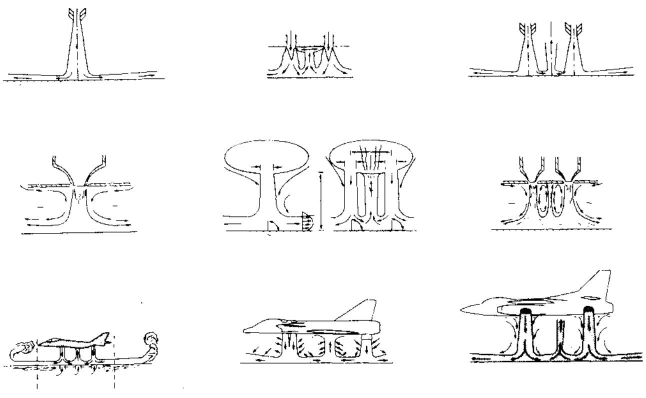

Fig. 6 The workshop provided many qualitative descriptions of flowfields during hover in ground effect.

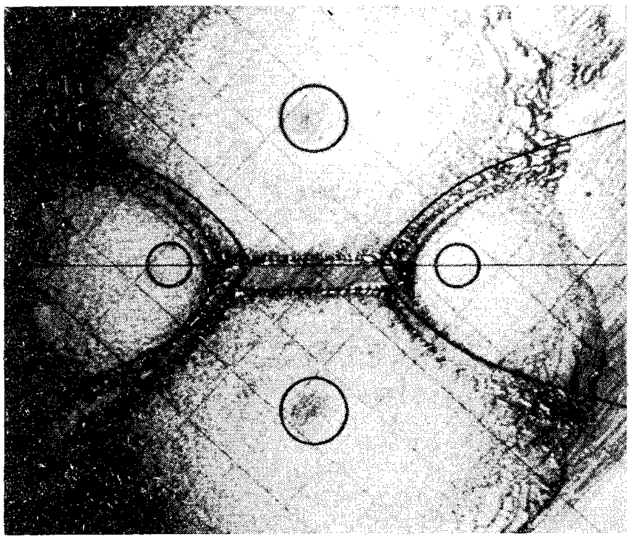

Fig. 7 Ground-plane oil flow obtained from a four-jet array with stagnation lines obtained from a potential-flow theory[6] (p. 381) superimposed.

precise description of these flowfields is not possible analytically. The most common procedure for studying such flowfields is to use experimental test facilities.[6, 7] However, some analysis of ground flow patterns has been successful.

The ground flow streamline patterns which result from jets impinging on the ground is illustrated in Fig. 7 for a four-jet pattern. This photograph was taken using an oil-flow technique and shows the lines of stagnation flow that were obtained by an accumulation of oil where the flow impinges and goes vertically upward toward the source of the jet. Superimposed on the stagnation lines are some calculated patterns using momentum relationships developed by Grumman.[57] The agreement for the four-jet pattern is quite impressive. This is one of the first computational procedures which gives an appreciation for recirculation flowfields on the ground. However, there is a significant amount of additional work required to describe the fountain flows and the impingement on the vehicle, and to ultimately provide a means of calculating the flows that might be induced into the inlet of an engine.

Exhaust Gas Ingestion

When exhaust gases impinge on the ground, complex flow patterns are often formed which can cause thermal effects on the ground and hot-gas ingestion in the engine inlet. There is qualitative data which describe the hot-gas ingestion problem. These data show that such effects are very dependent on airplane configuration and that very small changes (such as

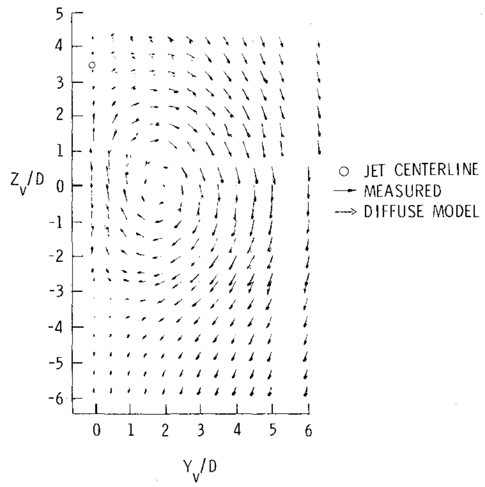

Fig. 8 Comparison of measured and calculated velocity fields for one vortex from the vortex pair generated by a lifting jet in a crossflow[6] (p. 918), $\delta_j = 90°$, $X/D = 8.3$, and $V_e = 0.125$.

location of jet exits, engine inlets, auxiliary inlets, or wings, or the division of thrust or nozzle deflections among the various lifting jets) can have very pronounced effects on the hot-gas ingestion. At the present time, there is no satisfactory method for estimating hot-gas ingestion. The only semiempirical method that comes close is that by Behnert.[58]

Induced Performance Effects in Hover

The induced performance effects sustained while an aircraft is hovering in ground effect often cause lift loss which is referred to as suckdown. However, for some configurations lift gains from fountain effect have been demonstrated at some heights. At the present time, satisfactory prediction of this phenomenon relies heavily on model measurements and no analytical methods are available for multijet configurations.

Characteristics of Lifting Jet in a Crossflow

In order to provide a simplified basis for the study of typical V/STOL flowfields, many investigators have concentrated on the turbulent flow of a subsonic round jet exhausting through a large flat plate into a uniform subsonic crossflow. Experimental results[6] (p. 918) presented in Fig. 8 indicate the following description of the vortices. The vortex pair is formed close to the jet orifice as relatively concentrated vortices with an initial strength that is roughly proportional both to the velocity of the jet at the orifice and to the diameter of the jet. The vortices are deflected by the crossflow, and they diffuse at a rate which is a function of the jet-injection angle and the arc length along the vortex curve, but which is a weak function of the effective velocity ratio. The vortices gradually weaken each other by the diffusion of vorticity across the symmetry plane.

One paper[6] (p. 918) presents some of the results of an extensive experimental investigation of the velocity field associated with a jet in a crossflow. Two simplified two-dimensional empirical models for the pair of contrarotating vortices are developed. The physical properties of the experimental vortices (their strength, location, and diffuseness) are inferred from selected velocity measurements taken in cross sections perpendicular to the trajectory of the vortex pair. The vortex flowfield results (Fig. 8) show the degree to which one empirical model describes the measured velocity field at 8.3 diam from the jet exit along the vortex curve for an effective velocity ratio of 0.125. The lightweight arrows are the velocities calculated from the diffuse empirical vortex model, and the heavy arrows are the measured velocities. The diffuse vortex model describes the projection of the velocity onto the cross section remarkably well.

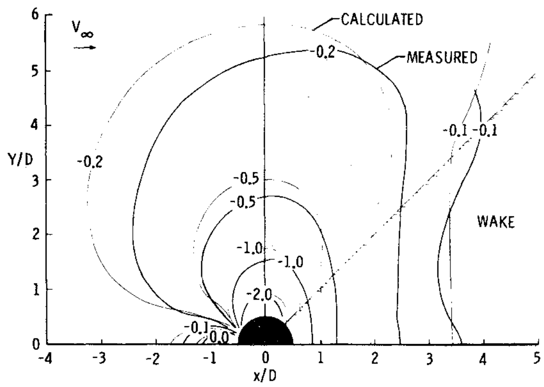

Fig. 9 Comparison between calculated and measured surface pressure distribution around a lifting jet in a crossflow[6] (p. 918), $\delta_j = 90°$ and $V_e = 0.125$.

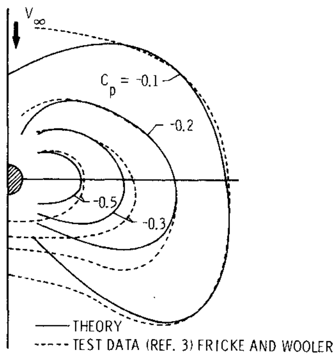

Fig. 10 Comparison of Wooler sink-doublet theory and measured surface pressure distribution around a lifting jet in a crossflow.[61] $\delta_j = 90°$ and $V_e = 0.125$.

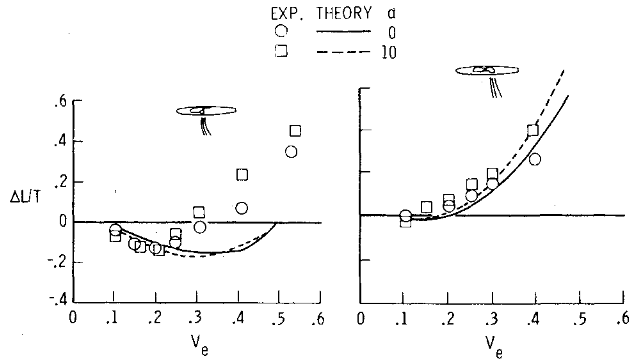

Fig. 11 Effect of chordwise position of lift/cruise nozzles deflected 90° on propulsion-induced lift increment including a comparison between Wooler's sink-doublet method and experimental data[6] (p. 1016).

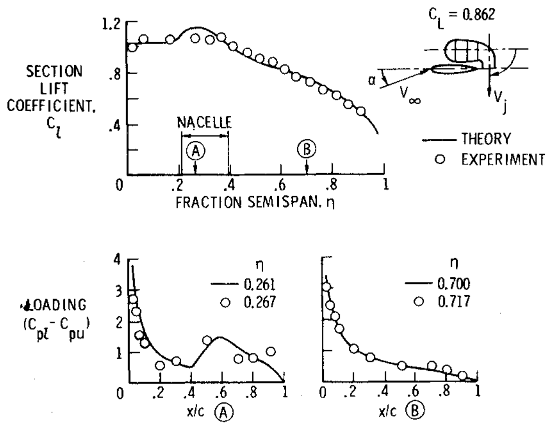

Fig. 12 Comparison of theory and experimental span and chord loadings where the experimental loadings were scaled to provide the same total lift coefficient as theory. The theory used a strip of horseshoe vortices for the jet model[6] (p. 743), $\delta_j = 90°$, $V_e = 0.2$, $C_L = 0.862$, $x_j/c = 1.39$, and $z_j/d_j = 0.44$.

The vortex models also provide a method of reasonably predicting the surface pressure distribution on the flat plate through a potential-flow approach which uses empirical equations to describe the vortex characteristics. Figure 9 shows a comparison between the potential-flow calculations and experimental results[59] at an effective velocity ratio of 0.125. The heavy lines are the experimental contours of the constant pressure coefficient on the flat plate. The light lines are the contours from the potential-flow calculations. These results show good agreement with experimental data.

A number of analytical formulations of the problem of a single jet exhausting into a crossflow exist.[60,61] In one of these references, an entrainment model was developed from dimensional analyses and physical considerations. The force on the jet boundary as a result of the pressure differential around the jet was accounted for by a crossflow drag. The geometry of the jet cross section was represented by an ellipse. Assuming constant and equal density for the crossflow, the continuity and momentum equations were solved for the jet path. The jet-induced velocity field was then determined by replacing the jet by a distribution of sinks and doublets. This sink-doublet model may then be used to compute the pressure distribution around a single lifting jet. Typical results[62] for Wooler's sink-doublet model are presented in Fig. 10 for the single jet exhausting at an angle perpendicular to the freestream at an effective velocity ratio of 0.125. This model is currently the most commonly used method for V/STOL aircraft aerodynamic predictions.

Snel[6] (p. 951) presented a calculation model for the prediction of the properties of a jet exhausting at an arbitrary angle into a three-dimensional nonuniform main flow. The model is based on the equations of conservation of mass and momentum, integrated across the jet cross section. A new semiempirical relation for axial velocity decay is derived and used. The model can be used to calculate properties of jets exhausting into the flowfield around aircraft configurations. The calculated properties serve as a basis for the determination of jet-induced velocity field by means of an extension of the "panel method"[8-10] for potential-flow calculations.

Aircraft/Jet Analysis Methods

There were seven workshop papers which dealt with aircraft/jet analyses for the transition flight regime. Efforts to determine the aerodynamic characteristics of configurations in forward flight date back to the 1960's. Hickey[63] derived a simplified equation for induced lift predictions based on a combination of two-dimensional jet-flap theory and three-dimensional flap-loading theory. Most of the methods are potential-flow models with empirical corrections. No attempts to date deal successfully with viscous flow through either patched, potential, or viscous-flow methods.

Mineck[6] (p. 1016) presented experimental comparisons with Wooler's sink-doublet model.[61] The lift interference on the wing for two vectored-thrust configurations is presented in Fig. 11. The experimental results for vectored thrust near the 10% wing chord show a lift loss at low effective velocity ratios and a lift augmentation at the larger effective velocity ratios. The theoretical results show the correct trends although the magnitudes differ because of the flow separation on the lower surface of the wing behind the jet. With vectored-thrust jets located near the wing trailing edge (Fig. 11), the lift interference is beneficial and the theoretical results are in good agreement with experiment.

Rozendaal et al.[6] (p. 743) presented a theoretical representation of a vectored-thrust V/STOL aircraft in transition which is made up of three building blocks. The airframe and nacelles are represented by a vortex lattice; the inlet mass flow ratios are controlled by disk sinks (or sources); and the deflected jets are modeled by strips of horseshoe vortices (similar to another jet method[60]). Interactions between the inlets and the airframe are accounted for by adjusting the disk sink strengths until a desired value of inlet mass flow ratio is achieved. Although the effects of lift jets on the airframe are, of course, included, the influences of the airframe flowfield upon jet plume development are not.

For a wing with a lift/cruise nacelle, theoretical and experimental wing load distributions have been calculated at the same value of total C_L. The results are compared in Fig. 12, for $\delta_j = 90°$ and $U_\infty / V_{jo} = 0.2$. The agreement in spanwise C_ℓ distribution is very good. The chordwise loading distributions at spanwise station A differ near the nacelle, but this may be due to the approximate nacelle fairing representation used in the theory. At spanwise station B, the theoretical and experimental chordwise load distributions are in good agreement. Although, as indicated in Fig. 11, the maximum nacelle diameter was less than 20% of the wing semispan, it was found that the jet induces lift along the entire semispan. The same was found to be true for the inlet.

A recently developed set of theoretical models for jet-induced effects on V/STOL aircraft was presented by Siclari et al.[57,6] (p. 998). A literature review was made of the empirical and theoretical data that describe the development of free jets, operating out of ground effect (OGE), and jets impinging on a ground plane, in ground effect (IGE). From this review, simplifying assumptions were made to obtain: a set of mixing data which simulate the viscous entrainment of ambient air for both free jets and wall jets; the free-jet shape in crossflow to provide the jet blocking effect; and the jet

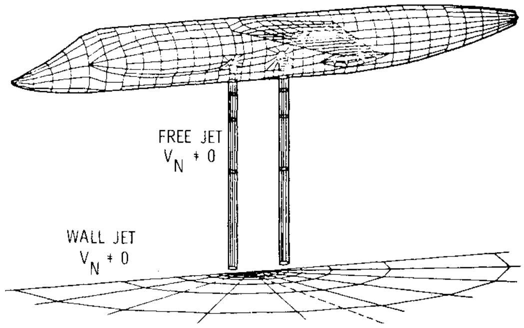

Fig. 13. Three-dimensional singularity panel approach to V/STOL aircraft prediction methods[6] (p. 998).

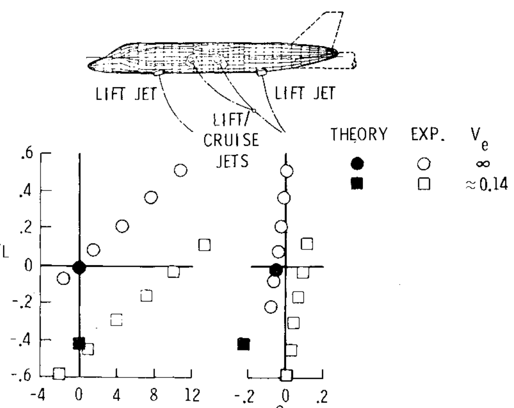

Fig. 14 Correlation between three-dimensional singularity panel method and experimental data[6] (p. 998).

formation along the ground which interacts with other jets leading to the "fountain" effect.

These jet data were combined with existing wing-body potential/theoretical methods in which the aircraft configuration is divided into a large number of panels, each containing a singularity (source/vortex) distribution (Fig. 13). The usual zero-normal-velocity boundary condition is used for the aircraft; the normal inflow velocity, simulating mass entrainment, is used for the jet and ground plane. The fountain momentum and incidence are calculated separately, with empirical data used to calculate additional fountain forces and moments.

Only a limited number of correlations with test data have been attempted. One correlation with VAK-191 is presented in Fig. 14 for lift and pitching-moment coefficients. These data are for a lift engine thrust deflection of 77.5 deg and a lift/cruise thrust deflection of 60 deg at approximately an effective velocity ratio of 0.14. For calculations at an angle of attack of 0 deg, the power-off results agree well with experiment and the power-on results agree well with lift coefficient. As in the results from Wooler's method (by Mineck,[6] p. 1016), the poor agreement for power-on pitching moment may indicate a need to include a separated flow representation in the wake portion of the jet.

Concluding Remarks

From July 28-31, 1975, the Naval Air Systems Command conducted a workshop to review the current status of prediction methods for propulsive flows and propulsion-induced aerodynamic effects which occur on jet V/STOL aircraft. The major topics covered included flows in propulsive ducts, propulsion-induced ground and thermal effects, aerodynamic loads induced during V/STOL and transition flight, flow-vectoring devices, thrust-augmenting ejector and lift-fan studies. The current status of prediction methods for V/STOL aircraft is best illustrated by the fact that over 80% of the workshop papers utilized experimental methods to study specific propulsive/aerodynamic flows and interaction effects. The existing prediction methods tend to be limited in their range of valid application and in the extent of their verification with experimental results. Prediction of viscous effects currently is quite primitive.

Most of the current analytic prediction methods are based on potential-flow methods which often use empirical procedures to represent propulsive flow characteristics. Near-term improvements in these methods will most likely use well-designed experimental investigations to improve the empirical representations and to include patched viscous/boundary-layer type solution methods.

Intermediate-term improvements will be achieved from the development of viscous models for specialized problems. These probably include solutions to simplified versions of the Navier-Stokes equations. Parabolic approximation solutions to Navier-Stokes equations have matured to the point of engineering application. Further work along these lines coupled with well-controlled experiments could provide significant progress.

For augmenter aircraft, there is need for both propulsion and aerodynamic experimental data and methods development. More fundamental experimental work is needed to better understand the entrainment and mixing mechanisms of primary jets and wall jets, especially in adverse pressure gradients, so that proper mathematical models of entrainment can be developed. Further experimental and analytical work is needed to extend current two-dimensional methods to account for three-dimensional effects, especially taper and end wall effects.

References

[1]Siewert, R. F., "U. S. Navy V/STOL Technology Assessment, Vol. I, Executive Summary," Naval Air Systems Command, Dept. of the Navy, June 1975.

[2]Barche, J., "Jet Lift Problems of V/STOL Aircraft," *V/STOL Aerodynamics*, AGARD CP-143, 1974.

[3]Hickey, D. H., "V/STOL Aerodynamics: A Review of the Technology," AGARD CP-143, 1974, pp. 1-1 – 1-13.

[4]"Military Applications of V/STOL Aircraft," AGARD CP-126, 1972.

[5]Margason, R. J., "Review of Propulsion Induced Effects on Aerodynamics of Jet V/STOL Aircraft," NASA TN D-5617, 1970.

[6]Platzer, M. F., ed., "Prediction Methods for Jet V/STOL Propulsion Aerodynamics," Vol. I and II, DDC AD-A024022 and A024023, 1975.

[7]Platzer, M. F. and Margason, R. J., "Prediction Methods for Jet V/STOL Propulsion Aerodynamics," AIAA Paper 76-932, Dallas, Texas, Sept. 1976.

[8]Rubbert, P. E., Saaris, G. R., Scholey, M. B., Standen, N. M., and Wallace, R. E., "A General Method for Determining Aerodynamic Characteristics of Fan-in-Wing Configurations. Vol. 1 - Theory and Applications," U. S. Army Aviation Laboratories, Tech. Rept. 67-61A, Dec. 1967.

[9]Hess, J. L., "Calculation of Potential Flow About Arbitrary Three-Dimensional Lifting Bodies, " Douglas Aircraft Co., Rept. MDC J5679-01 (Contract N 00019-71-C-0524), 1972.

[10]Tulinius, J. R. and Margason, R. J., "Aircraft Aerodynamic Design and Evaluation Methods," AIAA Paper 76-15, Washington, D.C., 1976.

[11]Cebeci, T. and Bradshaw, P., *Momentum Transfer in Boundary Layers*, McGraw Hill, New York, 1977.

[12]Peyret R. and Viviand, H., "Computation of Viscous Compressible Flows Based on the Navier-Stokes Equations," AGARDograph 212, Sept. 1975.

[13]Wirz, H. J., "Computational Fluid Dynamics," *AGARD Lecture Series No. 86*, AGARD-LS-86, 1977.

[14]Patankar, S. V., "Numerical Prediction of Three-Dimensional Flows," *Studies in Convection Theory, Measurements and Applications*, Vol I, edited by D. E. Launder, Academic Press, New York, 1975.

[15]Roache, P. J., *Computational Fluid Dynamics*, Hermosa Publishers, Albuquerque, N.Mex., 1972.

[16]Orszag, S. A. and Israeli, M., "Numerical Simulation of Viscous Incompressible Flows," *Annual Review of Fluid Mechanics*, Vol. 6, 1974, pp. 281-317.

[17]Shen, Shan-fu, "Finite Element Methods in Fluid Mechanics," *Annual Review of Fluid Mechanics*, Vol. 9, 1977, pp. 421-445.

[18]Launder, B. E. and Spalding, D. B., *Mathematical Models of Turbulence*, Academic Press, London, 1972.

[19]Spalding, D. B., "Turbulence Modeling, Solved and Unsolved Problems," *Proceedings of Workshop on Turbulent Mixing in Non-Reactive and Reactive Flows*, Project SQUID, Purdue Univ., Lafayette, Ind., May 20-21, 1974, Plenum Press, New York, 1975, pp. 85-115.

[20]Chu, C. K., "Physical and Numerical Dissipation and Dispersion in Viscous Compressible Flow Computation," *Proceedings of the Symposium on Unsteady Aerodynamics*, Univ. of Arizona, Vol. II, 1975, pp. 415-434.

[21]Stuart, A. R. and Hetherington, R., "The Solution of the Three Variable Duct Flow Equations," NASA SP-304, 1974, pp. 135-154.

[22]Patankar, S. V., Pratap, V. S., and Spalding, D. B., "Prediction of Laminar Flow and Heat Transfer in Helically Coiled Pipes." *Journal of Fluid Mechanics*, Vol. 62, Feb. 1974, p. 539.

[23]Briley, W. R., "Numerical Method for Predicting Three-Dimensional Steady Viscous Flow in Ducts," *Journal of Computational Physics*, Vol. 14, Jan. 1974, pp. 8-28.

[24]Zelazny, S. W. and Baker, A. J., "Modeling of Three-Dimensional Mixing and Reacting Ducted Flows," NASA, CR-2661, 1976.

[25]Briley, W. R., Kreskovsky, J. P., and McDonald, H., "Computation of Three-Dimensional Viscous Flow in Straight and Curved Passages," United Technologies Research Center Rept. R76-911841-9, Aug. 1976.

[26]Rushmore, W. L. and Zelazny, S. W., "A Three-Dimensional Turbulent Compressible Flow Model for Ejector and Fluted Mixer Characterization," NASA-CR, (in preparation).

[27]DeJoode, A. D. and Patankar, S. V., "Prediction of Three-Dimensional Turbulent Mixing in an Ejector, " AIAA Paper 77-706, Albuquerque, N. Mex., June 1977.

[28]Patankar, S. V. and Spalding, D. B., "A Calculation Procedure for Heat, Mass, and Momentum Transfer in Three-Dimensional Parabolic Flows," *International Journal of Heat Mass Transfer*, Vol. 15, Oct. 1972, pp. 1787-1806.

[29]*Internal Aerodynamic Manual,* Columbus Div., North American Rockwell Corp., Columbus, Ohio, Vol. I (AD723823) NR68H-434-Vol-1, Vol. II (AD723824) NR68H-434-Vol-2, Appendix (AD723841) NR68H-434-App., Contract No.-66-0460, June 1970.

[30]Miller, D., "Internal Flow – A Guide to Losses in Pipe and Duct Systems," British Hydromechanics Research Assoc., Cranfield, (Bedford), Eng., Nov. 1971.

[31]Kreskovsky, J. P., Paterson, P. W., and Bailey, D. A., "Analytical and Experimental Study of Turbine Endwall Flow and Heat Transfer," NASA NAS3-19752, 1977.

[32]Ghia, K. N. and Sokhey, J. S., "Analysis and Numerical Solution of Viscous Flow in Curved Ducts of Regular Sections, " *Numerical and Laboratory Computer Methods in Fluid Mechanics*, presented at the 1976 Winter Annual Meeting of the American Society of Mechanical Engineers, Dec. 1976, pp. 53-72.

[33]Rubin, S. G., Koshla, P. K., and Saari, S., "Laminar Flow in Rectangular Channels," *Numerical and Laboratory Computer Methods in Fluid Mecahnics*, presented at the 1976 Winter Annual Meeting of the American Society of Mechanical Engineers, Dec. 1976, pp. 29-52.

[34]Patankar, S. V., Pratap, V. S., and Spalding, D. B., "Prediction of Turbulent Flow in Curved Pipes," *Journal of Fluid Mechanics*, Vol. 67, Feb. 1975, p. 583.

[35]Pratap, V. S., and Spalding, D. B., "Numerical Computations of the Flow in Curved Ducts," *The Aero Quarterly*, Vol. 26, 1975, p. 219.

[36]Dodge, P. R., "A Numerical Method for 2-D and 3-D Viscous Flows," AIAA Paper 76-425, San Diego, Calif., 1976.

[37]Dodge, P. R., and Lieber, L., "A Numerical Method for the Solution of the Navier Stokes Equations with Separated Flow," AIAA Paper 77-170, Los Angeles, Calif., 1977.

[38]Humphrey, J. A. C., Taylor, A. M. K., and Whitelaw, J. H., "Laminar Flow in a Square Duct of Strong Curvature," *Journal of Fluid Mechanics*, (to be published).

[39]Quinn, B., "Experiments with Hypermixing Nozzles in an Area Ratio 23 Ejector," U. S. Air Force, Rept. ARL 72-0084 (AD 752207), 1972.

[40]Chigier, N. A. and Chervinsky, A., "Experimental and Theoretical Study of Turbulent Swirling Jets Issuing From a Round Orifice," *Israel Journal of Technology*, Vol. 4, Jan. 1966, pp. 44-54.

[41]Hill, W. and Greene, P., "Increased Turbulent Jet Mixing Rates Obtained by Self-Excited Acoustic Oscillations," ASME Paper 77-FE-18, June 15-17, 1977.

[42]Viets, H., "Development of a Time-Dependent Nozzle, " U. S. Air Force, ARL-TR-74-0113, Aug. 1974.

[43]Stewart, V. R., "A Study of Scale Effects on Thrust Augmenting Ejectors," Rockwell International, NR-76H-2, March 1976.

[44]Hasinger, S. H., "Performance Characteristics of Ejector Devices," U. S. Air Force, ARL-TR-75-0205, June 1975.

[45]Salter, G. R., "Method for Analysis of V/STOL Aircraft Ejectors," *Journal of Aircraft*, Vol. 12, Dec. 1975, pp. 974-978.

[46]von Karman, T., "Theoretical Remarks on Thrust Augmentation," *Reissner Anniversary Volume, Contributions to Applied Mechanics*, edited by J. W. Edwards, Ann Arbor, Mich., 1949, pp. 461-468.

[47]Bevilaqua, P. M. and McCullough, J. K., "Entrainment Method for V/STOL Ejector Analysis," AIAA Paper 76-419, *9th Fluid & Plasma Dynamics Conference*, San Diego, Calif., July 1976.

[48]Gilbert, G. B. and Hill, P. G., "Analysis and Testing of Two-Dimensional Slot Nozzle Ejectors With Variable Area Mixing Sections," NASA CR-2251, May 1973.

[49]Maroti, L. A., Hill, P. G., Armstrong, R. L., and Haines, D. M., "Analysis and Testing of Two-Dimensional Vented Coanda Ejectors With Asymmetric Variable Area Mixing Sections," NASA CR-2712, 1976.

[50]Launder, B. E. and Spalding, D. B., "The Numerical Computation of Turbulent Flows," *Computer Methods in Applied Mechanics and Engineering*, Vol. 3, No. 2, 1974, pp. 269-289

[51]Wilson, J. D. and Loth, J. L., "Thrust Augmented Wing Sections in Transition Flight," AIAA Paper 75-169, Pasadena, Calif., 1975.

[52]Goldhammer, M. I., Lopez, M. L., and Shen, C. C., "Methods For Predicting the Aerodynamic and Stability and Control Characteristics of STOL Aircraft," AFFDL TR-73-146, Dec. 1973.

[53]Mendenhall, M. R. and Spangler, S. B., "Prediction of Longitudinal Aerodynamic Characteristics of STOL Configurations with Externally Blown High Lift Devices," AIAA Paper 76-934, Dallas, Texas, Sept. 1976.

[54]Albers, J. A. and Stockman, N. O., "Calculation Procedures for Potential and Viscous Flow Solutions for Engine Inlets," *Journal of Engineering for Power*, Vol. 97, Jan. 1975, pp. 1-10.

[55]Chou, D. C., Luidens, R. W., and Stockman, N. O., "Prediction of Laminar and Turbulent Boundary Layer Flow Separation in

V/STOL Engine Inlets," AIAA Paper 77-144, Los Angeles, Calif., Jan. 1977.

[56]Boles, M. A. and Stockman, N. O., "Use of Experimental Separation Limits in the Theoretical Design of V/STOL Inlets," AIAA Paper 77-878, Orlando, Fla., July 1977.

[57]Siclari, M. J., Barche, J., and Migdal, D., "V/STOL Aircraft Prediction Technique Development for Jet-Induced Effects," Naval Air Propulsion Test Center, PDR-623-18, 1975.

[58]Behnert, R., "Bodeneffekte bei Strahlgestutzten Senkrechtstartern," Bericht FAG4, VFW-F4.10, ZTL, 1971.

[59]Fearn, R. L. and Weston, R. P., "Induced Pressure Distribution of a Jet in a Crossflow," NASA TN D-7916, 1975.

[60]Anon., "Analysis of a Jet in a Subsonic Crosswind," NASA SP-218, Sept. 1969.

[61]Wooler, P. T., et. al., "V/STOL Aircraft Aerodynamic Prediction Methods Investigation," U. S. Air Force Vols. I-IV, AFFDL TR-72-26, 1972.

[62]Fricke, L. B., Wooler, P. T., and Ziegler, H., "A Wind Tunnel Investigation of Jets Exhausting Into a Crossflow," Vols. I-IV, AFFDL-TR-70-154, U. S. Air Force Flight Dynamics Laboratory, Dec. 1970.

[63]Hickey, D. H. and Cook, W. L., "Aerodynamics of V/STOL Aircraft Powered by Lift Fans," AGARD CP-22, Sept. 1967, pp. 15-1 – 15-17.