

N 87-26011

COMPUTATIONAL FLUID DYNAMICS RESEARCH AT THE
UNITED TECHNOLOGIES RESEARCH CENTER REQUIRING SUPERCOMPUTERSAnton J. Landgrebe
Manager, Aeromechanics ResearchUnited Technologies Research Center
East Hartford, CT

ABSTRACT

An overview of research activities at the United Technologies Research Center (UTRC) in the area of Computational Fluid Dynamics (CFD) is presented. The requirement and use of various levels of computers, including supercomputers, for the CFD activities is described. Examples of CFD directed toward applications to helicopters, turbomachinery, heat exchangers, and the National Aerospace Plane are included.

Helicopter rotor codes for the prediction of rotor and fuselage flow fields and airloads have been developed with emphasis on rotor wake modeling. Airflow and airload predictions and comparisons with experimental data are presented. Work is currently underway at UTRC for the development and application of CFD codes related to the analysis of the inlet, scramjet combustor, and nozzle flow fields of the National Aerospace Plane. Examples are presented of recent parabolized Navier-Stokes and full Navier-Stokes solutions for hypersonic shock-wave/boundary layer interaction, and hydrogen/air supersonic combustion. In addition, other examples of CFD efforts at UTRC in turbomachinery Navier-Stokes methodology and separated flow modeling are presented.

A brief discussion of the 3-tier scientific computing environment at UTRC is also presented, in which the researcher has access to workstations, mid-size computers, and supercomputers. The recent UTRC link to the Numerical Aerodynamic Simulation (NAS) supercomputer system at the NASA Ames Research Center and a helicopter code demonstration on this supercomputer are described. A new activity to evaluate massive parallel processing supercomputers for CFD applications, using CFD codes for fluid flow model problems, is also described.

INTRODUCTION

The state-of-the-art of CFD and computer technology have simultaneously reached advanced levels such that new capabilities are providing unique opportunities for the CFD researcher. CFD problems that were impossible to consider just two years ago are now feasible, and as a result, progress is now accelerating rapidly in the CFD field. NASA has played a key role in leading the way for both CFD technology and the adaptation and provision of supercomputers for CFD research. UTRC is interacting with NASA, universities, computer organizations, and the Divisions of the United Technologies Corporation (UTC) (such as Pratt & Whitney, Sikorsky Aircraft, Hamilton Standard, and

Carrier) to adapt, develop, refine and evaluate CFD codes for application to gas turbines, helicopters, propellers, and heat exchangers. A wide range of CFD problems are being investigated which include the various levels of aerodynamics methodology (lifting line/surface, full-potential, Euler, and Navier-Stokes methods). Examples of UTRC CFD activities related to the forementioned variety of applications and methodology levels are presented herein.

Computer access and capability level have rapidly advanced at UTRC in the last year. UTRC's computing system is based on the 3-tier computer environment shown in figure 1. This consists of workstations

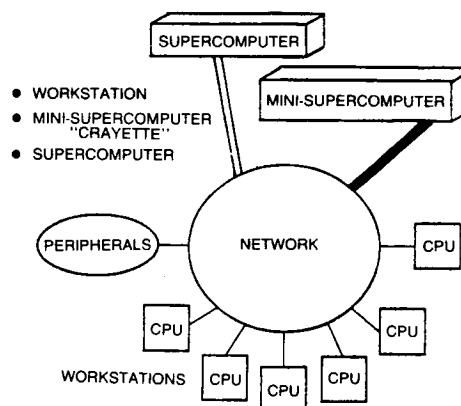


Figure 1. UTRC 3-Tier Computer Network

with built-in or accessible processors of limited power, mini-supercomputers ("Crayettes") which typically have one-tenth to one-third the computing power of the supercomputers, and supercomputers of varying types and capabilities. The computers have been specifically selected for performing CFD research. They cover the complete range of computer categories (scalar/vector and serial/parallel/massive parallel). Access is provided to the research engineers through an internal Ethernet network system. Many of the computers are off-site and are accessed through a variety of network systems which include high speed, long-haul communications networks. On-site computers include VAX and Apollo workstations, Perkin-Elmer computers, and an Alliant mini-supercomputer. Off-site computers are listed below. Of these, the UTC supercomputer is the CRAY X-MP, operational in 1987, and located at Pratt & Whitney Government Products Division in West Palm Beach, Florida. It is remotely linked to the local network systems at UTRC and the UTC Divisions. Several off-site computers

are being used to evaluate the use of massive parallel processors for CFD. In particular, a unique application of cellular automata techniques will be discussed in the last section of this paper.

Off-site computers

• UTC - West Palm Beach	Cray X-MP
• NASA-Ames	Cray - 2 (NAS)
• NASA-Lewis	Cray X-MP
• NASA-Langley	Cyber 205
• Cray Research	Cray X-MP
• Kirtland Air Force	Cray X-MP
• Air Force Weapons Lab	Cray - 1
• Los Alamos Nat'l. Lab	Cray X-MP
• IBM Kingston	LCAP
• Thinking Machine Corp	Connection Machine

UTRC was selected in 1986 as a remote site user/demonstrator of the new NASA Numerical Aerodynamic Simulation (NAS) supercomputer system. NAS is the computational facility at the NASA Ames Research Center that currently includes the CRAY-2 high-speed processor with an approximately 250 megaflops speed/256 megaword central memory capability. A high-speed network system linking the NAS supercomputer located in California, to UTRC in East Hartford, Connecticut, was made operational in 1986. Initial demonstration cases were successfully run over the 56 kb/sec link from a VAX workstation, as shown in figure 2. The initial demonstration cases consisted of computations of the influence of blade sweep on the airload distribution and sonic delocalization for a representative helicopter blade using a state-of-the-art 3-D aerodynamics analysis developed jointly by NASA and UTRC. This activity, endorsed by NASA and The American Helicopter Society, is serving to demonstrate the remote use of NAS for helicopter CFD codes. A description and results of this NAS/helicopter CFD activity will be presented herein. It is planned to continue this NAS activity at UTRC with the following general objectives: (1) to continue the use of NAS to develop, demonstrate, and evaluate advanced rotary-wing CFD codes for the prediction of helicopter/propeller airloads and performance, (2) to interact with NASA researchers on the use of NAS to adapt and integrate flow solver, wake, and rotary-wing CFD methodology, and (3) continue to perform NAS remote-site demonstrations of helicopter CFD codes for the helicopter industry.

Remote site (UTRC)/ NAS communications network

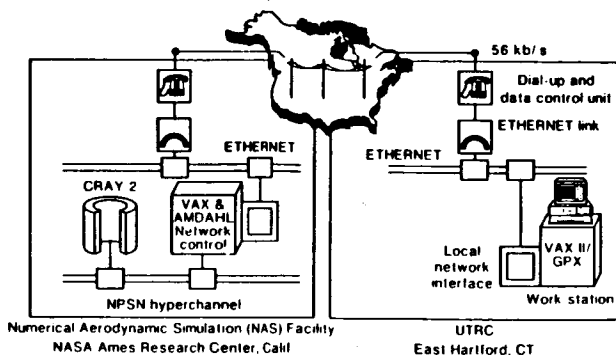


Figure 2. UTRC Remote-Site Link to NAS Supercomputer at NASA Ames Research Center

In addition to helicopter/propeller applications, the use of NAS for the development and evaluation of CFD codes related to the National Aerospace Plane (NASP) has been separately proposed. Examples of these codes and initial results are also presented herein.

The following are the major subject technology areas of this paper and the UTRC technical contributors to each technology area are indicated accordingly: Helicopter CFD (T. A. Egolf, P. F. Lorber, S. P. Sparks, and A. J. Landgrebe), CFD for NASP, Turbomachinery and Separated Flow Modeling (T. J. Barber, J. E. Carter, R. L. Davis, and D. E. Edwards), and Parallel Processing for CFD (A. F. Hought, and E. B. Smith). In addition to UTC Corporate sponsorship, NASA, Army, Navy, and Air Force contract support has contributed to several of the activities reported herein.

HELICOPTER CFD

The task of predicting the flow field and airloads of a helicopter rotor continues to be of primary importance for providing and evaluating improved rotor designs. Also, the capability to predict rotor induced flow velocities away from the main rotor is important for the calculation of aerodynamic interference effects at the fuselage, tail rotor, and tail surfaces. Technological advancement of rotor induced velocity and airloads methodology has focused on the analytical modeling of the rotor wake and the application of higher level computational aerodynamics techniques to represent the blades and their influence on the airflow. This has become possible with the advent of high-speed, large memory computers.

As part of a long-term effort to advance the aerodynamic technology of helicopter rotors, studies have been conducted at UTRC to develop methodology for predicting the rotor wake, flow field and blade airloads. Publications that document these studies, which span a period of twenty years, are listed in the author's overview of helicopter wake and airloads technology (Landgrebe (1985, 1986)). The scope of this effort has included analytical and experimental research programs, distorted and undistorted wake analyses, single and dual rotor configurations, hover and forward flight conditions, and blade airloads methods ranging from lifting line to full potential. Aerodynamic methods pertaining to these areas, developed or adapted at UTRC are presented herein. Examples of related methodology, developed by other researchers, are indicated within the references cited in the above overview paper and the references of McCroskey and Baeder (1985) and Davis and Chang (1986).

Rotorcraft Wake Analysis

Several methods have been consolidated and expanded at UTRC to form the Rotorcraft Wake Analysis, a comprehensive helicopter aerodynamic analysis to predict induced flow velocities both at and off the rotor (Landgrebe and Egolf (1976)). This family of component computerized analyses is normally used in conjunction with rotor performance and airloads analyses. A schematic of the Rotorcraft Wake Analysis showing the input, output, and capabilities of the computer program is presented in figure 3.

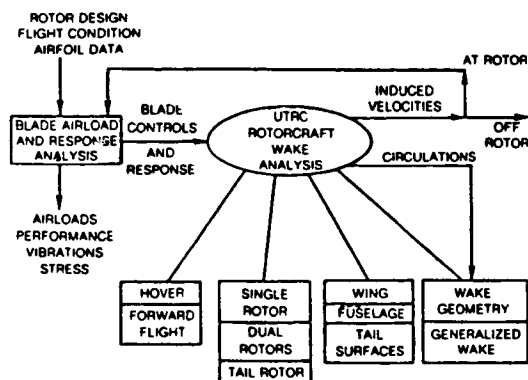


Figure 3. Schematic of Rotorcraft Wake Analysis Showing Input/Output and Capabilities of Component Codes

Briefly, the fundamental technical approach for the Rotorcraft Wake Analysis consists of the representation of each blade by a segmented lifting line, and the wake of the rotor by discrete segmented vortex filaments consisting of trailing vorticity, which result from the spanwise variation of blade bound circulation. The distribution of blade and wake circulation changes with azimuth position and is assumed periodic for each rotor revolution. The blades are divided into a finite number of radial segments, and the induced velocity at the center of each selected blade segment is computed by summing the contributions of all bound and trailing wake segments. The contribution of each vortex segment is obtained through use of the Biot-Savart law, which expresses the induced velocity in terms of the circulation strength of the vortex segment and its geometric position relative to the blade segment at which the induced velocity is desired. The blade bound circulation distribution is determined at each blade azimuth by expressing the wake circulations and induced velocities in terms of the unknown bound vortex strengths by means of the Biot-Savart law, and developing a set of simultaneous equations relating the bound circulation and local blade angle of attack at each blade segment. These equations thus involve the known flight condition, wake geometry, two-dimensional airfoil data, blade motion and control parameters, and the unknown blade bound circulation values. Solution of these equations yields the desired blade bound circulation values, which, when combined with the appropriate geometric relations in the Biot-Savart law, produce the required induced velocities at or away from the rotor blades. The influence of realistic airfoil data and blade motions and controls (e.g., for rotor trim) can be included through an iterative coupling with a blade airloads and performance program. The influence of the shed vorticity behind a blade, due to the time variation of blade bound circulation, can be included by the use of unsteady two-dimensional airfoil data in the airloads program.

Generality regarding the specification of the rotor wake geometry was retained in the computer programs by requiring only that the coordinates of the wake segment end points be stored with various options for obtaining the coordinates. Thus rotor wake geometry models ranging from the classical undeformed wake (figure 4) to the fully distorted

wake (figure 5) can be used. A computerized method for predicting the rotor distorted wake geometry is the UTRC Wake Geometry Analysis (Landgrebe (1969)). A generalized wake model has been made available which is based on generalized wake equations for hover (Landgrebe (1972)) and forward flight (Egolf and Landgrebe (1984)). A sample isometric view of the generalized wake model for forward flight is shown in figure 6.

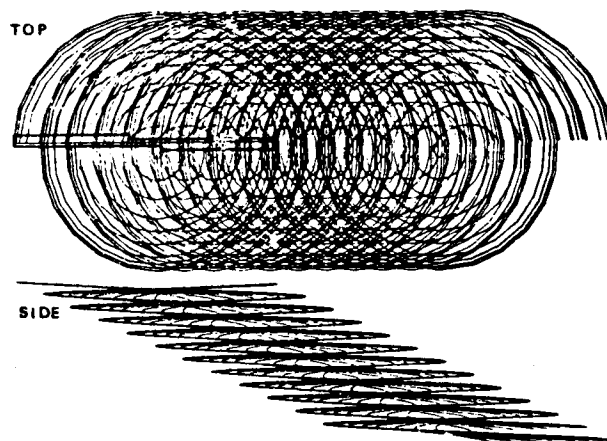


Figure 4. Classical Undeformed Wake Representation for a 30 Kt Forward Flight Condition

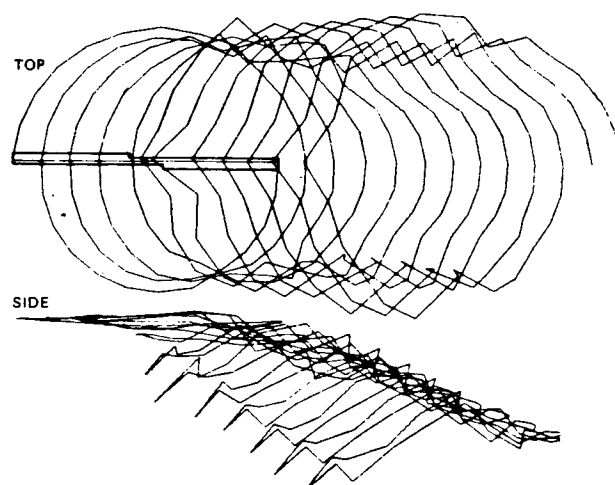


Figure 5. Distorted Tip Vortex Representation for a 30 Kt Forward Flight Condition

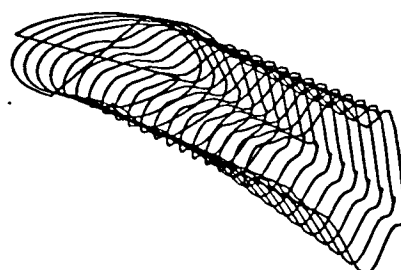


Figure 6. Isometric View of Generalized Distorted Tip Vortices

The ability of the component codes of the Rotorcraft Wake Analysis to predict rotor flow field velocities is shown for hover and forward flight conditions in figures 7 and 8 from Landgrebe and Egolf (1976). The experimental based generalized wake model was used for the hover condition and the predicted distorted wake geometry was used for the forward flight condition. It is noted that, although they are encouraging, these calculations are very sensitive to the wake model and wake geometry and the degree of correlation is not necessarily typical of all flight conditions and rotor blade designs, particularly some advanced tip designs.

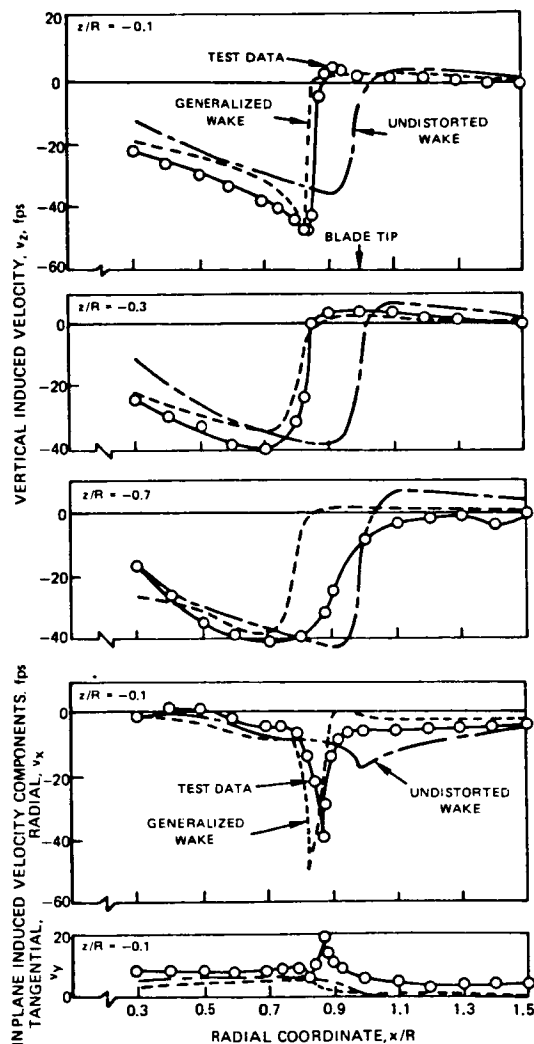


Figure 7. Predicted vs Measured Instantaneous Induced Velocity Components of a Full-Scale Hovering Rotor

In forward flight, the deformation of the forward and lateral sides of the wake toward the rotor disk, shown earlier in figures 5 and 6, can result in close blade-vortex passages which can introduce severe local azimuthal and spanwise gradients in blade airloads which are not predicted with uniform inflow and undistorted wake analytical models. This is shown in figure 9 where the predicted airloads (blade lift distributions) based on the different inflow/wake models are presented for a representative rotor in the form of surface contour plots. As

indicated, inclusion of tip vortex deformation in the wake model results in increased higher harmonic

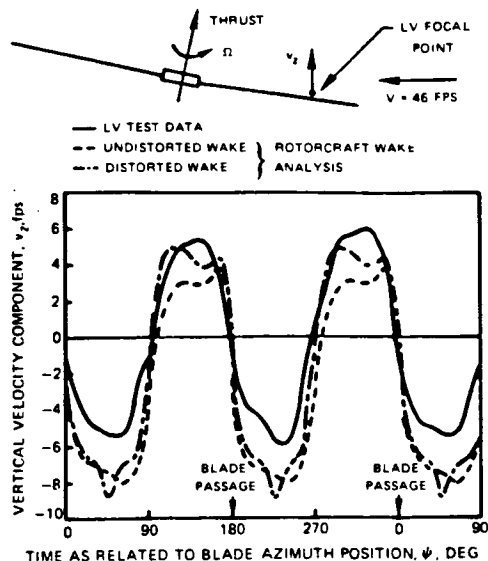


Figure 8. Predicted vs Measured Time Variations of the Flow Velocity at a Point Near a UTRC Model Rotor (Advance Ratio = 0.15)

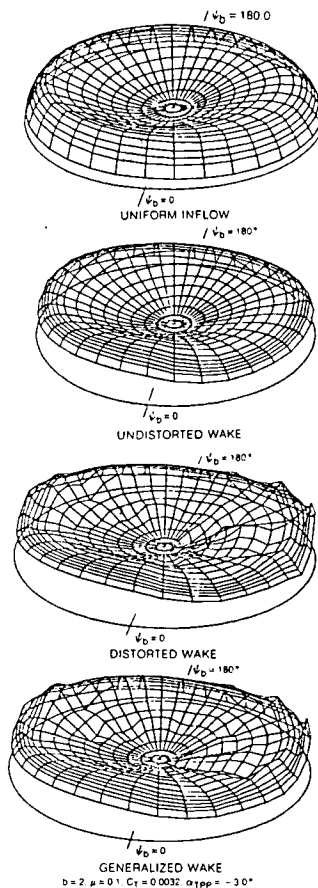


Figure 9. Blade Lift Distributions Predicted With Various Wake/Inflow Models

content in the airload prediction. The outboard advancing side of the rotor typically exhibits the most severe vibratory airload gradients with significant variations also on the outboard retreating side. These impulsive airload characteristics are of importance for helicopter vibration and noise as well as performance.

The requirement for supercomputers for helicopter airloads/wake predictions is well recognized when the computational interaction of thousands of wake elements with each other and with many blade elements at each rotor azimuth are considered. Also, rotor trim normally requires several airload/wake iterations. The computer requirements are further compounded when the aerodynamic interactions of the rotor with other helicopter components are computed (fuselage, tail rotor, tail surfaces etc.). Wake modeling for airload prediction of the various helicopter components is being pursued at UTRC. Earlier UTRC investigations of rotor-fuselage/wing/tail interaction are cited in the previously mentioned overview (Landgrebe (1985, 6)).

The influence of the rotor airflow on fuselage vibratory airloading is currently being considered by P. F. Lorber and T. A. Egolf under a NASA/Army contract. Experimental results have indicated that the rotor blades and wake can produce significant oscillating fuselage pressures at principally the fundamental blade passage frequency. These pressure pulses have been recognized as a fuselage vibration mechanism, but this mechanism and its effect have not been fully investigated analytically. Methodology is currently being developed at UTRC to approximate the rotor wake deformation due to the fuselage and to calculate the unsteady fuselage pressures. Thus far, provisions have been made for prescribed time variant local deformation of the rotor tip vortices around the fuselage surfaces and the calculation of the airflow and airloads on the panels representing the fuselage surfaces. Wake patterns and flow vectors for sample fuselage shapes and a 0.15 advance ratio condition are shown in figure 10.

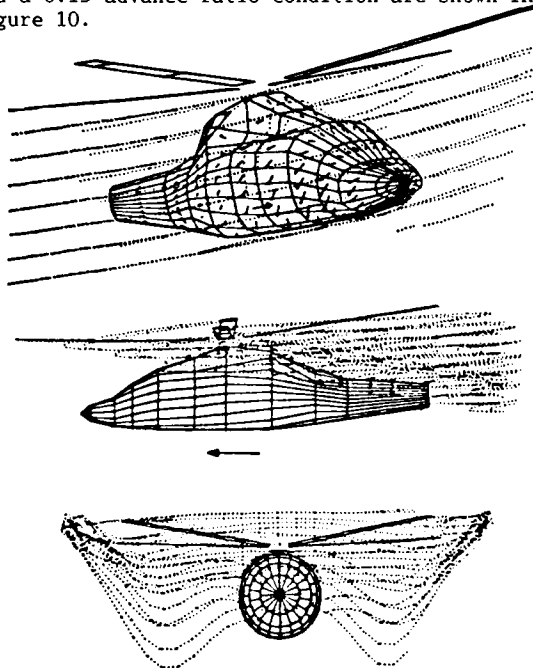


Figure 10. Wake Modeling for Rotor-Fuselage Aerodynamic Interaction

Full Potential Flow/Wake Analysis

The Rotorcraft Wake Analysis results presented earlier are based on a lifting-line representation for the blades. In order to provide a more accurate aerodynamic representation of the blades and more properly include chordwise airloading, blade thickness and compressibility effects, a full potential flow analysis has been extended at UTRC for the inviscid solution of the rotor and wake flow field. A NASA three-dimensional full potential analysis (ROT22), by Arieli and Tauber (1979), has been recently revised, under a NASA contract, at UTRC by Egolf and Sparks (1985, 1986) to include the rotor wake influence. The NASA analysis, based on an infinite computational domain, was revised to incorporate an inner and outer domain around each blade and match the domain boundary solutions in three dimensions. Blade surface flow pressure distributions can be computed for subsonic and transonic flow conditions.

Briefly, the ROT22/WAKE method consists of the use of a finite difference scheme to solve the nonlinear near blade flow (inner domain), while the remainder of the flow field (outer domain) is computed using a discrete vortex representation of the wake. A type of solution matching is employed for the resulting inner and outer domain problem. Although the solution is based on the full potential equation, details of the wake distortion are modeled using prescribed wake methods. This allows for the description of the wake to be as complex (or as simple) as desired while maintaining the relatively low cost full potential method for the detailed blade airloads. The finite inner domain and its computational boundary is shown in figure 11 with the hovering blade wake model represented by vortex filaments. The cut-surface that represents the blade wake in the inner domain for the full potential solution is adapted to the prescribed wake surface such as that determined from the generalized wake geometry described earlier. An embedded vortex technique has recently been developed to accommodate in the full potential solution the blade-vortex interaction for finite length vortex elements near the blade in the inner domain.

The analysis has been applied by Egolf and Sparks (1985) to compare with hover data from model rotors. A comparison of predicted and measured spanwise lift

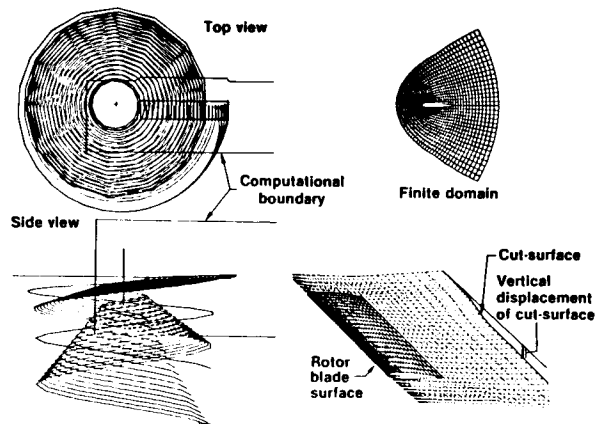


Figure 11. Schematics of Computational Domain Boundary and Wake Representation

distributions are presented for an Army rotor (Caradonna and Tung (1981)) in figure 12. The influence on the chordwise pressure correlation near the blade tip of increasing the tip Mach number to transonic conditions is shown in figure 13. The formation of a shock is evident at this location as the blade progresses from low to high tip Mach number.

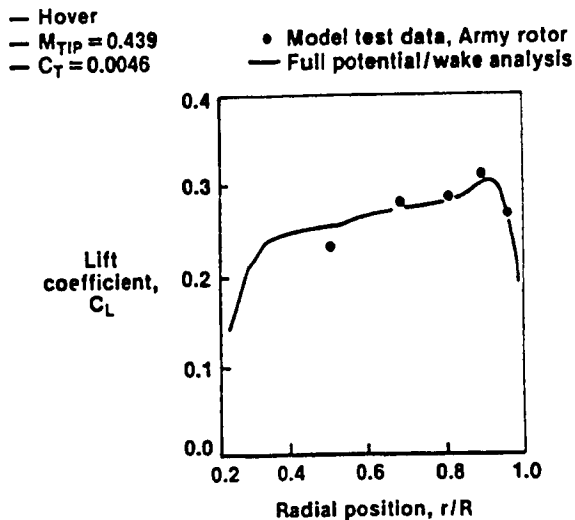


Figure 12. Comparison of Predicted and Measured Spanwise Lift Distribution

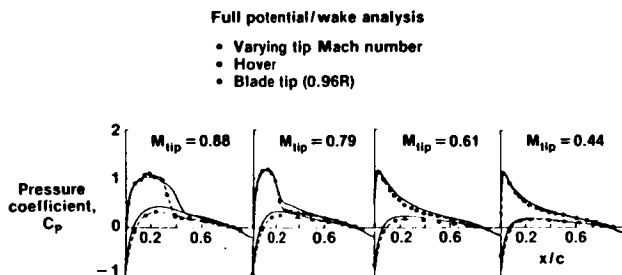


Figure 13. Comparison of Predicted and Measured Chordwise Pressure Distributions Near the Blade Tip for Varying Tip Mach Numbers

This analysis was recently extended and applied to forward flight conditions, and comparisons with test data are presented in Egolf and Sparks (1986). More recently the NAS supercomputer has been accessed remotely from UTRC, as described above, and used by T. A. Egolf to investigate the potential acoustic benefits of varying blade tip design. This is an extension of the work of Tauber (1984) of NASA, for the purpose of including the rotor wake influence. The smaller predicted supersonic zone for a swept tip with tapered thickness relative to that of a rectangular tip is shown in figure 14 in which Mach isobars are presented for a 4-bladed rotor ($B=4$) at a specified pitch angle. The results for the swept blade with only $1/4$ rev of flat wake are included in figure 14 to show that an enlarged supersonic zone is predicted when the influence of the other blades and the more complete generalized wake is included in the computation. Typically, the NAS CRAY-2 computer time requirement is between $1/2$ and 2 hours for predicting blade airloads and Mach isobars at one blade azimuth position. The requirement for a supercomputer for

rotor problems such as this is evident when it is considered that, even after the code is fully vectorized, many hours of NAS CRAY-2 time will be required for solutions over the rotor azimuth range.

The concepts involving the inclusion of the influence of the wake using an inner/outer solution domain are not limited to the quasi-steady full potential formulation. The method can be adapted to time-dependent rotor problems and more sophisticated numerical models such as Euler and Navier-Stokes analyses. This method of solution has the advantage of reduced volume for the nonlinear inner domain solution (finite difference methods) while treating the global influence of the complex rotor wake with a method based on prescribed wake models. The computational cost to obtain the complete solution is thus reduced as compared with an infinite domain solution approach. It is currently planned to apply the technique to other Ames Research Center full-potential codes by Chang (1984) and Strawn and Caradonna (1986) and include viscous airfoil characteristics using Navier-Stokes methodology (McCroskey, Baeder, and Bridgeman (1985)). It is planned to use our remote link to the NAS supercomputer for this activity. In addition to helicopter CFD, propeller CFD is of interest at UTRC to support the Prop-Fan activities of the Pratt & Whitney and Hamilton Standard Divisions of UTC. Various approaches to code development and evaluation are in progress based on 3-D lifting surface (vortex lattice), full-potential, and Euler methodology.

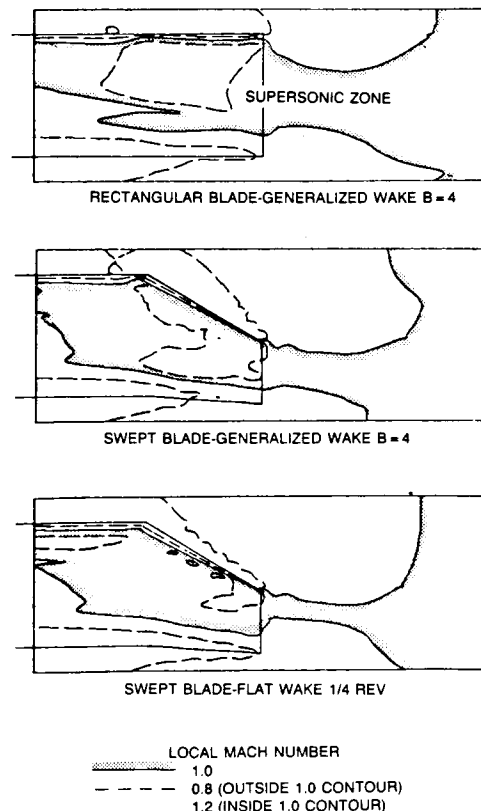


Figure 14. Predictions of Influence of Blade Design and Wake Model on Supersonic Zone

The design of the advanced high speed propulsion systems, such as scramjets and ramjets for a National Aerospace Plane (NASP) type vehicle operating at high Mach numbers and at extreme altitudes, cannot be based on conventional approaches utilizing extensive data bases and expensive experimental test simulations. The development of CFD techniques however has reached a state-of-the-art that will permit a designer to use Parabolized Navier-Stokes (PNS) and Navier-Stokes (NS) methods in the evolution of component designs. The current requirements for CFD codes in several high speed propulsion programs are being limited by the ability of the codes to model the relevant geometric features and physical processes.

A prime example of this problem is found in the analysis of new/novel combustor concepts. In this component, the flow is controlled by the complex three-dimensional interaction of the main stream fluid dynamics with reacting chemistry of the fuel. The flow simulation is significantly influenced by the introduction of multiple length and time scales that affect the stability of the flow as well as the numerical algorithms used in the simulation codes. Further complicating the design process is the lack of any meaningful data base to help reduce the simulation process to more tractable unit problems.

The use of CFD codes in the current NASP program is however predicated on using state-of-the-art codes with only a minimal degree of improvement. All codes related to the hypersonic propulsion problem can be categorized by their level of physical modeling, the completeness of data bases needed in these models, and by the type of numerical algorithm. In any application of these codes to the entire flight regime, the codes will have varying degrees of applicability and accuracy. A modular approach to classifying all codes helps a user to categorize a particular code, identify its capabilities and prioritize future code work.

Calibration of CFD codes for high speed applications is however, hampered by the limited availability of benchmark quality data for practical configurations. It is apparent that the need for modeling viscous and air/fuel chemistry effects over a three-dimensional hypersonic vehicle will require a blend of Parabolized Navier-Stokes (PNS) and full Navier-Stokes (NS) codes. Computer CPU and memory resources can represent a limiting factor forcing an engineer to trade geometric and physical modeling requirements with computer resource availability. Instead, our approach has been to identify unit problems that describe critical physical processes for which benchmark data are known to exist, i.e. include shock boundary layer interaction (Holden plate/ramp cases), flow three dimensionality (P8 inlet), viscous mixing induced combustion, etc. and then validate the codes against these data bases (as performed by T. J. Barber for the following examples).

Shock Wave/Boundary Layer Interaction

One such example is demonstrated in the analysis of a shock wave/boundary layer interaction problem. A good data base is found in the CALSPAN data of Holden (1970). In the CFD validation study shown on figure 15, a laminar $M = 14.1$ flow over a flat

plate/15 degree compression ramp combination is examined. Holden obtained surface static pressure, skin friction (C_f) and heat transfer rate (St) data. The static pressure contour plot shows the complexity of the flow physics in this benchmark case. A leading edge viscous interaction shock interacts with the compression corner shock. This shock/shock interaction controls the entire problem. Computed results from the SCRAMP (Krawczyk (1986)) (SAIC) PNS code and the NASCRIN (Kumar (1986)) (NASA LaRC) NS code are coplotted with the data. The NASCRIN two-dimensional code uses a MacCormack explicit time marching central difference scheme. In the calculations shown on figure 15, at least 15-20,000 time steps were required to reach a steady state. Good agreement is noted in the surface pressure and Stanton number comparisons. The wall skin friction data shows the expected drop into the compression corner. The single pass PNS results, as expected, do not reproduce this flow feature. Furthermore, with the relatively good agreement in the predicted Stanton number values on the ramp surface from the SCRAMP and NASCRIN and the poor ramp skin friction levels, one should question the applicability of Reynolds' analogy at these high Mach numbers.

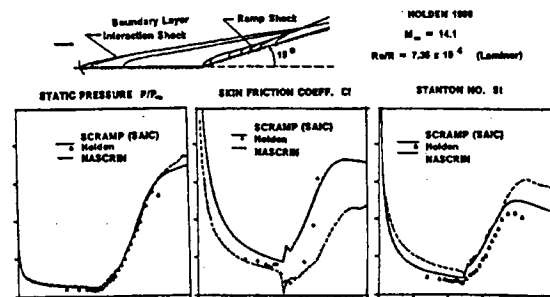
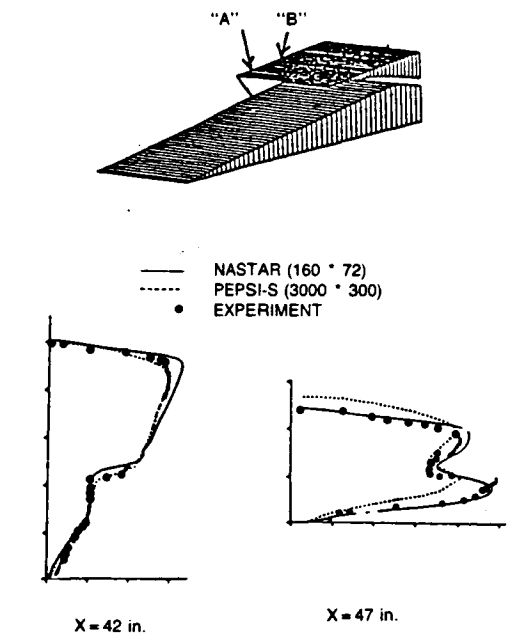


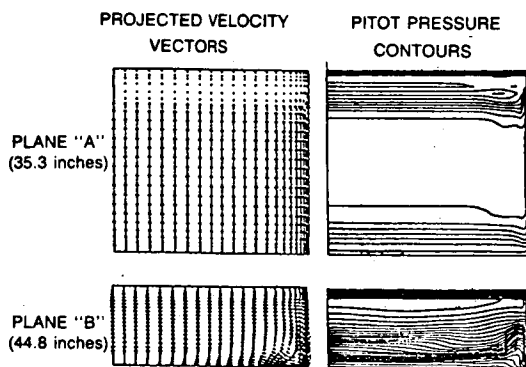
Figure 15. Unit CFD Problem: Shock Wave/Boundary Layer Interaction. (Validation of Viscous Inlet/Forebody Codes)

Flow Three-Dimensionality

Another relevant unit problem to investigate is flow three-dimensionality. The P8 inlet configuration of Seebaugh (1973), tested experimentally in the early 1970's, is a good three-dimensional configuration that does not require real gas modeling to simulate the flow physics. The incident flow Mach number was 7.4. Most of the available test data was measured along a narrow band inlet centerline. Because of flow symmetries, this problem can be initially examined in a 2D calculation. Predictions using the NASTAR (P&W) NS program of Rhie (1986) for pitot total pressure profiles are compared in figure 16(a) to the published results of Ng, Benson, and Kunik (1986) using the PEPSI-S (PNS) program and to experimental data. The PEPSI-S uses a substantially refined mesh (3000*300) to obtain the moderately better agreement at $X = 42$ ", however, its resolution of the reflected shock at $X = 47$ " is not as good as NASTAR's prediction.



(a) Two Dimensional P8 Inlet Calculation vs. Experimental Data



(b) Three-Dimensional P8 Inlet Calculation

Figure 16. Unit CFD Problem: Flow Three-Dimensionality. (Validation of Navier-Stokes Inlet Codes)

In the calculations shown in Figure 16(b), a full three-dimensional NASTAR (NS) solution is displayed. The computational mesh had 72 points in the axial direction, 36 points across the inlet duct height, and 28 points in the spanwise direction. A spanwise symmetry was assumed to conserve the number of grid points used in the calculation. The problem was run as a one zone domain, using a nonreflective boundary condition along the capture streamline. This grid setup assumes no internal interaction with the external flow. The implicit NASTAR procedure converged in 200 iterations, when started with an initial flow field generated from a two-dimensional calculation. The CPU required was 3.7 hours on an IBM 3090.

The results displayed on Figure 16(b) are shown at two cross-planes; one near the inlet entrance and the other substantially further in the inlet. Very little three-dimensional flow is seen to exist, and the symmetry line/2D comparisons presented previously are seen to be reasonable. The corner region of the downstream plane, however, shows some significant flow turning. This is more easily seen in the displayed velocity vector plots. The initial plane is basically 2D with upward pointing vectors corresponding to the ramp effect of the forebody. The downstream plane shows the kind of corner crossflow pattern observed by SRA/NASA LeRC in their PEPSI-S calculations.

Mixing Driven Supersonic Combustion

Finally, a mixing driven supersonic combustion validation study uses the Burrows & Kurkov (1973) experimental data to benchmark the UTRC version of the NASA LaRC SPARK NS code of Drummond (1985). In the experiment, shown schematically on figure 17, a sonic jet of hydrogen is injected tangentially into a supersonic air stream. Although the combustion efficiency is only 20 per cent, the experiment is unique in that species concentration data were obtained at an exit plane located approximately 90 slot heights downstream of the injection plane. SPARK NS calculations for the nonreacting and reacting cases are shown on figure 17. The

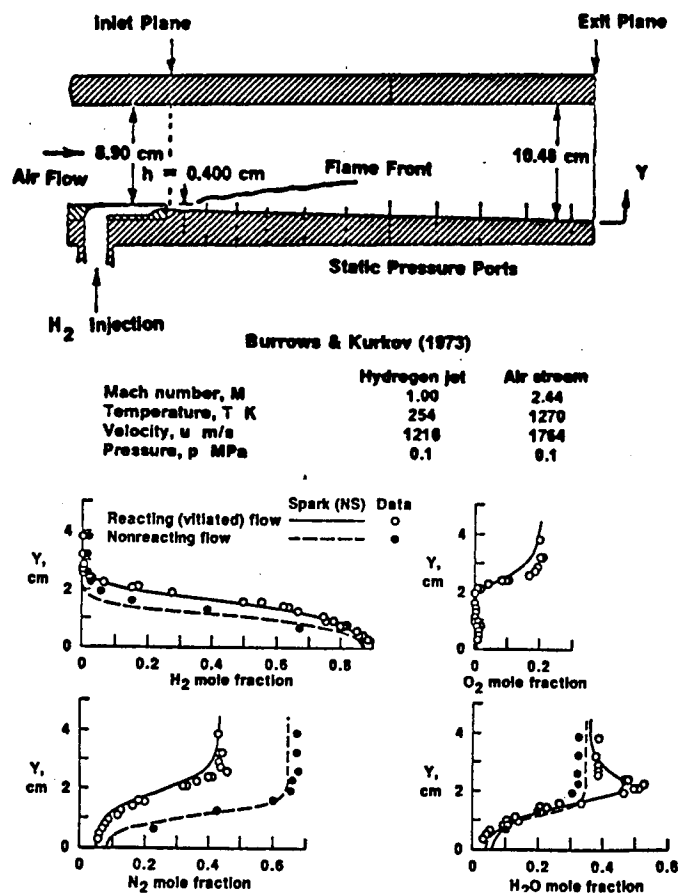


Figure 17. Unit CFD Problem: Mixing Driven Supersonic Combustion. (Validation of Navier-Stokes Combustor Codes)

nonreacting case represents the situation where the mainstream flow is heated through combustion in an upstream section, but the flow in the test section has no oxygen available for combustion. The reacting or vitiated case has the inlet air stream replenished with oxygen. The SPARK runs were made using a stretched 31×35 mesh. An algebraic model was used to simulate turbulence effects. No effort was made to model the upstream boundary layer in these calculations. Species comparisons with data for the reacting and nonreacting cases show excellent agreement in level as well as in the location of the flame front.

The ultimate goal of these types of CFD validation efforts is to apply state-of-the-art computational methods to complex hypersonic flow problems in support of the NASP design program. Frequently, critical design problems cannot be answered using classical design methodology. Since experimental verification is either difficult or too costly, a CFD based design procedure is necessary. Validated codes can then be applied with more confidence to the design of complex flow components such as a three-dimensional scramjet combustor. During portions of the flight envelope, fuel is injected at sonic velocities into a supersonic air stream, schematically shown on figure 18. The critical design issues of mixing efficiency (combustor length) and wall integrity (surface cooling) require a full three-dimensional reacting flow NS analysis. Computer main memory resources may be inadequate on most machines when sufficient mesh resolution (additional nodes) and full chemistry (additional words per node) are necessary. As part of UTRC's support of the NASP program, a new three-dimensional reacting flow Navier-Stokes Analysis will be developed from the NASCRIN/SPARK methodology. The NAS facility capabilities will be essential for expediting this effort and permitting use of the analysis in a realistic time frame.

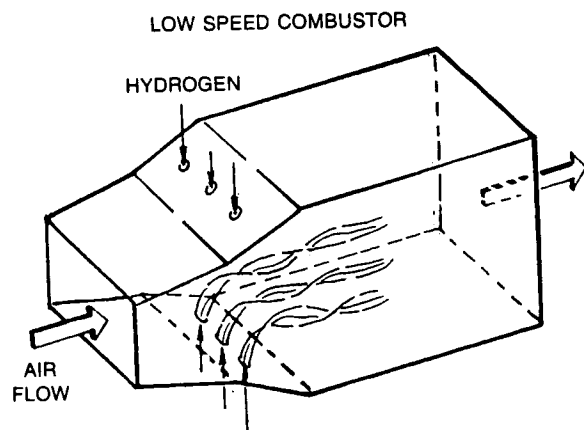


Figure 18. Three-Dimensional Scramjet Combustor Simulation: Normal Injection

TURBOMACHINERY CFD

Two-Dimensional Cascade Viscous Flow Analysis

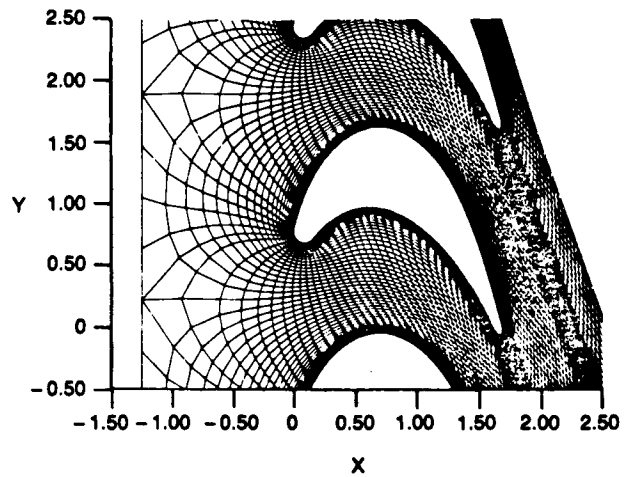
The drive towards higher jet engine efficiency has resulted in increased aerodynamic and heat transfer demands on the compressor and turbine sections of gas turbine engines. As loading requirements in the blade rows and turbine inlet stagnation temperatures increase beyond the range of previous applications, the designer's job becomes increasingly difficult without the use of sophisticated three-dimensional flow analyses. Separated flow regions and hot spots in turbomachinery blade passages occur more frequently as the operation envelope is pushed to the limit. In order to determine the existence and effects of these phenomena an accurate and reliable three-dimensional viscous flow procedure becomes necessary.

The analysis and prediction of three-dimensional viscous flows in blade rows has only recently been treated. Severe demands are placed on computer storage and computational time in order to accurately resolve the wall layers adjacent to the airfoil surface and endwalls. A logical starting point for the eventual development of a three-dimensional viscous analysis for turbomachinery is the development of a two-dimensional solution technique for the Navier-Stokes equations in which the wall layers are resolved with fine grid distributions.

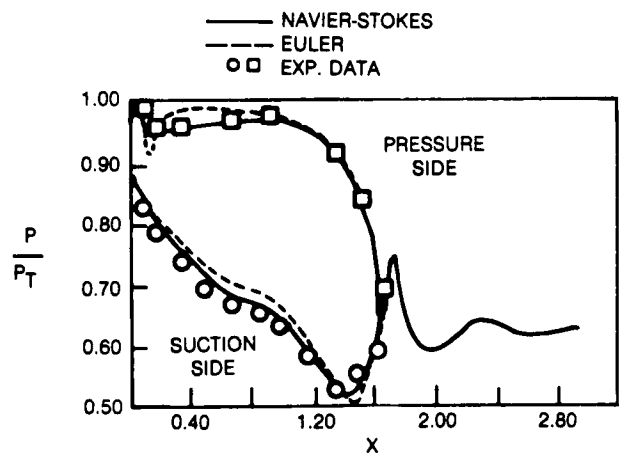
A technique recently developed for the solution of two-dimensional Navier-Stokes equations is a control volume algorithm presented by Davis, Ni, and Carter (1986). This algorithm uses an explicit, control volume approach combined with an efficient multiple-grid scheme, as described by Ni (1982) and Davis (1986), to update the flow variables in time until a steady state solution is obtained. The solution procedure is formally second order accurate in space for all terms in the Navier-Stokes equations. This time marching scheme is second order accurate in time for inviscid flow and first order accurate in time for viscous flow which is similar to other efforts. Several key features in this Navier-Stokes solution technique allow for the calculation of turbulent, high Reynolds number flow in cascades. A new compact operator for the calculation of the viscous terms in the Navier-Stokes equations provides high accuracy and stability in this numerical approach. A modified "C" grid generation procedure based upon the "GRAPE" Poisson solver of Sorenson (1980) and Steger (1979) is used to reduce the computational grid skewing in the mid-passage region compared to standard cascade "C" grid applications and thereby reduce numerical losses introduced by smoothing operators applied on skewed grids. The Baldwin-Lomax (1978) two layer algebraic turbulence model is implemented using a body normal grid which alleviates any apriori approximations as to the thickness of the boundary layer region adjacent to the solid surface. Finally, a combination of second and fourth difference numerical smoothing is used in this numerical procedure to dampen oscillations brought about by truncation errors on relatively coarse grids. Both second and fourth difference smoothing operators for the flow velocity variables are reduced to zero as the solid boundary is approached in order to accurately resolve the viscous layer region.

A case which demonstrates the ability of the current Navier-Stokes procedure to predict flows with significant separation is the subsonic turbine cascade tested experimentally by Stoeffer et. al. (1977). The 161×33 point grid shown in figure 19(a) was used in the viscous calculation whereas a 161×17 point mesh was used for the inviscid calculation. Both calculations were run with three levels of multiple-grid in addition to the fine grid solution. The viscous solution converged in 1500 time steps whereas the inviscid calculation reached convergence in 260 time steps. The flow on the suction side of the airfoil was assumed to be fully turbulent and for the pressure side of the airfoil, where a large separated region existed, transition was held fixed at $x = 0.24$ which was determined from the empirical correlation of Roberts (1979) for closed transitional separation bubbles. The inlet and exit aerodynamic conditions for this cascade are given in figure 19(b) along with the comparison between the predicted pressure distributions and the experimental data. Comparison of the predicted pressure distributions from both the Navier-Stokes and Euler analyses with the experimental data in figure 19(b) shows that the Navier-Stokes analysis has correctly predicted the change in the pressure due to the viscous effects. In this case it is quite clear that a significant inviscid-viscous interaction occurs due to the pressure side separation which effects the pressure distribution on both sides of the airfoil. The predicted streamline pattern shown in figure 19(c) clearly shows the extent of the separated flow region on the pressure side of the airfoil.

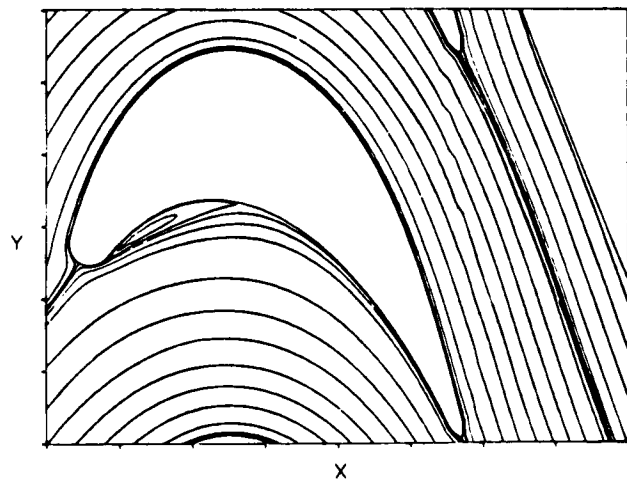
Accurate viscous solutions for two-dimensional cascades at near design incidence flow conditions are now obtainable with this Navier-Stokes technique of Davis, Ni, and Carter (1986). This numerical approach can now be the basis from which more challenging viscous turbomachinery flows can be attacked. For two-dimensional flows, high incidence cascade flow and total pressure loss performance prediction are two fluid dynamic problems which stretch existing numerical schemes as well as the present day computer technology to their limits. These problems are currently being pursued using the previously described Navier-Stokes approach using sophisticated supercomputer architectures. Research into the numerical approach and computational grid requirements to accurately predict cascade flows and corresponding total pressure loss over the entire incidence range is currently underway. Two types of supercomputer systems are being used in this research: a Cray XMP vector processor and the IBM LCAP parallel processing computer. The Cray XMP supercomputer has both vector and concurrent (multi-tasking) capabilities although current research will limit itself to using the vector processor of the computer. The IBM LCAP system is a research computer which consists of 10 floating Point System array processors tied in a ring to a scalar IBM host machine. A central storage area is used to communicate information between array processors. Parallel computing using this system is much different than multi-tasking on a Cray or IBM 3090/400 since data storage is local to the array processors and not in a common global storage location. The Cray and IBM LCAP computers have relatively the same peak performance. Supercomputers such as these are essential in computing the complex flow structures which arise in high incidence cascade flows.



(a) Computational Grid for Turbine Cascade



(b) Comparison Between Predicted Pressure Distributions and Experimental Data



(c) Predicted Streamline Pattern

Figure 19. Navier-Stokes Calculation for Turbine Cascade with Separated Flow

THREE-DIMENSIONAL SEPARATED FLOW CFD

Viscous effects can have a substantial impact on the aerodynamic performance of external and internal flow configurations. For a significant number of high Reynolds number flows of practical interest, the viscous effects, even in the presence of boundary-layer separation, are confined to a relatively small layer near the surface. As indicated in the McDonald and Briley (1984) survey, many two-dimensional aerodynamic problems have been represented by Interacting Boundary-Layer Theory (IBLT) whereby the flow is divided into regions of viscous and inviscid flows with the two regions coupled through the displacement thickness. However, since the majority of the fluid flows in nature are three dimensional, and with the rapid advances in the development of supercomputers, it is timely to turn our attention to the analysis of strongly interacting three-dimensional flows. An objective of this present investigation is to determine if the IBLT methodology, which has been applied successfully in analyzing two-dimensional flows, can be extended to analyze three-dimensional strongly interacting flows. A second objective has been to incorporate IBLT concepts in the development of a thin-layer Navier-Stokes (TLNS) procedure in order to take advantage of the efficiency which IBLT techniques offer but retain the generality and flexibility of Navier-Stokes procedures.

Interaction Boundary-Layer Theory

One procedure used at UTRC, in this study of three-dimensional separated flow, is Interaction Boundary-Layer Theory (IBLT) developed by Edwards (1986). In this procedure, the entire flow field is represented as regions of viscous and inviscid flow with the two regions coupled through the viscous displacement thickness. The inviscid solution is obtained from a three-dimensional small disturbance surface integral representation of an incompressible, irrotational flow field where the effect of the viscous flow is imposed by analyzing the inviscid flow over a displacement body instead of the actual geometrical surface. The flow in the viscous region is assumed to be governed by Prandtl's boundary-layer equations. The viscous equations are solved with an implicit finite difference approach, in which derivatives in the x -direction are discretized using first-order upwind approximations, and derivatives in the normal and z -directions are discretized using second order central differencing. Newton iteration is used to solve the viscous equations at each x and z location, as the solution is marched from a plane of initial data and a plane of symmetry. In this study, a three-dimensional extension of the quasi-simultaneous methodology of Veldman (1983) is used to couple the inviscid and viscous flow fields.

Thin-Layer Navier-Stokes Theory

A second procedure used in the study of three-dimensional separated flow is a thin-layer Navier-Stokes (TLNS) procedure developed by Davis, et al (1987). The objective in the development of this TLNS procedure has been to incorporate IBLT concepts in order to take advantage of the efficiency which IBLT techniques offer but retain the generality and flexibility of Navier-Stokes

procedures. This approach has been pursued by casting the 3-D TLNS equations into a streamlike function and vorticity transport form. In this formulation, two sets of streamlike function Poisson and vorticity transport equations are solved using an implicit line Gauss-Seidel relaxation procedure. The equations describing the axial component of flow are numerically uncoupled from the spanwise component flow equations allowing an efficient implicit block 2×2 tri-diagonal solution procedure to be used along lines of $x = \text{constant}$, $z = \text{constant}$ which are essentially perpendicular to the solid surface in a similar fashion as utilized by boundary layer techniques. Standard second order accurate central difference formula are used in the discretization of the Poisson and vorticity transport equations with the exception of the axial convection terms of the vorticity transport equations. For these convection terms, a first order windward difference operator, similar to that utilized for interacting boundary layer calculations, is used to model the parabolic nature of the vorticity transport equations. Thus, in the inviscid region where the Poisson streamlike function equations dominate the flow, central difference formula are used to model the elliptic nature of the equations, whereas in the viscous region which is dominated by the vorticity transport equations, appropriate one-sided windward differences are used to model the parabolic nature of these equations.

Application of IBLT and TLNS Methods

Results are presented for the analysis of three-dimensional incompressible separated flow using the the IBLT and TLNS methods. The model problem used in this study is laminar flow over a flat plate with either a dent or bump located downstream of the leading edge. The flat plate with the protuberance (or dent) is shown schematically in figure 20. A sign in the geometric equations determines if the flat plate has either a dent or bump. The oncoming flow for this model problem is aligned with the x coordinate. Along the vertical plane of symmetry, the surface geometry is identical to the planar trough shape studied by Carter and Wornom (1975) for two-dimensional separated flow.

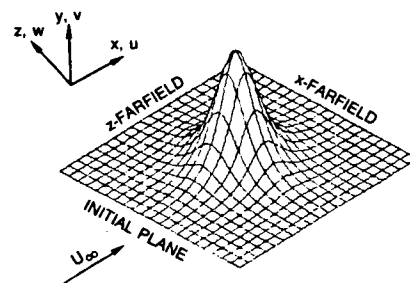


Figure 20. Schematic of Flow Over a Flat Plate With a Protuberance (Dent)

Several comparisons have been made between the present IBLT and TLNS methods (for example, Edwards (1986), Davis et al (1987), and Carter et al (1986)). One flow case is laminar flow over a flat plate with a dent ($t = -0.03$) at $Re = 8 \times 10^4$. Both analyses used the same uniform grid consisting of 121 points in the x-direction and 31 points in the z-direction where the x-z computational domain is $1 < x < 4$ and $0 < z < 1.5$. The IBLT method used 21 points in the normal direction across the boundary layer while the TLNS method used 81 points in the normal direction with at least 32 points in the viscous region. Both procedures have been tested with finer normal grids with little change in the computed results. Symmetry conditions are assumed along $z=0$ and the Blasius flat plate solution is assumed at $x=1$.

A comparison of the results of the two analyses is shown in figure 21 where contours from both analyses are shown for the axial component of skin friction. From this figure, it can be seen that the results of the two analyses show nearly similar trends for the flow quantities. Figure 22 shows the limiting streamlines near the vicinity of the dent (both analyses produced essentially the same result). These limiting streamlines are the paths taken by particles located an infinitesimal distance above the surface and are determined using a technique developed by Duck and Burggraf (1986). It is noted in figure 22 that the flow is moving from left to right. Figure 22 shows the formation along the symmetry plane of two singular points on the surface (saddle point of separation and nodal point of reattachment). Lighthill (1963) has shown that the number of nodal points must exceed the number of saddle points exactly by two for a closed body. Since the model problems in this study can be regarded as a local region on such a body, then any additional nodes and saddle points must cancel in number. It is seen from figure 22 that this condition is satisfied. Also in this figure, note that there is a dividing limiting streamline (sometimes referred to as a separation line) emanating from the saddle point of separation which results in forming a region that is not accessible from the upstream. Similar types of flow structure have been observed in the work of Duck and Burggraf (1986), where laminar flow over a flat plate with a protuberance at infinite Reynolds number was examined using triple-deck theory. This favorable comparison between the present IBLT and TLNS methods shows that the IBLT methodology can be applied to the analysis of strongly interacting three-dimensional flows. Further, it demonstrates that IBLT concepts should be incorporated in TLNS and Navier-Stokes procedures.

The 3-D separated flow calculations shown in this paper were made using the Cray X-MP at Cray Research, Inc. The TLNS procedure required approximately 3 hours of CPU time to obtain a converged solution which is based on a four orders of magnification reduction in the equation residuals. The IBLT procedure required approximately 7 minutes of CPU time to obtain a converged solution which is based on a four orders of magnification reduction in the maximum change per iteration of the displacement thickness.

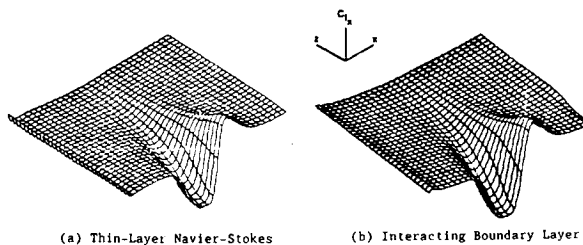


Figure 21. Flow Over a Flat Plate With a Dent. Global Comparison of Axial Skin Friction of IBLT and TLNS

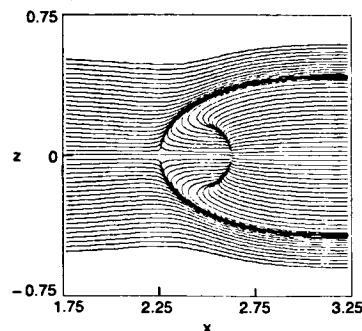


Figure 22. Limiting Streamlines for Flow Over A Flat Plate With a Dent ($Re = 80,000$)

PARALLEL PROCESSING FOR CFD

Computer codes for computational fluid dynamics can benefit from parallel processing. UTRC has recently established several studies to demonstrate and quantify the advantages of parallel processing for selected CFD problems with particular emphasis on evaluating new massive parallel processors. As examples of these activities, two current studies are described below.

Parallel Processing for a Combustor Flow Code

UTRC is participating in a joint program with Yale University's Computer Science Department, AFOSR and Apollo Computer, Inc. to investigate the applicability of parallel architecture computers to CFD problems. A model problem representing the energy equation for perfect gas flow through a cylinder has been solved using MacCormack's explicit difference scheme. (This problem was chosen because it contains the essential calculations for the Hankey-Shang combustor flow code.) The problem was first coded for an Apollo serial processor workstation computer at UTRC, then transferred to Yale where the code was modified for the Intel Hypercube and Encore parallel processor computers. Both the Intel Hypercube and the Encore are MIMD (Multiple Instruction stream/Multiple Data stream) machines however the hypercube has dispersed memory while the Encore has shared memory. The Intel Hypercube is basically 128 IBM PC computers linked together with a hypercube architecture (figure 23).

ORIGINAL PAGE IS
OF POOR QUALITY

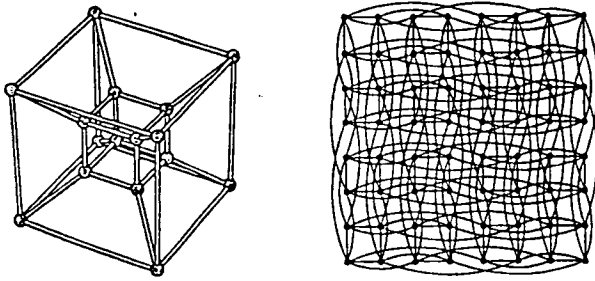


Figure 23. Hypercube Architecture

Numerous timing studies have been completed comparing the speedups for solving the model problem with different numbers of processors, different grid sizes and different domain decompositions. Basic theory such as Amdahl's Law (figure 24) is being extended to understand the difference in speedups and to permit evaluation of new and different parallel architectures. The theory shows the importance of communication time between processors in achieving desired speedups with additional processors. Future work will involve developing parallel code which is portable to different parallel architectures, evaluation of implicit solution algorithms and running the code on different parallel computers.

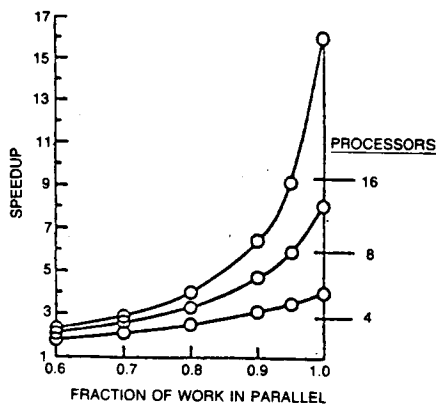
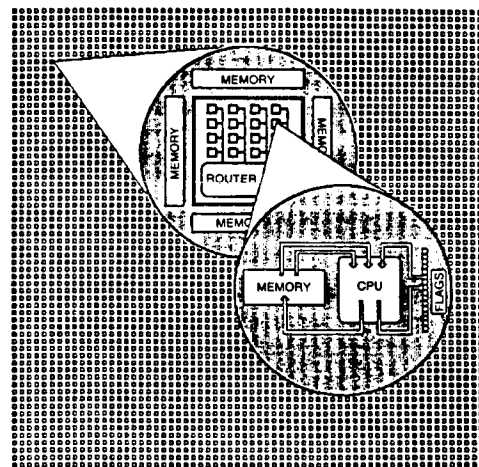


Figure 24. Computer Speedup as Determined by Fraction of Work in Parallel and Number of processors (Amdahl's Law)

Parallel Processing Using Cellular Automata Techniques for Flow/Heat Transfer Modeling

Massively parallel computer architectures containing tens of thousands of processors make practicable entirely new approaches to difficult but important problems. The Connection Machine, announced this year by Thinking Machines Corporation (TMC), Cambridge, Mass., contains 65,536 processors all operating in parallel. In a cooperative program with TMC, cellular automata techniques are being examined for fluid flow/heat transfer modeling required for heat exchanger applications and as an alternative to differential equations for representing physical systems.

The massively parallel architecture represented by the Connection Machine is intrinsic to implementation of these techniques. As shown in figure 25, the Connection Machine contains 4096 chips, each incorporating 16 one-bit processors, local memory for each processor, and a router which coordinates communication. Communication between any two of the 16 processors on a chip occurs in one machine cycle, which because of the simplicity of the processors is very short. The 4096 chips are connected as a 12 dimensional hypercube, i.e., the address of each chip may be represented as a 12 bit binary number and direct communication exists between any two chips whose addresses differ by one bit. Communication between any two chips is therefore accomplished in a maximum of 12 cycles. In the Connection Machine such communication is accomplished in parallel for all chips, via the routers, and in 12 machine cycles all 65,536 processors communicate with their target. With this organization, each processor can be assigned to an element of the problem input data set, and analysis applied simultaneously to all of the data. For very large data sets, the local memory is partitioned and the data is processed in "pages" of 65,536. Such data level analysis opens up radically new approaches to problems: for example, optical lens corrections can be effected by transferring each pixel of an image from a processor representing the input pixel location to another processor representing the pixel position had the lens been perfect. In a similar manner, the cellular automata modeling of fluid flow takes advantage of the extremely efficient communication between nearest neighbor processors of the massively parallel architecture.



Courtesy TMC

Figure 25. Connection Machine -- Processor

The cellular automata approach models with simple interacting elements the complex behavior of natural systems. For 2D fluid flow, the elements are unit mass, unit velocity particles constrained to move along the six directions of a hexagonal lattice.

The particles interact in accord with simple collision rules involving only particles on nearest neighbor lattice points (figure 26). Each lattice

- Unit mass, unit velocity particles
- Simple interactions with nearest neighbors

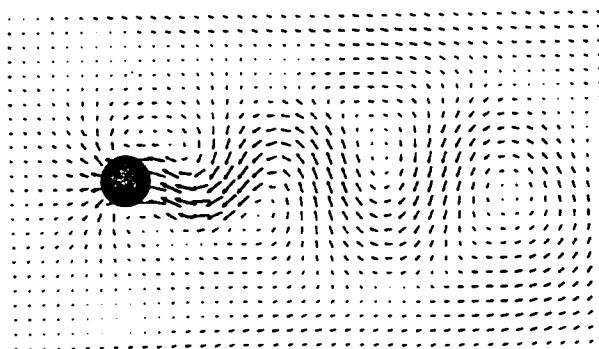


- Flow parameters determined by average over many cells
- Formal equivalence to Navier-Stokes equations

Figure 26. Cellular Automata Fluid Flow Modeling

point (cell) is represented by one processor which tracks the particles currently located at that position. The macroscopic flow parameters, density, velocity, etc., are determined by averages over many cells providing continuum results from the discrete particles. The behavior of this simple system is formally equivalent to the Navier-Stokes equations as described by Wolfram (to be published), and although the model is deterministic, the very large number of degrees of freedom gives rise to the rich array of phenomena observed in fluid flow. Figure 27 is a calculation performed at TMC of the 2D flow over a cylinder (in a coordinate system fixed on the cylinder) and clearly displays the vortex street observed in such flow. The UTRC contribution in the cooperative program involves validation of the cellular automata approach by calculation for known behavior test flow conditions and extension of the analysis to incorporate heat transport by analogy with the momentum transfer of the particle collisions. For example, as illustrated in figure 28, numerical experiments on the Connection Machine, of the 2D flow between parallel plates, have been used by A. Haught of UTRC to determine the effective fluid viscosity for comparison with the viscosity derived analytically from the cellular automata model. The effective fluid viscosity is evaluated from the fluid momentum decay and for the asymptotic exponential decay agrees closely with that derived by D. Levermore (B. Nemnich, TMC, private communications). This simple plate problem, when extended, will have relevance to the heat exchanger application.

The fact that simple interaction rules can reproduce complex behavior offers the exciting promise that cellular automata can be used as an alternative to differential equations for modeling a wide variety of physical systems (Vichniac (1984)). With cellular automata, limited, discrete information is evaluated for each element, and each "bit" calculated weighs equally in the results. With differential equations, very high accuracy is generally required during the analysis - e.g. evaluation of a derivative as the difference of two quantities - to obtain an answer of much lower accuracy. The "bit democracy" of the cellular automata approach results in efficient use of computer resources and represents a potential breakthrough for otherwise intractable problems. The prospect is intriguing, and the fluid flow/heat transfer modeling is an avenue to explore this potential.



Courtesy Bruce Nemnich, TMC

Figure 27. Two-Dimensional Flow Over a Cylinder

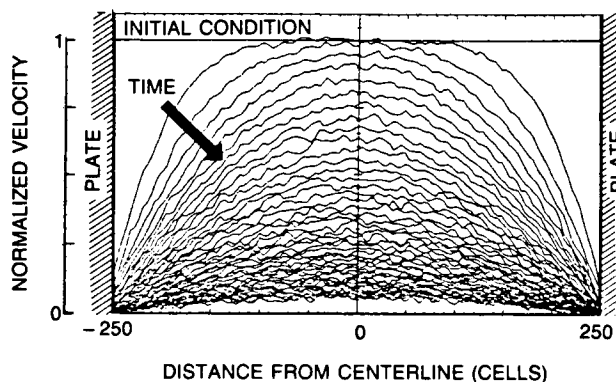


Figure 28. Two-Dimensional Flow Between Parallel Plates -- Velocity Profile Time History

CONCLUDING REMARKS

With the availability of advanced supercomputers and the rapid progress being made in computational fluid dynamics, considerable advances in the development of CFD codes applicable to complex aerodynamic problems of interest to the aircraft industry are foreseeable in the near future. Supercomputer systems such as those of the new NAS Facility and the new massive parallel processing systems are anticipated to provide computational tools for software which will significantly advance the state-of-the-art of CFD technology for a wide variety of design applications. Specific methodology related to helicopters, turbomachinery, heat exchangers, and the National Aerospace Plane have been presented herein, but these are just examples within a wide spectrum of potential applications. It has been recognized and demonstrated in the recent work that the code developers can now conceive of incorporating and combining the CFD features which they have had to separate or compromise on in the past, such as three-dimensional, viscous, compressible, unsteady, and heat transfer effects. This will result in more practical and accurate codes for both design and diagnostic investigations.

REFERENCES

- Arieli, R.; and Tauber, M. E.: Analysis of Quasi-Steady Flow About an Isolated Lifting Helicopter Rotor Blade, Joint Institute for Aeronautics and Astronautics TR-24, Aug. 1979.
- Burrows, M.C.; and Kurkov, A. P.: Analytical and Experimental Study of Supersonic Combustion of Hydrogen in a Vitiated Airstream, NASA TMX-2828, Sept. 1973.
- Baldwin, B. S.; and Lomax, H.: Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows, AIAA Paper 78-257, Jan. 1978.
- Caradonna, F. X.; and Tung, C: Experimental and Analytical Studies of a Model Helicopter Rotor in Hover, NASA TM 81232 (or USAAVRADCOM TR-81-A-23), Sept. 1981.
- Carter, J. E.; Davis, R. L; Edwards, D. E.; and Hafez, M. M.: Three-Dimensional Separated Viscous Flow Analyses, Presented at the Tenth International Conference on Numerical Methods in Fluid Dynamics, Beijing, China, June, 1986.
- Carter, J. E.; and Wornom, S. F: Solutions for Incompressible Separated Boundary Layers Including Viscous-Inviscid Interaction, NASA SP-347, 1975.
- Chang, I. C.: Transonic Flow Analysis for Rotors - Part 2 - Three-Dimensional, Unsteady, Full-Potential Calculation, NASA TP-2374, 1984.
- Davis, R. L.; Ni, R. H.; and Carter, J. E.: Cascade Viscous Flow Analysis Using the Navier-Stokes Equations, AIAA Paper 86-0033, Jan. 1986.
- Davis, R. L.; Carter, J. E.; and Hafez, M. M.: Three-Dimensional Viscous Flow Solutions with a Vorticity Stream Function Formulation, AIAA Paper 87-0601, 1987.
- Davis, S. S.; and Chang, I. C.: The Critical Role of Computational Fluid Dynamics in Rotary-Wing Aerodynamics, AIAA 24th Aerospace Sciences Meeting, Reno, NV AIAA-86-0336, Jan. 1986.
- Drummond, J. P.: Numerical Study of a Ramjet Dump Combustor Flow Field, AIAA Journal Vol. 23, No. 4, 1985, p. 604.
- Duck, P. W.; and Burggraf, O. R.: Spectral Solutions for Three-Dimensional Triple Deck Flow over Surface Topography, J. Fluid Mech., Vol. 162, 1986.
- Edwards, D. E.: Analysis of Three-Dimensional Separated Flow Using Interacting Boundary-Layer Theory, Presented at the International Union of Theoretical and Applied Mechanics (IUTAM) Symposium on Boundary-Layer Separation, London, U. K., August, 1986.
- Egolf, T. A.; and Landgrebe, A. J.: Generalized Wake Geometry for a Helicopter Rotor in Forward Flight and Effect of Wake Deformation on Airloads, Proceedings of the 40th Annual Forum of the American Helicopter Society, May, 1984.
- Egolf, T. A.; and Sparks, S. P.: Hovering Rotor Airload Prediction Using a Full Potential Flow Analysis with Realistic Wake Geometry, Proceedings of the 41st Annual Forum of the American Helicopter Society, May 1985.
- Egolf, T. A.; and Sparks, S. P.: A Full Potential Rotor Analysis with Wake Influence Using an Inner-Outer Domain Technique, Proceedings of the 42nd Annual Forum of the American Helicopter Society, June 1986.
- Holden, M. S.; and Moselle, J. R.: Theoretical and Experimental Studies of the Shock Wave/Boundary Layer Interaction on Compression Surfaces in Hypersonic Flow, Calspan Report AF-2410-A-1, Oct. 1969, also ARL 70-0002, Jan. 1970.
- Krawczyk, W. J.; Rahendran, N.; Harris, T. B.; York, B. J.; and Dash, S. M.: Computational Models for the Analysis Design of Hypersonic Scramjet Components - Part II, AIAA 86-1596 AIAA/ASME/SAE/ASEE 2nd Joint Propulsion Conference, Huntsville, AL, 1986.
- Kumar, A.: Numerical Simulation of Scramjet Inlet Flow Fields, NASA TP-2517, May, 1986.
- Landgrebe, A. J.: An Analytical Method for Predicting Rotor Wake Geometry, Journal of the American Helicopter Society, Vol. 14, No. 4, Oct. 1969.
- Landgrebe, A. J.: The Wake Geometry of a Hovering Helicopter Rotor and Its Influence on Rotor Performance, Journal of the American Helicopter Society, Vol. 17, No. 4, Oct. 1972, (also, Proceedings of the 28th Annual National Forum of the American Helicopter Society, May 1972.
- Landgrebe, A. J.; and Egolf, T. A.: Rotorcraft Wake Analysis for the Prediction of Induced Velocities, USAAMRDL TR-75-45, Jan. 1976. (Available from DTIC as AD A021, 202).
- Landgrebe, A. J.; and Egolf, T. A.: Prediction of Helicopter Induced Flow Velocities Using the Rotorcraft Wake Analysis, Proceedings of the 32nd Annual Forum of the American Helicopter Society, May 1976.
- Landgrebe, A. J.: Overview of Helicopter Wake and Airloads Technology, Proceedings of the American Helicopter Society/Nanjing Aeronautical Institute International Seminar - The Theoretical Basis of Helicopter Technology, Nanjing China, Nov. 1985, also Proceedings of the 12th European Rotorcraft Forum, Paper No. 18, Garmisch-Partenkirchen, FRG, Sept. 1986.
- Lighthill, M. J.: Attachment and Separation in Three-Dimensional Flow, Laminar Boundary Layers, Oxford University Press, 1963.
- McCroskey, W. J.; Baeder, J. D.; and Bridgeman, J. O.: Calculation of Helicopter Airfoil Characteristics for High Tip-Speed Application, Proceedings of the 41st Annual Forum of the American Helicopter Society, May 15-17, 1985.

- McCroskey, W. J.; and Baeder, J. D.: Some Recent Advances in Computational Aerodynamics for Helicopter Applications, NASA TM-86777, Oct. 1985.
- McDonald, H; and Briley, W. R.: A Survey of Recent Work on Interacted Boundary-Layer Theory for Flow with Separation, Numerical and Physical Aspects of Aerodynamic Flows II, Springer-Verlag, 1984.
- Ng, W. F.; Benson, T. J.; and Kunik, W. G.: Real Gas Effects on the Numerical Simulation of Hypersonic Inlet, J. Propulsion and Power, Vol. 2, No. 4, 1986, p. 381.
- Ni, R. H.: A Multiple Grid Scheme for Solving the Euler Equations, AIAA Journal, Vol. 20, Nov. 1982, pp. 1565-1571.
- Rhie, C. M.: A Pressure Based Navier-Stokes Solver Using the Multigrid Method, AIAA 86-0207, AIAA 24th Aerospace Sciences Meeting, Reno, NV, 1986.
- Roberts, W. B.: Calculations of Laminar Separation Bubbles and Their Effect on Airfoil Performance, AIAA Paper 79-0285, Jan. 1979.
- Seebaugh, W. R.: Hypersonic Flows in Large Scale Inlet Models, J. Aircraft, Vol. 10, No. 1, January 1973, p. 38.
- Sorenson, R. L.: A Computer Program to Generate Two-Dimensional Grids About Airfoils and Other Shapes by the Use of Poisson's Equation, NASA TM-81198, 1980.
- Steger, J. L. and Sorenson, R. L.: Automatic Mesh-Point Clustering Near a Boundary in Grid Generation with Elliptic Partial Differential Equations, Journal of Computational Physics, Vol. 33, No. 3, Dec. 1979.
- Stoeffler, R., Dring, R. P., and Kopper, F.: United Technologies Internal Report, 1977.
- Strawn, R. C.; and Caradonna, F. X.: Numerical Modeling of Rotor Flows with a Conservative Form of the Full-Potential Equations, AIAA 86-0079, Jan. 1986.
- Tauber, M. E.: Computerized Aerodynamic Design of a Transonic Low-Noise Blade, NASA Technical Memorandum 85928, March, 1984.
- Veldman, A. E. P.: A Numerical View on Strong Viscous-Inviscid Interaction, NLR MP 83048 U, 1983.
- Vichnia, G. Y.: Simulating Physics with Cellular Automata, Physics 10D, 1984, p. 96-116.
- Wolfram, S.: Cellular Automation Fluids 1: Basic Theory, The Institute for Advanced Study, Princeton, NJ, to be published.