Application of Computational Aerodynamics to Airplane Design

Luis R Miranda
Lockheed-California Company, Burbank California

Introduction

COMPUTATIONAL aerodynamics has been defined as the engineering discipline which deals with the simulation of flowfields about aircraft through the numerical solution of the equations of fluid motion using digital computers This simulation process is referred to as numerical flow simulation; another way of simulating flow about airplanes is the wind tunnel Wind tunnel technology is almost as old as aeronautics; wind tunnel testing became the principal tool of aerodynamic design and development very early in the history of airplane engineering Of course in addition to the experimental approach analytical methods based on aerodynamic theory always have been used to obtain valuable design information; but in order to reduce the computational effort to a humanly tolerable level, many simplifying assumptions had to be introduced These simplifications not only had to do with the mathematical modeling of the physical problem e g inviscid incompressible flow but they also involved the geometry of the configuration and flow condition e g the lifting line concept to represent a wing These simplications did not make possible the calculation of many aerodynamic characteristics and posed a high degree of uncertainty about the accuracy and validity of the numerical results Therefore these results could be used only as design guidelines and large emphasis had to be placed on wind tunnel testing

The advent of the digital computer and the impressive growth of its capabilities and cost effectiveness—as illustrated in Fig 1—now have made it possible to eliminate many of the simplifications that had been introduced in analytical methods This has increased the amount of information that can be extracted from a calculation and at the same time provides more accuracy and a greater range of validity for the results of numerical flow simulation This has turned computational aerodynamics into a major tool for airplane design

This paper surveys the use of computational aerodynamics as a design tool Much development work in computational aerodynamics is of a fundamentally scientific nature concerned with what type of numerical flow simulation is feasible within the boundaries of our knowledge of flow physics and numerical algorithms This paper deals with the engineering aspects of numerical flow simulation i e. what elements are practical in an environment where schedule and budget constraints predominate and where multidisciplinary interaction is the way of life and particularly with the factors that affect its effectiveness in such an environment Even though computer system characteristics do fundamentally affect the effectiveness of computational aerodynamics they are not the subject of this paper The survey presented here is of necessity, selective and consequently incomplete The main intent of this paper is to highlight those elements of computational aerodynamics that are important for airplane application what form they have taken and to indicate what future developments are considered necessary to increase the general utility of computational aerodynamics Three dimensionality geometrical complexity and computational efficiency are stressed as overdriving considerations for design application

First some general characteristics and needs of airplane design projects are discussed; within this context particular attention is given to the meaning of effectiveness and the identification of the computer code characteristics that affect it Next the code elements that are part of a typical numerical flow simulation are identified; the nature of these elements how they impact code effectiveness and how they are affected by considerations of effectiveness are addressed subsequently The author has drawn freely from developments at the Lockheed California Company to illustrate some points; for this he is indebted to many of his colleagues at Lockheed

Computational Aerodynamics in an Airplane Design Project: Requisites for Effectiveness

The many activities that take place in an airplane design project can always be considered as being part of a basic process loop consisting of three consecutive steps that may be repeated many times and at varying levels of complexity and

Luis R Miranda is presently Manager of the Computational Aerodynamics Department of the Lockheed California Company As such, he has responsibility for the development and implementation of computational methods for aerodynamic design and analysis He is also responsible for the wind tunnel testing that is required for validation of theoretical methodology

He joined Lockheed in 1966 when he was assigned to the L 2000 Supersonic Transport program aerodynamics staff In this position, he was involved in one of the first practical applications of computational aerodynamics performing supersonic wing design and sonic boom studies Subsequently he became a lead engineer in the L 1011 program participating in the analytical and experimental phases of the configuration development effort He has been principal investigator of research programs in transonic and supersonic aerodynamics and has also developed several numerical methods for compressible flows

Mr Miranda holds an Engineer's Degree in Aeronautics and Mechanics from the National University of Cordoba Argentina where he graduated with top honors in 1964 He has pursued postgraduate studies in fluid dynamics at the University of California at Los Angeles and at Davis Mr Miranda is an Associate Fellow of the AIAA

Presented as Paper 82 0018 at the AIAA 20th Aerospace Sciences Meeting Orlando Fla Jan 11 14 1982; received Feb 11 1982; revision received Jan 15 1983

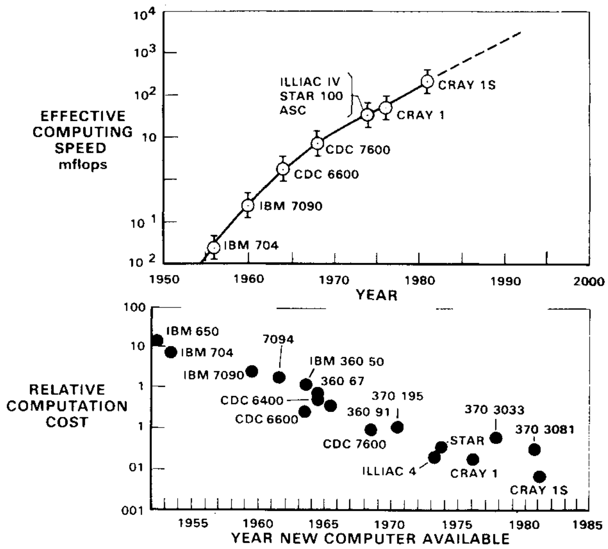

Fig 1 Progress in digital computer capability and cost effectiveness

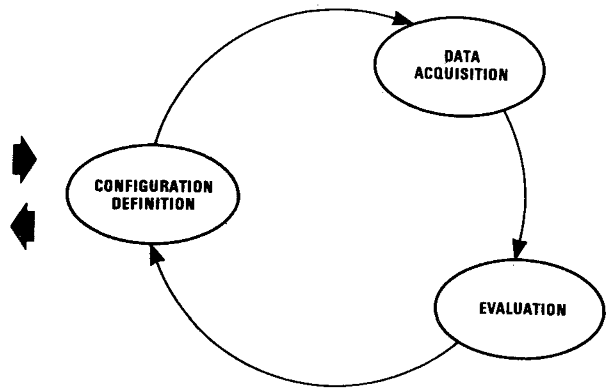

Fig 2 Design process loop

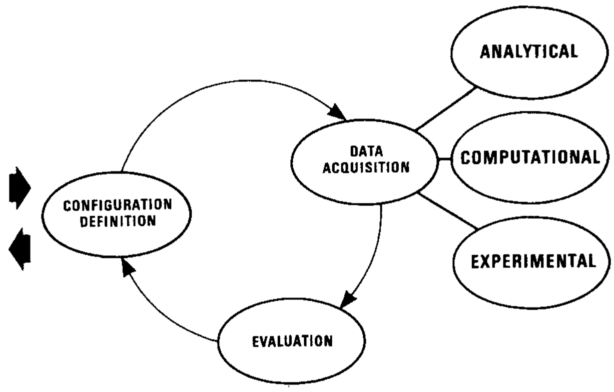

Fig 3 Means of data acquisition in the design process

detail: 1) configuration definition, 2) data acquisition and 3) evaluation (Fig 2) During the first step the geometry of the configuration is defined to the appropriate degree of accuracy and detail In the second step the required engineering data pertaining to the given geometry and design conditions are obtained The third step involves the evaluation of the design using the acquired data to determine to what degree the design objectives and constraints have been met and so specify what configuration changes are needed if any

Computational aerodynamics has affected this process loop by providing: 1) a new means of aerodynamic data ac quisition complementing the classical analytical and wind tunnel procedures (Fig 3) and 2) the unique capability of executing the design loop in an inverse manner i e specifying some desired flow characteristics and calculating the configuration geometry that yields such characteristics (Fig 4) something which is not feasible in the wind tunnel

The effectiveness of computational aerodynamics in a design environment will depend on the nature of the elements that constitute the computer codes used in a numerical flow simulation In order to understand the relationship between effectiveness and code characteristics it is useful to interpret effectiveness as the product of two factors as follows:

$$\text{Effectiveness} = \text{quality} \times \text{acceptance}$$

Although this expression has no actual quantitative value it serves to emphasize an often overlooked axiom: The impact that a given process has on the activity for which it is intended depends not only on how good the process itself is but also on how widely used or accepted it is

In the present context the quality factor refers to the ac curacy and realism of the numerical flow simulation; the acceptance factor has to do with the usability applicability and affordability of the code used in the simulation Some features of a given computational method will enhance the quality factor whereas others will improve the acceptance factor of the effectiveness equation Furthermore a given amount of research and development effort will not, in general produce equal amounts of improvement in both factors This is very important to keep in mind when one is concerned with the effectiveness of a computational procedure in a practical environment If increasing the ac curacy of a computational procedure will detract from its ease and economy of use the implied tradeoff between quality and acceptance should be considered carefully to determine if its effectiveness will actually be enhanced by the increase in accuracy

A quick look at the principal characteristics of the airplane design process will help delineate the requisites that a computational aerodynamics code has to satisfy for high effective ness and to which factor quality or acceptance each one of these requisites relates

First and foremost an airplane design project has to meet a set of milestones at predetermined time intervals and within specified budget allotments Failing to meet the milestones on schedule and/or exceeding the allocated budget may have serious consequences for the project and the organization conducting it The criticality of budget and schedule con straints is also manifested by the increase with time in the cost of configuration changes as exemplified by the diagram of Fig 5 In other words there is a compelling need to arrive at the correct and optimum aerodynamic configuration as early as possible

The design process requires aerodynamic data of varying degree of detail and different type in order to evaluate the vehicle's performance handling qualities structural integrity and the integration of systems such as powerplants and ex ternally mounted stores These data have to be obtained for symmetrical and asymmetrical flight conditions Typical data requirements for the various aerodynamic tasks that have to be performed as part of the design process are shown in Fig 6 Out of this bulk of data all of which is essential to the design effort there is one particular aerodynamic parameter whose accurate prediction is decisively important in an air plane project: the aerodynamic efficiency or lift to drag ratio

Another essential characteristic of the process is the geometrical complexity of its subject the airplane The airplane is a complex combination of three dimensional elements whose mutual interferences have to be accounted for properly The ability to represent the pertinent geometrical features of an airplane configuration accurately and to evaluate the tradeoffs due to the relative arrangements and sizes of the many constitutional elements is of paramount importance

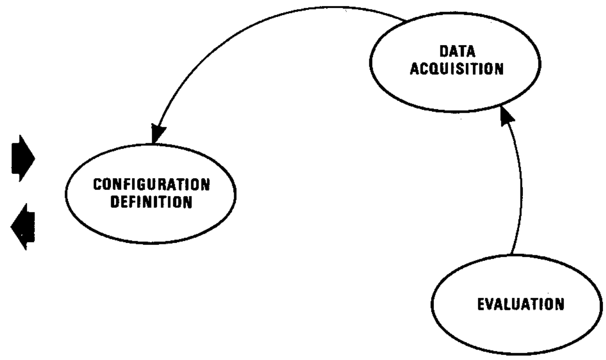

Fig 4 Inverse process loop

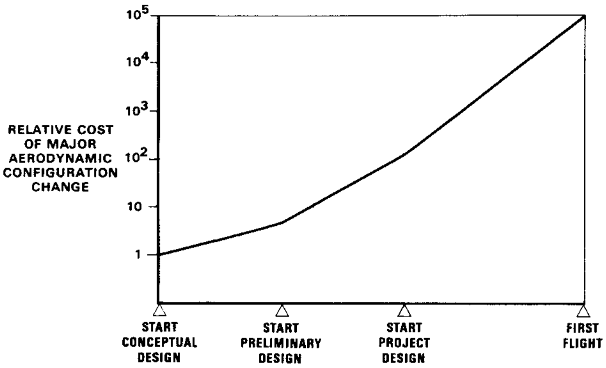

Fig 5 Relative cost of major aerodynamic configuration change

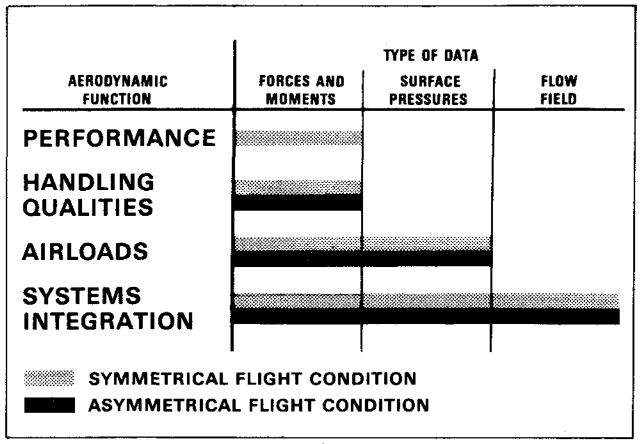

Fig 6 Aerodynamic data requirements for airplane design

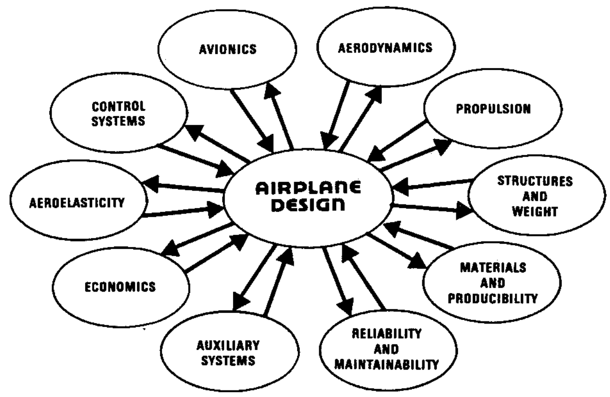

Fig 7 Interdisciplinary interaction in airplane design

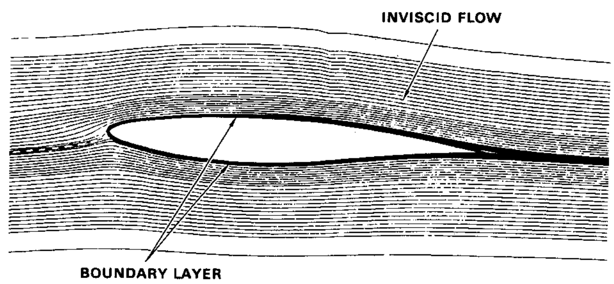

Fig 8 Zonal approach for numerical flow simulation

Last but not least, airplane design is a highly interactive interdisciplinary process involving quantitative tradeoffs among the various disciplines (Fig 7)

The above characteristics lead to the following list of requisites that a computer code should satisfy in order to achieve an acceptable degree of effectiveness

1) Adequate simulation of physics The mathematical model upon which the computer code is based must be capable of modeling those flow features that are significant to the problem under consideration, e g mixed elliptic hyperbolic domains and shock waves for transonic flows

2) Complex geometry The computer code should be capable of dealing with complete aircraft configuration or with major configuration elements, such as a wing and combination of such elements e g , wing and fuselage

3) Reasonable computer demand The code must be able to function on commercially available computer systems shared by other disciplines Adequate turnaround capability is important

4) Robustness The code should not require a high level of specialized expertise on the part of the user, nor should it rely on tinkering of input parameters and numerical experimentation to achieve a successful run

5) Friendliness The code should be user oriented with, its input and output as automated, clear, and logical as possible

Of these five requisites, the first two have a direct impact on the quality factor of the effectiveness equation whereas the last three affect mainly the acceptance factor

Computational Aerodynamics Code Elements

The above requisites and considerations of effectiveness have a major influence on determining the nature of the elements of a numerical flow simulation for airplane design application Presently such numerical flow simulation is performed by computer codes assembled within the framework of the zonal approach According to this approach the flowfield is divided into zones where significant simplications can be introduced locally; these zones are then linked by schemes of varying degree of interaction Along the lines of most common methods of numerical flow simulation, only two major zones are considered in the present discussion (Fig 8) as follows: 1) the inviscid zone where viscosity is neglected, and 2) the viscous zone usually a very thin region on the solid boundaries known as the boundary layer, where the viscosity of the fluid plays a significant role

In general, more zones may be present, such as inviscid zones of different energy levels and if Reynolds averaged Navier Stokes methods are resorted to zones of different turbulence models; this case is not considered in this review Here, a more restrictive meaning is given to the concept of zonal method; by its use we intend to exclude Navier-Stokes solvers entirely These solvers have not yet reached a state of development adequate for practical design application and consequently are not the subject of this discussion Global methods i e methods that are based on a single

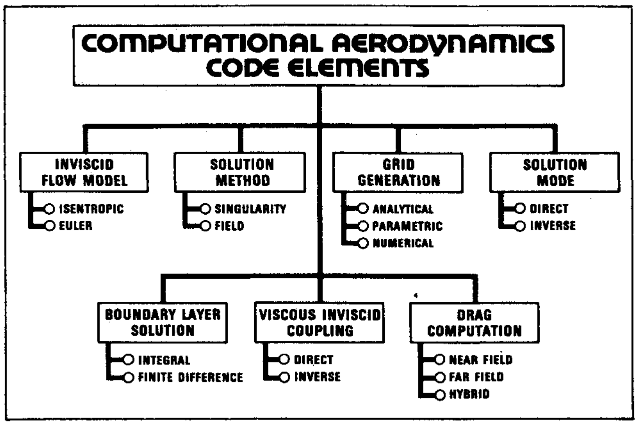

Fig 9 Computer code elements for numerical flow simulation

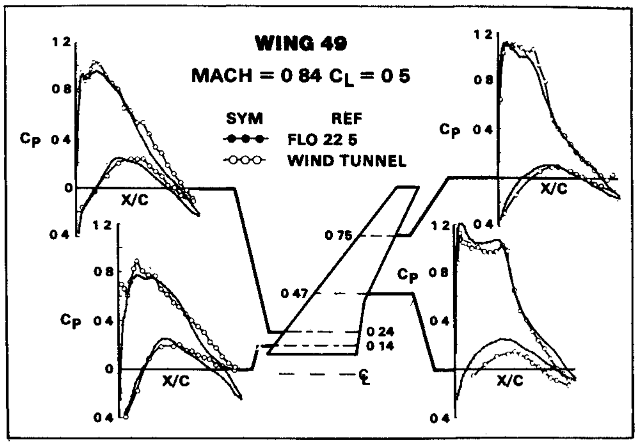

Fig 10 Example of theory experiment correlation for transonic flow using full potential method

mathematical model to simulate all features of the flowfield and therefore are essentially Navier Stokes solvers are at an even more primitive level of development

In a practical application we are then concerned with the mathematical modeling of the inviscid and viscous zones namely the determination of the equations to be used to represent the respective flows and with patching of the corresponding solutions Considerations regarding both the numerical methods used in solving the above equations and the geometrical problems of grid generation that arise in the implementation of these methods have great practical im portance Whether the numerical flow simulation is per formed in a direct mode to predict the aerodynamic characteristics of a given airplane configuration or whether it is used in an inverse mode to generate a geometry that con forms to a set of specified aerodynamic characteristics requires special attention The calculation of drag, an aerodynamic characteristic so important to airplane per formance deserves similar attention These various issues and areas of concern can be considered to be the key elements that determine the success of a computational aerodynamics code as a design tool and they are indicated in Fig 9 The breakdown into the elements shown is certainly not unique and it is to some extent arbitrary, but it identifies the major issues and areas of concern for airplane design application The adequacy of the techniques and method being used as the elements of numerical flow situation is addressed in what follows

The Inviscid Flow Model

Basically, there are two different mathematical models to describe the flow in the inviscid zone The first and simpler model, known as either isentropic or potential describes a flow where mass, entropy, and energy are conserved The fundamental partial differential equation governing this type of flow can be written for the steady state case as[1]

$$\nabla^2\phi = M^2\phi_{ss} \tag{1}$$

where ϕ is the velocity potential whose gradient yields the flow velocity $\bar{u}$ i e , $\bar{u}=\nabla\phi$ M is the local Mach number ∇^2 the Laplacian operator, and subscript s denotes the directional derivative along the streamline A very convenient sim plication of Eq (1) results when the local Mach number is assumed constant and equal to its freestream value, and the stream direction is almost parallel to the x axis at all points in the flowfield This yields the well known Prandtl Glauert equation

$$\nabla^2\phi = M_\infty^2\phi_{xx} \tag{2}$$

Other simplications of Eq (1) exist, the most notable one being the transonic small perturbation equation This par ticular approximation is considerably more complex than the Prandtl Glauert equation, but it retains the major nonlinear characteristics of Eq (1) while allowing a much easier ap plication of the boundary conditions

The second model expresses the conservation of mass, momentum, and energy and, therefore, it provides a more physically accurate representation of an inviscid flow The partial differential equations representing this flow model are known as the Euler equations; for steady state flow they take the following form[2]:

$$\frac{\partial\bar{E}}{\partial x}+\frac{\partial\bar{F}}{\partial y}+\frac{\partial\bar{G}}{\partial z}=0 \tag{3}$$

where

$$\bar{E}=\{\rho u \quad \rho u^2+p \quad \rho uv \quad \rho uw; \; (\rho e+p)u\}$$

$$\bar{F}=\{\rho v \quad \rho vu \quad \rho v^2+p \quad \rho vw \quad (\rho e+p)v\}$$

$$\bar{G}=\{\rho w \quad \rho wu \quad \rho wv \quad \rho w^2+p \quad (\rho e+p)w\}$$

$(x \; y \; z)$ are orthogonal Cartesian coordinates, ρ the fluid density, p its pressure e the internal energy per unit mass, and $u \; v \; w$ denote the flow velocity vector components along the coordinate axes

Except for some very special cases, codes based on the Euler equations have not been used in airplane design work the vast majority of applications have been conducted within the framework of the potential flow model The reason for this is the much higher mathematical complexity of the Euler equations; for a given flow configuration, this complexity translates into computational demands at least an order of magnitude higher than those of potential flow methods But the ever increasing power of computers is now making the use of Euler equations to model the inviscid flow a possibility being considered by several investigators Application of Euler codes to the analysis of supersonic flows about relatively simple geometries is commonplace but, recently, Euler codes have been developed to solve the more com putationally demanding problem of transonic flows about three dimensional wings as exemplified by the work of Rizzi and Eriksson [3] While these developments enrich the capabilities of computational aerodynamics, it is not a foregone conclusion that they will equally enhance its ef fectiveness as an airplane design tool If the simpler model of potential flow is sufficiently adequate for a given application, then the use of a more complex and, therefore, more com putationally expensive model may do little to increase such effectiveness

Obviously the adequacy of any model depends on the type of problem but by and large the potential flow model, particularly in its full expression, has been found to be quite adequate—in some cases surprisingly so—to represent the pertinent features of flow about airplane configurations. First, let us consider the case of transonic flows. Evaluations and review of how well potential flow codes compute subsonic and transonic flows abound in the technical literature; an example of a reasonable agreement between theory and experiment is shown in Fig. 10 for a typical transport wing. Examples of poor correlation between theory and experiment have also been published, but all of these failures have been attributed to one or more of the following causes:

1) Lack or inadequate treatment of boundary-layer effects.
2) Lack or inaccurate representation of the fuselage.
3) Incorrect or inaccurate boundary conditions.
4) Problems related to the numerical solution of the equations, e.g., convergence and numerical stability
5) Inadequate discretization of the partial differential equations, e.g., conservative vs nonconservative formulations, etc.
6) Uncertainty about the experimental data base.

None of the above causes of disagreement would be removed by switching from a potential to an Euler model for the inviscid zone. The argument which is often advanced to justify the use of Euler equations in transonic flow computations is that significant errors would be introduced if there are strong shock waves in the flowfield. While true in a strict sense, in practice this argument may not hold. Steger and Baldwin[4] have thoroughly studied the matter of shock waves and drag in the numerical calculation of potential transonic flow They have shown that if the Mach number normal to the shock is less than 1.3 then there is very little difference between the jump relations for a normal shock in Rankine-Hugoniot flow or in potential flow (Fig. 11). Furthermore, they have explained the mechanism of wave drag production in isentropic flow and how drag computations based on the isentropic flow results relate to those obtained by using the Rankine-Hugoniot relations. For a supersonic free stream, isentropic and Rankine-Hugoniot results should yield approximately the same value of wave drag for weak oblique shocks. For a subsonic freestream, this is no longer true, but the isentropic results can be rendered adequately accurate by the use of a simple correction factor if the normal shock waves remain weak.

The fact that for normal Mach numbers less than 1.3, the Rankine-Hugoniot and the isentropic shock jump relations are nearly equivalent has great practical significance; this implies that up to that point, Euler and potential flow calculations should yield approximately the same shock position and shock strength. Experience has shown that whenever the normal Mach number exceeds 1.3, shock-induced separation of the boundary layer will occur. Therefore, the predominant problem will be the modeling of the flow separation and the ensuing strong viscous-inviscid interaction; mathematical flow models that do not take this into account will be in error, regardless of whether they are based on the potential flow or the Euler equations.

Another problem for which Euler methods have been proposed is the analysis of flows where vorticity is being shed from other than trailing edges and where the convolution of this shed vorticity is of importance, such as the leading-edge vortex flow about highly swept wings. Likewise, several potential flow methods have been developed, or are under development, to deal with this problem, as exemplified by the work of Johnson et al.,[5] Kandil et al.,[6-8] Nangia and Hancock,[9] Mehrotra and Lan,[10] and others. Much work remains to be done in this area, and it is still too early to say which approach will prove more adequate.

While some investigators are working on the solution of the Euler equations for transonic application, others are proposing to use the full potential equation (1) to model supersonic flows about airplanes. The work of Grossman and Siclari,[11] and Shankar and Chakravarthy[12] are representative

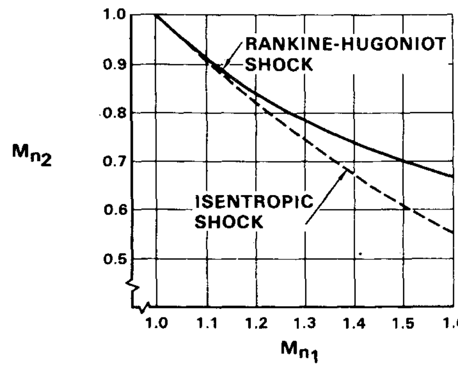

Fig. 11 Mach number variation for Rankine-Hugoniot and isentropic normal shocks.

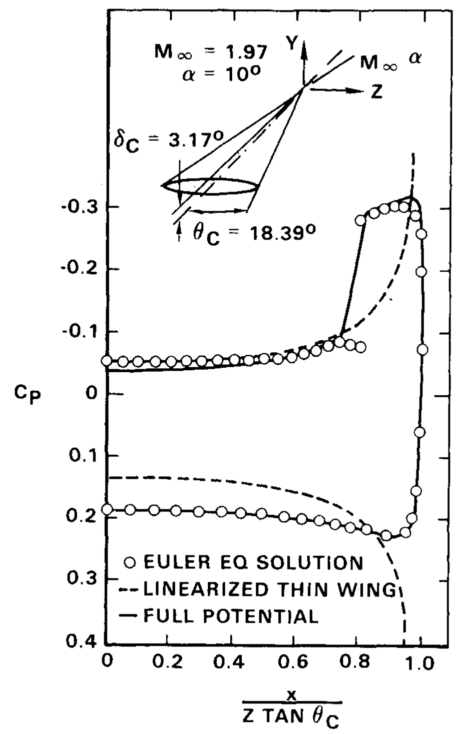

Fig. 12 Comparison of Euler, full potential, and linearized potential solutions for supersonic flow.

of this rather recent trend. While there may be some doubt about the effectiveness of a transonic application of the Euler equations, there is little doubt that a full potential method for supersonic analysis will add greatly to the effectiveness of computational aerodynamics provided they represent the physics of the problem adequately While this latter point has not yet been settled sufficiently there is encouraging evidence that a full potential model will offer a significant improvement in accuracy over the standard linearized supersonic potential methods at a small fraction of the cost of Euler methods, as illustrated by the example of Fig. 12 taken from Ref. 11.

From the preceding discussion it appears that a proper potential flow model should be sufficient to represent the inviscid zone for most aeronautical applications. It is the author's opinion that, at the present time, effectiveness can be better served by improving the efficiency and extending the geometrical range of applicability of potential flow solvers rather than by developing more complex codes.

The Solution Method

Once the appropriate flow model to represent the inviscid zone has been selected, the next element to be considered is the

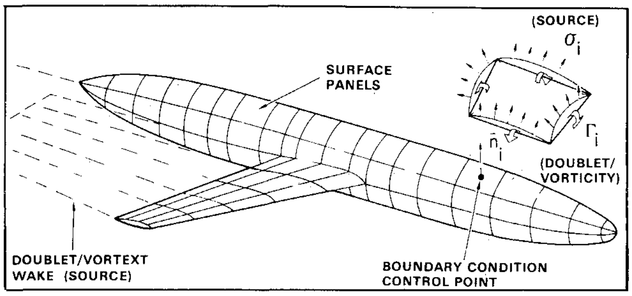

Fig 13 Concept of panel methods

Table 1 Principal solution features of panel methods

Characteristic	1	2	3
Panel layout	Mean surface	Hybrid	Actual surface
Panel	Planar	Hyperbolic	Arbitrarily curved
Singularity type	Source	Doublet/ vorticity	Source and double/vorticity
Singularity distribution	Constant	Linear	Higher order
Boundary condition	Dirichlet	Neumann	Mixed
Solution formulation	All unknown	Doublet var pre specified	Source deter mined by Green s theorem

numerical method used to solve the partial differential equations implementing the chosen flow model Two dif ferent solution methods have become common in aerodynamic applications: singularity and field methods

Singularity methods also known as boundary integral methods are generally derived from the application of some analog of Green's theorem to the partial differential equation they intend to solve [13] If this equation can be arranged as a homogeneous Laplacian such as the Prandtl Glauert equation then the solution at any point in the flowfield is expressed as an integral of its value on the boundary surfaces Lattice quasilattice and panel methods in general belong to this class of solution In contrast field methods require the knowledge of the solution inside the flowfield as well as on the boundary surfaces in order to determine its value at any arbitrary point Finite difference finite volume and finite element methods belong to this solution class Until now singularity techniques have been applied to the solution of linear partial differential equations whereas nonlinear partial differential equations such as the full potential and transonic small perturbation equations have been almost exclusively solved by field methods

Three dimensional panel methods are probably the solution techniques most widely used in airplane design They have been under development for nearly two decades; their popularity stems mainly from their ability to handle con figurations of arbitrary geometric complexity Their major shortcoming is their limitation to the solution of the Prandtl Glauert equation; but the ingenuity of many investigators has made possible the application of these methods to many problems of practical interest while still remaining within the framework of the Prandtl Glauert equation Examples of such applications are the already mentioned analysis of leading edge vortex flows and the modeling of separated flows along the lines proposed by Maskew and Dvorak[14 15] and Rao et at [16] In depth expositions of this and similar types of applications of singularity methods have been presented recently by Smith [17] Hunt [18] and Butter and Mobbs [19]

The basic concept of panel methods is illustrated in Fig 13 The configuration is modeled by a large number of elementary quadrilateral panels lying either on the actual aircraft surface or on some mean surface or a combination thereof To each elementary panel there is attached one or more types of singularity distributions such as sources vorticity and doublets of dipoles These singularities are determined by specifying some functional variation across the panel (e g , constant linear quadratic etc) whose actual value is set by corresponding strength parameters; these parameters are determined by solving appropriate boundary condition equations Once the singularity strengths are known the velocity field can be directly calculated

A large variety of panel methods has been developed, the variations depending mainly on the choice of type and form of singularity distribution the geometric layout of the elementary panels and the type of boundary condition being imposed The major variations that have been incorporated in panel codes are indicated in Table 1 The choice of com binations is by no means a trivial matter; although many different combinations are in principle mathematically equivalent their numerical implementation may yield significantly different results from the point of view of numerical stability computational economy, accuracy and overall code robustness The early generation of panel methods, as exemplified by Smith and Hess [20] Rubbert and Saaris [21] and Woodward et al [22 23] were characterized by planar panels located on mean surfaces or on a combination of mean and actual surfaces with constant singularity strengths and boundary conditions of the Neumann type Continuing evolution has led to improvements in many areas A major development has been the introduction of higher order distributions of singularity strengths such as linearly varying source intensity and splined doublets This development was needed to improve the supersonic ap plication of panel methods: Constant strength panels in troduce infinite perturbations at the panel edges which propagate along the Mach cones; this generates spurious waves in the numerical solution, sometimes with catastrophic results The PAN AIR[24] code represents the most advanced implementation of higher order singularity distributions Unfortunately this implementation has resulted in a significant increase in the computational cost

Because of the large increase in computational cost caused by the higher order panel methods it is important to note that there have been other developments and technique im provements that, when properly implemented within the framework of a low order, constant singularity strength panel method yield subsonic flow results of an accuracy comparable to that of the higher order panel methods at a fraction of the cost This point has already been argued at some length by Maskew [25] and it is further illustrated herein by briefly describing an advanced low order panel method QUADPAN [26] developed at the Lockheed California Company and showing some comparative results

Even though the geometry package of the QUADPAN code is quite sophisticated allowing the description of very complex geometries with a relatively small amount of data input its fundamental geometric element is a flat (planar) quadrilateral panel located on the actual airplane surface A source and a doublet distribution of constant strength are assigned to each panel; therefore this method represents the lowest possible order of singularity distribution The following features are considered to be important factors in determining the accuracy and overall effectiveness of the code

1) The boundary condition equations are formulated in terms of the velocity potential i e the aerodynamic influence coefficients are induced potential coefficients rather than induced velocity coefficients

2) Internal flow Dirichlet boundary condition is applied The fictitious flow in the interior of the body is equated to the freestream flow; i e the perturbation potential in the interior is null

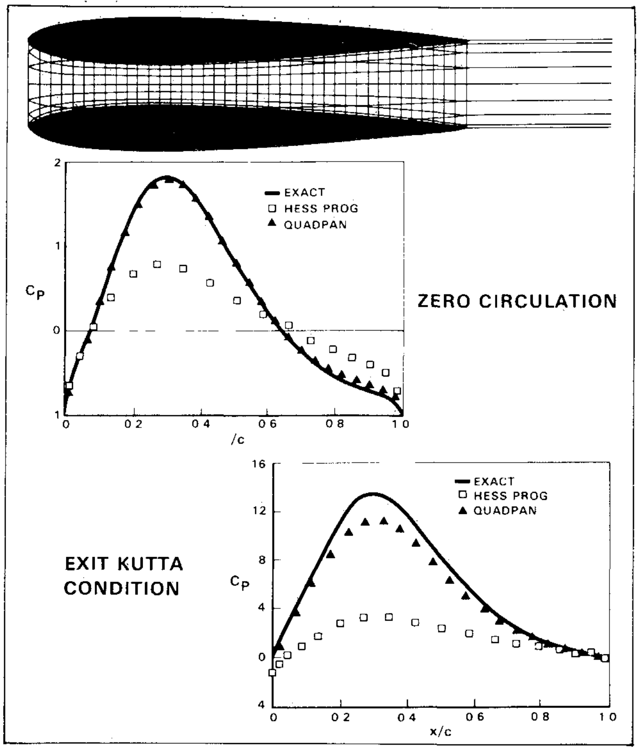

Fig 14 Comparison of theoretical results for nacelle internal flow

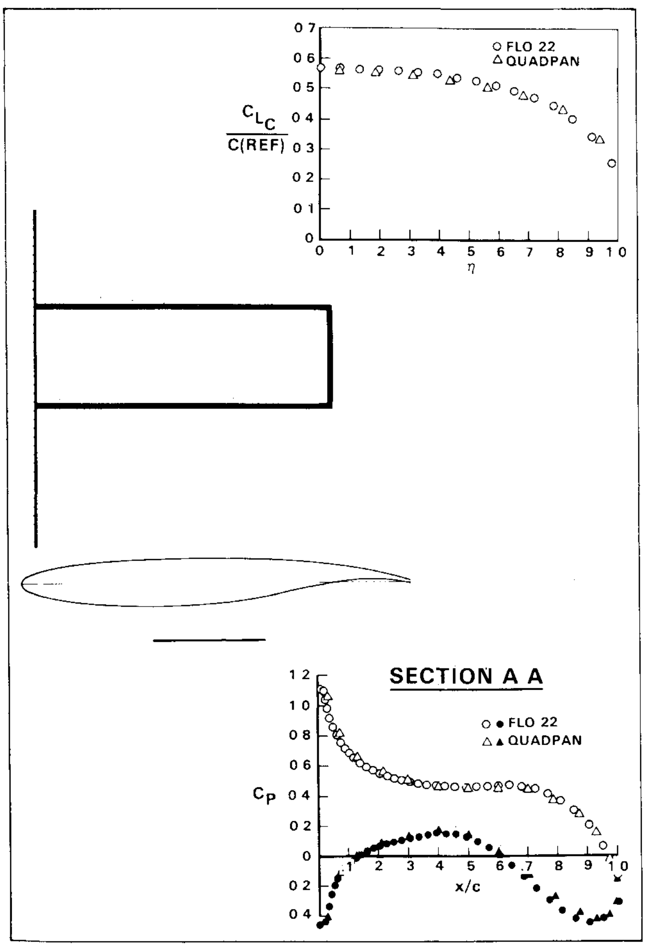

Fig 15 Comparison of theoretical results for highly aft loaded wing

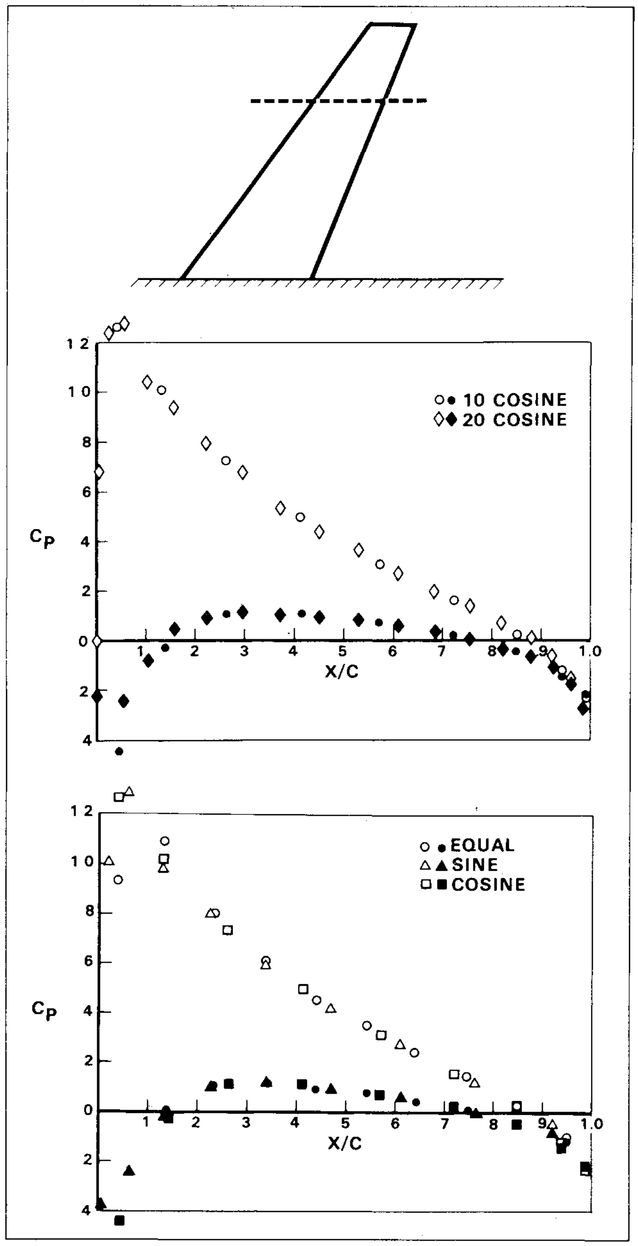

Fig 16 Effect of paneling spacing on QUADPAN results

3) The unknowns solved for the boundary condition equations are the doublet strengths The source strengths are determined by the component of the onset mass flux along the normal to the panel as specified by using an analog of Green's theorem

4) Surface velocities are calculated by finite differencing the value of the potential at the body surface and the corresponding pressure coefficients are computed by applying the quadratic approximation to Bernoulli's equation [13]

5) Great attention has been devoted to input and output automation error check features and data management considerations in general

The QUADPAN code has proven to be quite accurate when compared with results obtained using higher order codes An example of this is illustrated in Fig 14 which pertains to the flow through a high contraction ratio nacelle This is a very difficult problem due to the sensitivity of internal flows to leakage through the boundaries a common defect of older low order panel methods As can be seen QUADPAN results are much closer to the exact potential flow solution than those obtained by Hess' higher order method [27] Another difficult

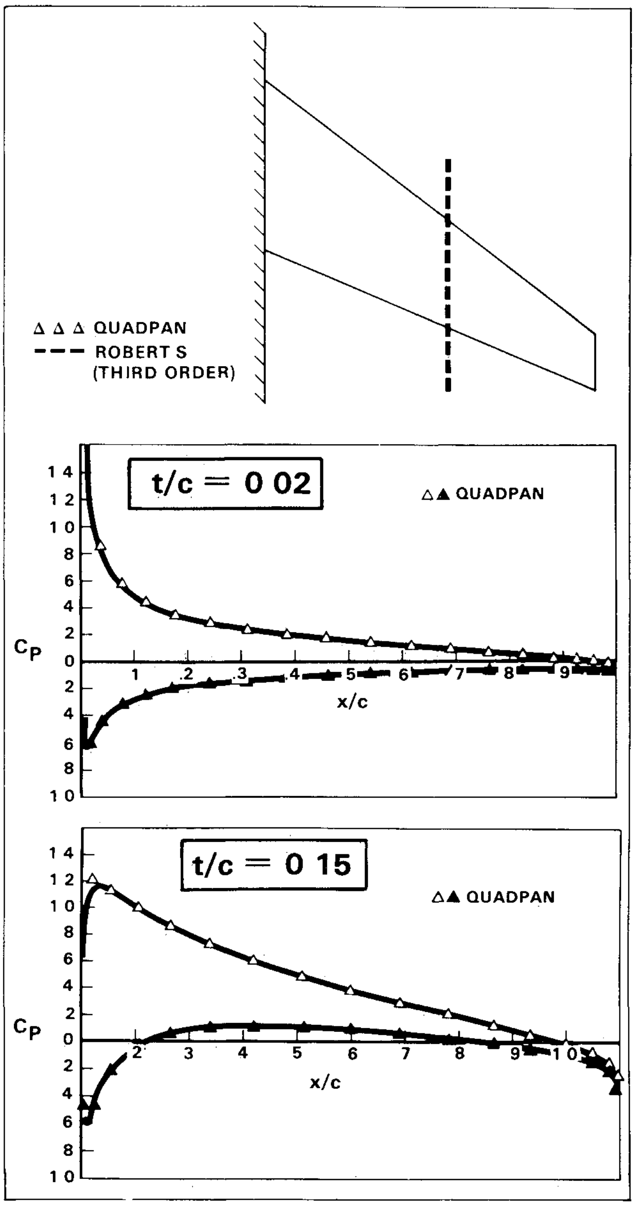

Fig 17 Comparisons between QUADPAN and higher order method for lifting wings

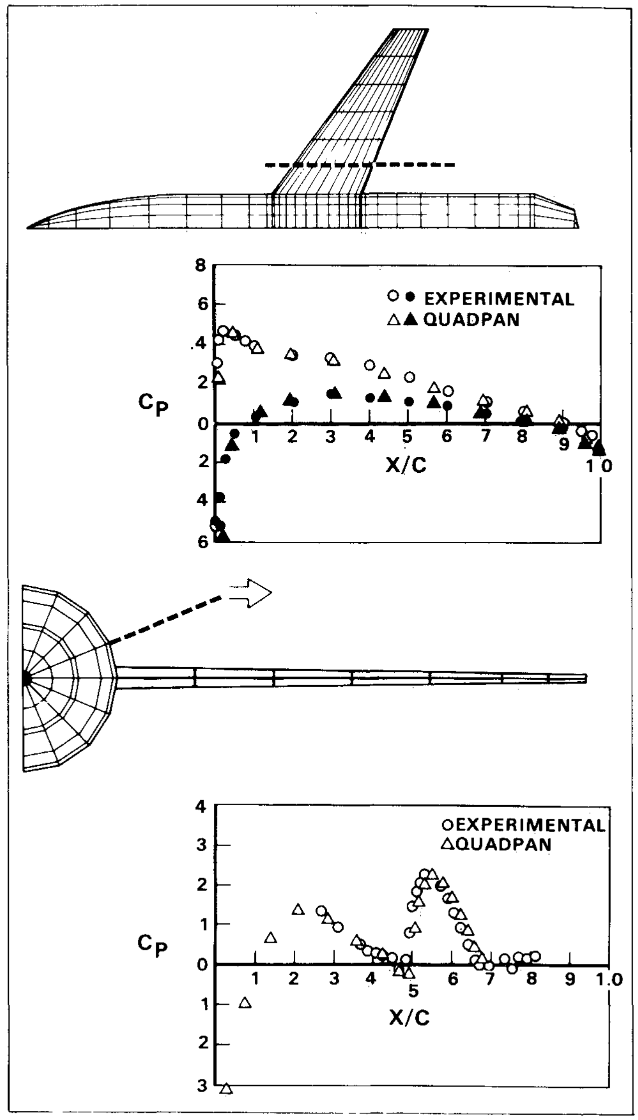

Fig 18 QUADPAN experiment comparison for wing body com bination

case even for some of the higher order panel methods is the computation of surface pressure distributions for highly aft loaded wing sections; the accuracy of the QUADPAN code for this particular case is illustrated in Fig 15 where it is compared with results from a finite difference solution which for practical purposes may be considered exact First generation low order panel methods were quite sensitive to panel spacing; the elimination of this drawback was one of the principal motives behind the development of higher order panel methods The low sensitivity of QUADPAN to panel spacing is shown in Fig 16 A representative example of the accuracy of QUADPAN results is given by the comparison with Roberts third order panel method (see Ref 28) presented in Fig 17 An additional example is the correlation with experimental results for a wing body configuration [29] Fig 18 Such satisfactory performance is achieved at a small fraction of the computational cost of the higher order panel methods The important point here is that for many applications significant gains in accuracy can be obtained by improving a simple theoretical model rather than by switching to a more complex one

Finite difference methods have also become widely used in design applications particularly in the area of transonic wing design In this context, Hicks[30] has discussed the successes and failures of finite difference potential flow codes ex tensively This application was also the subject of several papers at the Transonic Perspective Symposium held at NASA Ames in 1981

Both the full potential equation (FPE)[31] and the transonic small perturbation (TSP)[32] equation have been solved by finite difference methods TSP methods are capable of dealing with more complex geometries than FPE methods due to the simplifications introduced in the application of the boundary conditions; these simplifications allow a more versatile grid structure The methods of Bailey and Ballhaus[33] and Boppe[34] are representative of this type of solution The relative accuracies of FPE and TSP methods have been evaluated by Hinson and Burdges[35] and others In general FPE codes have been found to be more accurate than TSP codes This is particularly true for transport type wing configurations with relatively blunt edges. Furthermore this increase in accuracy results in no significant computational cost penalty The major drawback of FPE codes is the need to generate a body fitted computational mesh The geometrical and complexities of this problem have seriously restricted the range of geometric configurations that can be analyzed

The state of the art of FPE transonic flow methodology is best represented by three codes developed by Jameson and Caughey [36 37] known as FLO 22 FLO 28 and FLO 30 The

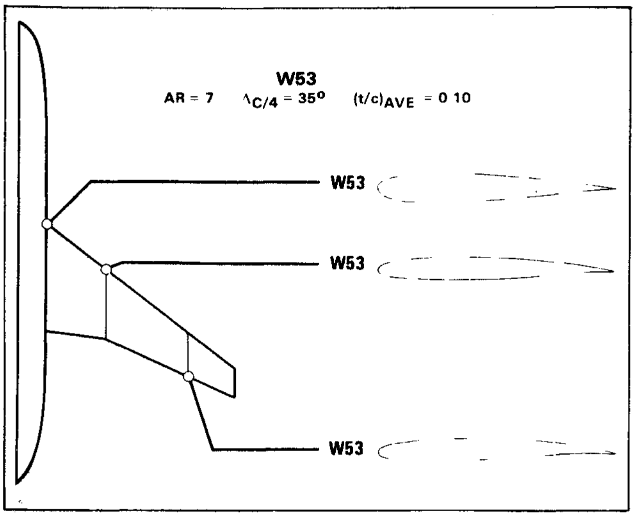

Fig 19 Wing body configuration used in transonic method assessment

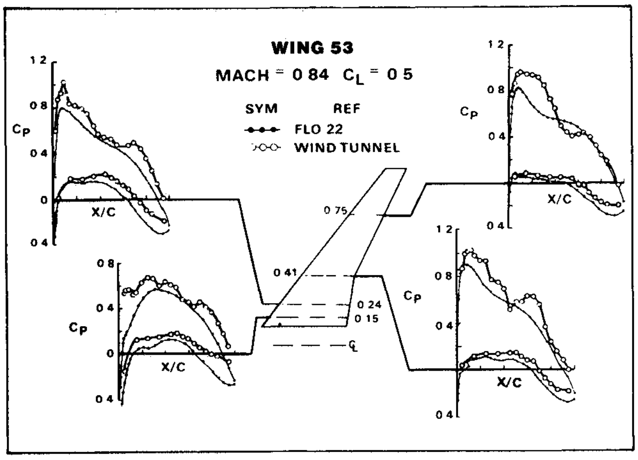

Fig 20 Comparison of predicted and experimental wing surface pressure distributions, FLO 22

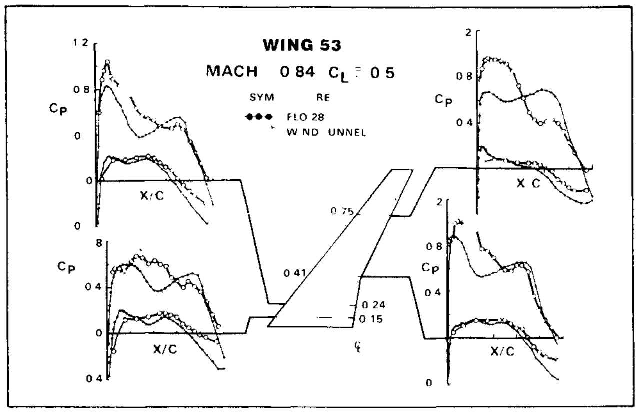

Fig 21 Comparison of predicted and experimental wing surface pressure distributions, FLO 28

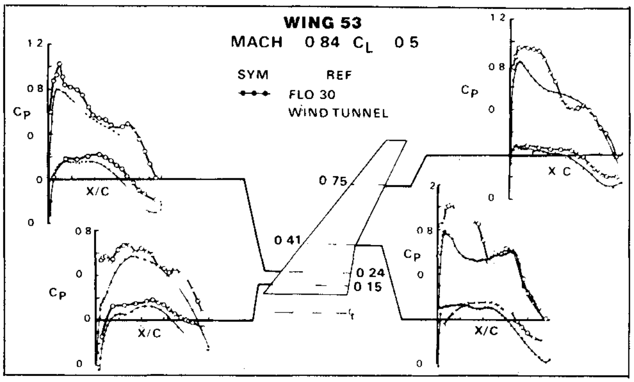

Fig 22 Comparison of predicted are experimental wing surface pressure distributions, FLO 30

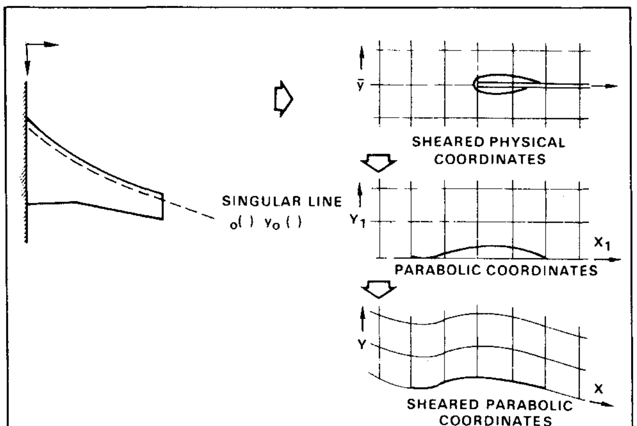

Fig 23 Analytical grid generation

first one FLO 22, deals with isolated or wall-mounted wings, whereas the other two can compute the flow about wing body combinations FLO 22 uses a nonconservative finite dif ference scheme and it generates the computational grid through a sheared parabolic coordinate transformation FLO 28 and FLO 30 are both based on the finite volume con servative difference scheme which essentially decouples the flow solution and the grid generation processes The principal difference between FLO 28 and FLO 30 is the manner in which the computational grid is generated FLO 28 uses a combination of sheared parabolic and Joukowsky trans formations to map the physical space into the computational domain Cylindrical and logarithmic type transformations are used in FLO-30 to accomplish such mapping All three codes calculate the surface pressure distribution in a potential flow; the aerodynamic force and moment coefficients are obtained by the corresponding surface integrations of the computed pressure distribution

The relative performances of these codes were evaluated as part of a cooperative program between the NASA Ames Research Center and the Lockheed California Company The highlights of this evaluation are presented in the following paragraphs

The FLO codes were applied to the analysis of the wing body configuration shown schematically in Fig 19; the configuration has supercritical airfoil sections with a large chordise extent of upper surface supercritical flow and moderate aft loading A full span, sting mounted model of this configuration was built and tested in the NASA Ames 14 ft Transonic Tunnel The model was equipped with about 150 surface pressure taps distributed on both upper and lower surfaces of the wing at four span stations; a six component strain gage balance was internally mounted in the body The data were obtained at a Reynolds number of approximately $3\ 5\times10^6$ based on the model mean aerodynamic chord Boundary layer transition was artificially tripped by car borundum strips located near the leading edge on both upper and lower surfaces

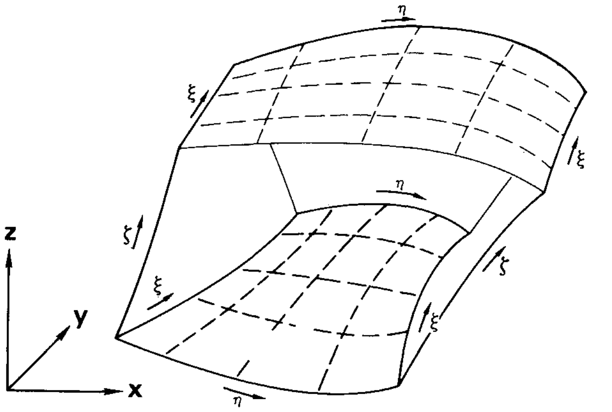

Fig 24 Parametric grid generation

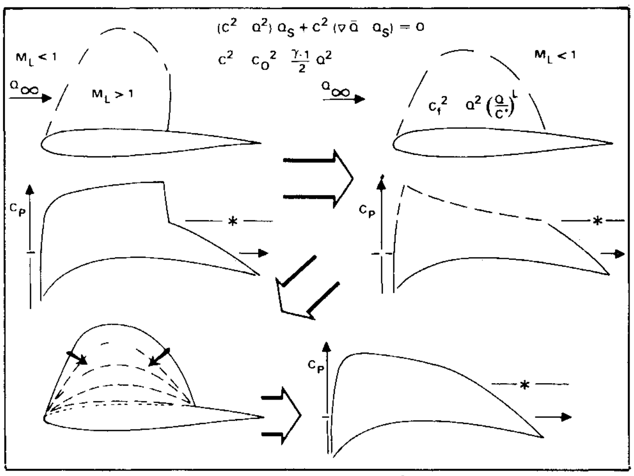

Fig 25 Shock free design process using fictitious gas concept

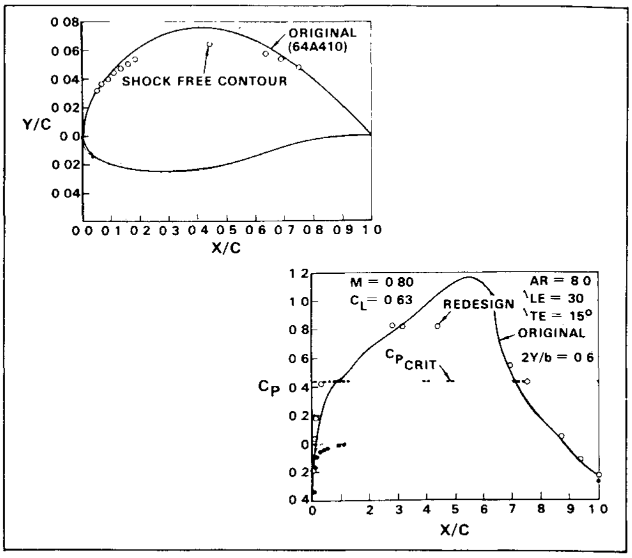

Fig 26 Example of shock free design using fictitious gas concept

Only the exposed wing was modeled in FLO 22 namely it was analyzed as a wall mounted wing Comments about the comparison between the predicted and measured pressure distributions at the wing design point Fig 20 are as follows

1) Good agreement exists between experimental and the calculated values of sectional lift coefficient given by the area enclosed by surface pressure distributions

2) The theoretically computed values of the upper surface pressure coefficients are less negative than the experimental values in the supercritical region of the wing

3) No shock waves are clearly discernible in the theoretical pressure distributions whereas the presence of a moderate shock running approximately along the 40% chord line is apparent in the experimental pressure distributions

The analysis of the same wing body configuration at the same test condition was then repeated using the more ad vanced codes FLO 28 and FLO 30 This time the geometry of the finite fuselage was included in the mathematical model The theoretical results are compared with the experimental pressure distributions in Fig 21 for the FLO 28 case, and in Fig 22 for the FLO 30 case The following observations can be made regarding the following comparison

1) Like FLO 22 both FLO 28 and FLO 30 predict the sectional lift coefficient values accurately

2) The underestimation of the supercritical levels of the pressure coefficient is even more pronounced for FLO 28 and FLO 30 than it was for FLO 22

3) Likewise, the overprediction of the aft loading is more pronounced in the FLO 28 and FLO 30 pressure dis tributions

4) The predictions of the pressure distribution at the wing station next to the fuselage are substantially different between FLO 28 and FLO 30 The FLO 30 distribution is in closer agreement with the experimental data at least in the general shape of the distribution if not in actual details and levels It should be pointed out that FLO 22 showed the same kind of agreement with the experimental results shown by FLO 30 at this station even though it did not incorporate any simulation of body effects

From the preceding discussion it is obvious that no im provement in the agreement between theoretical and ex perimental pressure distribution accrued from the use of more advanced codes FLO 28 and FLO 30 The major dis crepancies between the theoretical and experimental results were not resolved by the capability provided by both FLO 28 and FLO 30 to model the fuselage In the present case viscous effects appear to predominate, overshadowing any improvement in accuracy that might result from the inclusion of fuselage effects On the other hand, the use of the FLO 22 code showed some significant practical advantages namely it was at least six times less expensive in computer cost and it was easier and more reliable to run, requiring far less manipulation of the input data to achieve a successful run

This comparative example serves to stress the point that the numerical simulation of flows about airplane configurations is an aggregate of elements and that in general, improving or increasing the sophistication of one of the elements while neglecting the others will not necessarily increase the overall accuracy and effectiveness of the simulation; quite often the contrary occurs In the particular case under consideration, the neglected element was the calculation of the boundary layer effect i e , the viscous zone and its interaction with the inviscid zone This should not be taken to imply that fuselage effects are not significant in general; for many configurations they may indeed be predominant

In summary panel and finite difference/volume methods have become the standard solution techniques in applied computational aerodynamics The extension of panel methods to supersonic flow requires the use of higher order singularity distributions, whereas for subsonic flow properly formulated low order panel methods may be quite adequate for most practical applications Finite difference/volume techniques will most likely remain the principal solution tools for transonic flows, but their applicability will be determined by progress in the areas of computational efficiency and grid generation for arbitrary geometries the subject of the following section If adequate computational grids can be generated for complex geometries then finite dif ference/volume techniques will become very competitive with higher order panel methods This will most likely result in the exclusive use of field methods over singularity methods particularly for supersonic applications

Grid Generation

The problem of grid generation is a crucial one for the practical application of numerical flow simulation Both methods of solution, singularity and field require the con struction of a computational mesh—surface grids for singularity methods and spatial grids for field methods Although the former problem i e the generation of surface grids or lattices is by no means a trivial one it is relatively simple by comparison with the latter namely the generation of a spatial coordinate mesh around the airplane con figuration The capability to generate adequate com putational grids about arbitrarily complex configurations constitutes the single most important problem from the standpoint of practical applications of finite difference and finite volume methods This is the problem that is discussed in this section

One of the difficulties that make the grid generation problem so difficult is the fact that at the present time there is little systematic knowledge defining what is considered an adequate computational grid for a finite difference or finite volume method In spite of this three grid characteristics are generally accepted as sine qua non for computational adequacy as follows:

1) Although an orthogonal grid system is not required in most applications it is highly desirable to have nearly or thogonal grid lines close to the body surfaces

2) The grid must be body fitted i e a set of grid lines must lie on the surfaces of the body or bodies

3) Grid line spacing must be dense in regions where the solution is expected to exhibit large gradients

Three different methods of grid generation have evolved during the years: analytical parametric and numerical

Analytical grid generation techniques are by far the most computationally efficient and the simplest to implement Usually they consist of a combination of relatively simple coordinate transformations some of them based on con formal mapping concepts Typical of this type of grid generation technique is the sheared parabolic coordinate method of Jameson and Caughey [36] (Fig 23) This method generates a body fitted computational grid about an isolated three dimensional wing It consists of a series of trans formations; the first one is a shearing of the wing planform to unsweep a prespecified singular line which runs close to and along the wing leading edge The second one unwraps the wing surface about this singular line by means of a parabolic (square root) transformation The third and last tran sformation is a simple shearing of the coordinate lines The great computational efficiency of analytical methods of grid generation is offset by a very serious drawback: It is very difficult to generate sufficiently adequate grids for other than simple wing-body combinations

Somewhat akin to the concept of analytical grid generation is the parametric method; in fact parametic grid generation may be considered an extension or generalization of the analytical method The basic idea of the parametric method is as follows: The boundary surfaces of the computational domain are specified as functions of two parametric coor dinates; these surfaces are then connected by univariate in terpolating functions of a third parameter (Fig 24) The grid points are obtained by given specific values to each one of the three parameters Eriksson's[38] method of transfinite mesh generation is a recent example of this type of technique

The third method of grid generation obtains the coordinates of the grid points by numerically solving some appropriate elliptic partial differential equations which relate the physical coordinates to the computational coordinates The method of Thompson et al [39] exemplifies this approach; more recent developments along this line, with extentions to three dimensions, have been presented by Thomas [40]

Of the above three methods so far only the first one has been implemented in the codes used in practical applications Recently a great proliferation of various grid generation procedures has taken plance as demonstrated by the workshop on grid generation held at NASA Langley in 1980 There is yet little experience as to how well the various full potential flow solvers perform in practice with these grids: many questions of stability convergence and accuracy remain to be resolved It is quite appropriate to close this section by reiterating the importance for practical ap plications of adequate grid generation about arbitrarily complex geometries; the satisfactory solution of this problem will greatly expand the effectiveness of computational aerodynamics

Solution Mode

Computational aerodynamics has a very special advantage over the wind tunnel: Its ability to reverse the design process by computing the geometry that yields some desired aerodynamic characteristics rather than calculating the characteristics for a given geometry This mode of solution known as inverse, has obvious and great significance for design applications

Inverse solution modes have been developed for both singularity and field methods Singularity methods are ideally suited for this mode of solution; corresponding inverse methods abound and work will continue to be pursued in this area because of its great practical value More difficult is the development of inverse methods for finite volume techniques; this is due not only to the nature of such techniques but also to the fact that they are usually applied to nonlinear transonic flow problems In addition to the classical closure problem the latter consideration poses questions of existence of solutions and in some cases, of ill posed boundary con ditions In spite of these difficulties several inverse finite difference methods have been developed and used successfully in transonic applications Of these methods only those ap plicable to three dimensional full potential flow are addressed herein

The several approaches that have been developed for three dimensional transonic flow application can be grouped in two broad categories as follows: 1) methods based on the fictitious gas concept and 2) methods based on some pertubation redesign or geometry relaxation scheme

The fictitious gas concept was originally proposed by Sobieczky et al ,[41 42] and several variations and refinement of the method have been implemented in practical computer codes An example of this type of technique is the one im plemented at the Lockheed California Company and described in what follows [43]

In the present implementation of the fictitious gas ap proach a general characteristic of the flow rather than a definite pressure distribution is specified namely that the flow should be free of shocks To obtain the corresponding geometry a baseline configuration is analyzed at the desired design condition as defined by a Mach number and angle of attack combination using a version of the full potential transonic code FLO 22 which has been modified to solve the flow equation of a fictitious gas This fictitious gas is characterized by a law which gives the speed of sound as function of the flow velocity in a physically correct manner wherever the flow is subsonic but which renders the local flow subsonic when the velocity exceeds the physically correct speed of sound Consequently the entire flowfield en compassing subsonic and supersonic regions is elliptic Thus the solution obtained for the velocity potential is correct in all subsonic regions up to the sonic surface whose shape and extent depend upon the choice of the fictitious gas model After obtaining the fictitious gas solution the flowfield within the supersonic domain is recomputed by a marching procedure using real gas flow equations and the sonic surface data By appropriate integration of the supersonic velocity field, a new stream surface and hence wing geometry is obtained under the supersonic enclosure The overall process is schematically illustrated in Fig 25 and typical results are shown in Fig 26

Methods based on perturbation redesign are exemplified by the approach proposed by Garabedian and McFadden[44] whose principal features are outlined next in terms of its implementation at the Lockheed California Company as part of an analysis and design code known as FLO 22 5 [45]

Starting from a baseline wing geometry the surface pressure distribution is computed at the design condition The next step in the design procedure is the generation of a "target" pressure distribution This is obtained as a modifi cation of the previously computed pressure distribution The designer examines the initial pressure distribution and selects the span stations that need modification At these selected stations the chordwise extent of the regions to be modified on the upper and lower surfaces are delineated A small number of target pressure coefficient values and their corresponding chordwise locations are input for the region to be modified; the target pressure distribution is approximated by a spline fit through the input values This spline fit is then used to calculate the local flow velocity at the mesh points on the wing surface; the difference between the initial surface velocity distribution and the target surface velocity distribution provides the driving force for the surface modification The mathematical formulation of the problem leads to the following partial differential equation that governs the geometry modification:

$$F_{20}\frac{\partial}{\partial t}\frac{\partial 2S}{\partial X_1^2}+F_{10}\frac{\partial}{\partial t}\frac{\partial S}{\partial C1}+F_{00}\frac{\partial S}{\partial t}=(Q^2-q^2) \qquad (4)$$

Here, $S(X_1\ \ Z_1)$ defines the wing surface in the sheared parabolic coordinate system $(X_1\ \ Y_1\ \ Z_1)$ The pseudotime coordinate t, is analogous to the iteration level The symbol Q denotes the target velocity and q denotes the value of the velocity for any iteration level The coefficients F_{00} F_{10} and F_{20} are constant whose values are determined by con siderations of numerical stability and convergence

If superscript n denotes the value of a variable at nth level of iteration, one can write

$$S^n = S^{n-1} + \Delta S$$

and an equation for ΔS may be derived as

$$F_{20}\frac{\partial^2(\Delta S)}{\partial X_1^2}+F_{10}\frac{\partial(\Delta S)}{\partial X_1}+F_{00}\,\Delta S=\Delta t(Q^2-q^2) \qquad (5)$$

This equation is integrated by marching in X_1 direction using a central difference approximation to the second derivative and forward difference to the first derivative Once the initial surface is updated the flow is computed on this modified surface The new values of the surface velocity coupled with the target values are used to recompute the modifications to the surface geometry The cycle is repeated until covergence is achieved or a prescribed number of cycles is completed (Fig 27) Chordwise pressure distributions corresponding to the initial data target values and the final redesigned values together with the corresponding initial and modified surface geometries, are shown in Fig 28 In this particular example the target values were specified for the upper surface only

The development of inverse methods is, and will continue to be the one area of computational aerodynamics offering a unique and most valuable benefit for the design process Even though substantial effort has already been dedicated to this area future possibilities and the corresponding payoffs will justify an even higher level of dedication

Boundary-Layer Solution

Boundary layer calculations were probably one of the earliest activities of computational aerodynamics and many methods and corresponding computer codes have been developed for the analysis of two and three dimensional laminar and turbulent boundary layers In spite of the significant amount of effort that has gone into this area, progress has been slow; here the computational aero dynamicist is frustrated not only by the difficulty of the problem but also by the scarcity of good quality reliable experimental data which are so necessary for the development and evaluation of the theoretical models

From the standpoint of most pratical applications the prediction of five boundary layer characteristics are im portant, as follows:

1) Skin friction
2) Heat transfer
3) Laminar to turbulent flow transition
4) Displacement effect
5) Separation

The prediction of transition and separation based solely on numerical and theoretical methods has no satisfactory solution at the present time and empirical criteria have to be

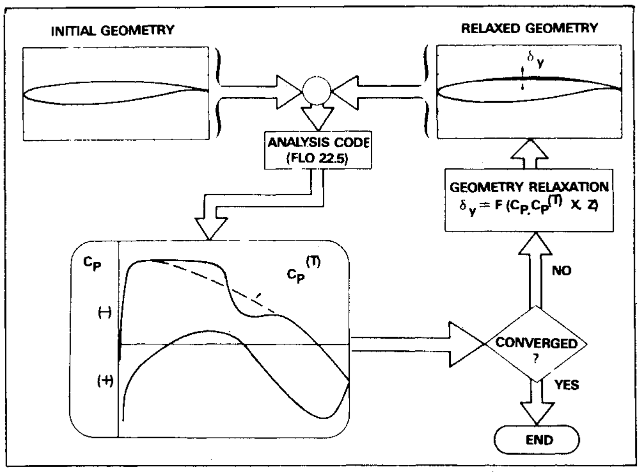

Fig 27 Design process using geometry relaxation

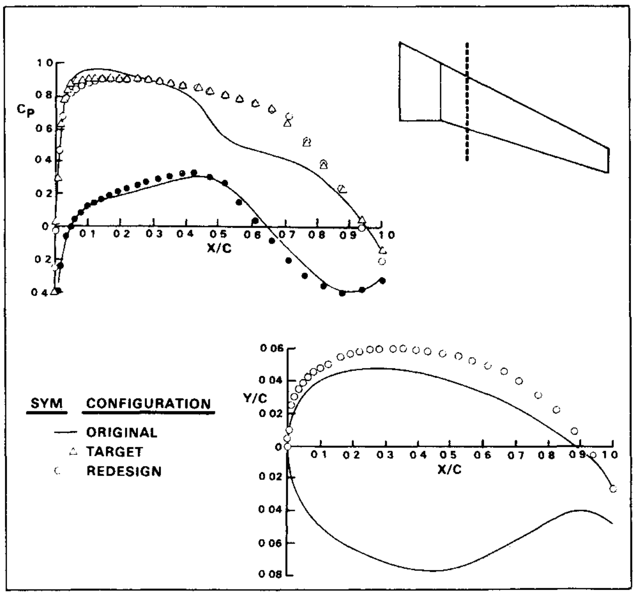

Fig 28 Example of design by geometrical relaxation

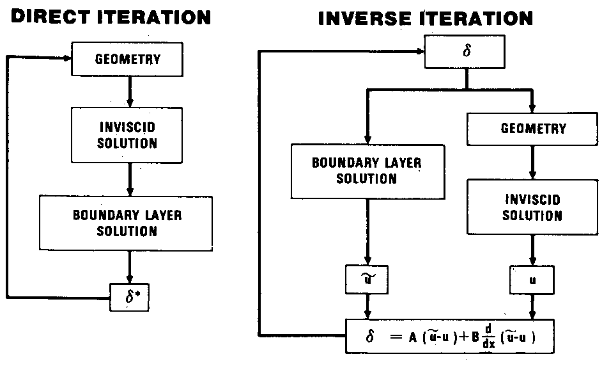

Fig 29 Methods for viscous inviscid coupling

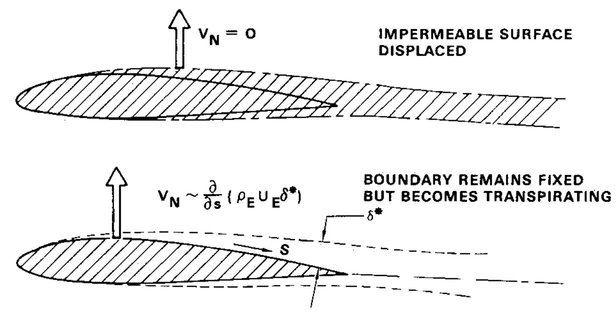

Fig 30 Methods for simulating boundary layer displacement effect

resorted to On the other hand the calculation of skin friction and heat transfer, although based on semiempirical approaches is fairly adequate and straightforward The prediction of the boundary layer displacement effect has been found to be most crucial for the calculation of real transonic flows about wings with highly aft loaded airfoil sections Even though it is not trivial the computation of the boundary layer displacement is amenable to numerical solution and it is the subject of the present section

Two types of methods for solving the boundary layer equations have evolved over the years; integral and finite difference The majority of practical calculations of boundary layers has been performed by resorting to stripwise application of two dimensional integral methods such as those of Truckenbrodt [46] Scholz [47] Nash and MacDonald [48] Green et al ,[49] and East et al [50] Only fairly recently have three dimensional boundary layer methods been developed and coupled with panel or finite difference potential flow methods

The use of finite difference boundary layer codes as part of a coupled viscous inviscid computation still remains unsatisfactory for practical application due to the ' fragility'' and cost of these codes Presently available finite difference boundary layer codes cannot continue the solution process through any separation point or line no matter how localized this separation may be; this introduces a high sensitivity to the pressure distribution imposed by the inviscid flow upon the boundary layer Furthermore finite difference codes have not proven to be generally more accurate than integral codes in the computation of the boundary layer displacement thickness, the key parameter in the viscous inviscid coupling The cost of running a finite difference computation of the entire inviscid flowfield In a typical iterative calculation the boundary layer computation has to be repeated two to five times; this results in a cost which is too excessive for routine design application

The development of integral methods for three dimensional boundary layer calculations has suffered some neglect partly due to the emphasis placed on finite difference solutions The present inadequacy of the latter within the framework of a coupled viscous inviscid computation already referred to has recently led to a reconsideration and revival of integral approaches Presently the best developed integral methods for three dimensional boundary layers are those due to Smith[51] for turbulent flow and Stock[52] for laminar flow Computer routines implementing this method have been coupled successfully with potential flow solvers such as FLO 22 yielding fairly robust codes

The conventional application of the method of integral relations to the solution of the boundary layer equations involves the assumption of some parametric velocity profiles Herein lies the main source of inaccuracy for this type of approach; if the assumed shape of the boundary layer velocity profile does not correspond to physical reality then the method cannot be expected to yield reliable answers But integrating the boundary layer in multiple, instead of a single layers provides additional equations which can be used to determine additional parameters to define the velocity profile within the boundary layer thus increasing the generality of the technique Although this approach has already been successfully tried by Moses[53] in two dimensional flows it has not yet been implemented in three dimensions Such implementation is worthy of investigation; although the computational cost will be greater than single-layer integral methods, it will still remain substantially lower than that of differential methods and it will probably result in a significant increase in the reliability and accuracy of integral techniques

Viscous-Inviscid Coupling

Two basic types of viscous inviscid interaction models have been developed: 1) the direct iteration and 2) the inverse iteration, Fig 29 In the direct iteration approach, the inviscid flowfield is calculated for the given geometry; the results of the inviscid calculations are used in the solution of the boundary layer equations to determine a displacement thickness distribution This displacement thickness distribution is then used to update the boundary condition of the inviscid calculation and the process is repeated until convergence is achieved

The inverse iteration uses the boundary layer equations to determine a velocity distribution at the body surface which corresponds to a specified displacement thickness distribution Le Balleur[54 55] has presented the basic concepts underlying this approach and further developments have been reported by Wigton and Holt [56] The inverse iteration concept yields an updating scheme for the displacement thickness of the boundary layer of the form

$$\delta^{*(n+1)} - \delta^{*(n)} = A(\tilde{u}-u) + B\frac{d}{dx}(\tilde{u}-u) \qquad (6)$$

where $\tilde{u}$ is the solution of the boundary layer equation for a given $\delta^{(n)}$ u the inviscid flow solution the superscript n denotes the iteration cycle, and the values of the A and B parameters depend on the local Mach number

The direct iteration is the more highly developed method and has been found to be adequate as long as the boundary layer is attached and there is no strong interaction such as shock induced separation or trailing edge separation Two ways of implementing this model have been tried as indicated in Fig 30: 1) modifying the geometry by the appropriate displacement surface, or 2) modifying the boundary condition, i e the surface transpiration concept The latter approach is more satisfactory because of the surface geometry and the computational grid are not affected by the boundary layer This means that for panel methods the aerodynamic influence coefficients and for finite difference methods the computational grid do not have to be recomputed at each iteration

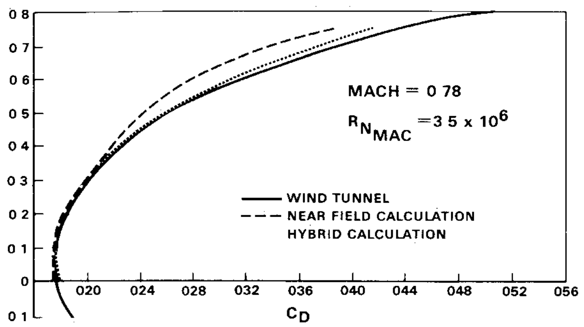

Fig 31 Comparison of techniques for drag computation with ex perimental data; supercritical wing body configuration

The inverse iteration method is fairly new and it evolved from the need to handle separated flows more effectively Only two dimensional versions have been developed so far and they have not yet been used extensively in design work but they offer the potential of modeling flows with extensive separation without having to resort to Navier Stokes solu tions

Drag Computation

The computation of drag is considered here as a principal and separate element of a numerical flow simulation in order to emphasize the practical importance that this aerodynamic parameter has in airplane design In many cases this is the bottom line quantity which determines the goodness of any given design Therefore its accurate calculation as part of the numerical flow simulation deserves special consideration

Drag can be considered to be made up of two components: friction drag and pressure drag If $\bar{t}$ denotes the tangential shear stress vector at a point on the body surface p the static pressure and $\bar{n}$ the external normal to the element of surface dS then the drag can be formally expressed as

$$\bar{D}=\int_{S_B}\bar{t}\;e_\infty \mathrm{d}S-\int_{S_B}p\bar{n}\;\bar{e}_\infty \mathrm{d}S \qquad (7)$$

where S_B stands for the totality of the body surface wetted by the fluid and $\bar{e}_\infty$ is a unit vector parallel to the freestream direction The first integral represents the friction component and the second integral is the negative of the pressure drag The latter integral poses numerical difficulties which require special techniques for its computation.

The most straightforward approach to computing pressure drag is to perform the numerical integration indicated in Eq 7 Usually as the result of the numerical flow simulation the pressure distribution is known at the body surface con sequently, all of the elements needed for the numerical in tegration are known; this approach is usually called the near field method of drag computation Unfortunately this is a very inaccurate procedure The reason for this is that the pressure drag integral is the difference between the integration performed on forward facing and rearward-facing surface elements this difference being a small second order quantity for slender bodies

This difficulty can be circumvented by applying the momentum theorem and calculating the momentum flux at some arbitrary but convenient control surface enclosing the configuration. This approach is known as the far field method of drag computation It has been found to be quite accurate for subsonic and supersonic applications but a problem has arisen in its implementation in full potential transonic flow codes such as FLO 22 FLO 28 and FLO 30 namely the computed value of drag varies with the choice of control surface The calculated momentum flux should be constant for any control surface which entirely encloses the configuration and all associated shock waves but in numerical experimentations conducted at the Lockheed California Co it was noticed that this is not generally true

An alternative approach that appears to be quite satisfactory is a hybrid method whereby the pressure drag is determined by evaluating the vortex drag and the wave drag[45] separately The vortex drag results from the shed vorticity sheet whose strength is related to the spanload distribution; once the spanwise lift distribution is computed, the calculation of the vortex drag is performed in the Trefftz plane—a far field technique Calculation of the wave drag follows the approach proposed by Garabedian and Mc Fadden[44]—essentially a near field method In this approach the wave drag is calculated by numerically integrating a term proportional to the square of the second derivative of the perturbation potential along the streamwise direction; this term is also related to the artificial viscosity associated with the rotated difference scheme used in the supercritical flow region

Drag predictions for a wing body configuration by near field and hybrid methods are compared with experimental results in Fig 31 The comparison is shown at the design Mach number of the subject configuration which has a moderately thick sweptback wing of high aspect ratio designed for a cruise lift coefficient of 0 60 As can be ob served the hybrid approach offers a remarkable improvement over the computation of drag by direct surface pressure in tegration This improvement existed at all Mach numbers below the onset of steep drag rise At the higher Mach numbers the agreement between the hybrid technique and the experimental results deteriorated due to the onset of strong shock/boundary layer interaction and separation phenomena which were not modeled adequately in the numerical computations

In summary the prediction of airplane drag is of paramount importance in airplane design and the numerical difficulty of the task requires that careful attention be given to the techniques used The accuracy and completeness of the drag prediction is, of course limited by the degree of realism of the numerical flow simulation but significant improvement can be achieved within the present capabilities of com putational aerodynamics A large amount of detailed flowfield information is computed as part of the solution process of presently used computer codes This information should be sufficient to calculate drag with generally adequate accuracy; thus the problem is not the lack of information but the manner of using it

Concluding Remarks

An overview of the various elements of computational aerodynamics codes that are the main contributors in determining the practical effectiveness of numerical flow simulation has been presented Computational aerodynamics has grown so rapidly over the past few years that a com prehensive survey is beyond the scope of any reasonable length paper therefore this has not been attempted here On the contrary the survey that has been presented is selective and in may respects biased by the author's own experience; areas of importance for some applications have been left untouched But it is hoped that this survey has served to clarify and emphasize some points of significant practical value as follows:

1) The effectiveness of computational aerodynamics depends not only on the accuracy of the codes but to a very large degree—perhaps more than is generally appreciated—on their robustness ease and economy of use

2) Potential flow models appear to be quite adequate for the vast majority of airplane applications provided appropriate viscous corrections are incorporated in the transonic regime

3) Greater gains in effectiveness are more likely to come from the expansion of the geometrical capabilities of inviscid

flow codes rather than from the introduction of more ac curate flow models such as Navier Stokes solvers

4) Adequate modeling of nonpotential flow pheno mena—such as separation and leading edge vortex flows—can be accomplished within the general framework of potential methods This requires ingenuity on the part of the methods developer but the practical benefits are worth the effort

Finally computational aerodynamics and wind tunnel testing should not be viewed as competing technologies trying to do the same job. Rather they should be used in a com plementary fashion: due to its unique advantages in time cost inverse solution capability, and the relative ease of including aeroelastic effects, computational aerodynamics is an optimization and preliminary design tool par excellence On the other hand, the general accuracy of the wind tunnel remains unmatched and it still is the best tool for verification and validation and it is likely to remain so for the foreseeable future

References

[1] Garrick I E 'Nonsteady Wing Characteristics ' Section F of Aerodynamic Components of Aircraft at High Speeds Volume VII of *Princeton Series in High Speed Aerodynamics and Jet Propulsion* Princeton University Press Princeton, N J 1957

[2] Richtmyer R. D and Morton K W *'Difference Methods for Initial Value Problems* John Wiley & Sons New York 1967

[3] Rizzi A and Eriksson L E , 'Transfinite Mesh Generation and Damped Euler Equation Algorithm for Transonic Flow Around Wing Body Configurations AIAA Paper 81 0999 1981

[4] Steger, J L and Baldwin B S Shock Waves and Drag in the Numerical Calculation of Isentropic Transonic Flow NASA TN D 6997 Oct 1972

[5] Johnson F T , Lu P , Tinoco E N and Epton M A ' An Imposed Panel Method for the Solution of Three Dimensional Leading Edge Vortex Flows NASA CR 3278 July 1980

[6] Kandil O A Mook D T and Nayfeh A H Nonlinear Prediction of the Aerodynamic Loads on Lifting Surfaces ' *Journal of Aircraft* Vol. 13, Jan 1976, pp 22 28

[7] Kandil O A Mook D T and Nayfeh A H A Numerical Technique for Computing Subsonic Flow Past Three Dimensional Canard Wing Configurations with Edge Separations AIAA Paper 77 1, 1977

[8] Kandil O A Mook D. T and Nayfeh, A H Three Dimensional Steady and Unsteady Asymmetric Flow Past Wings of Arbitrary Planforms AGARD CP 227, Sept 1977

[9] Nangia, R K and Hancock, G J ' A Theoretical Investigation for Delta Wings with Leading Edge Separtion at Low Speeds; ' ARC CP 1086 1968

[10] Mehrotra S C and Lan C E A Theoretical Investigation of the Aerodynamics of Low Aspect Ratio Wings with Partial Leading Edge Separation, NASA CR 1453304 May 1978

[11] Grossman B and Siclari M J 'The Nonlinear Supersonic Potential Flow over Delta Wings, AIAA Paper 80 0269 Jan 1980

[12] Shankar V and Chakravarthy S , ' An Implicit Marching Procedure for the Treatment of Supersonic Flow Fields Using the Conservative Full Potential Equation AIAA Paper 81 1004 1981

[13] Ward G N , *Linearized Theory of Steady High Speed Flow* Cambridge University Press Cambridge Mass 1955

[14] Maskew B Rao B M , and Dvorak F A Prediction of Aerodynamic Characteristics for Wings with Extensive Separations Paper No 31 in *Computational of Viscus Inviscid Interactions* AGARD CP 291 Sept 1980, pp 31 1 31 15

[15] Maskew B and Dvorak F. A The Prediction of CLMAX Using a Separated Flow Model, *Journal of American Helicopter Society* Vol 23 April 1978 pp 2 8

[16] Rao, B M Maskew B , and Dvorak F A , Prediction of Aerodynamic Chracteristics of Fighter Wings at High Lift ONR Rept CR 2115 258 1 July 1979

[17] Smith J H B Vortical Flows and Their Computation *Computational Fluid Dynamics,* VKI LS 1980 Vol 1 1980

[18] Hunt B , 'Recent and Anticipated Advances in Panel Methods: The Key to Generalized Field Calculations? *Computational Fluid Dynamics* VKI LS 1980 Vol 1 1980

[19] Butter D J and Mobbs D Viscous Flow Modeling in Panel Methods *Computational Fluid Dynamics* VKI LS 1980 Vol 1 1980

[20] Smith A M O and Hess J L , Calculation of the Non Lifting Potential Flow about Arbitrary Three Dimensional Bodies Douglas Rept E S 40622 1962

[21] Rubbert P E and Saaris, G R A General Three Dimensional Potential Flow Method Applied to V/STOL Aerodynamics, SAE Paper 680304, 1968

[22] Woodward, F A Tinoco, E N and Larsen J W Analysis and Design of Supersonic Wing Body Combinations Including Properties in the Near Field NASA CR 73106 1967

[23] Woodward F A A Unified Approach to the Analysis and Design of Wing Body Combinations at Subsonic and Supersonic Speeds AIAA Paper 68-55, 1968

[24] Carmichael R L and Erickson, L L , ' PAN AIR—A Higher Order Panel Method for Predicting Subsonic and Supersonic Linear Potential Flows About Arbitrary Configurations ' AIAA Paper 81 1255, 1981

[25] Maskew B. ' Prediction of Subsonic Aerodynamic Charac teristics—A Case for Low Order Panel Methods ' AIAA Paper 81 0252 1981

[26] Coopersmith R M , Youngren H H , and Bouchard E E ' Quadrilateral Element Panel Method (QUADPAN) Lockheed California LR 29671, 1981

[27] Hess J L "A Higher Order Panel Method for Three Dimensional Potential Flow NADC 77166 30 1979

[28] Sytsma H S Hewitt, B L and Rubbert P E 'A Com parison of Panel Methods for Subsonic Flow Computation AGARD AG 241, 1979 1979

[29] Fluid Dynamics Panel Working Group 04, Experimental Data Base for Computer Program Assessment, ' AGARD AR 138 1979

[30] Hicks R M 'Transonic Wing Design Using Potential Flow Codes—Successes and Failures, SAE Paper 810565 1981

[31] Jameson A , 'Transonic Flow Calculations,' *Numerical Methods in Fluid Dynamics* edited by H J Wirz and J J Smolderen McGraw Hill Book Co New York 1978

[32] Ballhause W F *Some Recent Progres in Transonic Flow Computations,* edited by H J Wirz and J J Smolderen McGraw Hill Book Co New York, 1978

[33] Bailey F R and Ballhaus W F Relaxation Methods for Transonic Flow about Wing Cylinder Combinations and Lifting Swept Wings *Lecture Notes in Physics* Vol 19, Springer New York, 1972

[34] Boppe C W Calculation of Transonic Wing Flows by Grid Embedding ' AIAA Paper 77 207 1977

[35] Hinson, B L and Burdges K P An Evaluation of Three Dimensional Transonic Codes Using New Correlation Taiload Test Data, AIAA Paper 80 0003 1980

[36] Jameson A and Caughey, D A , Numerical Calculation of the Transonic Flow Past a Swept Wing ' ERDA R&R Rept C00 3077 140, New York University 1977

[37] Jameson A and Caughey D A. Progress in Finite Volume Calculations for Wing Fuselage Combinations ' *AIAA Journal* Vol 18, Nov. 1980 pp 1281 1288

[38] Ericsson L E Three Dimensional Spline Generated Coor dinate Transformations for Grids Around Wing Body Con figurations *Numerical Grid Generation Techniques* NASA CP 2166, 1980

[39] Thompson J F , Thames F C and Martin C W ' Boun dary Fitted Curvilinear Coordinate Systems for Solution of Partial Differential Equations on Fields Containing Any Number of Ar bitrary Two Dimensional Bodies, NASA CR 2729, 1977

[40] Thomas, P D 'Construction of Composite Three Dimensional Grids from Subregion Grids Generated by Elliptic Systems AIAA Paper 81 0996 1981

[41] Sobieczky H Yu N J. Fung, K -Y and Seebas A R New Method for Designing Shock Free Transonic Configuration *AIAA Journal* Vol 17 July 1979, pp 722 729

[42] Fung K Y Sobieczky H and Seebass, A R Shock Free Wing Design *AIAA Journal* Vol 18 Oct 1980 pp 1153 1158

[43] Raj P Miranda, L R and Seebass A R A Cost Effective Method for Shock Free Supercritical Wing Design AIAA Paper 81 0383 1981

[44] Garabedian, P and McFadden G., Computational Fluid Dynamics of Airfoils and Wings *Proceeding of the Symposium on Transonic Shock and Multidimensional Flows* University of Wisconsin Madison 1981

[45] Raj P and Reaser J S An Improved Full Potential Finite Difference Transonic Flow Code (FLO 22 5) for Wing Analysis and Design Lockheed California LR 29759 1981

[46]Truckenbrodt, E , ' A Method of Quadrature of Calculation of the Laminar and Turbulent Boundary Layer in Case of Plane and Rotationally Symmetrical Flow NACA TM 1379, May 1955

[47]Scholz, N Zur rationallen Berechnung laminarer and tur bulenter kompressibler Grenzschitchen mit Warmeubergant *Zeitschrift fur Flugwissenschaften,* Vol 7, Oct 1959 pp 33 39

[48]Nash J F and Macdonald, A G J. The Calculation of Momentum Thickness in a Turbulent Boundary Layer at Mach Numbers up to Unity British ARC CP 963 1967

[49]Green, J E Weeks D J and Brooman J W F. Prediction of Turbulent Boundary Layers and Wakes in Incompressible Flow by a Lag Entrainment Method, ' British ARC R&M 3791 1977

[50]East L F Smith P D and Merryman P J ' Prediction of the Development of Separated Turbulent Boundary Layers by the Lag Entrainment Method British Royal Aircraft Establishment Tech Rept 77046 1977

[51]Smith P D An Integral Prediction Method for Three Dimensional Compressible Turbulent Boundary Layers ARC R&M 3739 1974

[52]Stock H W Integral Method for the Calculation of Three Dimensional Laminar and Turbulent Boundary Layers NASA TM 75320 1978

[53]Moses H L A Strip Integral Method for Predicting the Behavior of Turbulent Boundary Layers, *Computation of Turbulent Boundary Layers 1968 AFOSR IFP-Stanford Conference Proceedings* Vol 1 1968 pp 76 82

[54]Le Balleur J C , 'Couplage Visqueux Non Visqueux: Methode Numerique et Applications, Aux Ecoulements Bidimensionnels Transsoniques et Supersoniques, ' *La Recherche Aerospatiale* Vol 1978 2 March April 1978 pp 67 76

[55]Le Balleur, J C 'Calcul des Ecoulements a Forte Interaction Visqueuse au Moyen de Methodes de Couplage AGARD Conference on Computation of Viscous Inviscid Interactions Colorado Springs Preprint 291, Paper 1 1980

[56]Wigton, L B and Holt M Viscous Inviscid Interaction in Transonic Flow AIAA Paper 81 1003 1981