PROGRESS IN THE DEVELOPMENT OF COMPUTATIONAL AERODYNAMIC DESIGN METHODS: INVERSE PROCEDURES

John B. Malone
NASA Langley Research Center
Hampton, Virginia 23681

Abstract

This paper describes some recent progress in the development of a class of computational aerodynamic design methods referred to as inverse procedures. The discussion is directed towards a selection of those inverse methods which incorporate computational fluid dynamics methodologies based on either the Euler or Navier-Stokes equations. Two classifications of inverse design algorithms are presented, those based on a coupled flow/geometry solution procedure, and those based on a decoupled flow/geometry solution procedure. Each of the methods described have been successfully used for the aerodynamic design of two-dimensional or three-dimensional aerodynamic configurations, such as airfoils, wings, wing-body configurations, and wing/pylon/nacelle configurations. Two hybrid aerodynamic design procedures, coupling an inverse design method with an optimization procedure, are also discussed. Brief descriptions are provided for each method. Information is given on any unique features, the computational effort required to use the different design techniques, the range of computational problems to which the methods have been applied, and, where known, any practical limitations on the application of the design procedures.

Introduction

The creation of successful aircraft designs in the modern engineering environment requires the application of computational methods and other analysis techniques during all phases of the design process, including the conceptual, preliminary and production, or detailed design, stages. During the conceptual phase, many important vehicle aerodynamic parameters are developed, such the lifting-surface arrangement to be used (i.e. wing/tail vs. wing/canard, etc.), the wing placement relative to the center-of-gravity of the vehicle, the wing aspect ratio, the wing sweep, the wing placement on the fuselage (e.g. high, mid, low), as well as "first-cuts" at defining fuselage cross-sections from nose-to-tail to satisfy geometric and wave-drag constraints. In addition to these types of aerodynamic parameters, other discipline requirements are also developed at this early stage in the design, primarily in the areas of structures and propulsion. These additional design requirements often impact the aerodynamic design as well. For example, structural considerations such as the wing-box volume required to allow space for fuel tanks, hydraulic-fluid pipes, control-surface actuators and electrical wiring can impact the final aerodynamic shape of the new vehicle. Thus, successful aircraft designs are arrived at through compromises made within a multidisciplinary context.

The details of the vehicle external lines or contours are generated during the preliminary design phase. During the design process, which can consume many months of engineering design effort [1], analytical and numerical predictions are often combined with wind tunnel model test results to arrive at a viable vehicle configuration. In the past, the computational portion of this process often consisted of a "cut-and-try" approach. The "cut-and-try" approach consists of multiple applications of aerodynamic analysis methods (i.e. the aerodynamic surface shape is assumed to be known and fixed during the computation). Using this approach, the design engineer must use past experience and training to guide changes to the geometry definition in hopes that aerodynamic improvements are demonstrated by the next analysis. This approach to aerodynamic design is time consuming, and usually requires considerable expertise to produce useful configuration shapes.

In an effort to improve the aerodynamic design process, computerized methods have been developed over the past three decades which attempt to automate, to some degree, the process of generating geometric surfaces with prescribed or, at least, favorable, aerodynamic characteristics. These methods are generally separated into three categories: indirect methods; inverse methods; and optimization methods. Each of the three design methods have benefited from advances in Computational Fluid Dynamics (CFD) algorithms which were originally developed for analysis applications. The goal of developing and using CFD-based design methods is both to increase the accuracy of the predicted performance of the vehicle design, as well as to ultimately decrease the calendar time required to achieve specified design goals.

Prior to the last decade, computerized aerodynamic design procedures were based upon methods which solved the linearized form of the potential-flow equation. These design procedures include linear-equation techniques such as the supersonic Vortex-Lattice-type method of Woodward and Tinoco [2], and the subsonic Surface-Singularity Panel methods, such as those developed by Bristow [3], Fray and Slooff [4] and Malone[5]. Later, design procedures based on the Full-Potential Equation (FPE) were developed Tranen [6], Shankar and Malmuth [7] , Garabedian and McFadden [8] , Malone, Vadyak and Sankar [9], and Campbell and Smith[10]. The FPE-based methods could account for the non-linear effects of shockwaves on the airload distributions, and with use of coupled boundary-layer computational methods, could account for some viscous effects as well.

Although the cost, in terms of computer resources, is greater for higher-order CFD methods based on the Euler Equations (EE) and the Reynolds-Averaged Navier-Stokes (RANS) equations than for those methods based on potential-flow formulations, there is now an increasing interest in developing aerodynamic design procedures that use the higher-order algorithms. If methods incorporating more advanced CFD algorithms can be developed, validated, and then actually used during the design process, the aerodynamicist can account for the occurrence of important fluid dynamic phenomena which cannot be adequately predicted with potential-flow-based methods. These phenomena include rotational flow effects, vortical flow field structures, and a number of significant viscous effects such as flow separations and strong shock/boundary-layer interactions. The flight conditions where these non-linear fluid dynamic phenomena occur are often the most critical from an aerodynamic loads perspective, and thus can significantly effect the performance and structural weight of the vehicle's final design.

A number of excellent papers are available which review the state-of-the-art in computational aerodynamic design methods. Among these are papers by Slooff [11], Dulikravich[12], Labrujere and Slooff [13], Labrujere and Slooff [14], and Dulikravich [15]. These papers survey selected publications in each of the major classifications of aerodynamic shape design methods: indirect, inverse and optimization methods. Additionally, several technical conferences have been held during the past decade which address this topic in considerable detail. Among these conferences are two AGARD meetings [16,18], two AIAA meetings [19,20], and one organized by Dulikravich [17]. Readers are directed to the aforementioned publications to obtain information on the complete range of design methods. In contrast, this paper seeks to highlight recent progress in one specific technical area related to aerodynamic design methodology. Specifically, the paper describes a selection of inverse design methods which incorporate EE and RANS CFD algorithms as part of the aerodynamic shape design methodology.

Inverse Methods

As discussed in reference 14, inverse methods may be divided into two major categories: coupled-solution methods and decoupled-solution methods. Coupled-solution methods are those in which the geometric boundary is usually obtained directly, along with the flowfield variables, as part of the solution of a nonlinear boundary-value problem. With these methods, the usual analysis procedures for solving the fluid dynamic equations are reformulated so that the flowfield and geometric variables are computed in a simultaneous fashion. In this sense, the two different types of variables, flow and geometry, are "tightly coupled" in the numerical procedure. With these methods, the final geometry is either arrived at directly as part of the design process, or computed using geometric transformations between solution variables at the end of the calculation. This feature eliminates the need to compute interim geometry shapes during the design process. A flow diagram of a typical coupled-solution inverse method is illustrated in Figure 1.

Decoupled-solution methods, on the other hand, separate the design process into repetitive cycles consisting of two distinct steps. First, an analysis is performed of the flowfield corresponding to a specified geometry. Then, a geometry update is determined using the flowfield properties computed in the first step to predict a new geometry which should provide a better match with the design objectives. The execution of this pair of steps in the design process is referred to as a design cycle. To use these decoupled design procedures, one must then iterate between the two steps until a satisfactory solution is obtained.

Decoupled methods are themselves separated into three categories: Neumann methods, Dirichlet methods and variational methods [14]. Only the Neumann and Dirichlet methods will be addressed in this paper. In the Neumann methods, a pressure distribution is computed in step one, and then used to predict a new geometry in step two. One benefit of the Neumann-type of decoupled procedures is the possibility to use CFD analysis methods, as virtually unchanged, "black boxes", to provide the necessary flowfield information for the first step of the process. Thus, new CFD methods can be easily incorporated into the design procedure as they become available. Figure 2 illustrates a typical Neumann-type of decoupled inverse procedure.

In the Dirichlet methods, a non-zero normalwash is computed in the first step and subsequently used to update the geometry in the second step. Two methods used to update the geometry using the computed normalwash are the conservation of mass flux (transpiration model) and a streamline alignment procedure.

Regardless of the type of inverse method under consideration, the objective of the design process is to achieve a given pressure distribution or velocity distribution imposed along all, or some part of the geometry boundary. This distribution of pressure (or velocity) is often referred to as the "target" distribution. Even for some methods which allow the use of integrated aerodynamic load coefficients (lift, moment or drag) as the design objective(s), a non-unique, but realistic, pressure distribution is actually computed from these design objectives and subsequently imposed as the target upon which the inverse design procedure operates. Thus, to effectively use inverse methods, the designer should be able to answer the question: "What desirable characteristics should the target pressures possess"? For example, the designer may require a shockless airfoil, or airfoil with only a weak shock, in order to reduce wave drag at transonic cruise speeds. Or, in another case, the design requirements might call for a favorable pressure distribution near the leading edge to promote laminar flow conditions in the boundary layer region. However, it is also possible to impose pressure distributions which do not lead to useful airfoil shapes (i.e. surface crossover, slope discontinuities, etc.). Volpe [21] discusses certain constraints upon the choice of pressure distributions which, if satisfied, insure the existence of useful airfoil shapes corresponding to the target distribution.

Coupled-Solution Methods

Giles and Drela [22, 23, 24] have developed a coupled-solution two-dimensional design method for the aerodynamic design of airfoil configurations. The method is based on a finite-volume, conservative formulation of the two-dimensional Euler equations. The streamwise coordinates of the computational mesh are aligned with streamtubes of the flowfield, while the remaining coordinate directions are aligned normal to the streamlines. Because of this choice of computational coordinates, the mass and energy conservation equations can be integrated exactly, and the unknown solution variables are reduced to the flowfield pressure and coordinate locations in the normal (to the streamline) direction. Numerical results are obtained from a Newton iterative procedure used to solve the resulting system of algebraic equations. The implementation of farfield boundary conditions permits the simulation of isolated airfoils, solid-wall wind tunnel flow, and freejet flow.

This method has been developed into a computer program, referred to as the ISES code, which can be used to perform aerodynamic analysis, full-design and mixed analysis-design computations. In the full-design mode, the complete airfoil shape is designed. In the mixed analysis-design mode, a portion of the suction side of the airfoil can be designed while keeping the remainder of the surface geometry fixed in a specified shape. To insure smooth geometry between the fixed and designed airfoil boundaries in the mixed analysis-design mode, two additional free parameters are introduced into the target pressure distribution and solved for during the computation. The authors indicate that although their method is efficient as an iterative procedure, converging in from three to fifteen iterations in the analysis mode, and four to six iterations in the design mode, the overall computational work is proportional to the product of the number of streamwise coordinates times the cube of the number of streamlines, thus limiting the number of streamtubes that can be used and still retain computational efficiency.

Drela and Giles [23] also describe a viscous-flow capability incorporated in the ISES code. A two-equation, integral boundary-layer model is used to predict the effects of laminar or turbulent flows. Transition is predicted using an amplification-factor technique based on the Orr-Sommerfeld equation. The integral boundary-layer equations are formulated to permit stable computations through separated-flow regions on the airfoil surface. For viscous-flow calculations, a Newton procedure is used to provide simultaneous solutions of both the Euler and boundary-layer equations in a strongly-coupled manner. The authors presented numerical results for airfoils in viscous-flow at low-Reynolds number and transonic flow conditions [23], and for airfoil and compressor-cascade configurations [24]. In reference 24, the author points out that using the method in the full-design mode can cause difficulties near blunt leading-edges, and recommends the use of the method in the mixed analysis/design mode with fixed leading-edge geometries.

Drela [24] also describes a modal-inverse formulation, which removes some of the difficulties associated with designing an airfoil when the targets contain a pressure discontinuity along the airfoil surface. Without some sort of shape control, airfoil surface-slope discontinuities can occur on the airfoil surface beneath these pressure-discontinuity locations. In the author's modal-inverse formulation, the surface in these regions is represented by a summation series of smooth shape functions, the participation coefficients of which must be solved for as part of a modified Newton iterative procedure. Although the modal-inverse option reduces the convergence rate of the design method from quadratic to linear, converged designs are still computationally efficient, requiring only three-percent extra computational effort for each shape function used in the design.

A transonic airfoil design problem was used to demonstrate the modal-inverse method in reference 24. For this case the target pressure distribution contains a moderate-strength shockwave on the upper surface. The final designed-geometry, which was obtained after five Newton-iterations, does not exhibit a surface-slope discontinuity at the shock location.

Neumann-Type Decoupled-Solution Methods

Three Neumann-type decoupled-solution methods described below are formulated such that the difference between computed and target surface pressures at interim stages of the design calculation are used to predict an updated geometry which should provide a better match to the target pressure distributions. This pressure difference function, or distribution, is referred to as the "residual".

Integral-Equation Method

Takanashi [25] has developed a residual-correction design method that can be used for three-dimensional wing design applications. A three-dimensional FPE direct-analysis method is used to obtain estimates of the actual pressures corresponding to a specified wing geometry. The difference between the user-specified target pressures and the calculated pressures becomes the residual distribution required to predict a new wing shape. The method used to update the surface geometry is derived from an integral-equation form of the three-dimensional, full-potential transonic-flow equation. After some rearrangement and certain small-disturbance assumptions are made, an expression which relates pressure residuals on the wing surface to changes in symmetric and antisymmetric components of the velocity potential is obtained. Since wing normal velocity can be related to derivatives of the velocity potential, the corrections, or updates, to the wing surface coordinates can be obtained by spatial integration of the calculated velocity components obtained after each design iteration. In deriving the integral geometry-correction equation, the assumption is made that trailing coordinates do not change value, which requires a starting, or initial geometry with the desired, final value of trailing-edge thickness. Since the method treats the direct-solver, or analysis method, as a "black box", virtually any CFD method can be incorporated into the design procedure. For example, the effect of fuselages can be approximated by using a wing-body analysis method to provide wing pressures for computing the residual distributions.

Hirose, Takanashi and Kawai [26] demonstrated the use of the Takanashi Method for airfoil design using a two-dimensional RANS airfoil analysis code. In order to utilize the three-dimensional design procedure for two-dimensional applications without making any coding changes to the integral method, a large aspect-ratio unswept wing with the appropriate airfoil shapes was used to generate geometry corrections. The numerically-generated computational mesh required for the two-dimensional airfoil analysis was then recomputed at each design step to provide a body-fitted grid.

In reference 26, five airfoil design cases were presented to demonstrate the use of the design algorithm. Reasonably converged results were achieved with from four to ten complete (i.e. fully converged) airfoil analyses for the cases presented. The large variation in design-iterations required for the example problems was attributed to the fact that, in some cases, of previously-converged flowfield results were used as starting conditions for the RANS code, thus reducing the total computational effort required.

Bartelheimer [27] has developed an improved integral-equation method which is suitable for the design of two-dimensional airfoils and three-dimensional wings in transonic flow. A multigrid, Runge-Kutta-type, Euler / Navier-Stokes solution procedure was used to obtain surface pressures during the analysis phase of the design process. The geometry corrections were obtained using a modified Takanashi integral-equation algorithm. However, unlike the results report in Reference 26, the author found that the integral-equation method would not converge for cases with supersonic flow on the upper surface. In order to achieve converged designs, upwinded derivative terms were substituted for the streamwise derivatives of velocity potential in the geometry-correction equation. Also, in order to reduce the computational effort needed to produce a body-fitted mesh for the flow solver, the grid was modified using a decaying-function, algebraic technique driven by the proposed incremental changes to the previous surface geometry. Finally, it was found necessary to use a high-order Bezier polynomial technique to smooth the geometry resulting from applying the integral-equation procedure.

Computed results for both inviscid and viscous two-dimensional design cases show improved convergence compared to the original integral-equation formulation. Converged results were obtained after eighty design iterations, with between eleven and thirty multigrid cycles used for each airfoil analysis. Two, three-dimensional wing-design cases were also presented showing converged results could be obtained over a range of between ten to thirty design iterations. The range of convergence might depend upon how significant a difference existed between the target pressures and the initial analysis results. Although not stated explicitly, it is assumed by this author that each design iteration required one converged direct-analysis result to predict the residuals for a subsequent design cycle. If this assumption is correct, a possible savings in computational effort might be achieved by using partially-converged analysis results to update the geometry for a subsequent design cycle.

Modified Garabedian-McFadden Method

Malone, Vadyak and Sankar [9] extended the original Garabedian-McFadden design procedure [8] to permit the aerodynamic design of complete nose-to-tail airfoil shapes. The design procedure, referred to as the Modified Garabedian-McFadden (MGM) design method, is classified as a decoupled-solution procedure of the Neumann-type[13]. As with the integral-equation method mentioned previously, the MGM method is also referred to as a residual-correction technique. The method utilizes an empirically-derived auxiliary equation, referred as a flexible membrane equation [15], to update the surface geometry between partially-converged flowfield solutions obtained from an available CFD flowfield analysis method. The form of the auxiliary equation combines terms for the first and second derivatives of airfoil surface ordinate with respect to the streamwise coordinate, together with user-specified multiplicative constants used to enhance numerical stability. The concept used for the auxiliary geometry correction equation is general enough to admit additional mixed derivative terms in the spanwise coordinate direction for three-dimensional wing design applications [8]. The technique is capable of producing useful (i.e. manufacturable) airfoil shapes which exhibit prescribed airfoil pressure distributions at a given flowfield Mach number and airfoil angle-of-attack. Airfoil ordinate smoothing can be used between each design cycle, but the method has been used successfully without smoothing when required for particular design applications. As originally formulated, the MGM method provides for user-specified constraints on maximum thickness-to-chord-ratio and trailing-edge thickness. The method also permits non-zero changes to the leading-edge normal coordinate, thus allowing for leading-edge camber changes, as well as relative angle-of-attack changes with respect to the original value input to the design procedure.

The MGM design method was recently used to develop FPE aerodynamic design procedures for three-dimensional wings by Silva and Sankar [28], for three-dimensional helicopter rotor blades by Tapia, Sankar and Schrage [29], and for three-dimensional blended wing-body configurations by Hazarika and Sankar[30]. Since the CFD analysis procedure used in each case was kept as a separate computational module, higher order CFD capabilities could be substituted in a straight-forward manner[30].

Malone, Narramore and Sankar [31] coupled the MGM design procedure with an existing two-dimensional RANS airfoil analysis method to produce an airfoil-design computational method which could account for strong viscous flow effects. Three transonic airfoil design cases were presented in reference 31. Converged airfoil designs were obtained for a computational effort of from between one to four times the computational effort required for a single direct analysis. One airfoil design case demonstrated that a useful airfoil shape could be designed that corresponded to a surface-pressure target distribution for which an alternate, FPE-based airfoil-design procedure, had previously proven to be numerically non-convergent. This example case provides some justification for the judicious use of higher-order CFD design methods, when required, to achieve certain design objectives which may be untractable for lower-order methods. Reference 31 also described the details of a knowledge-based technique, originally developed by Narramore and Yeary [32], which can be used to generate realistic target pressure distributions with required aerodynamic force, moment and drag coefficients.

Birckelbaw [33] also coupled the MGM design procedure to an existing two-dimensional RANS method. This application of the MGM algorithm included two new features to the design method. First, a grid-shearing procedure was added to permit the cost-effective re-generation of new grids throughout the airfoil design computation. Second, a Bezier curve fitting routine was implemented to insure that airfoil shapes with smooth slopes and curvature were generated throughout the process. Two sample airfoil-design cases were described in reference 33. Converged design results for these cases were obtained with a computational effort equivalent to between twelve and eighteen complete direct analyses with the RANS code.

Finally, Malone and Swanson [34] demonstrated that the MGM algorithm, like other decoupled design techniques, are relatively easy to implement as different CFD methods become available. In this case, the transfer of the MGM module used in Reference 31 into a new multigrid two-dimensional RANS solver took less than eight hours of design and coding effort. For the three airfoil design cases presented in reference 34, partially-converged direct-analysis results were used to compute pressure residuals. Fully converged designs were obtained after one-hundred and sixty design cycles, using five-cycle multigrid iterations for the direct-solver each design cycle. This corresponds to an equivalent computational effort of from between eight and ten direct analyses with the RANS CFD code to obtain a converged design result. This research effort indicates that future study of the MGM algorithm is needed relative to the multiplicative constants used in the auxiliary geometry-correction equation. Currently, these constants must be supplied by the user. Although the method has been shown to be robust in the applications studied to date (in that the same values of coefficients seem to provide stable computations when used with different direct-solvers), nevertheless, more computational efficiency is possible if an automatic selection criteria and rescaling algorithm could be developed for these coefficient terms.

Direct Iterative Surface Curvature Method

Campbell and Smith [10] have developed a versatile computational design method, referred to as the Direct Iterative Surface Curvature (DISC) method. Their procedure is an extension of the streamline curvature method of Barger and Brooks [35], in which the pressure residuals along an airfoil surface are assumed to be proportional to an incremental change in airfoil surface curvature. By assuming small changes in values of surface slope, the curvature distributions are written in terms of the second derivative of airfoil ordinates with respect to the streamwise coordinate. At each design cycle, a new airfoil shape is calculated by spatially integrating the incremental ordinate derivatives in a streamwise direction from the leading edge to the trailing edge [36]. Smoothing of the resulting airfoil shape is performed at each design cycle using several piecewise polynomial expressions, evaluated by performing a least-squares fit to the computed airfoil ordinates.

In reference 37, Campbell describes a constrained DISC method, referred to as CDISC. Other features of the CDISC design method [37] include: shearing of the CFD grid between design cycles to insure adequately clustered, body-fitted computational grids; knowledge-based generation of initial target-pressure distributions; automatic modification of target pressures during the design process to meet flowfield or geometric constraints; successive constraint release for problems which are over constrained; and imposition of geometric constraints on airfoil thickness or surface boundary limits (glove option) at specified chordwise locations

The DISC method has been coupled with both EE and RANS CFD analysis procedures to solve two-dimensional and three-dimensional aircraft design problems [37-41]. In reference [37], a structured-mesh, two-dimensional Euler solver was used. Weak interaction viscous effects were simulated by adding a calculated boundary-layer displacement thickness distribution to the airfoil design ordinates.

Chen [38] used the DISC method to design isolated and installed nacelles using a three-dimensional multiblock, multigrid Euler solver. The multiblock approach was used to permit aerodynamic modeling of a nearly complete transport aircraft configuration, including fuselage, wing, pylon, and nacelle. To avoid recomputing the computational mesh during the design process, a surface-transpiration technique was used to simulate the effect surface geometry changes would have on the flowfield. For nacelle geometries, surface ordinate smoothing was performed, using least-squares fit polynomials in both the nacelle circumferential and streamwise directions. In one case out of the three presented by Chen, the design objectives were not totally achieved because of certain aspects of the target pressure distributions used for the design. However, for the three design problems discussed, the DISC/Euler method produced improved designs (relative to the initial geometries) for about the same cost as an analysis of the same configuration.

Other examples of the use of the DISC method with higher-order CFD methods can be found in Potsdam, et. al.[39], Campbell [40], Naik, et. al.[41] and Bell and Cedar[42]. In Reference 39, two different three-dimensional, unstructured mesh Euler solvers were coupled with the DISC method and used to study a design problem for the leading-edge fillet region of an installed nacelle. In Reference 40, the CDISC design method was used with both a two-dimensional and a three-dimensional thin-layer Navier Stokes code. Each code used different turbulence models, but utilized similar numerical solution procedures (i.e. Runge-Kutta integration with multigrid cycles). An interesting feature of this paper is that although numerous results are presented for improvements in aerodynamic force coefficients, no surface pressure distributions are reported, since the emphasis of the work is on meeting flow and geometry constraints. Computer resources required for the design work were reported to be only 1.5 times the cost of an aerodynamic analysis.

Naik et al.[41] report the coupling of CDISC with two different three-dimensional multiblock RANS codes and applied the methods to nacelle design problems. Different approaches for modeling complex aircraft configurations, non-overlapping point-matched grid blocks versus overset grid blocks, were used for the two design methods. Finally, Bell and Cedar [42] describe a three-dimensional design method for aircraft nacelles derived from coupling the DISC inverse method to a three-dimensional RANS code. To account for circumferential effects, the DISC method is used to design three different two-dimensional slices, at the crown, side and keel of the nacelle. Then parabolic interpolation is performed using the incremental changes in the radial coordinates at these three locations, in order to obtain nacelle cross-sectional shapes at other intermediate circumferential locations. The authors report that the nacelle design procedure requires computer resources equivalent to four analysis calculations.

Dirichlet-Type Decoupled-Solution Method

Leonard and Van den Braembussche [43] presented an inverse design method for two-dimensional compressor and turbine blades. In their method, a time-marching, finite-volume formulation of the Euler equations is solved using a Runge-Kutta algorithm. The boundary conditions are formulated to permit the imposition of a prescribed Mach number distribution along the blade surface. The blade surface is treated as a permeable surface, allowing for the generation of a non-zero normal velocity at the boundary surface. Geometry coordinates are updated using the transpiration method.

The authors presented three design cases for compressor and turbine blade configurations, one of which required the redesign of an initial geometry with a strong shock wave. Converged results are reported with only a few geometry updates being required (only two iterations in one case requiring a redesign of the trailing-edge pressures). The authors also report off-design calculations which showed the reappearance of a shockwave for a "shock-free" blade design.

Hybrid Inverse/Optimization Methods

Although inverse design methods can be quite computationally efficient, one consequence of using them is the relative difficulty of incorporating completely arbitrary geometric and flowfield constraints into the design process. As a result, optimization methods are often used instead of inverse methods when constraints must be applied, but can also be considerably more expensive to use than the unconstrained inverse methods. The two design procedures described below attempt to create a design tool which incorporates relative strengths of each type of procedure, inverse and optimization, into one integrated approach.

Coupled-Solution Method

Drela [24] describes the use of the ISES code as part an optimization procedure to minimize global aerodynamic quantities such as airfoil drag. In this optimization scheme, the Newton inverse iteration method can be used to efficiently compute aerodynamic sensitivities (i.e. gradient directions) required as input to the external optimization loop. Optimization algorithms such as the steepest descent and conjugate direction methods are used in the procedure.

The optimization procedure makes use of the mode-inverse option, where the modal participation coefficients are solved for in the external optimization loop driven by the cost-function, for example, the airfoil drag. The computed results of this optimization loop generate pressure and nodal locations, which, in turn, can be used in the original Newton iteration scheme to compute an airfoil geometry. The author presented several example optimization design cases for airfoils and cascade flows. In these cases, approximately five optimization cycles with three Newton iterations between each optimization cycle were sufficient to achieve significantly improved, single-point designs. Cost functions used included airfoil drag and cascade total pressure loss. The author also discusses briefly the importance of investigating off-design conditions which can be worsened, not improved by decisions made in arriving at a single-point design configuration.

Decoupled-Solution Method

Santos and Sankar [44] reported the development of a hybrid inverse-design / numerical optimization procedure. The MGM inverse-design procedure was used to provide a cost-effective computational framework for the design method, and a gradient-directed optimization technique was used to enforce geometric constraints on the converged airfoil design shape. At a given point in the design process, the MGM geometry-correction equation is solved with finite differences and results in a tridiagonal set of linear algebraic equations. The solution of these equations gives the geometric shape corrections corresponding to the magnitude of the residual between the target and computed pressure distributions. Instead of solving this system of equations using conventional linear-equation techniques, such as the Thomas algorithm[45], the authors reformulated the auxiliary equations as a minimization problem, and then used the Fletcher-Reeves [46] conjugate-direction method to solve this equation set. A penalty-function approach is used to enforce thickness constraints at user specified locations, as well as a maximum thickness constraint. The authors demonstrated their hybrid design procedure using a three-dimensional FPE code for wing design and a two-dimensional RANS code for viscous-flow airfoil design. Numerical results indicate that the method can be used to effectively impose geometric constraints on the airfoils and wings during the design process. In both cases, the numerical results presented were obtained on workstations, rather than mainframe computers.

Concluding Remarks

Several inverse methods, suitable for use with higher-order CFD methods, have been described which may be used for the aerodynamic design of two-dimensional and three-dimensional aerodynamic configurations. Where possible, a description has been given of any unique features of the methods, the computational effort required to use the methods, the range of computational problems to which they have currently been applied, as well as the practical limitations on their application as identified within the references. These design methods have yet to reach the maturity levels already attained by methods based on linear aerodynamic theories or full-potential formulations. Except for the DISC method, the computational effort required to use the inverse procedures reviewed here is still significantly more than required for a single analysis effort. However, all of the methods discussed here seem to provide the aerodynamic designer with automated tools which would be far more computationally efficient that using a "cut-

22. Giles, M.B. and Drela, M., "Two-Dimensional Transonic Aerodynamic Design Method," AIAA Journal, Vol. 25, No. 9, September 1987, pp. 1199-1206.

23. Drela, M., and Giles, M.B., "Vicous-Inviscid Analysis of Transonic and Low Reynolds Number Airfoils," AIAA Journal, Vol. 25, No. 10, October 1987, pp. 1347-1355.

24. Drela, M., "Viscous and Inviscid Inverse Schemes Using Newton's Method," Special Course on Inverse Methods for Airfoil Design for Aeronautical and Turbomachinery Applications, AGARD Report No. 780, 1990.

25. Takanashi, S., "An Iterative Procedure for Three-Dimensional Transonic Wing Design by the Integral Equation Method," AIAA Paper No. 84-2155, August 1984.

26. Hirose, N., Takanashi, S. and Kawai, N., "Transonic Airfoil Design Based on Navier-Stokes Equation to Attain Arbitrarily Specified Pressure Distribution - an Iterative Procedure, " AIAA Paper No. 85-1592, July 1985.

27. Bartelheimer, W., "An Improved Integral Equation Method for the Design of Transonic Airfoils and Wings," AIAA CP 95-1688.

28. Silva, D.H. and Sankar, L.N., "An Inverse Method for the Design of Transonic Wings," AIAA Paper No. 92-1025, February 1992.

29. Tapia, F., Sankar, N.L., and Schrage, D.P., "An Inverse Aerodynamic Design Method for Rotor Blades in Forward Flight," 51st AHS Forum, May 9-11, 1995.

30. Hazarika, N., "An Efficient Inverse Method for the Design of Blended Wing-Body Configurations," Ph.D. Thesis, Georgia Institute of Technology, June 1988.

31. Malone, J.B., Narramore, J.C. and Sankar, L.N., "An Efficient Airfoil Design Method Using the Navier-Stokes Equations," AGARD CP 463, Paper No. 4, May 1989

32. Narramore, J.C. and Yeary, R.D., "Airfoil Design and Analysis Using an Information Systems Approach," AIAA Paper No. 80-1444, 1980.

33. Birckelbaw, L., "Inverse Airfoil Design Using the Navier-Stokes Equations," AIAA Paper No. 89-2202, July 1989.

34. Malone, J.B. and Swanson, R.C., "Inverse Airfoil Design Procedure Using a Multigrid Navier-Stokes Method," Proceedings of ICIDES-III, October, 1991, pp. 55-66.

35. Barger, R.L. and Brooks, C.W., "A Streamline Curvature Method for Design of Supercritical and Subcritical Airfoils," NASA TN D-7770, September 1974.

36. Smith, L.A. and Campbell, R.L., "A Method for the Design of Transonic Flexible Wings," NASA TP 3045, December 1990.

37. Campbell, R.L., "An Approach to Constrained Aerodynamic Design with Application to Airfoils," NASA TP 3260, November 1992.

38. Chen, H.C., "An Installed Nacelle Design Method Using Multiblock Euler Solver," AIAA Paper No. 93-0528, January 1993.

39. Potsdam, M.A., Intemann, G.A., Frink, N.T., Campbell, R.L., Smith, L.A., and Pirzadeh, S., "Wing/Pylon Fillet Design Using Unstructured Mesh Euler Solvers," AIAA Paper No. 93-3500, August 1993.

40. Campbell, R.L., "Efficient Constrained Design Using Navier-Stokes Codes," AIAA Paper No. 95-1808, June 1995.

41. Naik, D., Krist, S., Campbell, R.L., Vatsa, V., Buning, P. and Gea, L., "Inverse Design of Nacelles Using Multi-Block Navier Stokes Codes," AIAA Paper No. 95-1820, June 1995.

42. Bell, R.A. and Cedar, R.D., "An Inverse Method for the Aerodynamic Design of Three-Dimensional Aircraft Engine Nacelles," in Proceedings of 3rd International Conference on Inverse Design Concepts and Optimization in Engineering Sciences, ICIDES III, Washington, D.C., 1991, pp. 405-417.

43. Leonard, O. and Van den Braembussche, R.A., "Blade Design and Analysis Using a Modified Euler Solver," in Proceedings of 3rd International Conference on Inverse Design Concepts and Optimization in Engineering Sciences, ICIDES III, Washington, D.C., 1991, pp. 325-336.

44. Santos, L.C.C. and Sankar, L.N., "A Hybrid Inverse Optimization Method for Aerodynamic Design of Lifting Surfaces," AIAA-94-1895-CP, pp. 672-685.

45. Anderson, D.A., Tannehill, J.C. and Pletcher, R.H., Computational Fluid Mechanics and Heat Transfer, McGraw-Hill, 1984.

46. Vanderplaats, G.N., Numerical Optimization Techniques for Engineering Design with Applications, McGraw-Hill, 1984.

and-try" approach. In fact, many of the aerodynamic design examples given in these references demonstrate the future potential benefits to the design engineer such automated methods will bring, and point to the need for continued further research in this area.

Acknowledgement

The author would like to acknowledge several experts in the area of computational aerodynamic design methodology who kindly provided copies of their research publications and other materials which were of great assistance in producing this paper. Special thanks are given to George S. Dulikravich of the Penn State University, Mark Drela of the Massachuetts Institute of Technology, Manuel D. Salas of the Institute for Computer Applications in Science and Engineering (ICASE) at NASA Langley Research Center, and Richard L. Campbell, also of NASA Langley Research Center.

References

1. Raj, P., "Requirements for Effective Use of CFD in Aerospace Design," NASA CP 3291, pp. 15-28.

2. Woodward, F.A. and Tinoco, E.N., "Analysis and Design of Supersonic Wing-Body Combinations, Including Flow Properties in the Near Field," NASA CR-73106, 1967.

3. Bristow, D.R., "A New Surface Singularity Method for Multi-Element Airfoil Analysis and Design," AIAA Paper 76-20, January 1976.

4. Fray, J.M.J. and Slooff, J.W., "A Constrained Inverse Method for the Aerodynamic Design of Thick Wings with Given Pressure Distribution in Subsonic Flow," AGARD CP No. 285, Paper 16, 1980.

5. Malone, J.B., "An Optimal-Surface-Transpiration Subsonic Panel-Method for Iterative Design of Complex Aircraft Configurations," AIAA Paper No. 81-1254, 1981.

6. Tranen, T.L., "A Rapid Computer Aided Transonic Airfoil Design Method," AIAA Paper No. 74-501, June 1974.

7. Shankar, V. and Malmuth, N.D., "Computational Transonic Design Procedure for Three-Dimensional Wings and Wing-Body Combinations," AIAA Paper No. 79-0344, January 1979.

8. Garabedian, P. and McFadden, G., "Design of Supercritical Swept Wings," AIAA Journal, Vol. 20, No.3, 1982, pp. 289-291.

9. Malone, J.B., Vadyak, J. and Sankar, L.N., "A Technique for the Inverse Aerodynamic Design of Nacelles and Wing Configurations," AIAA Paper No. 85-4096, October 1985.

10. Campbell, R.L. and Smith, L.A., "A Hybrid Algorithm for Transonic Airfoil and Wing Design," AIAA Paper No. 87-2552, August 1987.

11. Slooff, J.W., Computational Methods for Subsonic and Transonic Aerodynamic Design, AGARD Report 712.

12. Dulikravich, G.S., "Aerodynamic Shape Design and Optimization: Status and Trends," Journal of Aircraft, Vol. 29, No. 6, 1992, pp. 1020-1026.

13. Labrujere, Th.E., Slooff, J.W., "Computational Methods for Aerodynamic Design of Aircraft Components," NLR TP 92072 U, 1992.

14. Labrujere, Th.E., Slooff, J.W., "Computational Methods for Aerodynamic Design of Aircraft Components," Annual Review of Fluid Mechanics, Vol. 25, 1993, pp. 183-214.

15. Dulikravich, G.S., "Shape Inverse Design and Optimization for Three-Dimensional Aerodynamics," AIAA Paper No. 95-0695, January 1995.

16. AGARD, Computational Methods for Aerodynamic Design (Inverse) and Optimization, Leon, Norway, 1989, AGARD CP-463.

17. Dulikravich, G.S., ed., Proceedings of 3rd International Conference on Inverse Design Concepts and Optimization in Engineering Sciences, ICIDES III, Washington, D.C., 1991.

18. AGARD, Optimum Design Methods for Aerodynamics, Rhode-Saint-Genese, Belgium, 1991, AGARD-R-803.

19. AIAA, Proceedings of the 5th AIAA/NASA/USAF/ISSMO Symposium on Multidisciplinary Analysis and Optimization, Panama City, Florida, 1994, CP 9413.

20. AIAA, Proceedings of the12th AIAA Computational Fluid Dynamics Conference, San Diego, California, June 19-22, 1995.

21. Volpe, G., "Inverse Design of Airfoil Contours: Constraints, Numerical Method, and Applications," Paper No. 4, AGARD-CP-463.

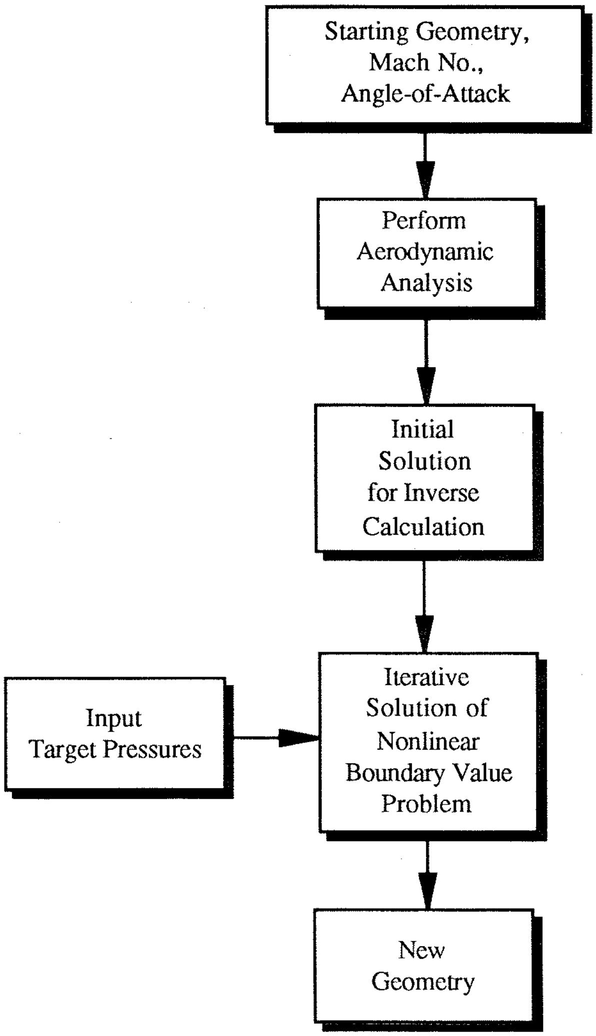

FIGURE 1 - Flow Diagram for a Typical Coupled-Solution Inverse Aerodynamic Design Method

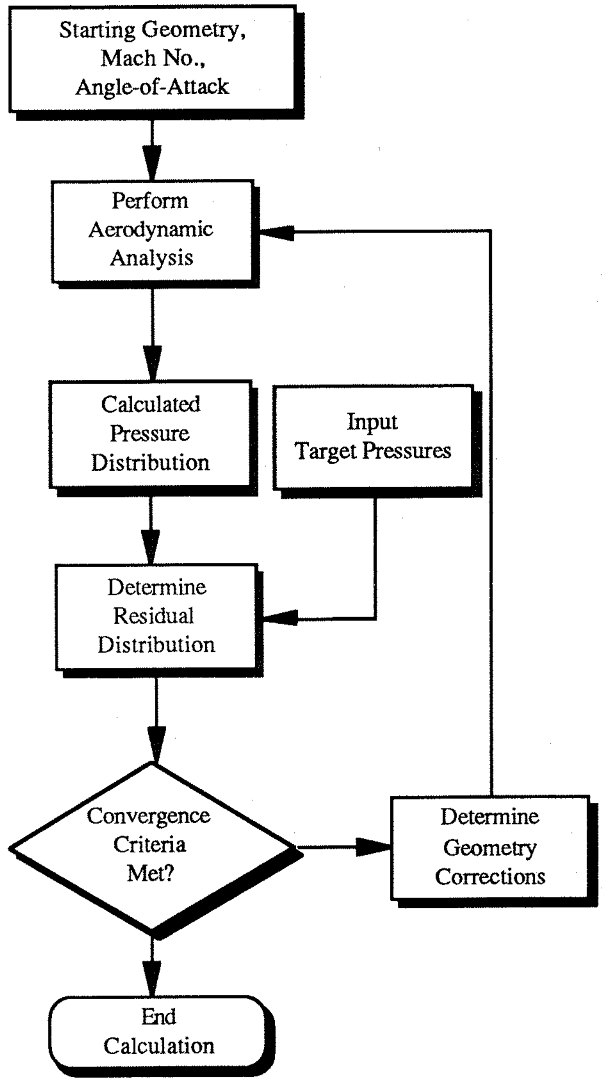

FIGURE 2 - Flow Diagram for a Typical Neumann-Type Decoupled-Solution Inverse Aerodynamic Design Method