NASA Technical Memorandum 100066

USAAVSCOM Technical Report 88-A-001

NASA-TM-100066

19880015217

Some Rototcraft Applications of Computational Fluid Dynamics

W. J. McCroskey

March 1988

National Aeronautics and
Space Administration

US ARMY
AVIATION
SYSTEMS COMMAND

AVIATION RESEARCH AND
TECHNOLOGY ACTIVITY

NASA Technical Memorandum 100066
USAAVSCOM Technical Report 88-A-001

Some Rototcraft Applications of Computational Fluid Dynamics

W. J. McCroskey
Ames Research Center and Aeroflightdynamics Directorate, U.S. Army Aviation Research and Technology Activity, Ames Research Center, Moffett Field, California

March 1988

NASA
National Aeronautics and Space Administration

Ames Research Center
Moffett Field, California 94035

US ARMY
AVIATION
SYSTEMS COMMAND

AVIATION RESEARCH AND
TECHNOLOGY ACTIVITY
MOFFETT FIELD, CA 94305-1099

N88-24601#

SOME ROTORCRAFT APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS

W.J. McCroskey
Ames Research Center
and
Aeroflightdynamics Directorate
U.S. Army Aviation Research and Technology Activity

ABSTRACT

The growing application of computational aerodynamics to nonlinear rotorcraft problems is outlined, with particular emphasis on the development of new methods based on the Euler and thin-layer Navier-Stokes equations. Rotor airfoil characteristics can now be calculated accurately over a wide range of transonic flow conditions. However, unsteady three-dimensional viscous codes remain in the research stage, and a numerical simulation of the complete flow field about a helicopter in forward flight is not currently feasible. Nevertheless, impressive progress is being made in preparation for future supercomputers that will enable meaningful calculations to be made for arbitrary rotorcraft configurations.

I. INTRODUCTION

The flow fields of rotorcraft provide a rich variety of challenging problems in applied aerodynamics. It is well known that much of the flow of the rotating blades is nonlinear, three-dimensional, and often unsteady, with periodic regions of transonic flow near the blade tips, and with dynamic stall pockets inboard. The blades also shed complex vortical wakes, and serious aerodynamic interactions may arise between the major rotating and nonrotating components.

For many years, helicopter engineers have addressed these difficult problems with a mixture of simplified linear aerodynamic theories, wind tunnel data, and design charts. At the same time, a small community of research scientists has systematically explored the details of individual pieces of the overall problem, indicated by the sketches in Figure (1). References (1) and (2) provide an overall picture of the practical side of helicopter aerodynamics, and References (3), (4), and (5) summarize many of the recent studies of these simpler "building blocks." In recent years, computational fluid dynamics (CFD) has begun to offer new tools to the rotorcraft community; and Reference (5), in particular, emphasizes some pioneering applications of CFD to transonic tip flow on the advancing side of the rotor disc.

Computational fluid dynamics has had a major impact on fixed-wing aircraft design, but for a variety of reasons, comparable applications to and payoffs for rotorcraft lag by a decade or more. Nevertheless, the combination of fixed-wing successes and rapid advances in modern supercomputer software and hardware have begun to attract the attention of both government and industry management. Consequently, fresh challenges and new opportunities are arising for scientists and engineers engaged in rotorcraft research and development.

Significant applications of CFD are already under way at several laboratories, universities, and helcopter companies, including investigations described at this conference. This paper is not intended to be a review of these activities. Instead, it selectively highlights a few of the joint Army/NASA efforts by the author and his colleagues to develop, adapt, and apply the latest CFD technology to rotorcraft. The overall, long-term objective of this work is to develop and validate advanced CFD codes for three-dimensional, unsteady viscous flows about arbitrary, elastic rotorcraft configurations. It must be noted that we have no illusions of building an all-inclusive, comprehensive analysis system based on CFD. Rather, the goal is to provide a complete numerical simulation of realistic rotorcraft flow fields with as few physical approximations as possible. This predictive capability will help to reduce the risks for new rotorcraft designs and will provide improved tools to help engineers increase performance and efficiency and reduce noise and vibrations.

In pursuing this goal, a twofold approach is currently being taken at the Ames Research Center:
1) Mature CFD methods are being coupled with integral wake models and simplified structural models for engineering design and analysis using current or almost-available supercomputers.
2) New CFD codes are being developed and validated

Presented at the Second International Conference on Basic Rotorcraft Research, College Park, Maryland, February 1988.

for selective calculations of rotorcraft configurations using future supercomputers.

By way of explanation, the relatively mature computational methods used in the first approach include the transonic small-disturbance and full-potential flow solvers that have been used routinely by the fixed-wing industry for years, but which are only now becoming useful to helicopter engineers. Their approximations may limit their scope and accuracy, but they run fairly easily and efficiently, and they can provide extremely valuable information to a skilled user.

In the second approach, which is the subject of this paper, the new codes and their associated technology employ Euler and Navier-Stokes flow solvers. Today these are mostly being implemented in a pure research mode. That is, they are currently nursed along by CFD specialists for highly-specialized cases, at an inordinate expense in CPU time on the most powerful supercomputers available, and the results may or may not be realistic. Whenever possible, useful and practical information is derived from these exploratory results, but the main purpose is to build toward future capabilities. While we must refrain from unrealistic projections, such as replacing wind tunnels by computations, or designing new rotorcraft automatically by expert systems, the fixed-wing legacy clearly shows that today's CFD research codes become tomorrow's design tools.

II. INDIVIDUAL COMPONENTS

The building-block approach depicted in Figure (1) provides a framework for building sophisticated CFD codes in manageable steps, with some early applications and practical dividends along the way. This section describes some of the recent CFD developments at the Ames Research Center for these individual components.

A. Static Airfoil Characteristics

The author and his colleagues began their long-range program of applying Navier-Stokes algorithms to rotorcraft problems with the two-dimensional airfoil calculations described in Reference (6). In this study, the aerodynamic section characteristics of several helicopter profiles were computed using the existing NASA-Ames code ARC2D [7], which solves the Reynolds-averaged, thin-layer Navier-Stokes equations with the Baldwin-Lomax eddy-viscosity model to approximate boundary-layer turbulence [8]. A wide range of Mach number, angle of attack, and Reynolds number was examined in Reference (6), with particular emphasis on transonic separated flow conditions relevant to the advancing blade in high-speed forward flight. An extensive study was made of the accuracy and sensitivity of the results to the various numerical parameters and approximations, and favorable comparisons were made with a large body of experimental data.

Figures (2) and (3) give representative examples from Reference (6) for lift, drag and pitching-moment coefficients. Of particular interest to the helicopter community is the variation of C_m with Mach number. For example, the large negative peak in the value of C_m at $M \approx 0.88$, called "Mach tuck," is well predicted by ARC2D, as shown in Figure (3). Stall and maximum lift at low Mach number were not addressed, however, and this important problem remains unsolved for high Reynolds-number flows. However, this is mainly because of the turbulence modeling, and not because of computational barriers.

More recently, Holst [9] examined in detail the capabilities and limitations of ARC2D, along with numerous other transonic airfoil codes, and he concluded that codes of this type are capable of producing airfoil data that are as accurate as wind tunnel measurements. This new capability is beginning to be exploited by specialists in the helicopter industry, as a tool complementary to airfoil testing.

B. Airfoil-Vortex Interactions

The lessons learned from the preceding experiences with the quasi-steady version of ARC2D and from the adaptation of the unsteady transonic small-disturbance code ATRAN2 to inviscid airfoil-vortex interactions [10] led us to apply time-accurate Euler and Navier-Stokes algorithms [11-13] to this two-dimensional approximation of blade-vortex interaction. Figure (4) shows the Navier-Stokes results from Reference (11), using the so-called prescribed vortex approach and unsteady solution-adaptive gridding. In this example, the structure of the vortex was prescribed and maintained throughout the calculation, but its convected path in space was computed as part of the solution. This technique, which is somewhat analogous to shock fitting in the numerical analysis of transonic and supersonic flows, is designed to overcome the tendency of most numerical schemes to dissipate artificially the steep gradients within the vortex.

There was no significant shock-wave/boundary-layer interaction in the case shown in Figure (4). Therefore comparisons with other numerical techniques [14] revealed that the Navier-Stokes equations were not necessary to obtain the correct time-dependent airloads in this case - they just required much more CPU time to solve. On the other hand, the solution-adaptive grid used in Reference (11) provided such fine resolution near the moving shock wave that a lambda structure was observed during the vortex encounter, whereas this structure was not revealed in other calculations.

The prescribed-vortex approach of References (10), (11), and (14) is appropriate to cases in which the core of the vortex bypasses the airfoil or rotor blade, but it cannot treat a direct, head-on encounter that would significantly alter the vortex structure. However, Rai [13] has developed a vortex-capturing method with very low numerical dissipation for such cases, using a higher-order differencing algorithm for the Euler or Navier-Stokes equations and a special grid structure. Rai's method is also suitable for capturing weak disturbances, such as acoustic waves.

In fact, a major motivating factor in studying airfoil-vortex interactions is the acoustics of helicopter blade-vortex interaction, and a central question here is the capability of "conventional" CFD techniques to capture the far-field acoustic radiation due to the nonlinear shock-wave motion. This issue is being addressed by James Baeder, with the aid of Rai's method described above, and he has reported his preliminary findings in References (12) and (15). Baeder is currently comparing the effects of the numerical algorithms on the results in the near field of the airfoil, where the reactive components of the fluctations are dominant, and in the intermediate field, where the radiative components begin to dominate. Although the surface airloads are not significantly different, Figure (6) from Reference (12) shows that the different levels of sophistication in the various codes produce differences in the acoustic pressure throughout the field. At present, a fifth-order-accurate Euler code seems to be the best overall compromise between accuracy and cost of the computations. Baeder [12] also studied the effect of vortex core size, miss distance (including head-on encounters), and Mach number, and he is currently examining various airfoil shapes to see whether BVI noise can be alleviated by changing the airfoil geometry. The results so far indicate that the Mach number and miss distance are the most important parameters in determining the radiated noise.

C. Tip Flows and Tip-Vortex Formation

The formation and subsequent rollup of tip vortices from blade tips is an important fundamental problem that provides an ideal opportunity to extend the capabilities of the Navier-Stokes codes to a practical component of the total rotor flow field. It is well known that tip vortices play a crucial role in helicopter aerodynamics and that their structure is closely related to the blade-tip geometry. However, past efforts to alter tip shapes and vortex structure have been largely by trial and error, with mixed results, and no consensus has emerged on the optimum blade geometry. This seems to be an area where computational fluid dynamics can improve our understanding of the basic phenomena and help design and analyze better blade tips. In addition, we envision that accurately-computed vortex structures will be used as inputs to wake modeling efforts and/or prescribed-vortex methods for calculating the complete vortical wake development for several rotations of the blades.

Srinivasan, et al. [16,17] have extended the pioneering efforts of Mansour [18] and Kaynak, et al. [19,20] for viscous three-dimensional wings by concentrating on the tip region and the vortex formation process, using the Reynolds-averaged thin-layer Navier-Stokes equations. The planforms shown in Figure (7) have already been investigated, and the British Experimental Rotor Program (BERP) configuration, Figure (8), is currently undergoing detailed study.

Figure (9) shows the computed streamline pattern for a swept-tip configuration designed by ONERA. This example is nonrotating and is at a freestream Mach number of 0.85. The swept tip reduces the shock wave strength, but it increases the concentration of vorticity and the peak velocities in the tip vortex.

Representative results for a rectangular wing at low speeds are shown in Figure (10) from Reference (16). An interesting result of this series of calculations was the sensitivity of the detailed flow features to the geometry of the tip cap, and the importance of using different computational grid topologies in modeling the different tip-cap shapes, e.g. Figure (11). Generally, good agreement was obtained with experimental results, including the qualitative differences between the different tip caps. However, the peak suction levels that were measured in the extreme tip region were not obtained in the calculations. The reason for this clear but relatively minor discrepancy in the numerical results has not been identified; it may be due to the turbulence model, to inadequate local grid resolution, or to the thin-layer approximation in the governing equations.

Another shortcoming of the tip-vortex calculations done up to now is the distortion and diffusion of the vortex structure downstream of the blade tip, due to the effects of the sparse grids in the far field and the numerical dissipation used to enhance the stability of the code. For this reason, the vorticity contours in the wake and the calculated rollup process can only be considered qualitative at this time. New, special grid topologies and solution-adaptive grid generation strategies will be required to overcome this limitation, as discussed in Section V.

III. ISOLATED ROTOR BLADES

The numerical methods of the preceding section for studying blade tip flows and tip vortices and the experience gained in working with them are now being applied to rotating blades. However, the extension is not a small step, as the complexities of the flow field and the computer resources required are significantly greater in these cases. In this section, a brief overview is given of some recent results, current activities, and future directions for several aspects of flows on rotor blades.

Two aspects of our current research should be pointed out. First, the codes are being developed to treat hover and forward flight in a unified manner, for arbitrary blade motion. Therefore, as explained in References (17), (21), and (22), we have chosen to solve the unsteady Euler and thin-layer Navier-Stokes equations in an inertial frame of reference, rather than in blade-fixed coordinates. This results in some increases in CPU time for hovering cases, where non-time-accurate convergence-acceleration techniques might otherwise be used, but it allows the same general formulation and code to be used for both hover and forward flight.

Second, as discussed in Section IV, our long-term goal is to include arbitrary rotor-body combinations and to calculate the aerodynamic interactions between rotating and non-rotating components. This has influenced our preliminary choice of grid topologies to insure this capability. Also, the topology chosen readily lends itself to a straightforward treatment of multiple-blade rotors and to wake-capturing in hover.

A. Results for Nonlifting Hover and Forward Flight

In the absence of lift on the rotor, the vexing problems of the wake are avoided. Figure (12) from Reference (21) shows two Euler solutions for a six-blade rotor in hover, at transonic tip speeds just below and just above the conditions for the onset of the aeroacoustic phenomenon known as delocalization [23]. The effects of blade-to-blade thickness interference were studied for two, four, and six blades.

Figure (13) shows the instantaneous pressure distributions computed with the same Euler solver for a particularly difficult forward flight case. The results, particularly the unsteady shock-wave formation and its rapid decay in the second quadrant, agree better with the experiment than in any previous numerical study.

B. Results for a Lifting Rotor in Hover

As reported in Reference (22), C.L. Chen recently computed the flow for a lifting two-bladed rotor with no wake modeling. Instead, the vortical wake was captured approximately by his Euler code through the use of a periodic boundary condition between grid zones containing the blades. Figure (14) summarizes his solution and shows the importance of this periodic boundary condition. After several revolutions of the blades, solution converged with the several individual tip vortices more or less merged into one vortical blob that induced approximately the correct downwash near the blade tips. Consequently, the agreement with experimental data in the tip region is quite good. However, the accuracy of the solution deteriorated in the inboard region, where the induced downwash was too small and the resulting blade-element lift was too large. Much more work remains to be done on this problem, but these preliminary results with no wake modeling are encouraging.

Chen and McCroskey [22] calculated the flow past this hovering rotor for $M_T = 0.44$, 0.794, and 0.877, with the best agreement with the experimental results occurring at the two lower Mach numbers. The results at $M_T = 0.877$ were in close agreement with the Euler solution of Agarwal and Deese [24], which was obtained using a simple wake model. Both Euler solutions produced a strong shock wave that was significantly farther rearward than in the experiment. However, Srinivasan and McCroskey [17] computed this same case using a Navier-Stokes code and Agarwal's wake model, and they found almost perfect agreement with the experimental shock wave position and strength, as shown in Figure (15). It should be mentioned that the three solutions [17,21,24] were obtained with three different codes on three different grids, and differences due to these purely numerical factors cannot be discounted. Nevertheless, these are strong-shock conditions that would be expected to produce significant shock-wave/boundary-layer interaction, and it is our opinion that these comparisons substantiate this speculation. Of course, it remains puzzling that Strawn and Caradonna [25] achieved almost as good agreement with the experimental results using a full-potential code, neglecting viscous effects entirely.

Srinivasan and McCroskey [17] also examined several nonrotating configurations that are sometimes proposed to simulate rotor blade tips in conventional wind tunnels. In this exercise, the spanwise distribution of circulation of the hovering rotor described in the preceding paragraph was approximately matched in two nonrotating simulations: 1) by a linear variation in the spanwise Mach number distribution with a constant blade pitch angle of 4.2°, and 2) by a linear spanwise twist distribution at a constant Mach number of 0.877.

The computations for this strong transonic case are shown in Figure (16). These results show that the variable Mach-number simulation is a good representation of the rotating blade in the tip region. However, the more conventional approximation of variable twist at constant Mach number is not even qualitatively correct anywhere on the blade, under these transonic conditions. The differences inboard are to be expected, since the flow on the rotating blade is completely subsonic there, but it is rather surprising that the nonrotating solution is so different in the tip region. On the other hand, it should be noted that this overall conclusion is limited to transonic conditions; both types of nonrotating simulations agreed well with the rotating solution and with the rotor test data at subsonic tip speeds. In any case, this investigation was a striking demonstration of the flexibility and power of CFD to gain physical insight, study novel ideas, and examine various possibilities that might be difficult or even impossible to set up in physical experiments.

IV. ROTOR - BODY COMBINATIONS

The aerodynamic interaction between rotating and nonrotating components is an important distinguishing feature of rotorcraft that produces considerable additional complications for CFD analyses. This is because the inner boundaries of the computational grids of most codes today conform to and move with the surface of the respective components, and the flow field around each object is contained within the interior of one or more of these grid zones. The art of transferring numerical data accurately across grid zone boundaries as the solution develops is by no means mature for fixed-wing configurations, and it is in its infancy for problems with relative motion between the various grid zones around the individual components. Therefore, much creative work remains to be done in this aspect of CFD for rotorcraft applications.

Fortunately, other engineering problems have spawned research activities that are relevant here; for example, rotor-stator interactions in turbomachinery and store separations of fixed-wing aircraft. Representative two-dimensional examples of grid topologies and data-transfer strategies for these two problems [26,27] are indicated schematically in Figure (17). The patched-grid method on the left, in which a single plane separates the two zones, seems intuitively attractive for many rotorcraft configurations. However, the overset-grid method on the right, in which one grid system sweeps through the interior of another, may allow more flexibility for the grid topologies for the individual components in some cases. The idea of overset grids is also attractive for capturing tip vortices that move relative to the blade-fixed grid zones, such as for the wakes in forward flight. Both strategies are being studied at the Ames Research Center and elsewhere.

The research of C.L. Chen [21,22] described in Section III has been tailored to the patched-grid concept, particularly regarding the grid topologies. This is illustrated in Figure (18). Here the "outer" cylindrical grid zones have identical circumferential grid lines at the interface, thereby minimizing the complexities at the moving boundary between the rotating and nonrotating outer zones. "Inner" zones that are more appropriate for the local flow past the actual blades, fuselage, wing, etc. can be imbedded within each cylindrical block. For example, a C-mesh topology for the viscous region on the blade and in the near wake is shown. In such cases, the interfaces between the inner and outer zones are likely to be more complex; but as they are not in relative motion, they can be treated by conventional zonal methods [28].

The implementation of this overall strategy has not yet been accomplished. The work described in References (21) and (22) is a limited first step; the Euler equations have been solved in the outer rotating zone only. However, the inclusion of inner, viscous-layer and vortical-wake zones, as well as nonrotating components, has high priority in our research plans.

V. LIMITATIONS AND CRITICAL ISSUES

Although the growth trends of supercomputer technology and overall CFD capability are well established, it will be some time before the Euler and Navier-Stokes methodologies outlined in Sections III and IV will offer a significant competitive edge over simpler approaches. As noted in the Introduction, the two overriding issues today are, first, that the CPU time and memory are excessive and, second, that the accuracy of the results is often questionable. Major improvements in supercomputer hardware and software, numerical grids and algorithms, turbulence models, and vortical wake-capturing capabilities are urgently required, as briefly discussed below. Other essential capabilities which are not addressed in this paper include improved graphical displays, better pre- and post-processing of the enormous data structures, and aerodynamic/structural coupling. Finally, there is an acute shortage of both structural-dynamics engineers and technical managers who understand CFD.

A. Computer Speed and Memory Requirements

In addition to crucial three-dimensional effects, unsteadiness is an important, complicating aspect of flows past rotor blades. This feature is shared by the fixed-wing aeroelasticity and turbomachinery communities. These groups have helped to extend the methodologies of quasi-steady aerodynamics, generally a few years after they were first introduced. However, existing time-accurate codes tend to have stability restrictions that restrict the time steps to values which are much smaller than necessary for accurate resolution of the relevant unsteady physics of the flow. As discussed in Reference (29), such restrictions increase the CPU time by an order of magnitude or more for Euler and Navier-Stokes calculations; and therefore, they must be overcome before complete rotor flow fields can be computed on a routine basis.

It is instructive to examine the factors that determine the CPU time for various CFD approaches, which can be estimated from the following formula from Reference (29):

$$CPU = A \times W_{GT} \times N_G \times N_T/FLOPS \qquad (1)$$

where: A = "numerical inefficiency" factor, >1.0

W_{GT} = number of floating-point operations per grid point per time step

N_G = no. grid points

N_T = no. time steps

= (no. ref. lengths/revolution) × (no. rev.)/$\Delta\tau$

$\Delta\tau$ = nondimn. time step

FLOPS = no. floating-point arithmetic operations per unit time

The efficiency factor, A, is introduced to emphasize that the code may not take full advantage of the computer being used. In practice, it is a function of the programming efficiency, the degree of vectorization, the coupling between the grid and the solution algorithm, user experience, etc. Ideally, its value should approach unity; but especially with the advent of supercomputers with novel architecture, it could be much larger.

The number of arithmetic operations per grid point per time step, W_{GT}, is a strong function of the numerical method; that is, of the flow equations, the boundary conditions, the solution algorithm, and the grid. The quantity N_G represents the number of grid points for a finite-difference method, the number of elements for a finite-element method, or the number of panels for a panel method. Consequently, $W_{GT}\, N_G$ represents the number of arithmetic operations that must be performed at each iteration or time step, although in some instances with panel methods, $N_G \log N_G$ or N_G^m is a more accurate representation than N_G.

Ideally for rotorcraft applications, the total number of time steps, N_T, would simply be the number of time steps per blade revolution multiplied by the number of revolutions needed to determine the aerodynamic characteristics. However, many nonlinear aerodynamics codes have stability or accuracy limits that are determined by a nondimensional time step, $\Delta\tau = U\Delta t/L$. Thus the maximum permissible value of $\Delta\tau$ typically depends upon the complexity of the problem, the algorithm, the grid, and the desired accuracy.

Finally, the computing speed, FLOPS, is a function of the computer clock speed and architecture, the data management techniques of the code, the memory requirements (in-core or external memory), and the solution algorithm. Thus it is clear that many different factors determine the CPU time, and hence the cost, of an aerodynamic calculation.

In Reference (29), a detailed analysis was made of the solution times that would be required to perform fixed-wing flutter analyses, using a wide range of contemporary time-accurate methods on modern supercomputers. This information is at least qualitatively relevant to rotorcraft applications. Table 1, reproduced from that investigation, shows a breakdown of the factors in Equation 1 for a wing of moderate complexity undergoing 3 cycles of oscillatory motion and 25 chord-lengths of travel per cycle, running on a computer with a nominal sustained rate of 80 million

Table 1. Computational Requirements for Various 3-D Time-Accurate Methods

Flow model	W_{GT}	N_G	$\Delta\tau$	CPU, minutes	Memory, million words	Notes
Nonlinear panel	$W_{GT} \times N_G^m \approx 2 \times 10^6$		0.05	60	2.0	*a,c*
Small disturbance	100	10^5	0.06	8	0.6	*b,d*
Full potential	600	10^5	0.04	23	2.0	*d,e*
Full potential and integral B.L.	630	10^5	0.02	50	2.0	*d,e,f*
Euler	3000	10^5	0.01	450	3.0	*d,e*
Euler and finite difference B.L.	2000	2×10^5	0.01	600	6.0	*d,e,g*
Thin-layer Navier-Stokes	3600	10^6	0.005	11,000	30	*d,e*
Reynolds-averaged Navier-Stokes	4500	2×10^6	0.004	35,000	60	*d,e*

Notes: *a*. $A = 2$
b. $A = 3$
c. $\Delta\tau$ for time accuracy
d. $\Delta\tau$ for stability limitations
e. W_{GT} includes 100 for grid generation
f. 5% increase in W_{GT} for boundary layer
g. $W_{GT} = 500$ in viscous layer, 3000 in inviscid region

floating-point operations per second. Several of the additional assumptions are noted in the footnote to Table 1. The solution-time requirements are compared graphically in Figure (19). Of course, all of these results are very approximate, accurate to one significant figure at best.

It should be mentioned that a time step limit of $\Delta\tau = 0.05$ was assumed as a rather subjective estimate of what is required to resolve accurately unsteady transonic effects on a wing, including significant shock wave motion. Also, the estimates of the stability limitations on $\Delta\tau$ are very approximate, and the numbers given are intended more to give a sense of the relative values of the various methods, than the absolute values. The important point is that, the more sophisticated the method, the more severe is the stability restriction on $\Delta\tau$ for highly nonlinear problems.

Estimates for rotorcraft applications are more difficult to determine. For two revolutions in forward flight of a two-blade rotor with blades of aspect ratio 10 above a simple fuselage, and for a typical implicit thin-layer Navier-Stokes code with algebraic eddy-viscosity modeling of turbulence, the following values might be (optimistically) appropriate:

$$A = 1.5$$

$$W_{GT} = 4000$$

$$N_G = 10^6$$

$$\Delta\tau = 0.025$$

$$N_T = 5000$$

Then Equation 1 yields CPU $\approx$ 80 hours for a 100-megaflop supercomputer, and approximately 30 million words of memory would be required for this problem. At the present time, everyone has difficulty achieving even this level of performance on simpler problems. These results illustrate the magnitude of the challenge. Only the largest machines, such as the Cray 2, have adequate main memory, and its speed is insufficient for more than a few showcase solutions. It seems clear that the megaflop rates of even the next generation of supercomputers will be a limiting factor for practical CFD computations of helicopter aerodynamics.

B. Improvements in Grid Generation

An essential step in solving rotorcraft problems using computational aerodynamics is the generation of a suitable grid. Although a significant advantage of some panel methods and of the transonic small-disturbance method is that the appropriate boundary conditions can be applied on nonaligned grids, most of the more sophisticated methods rely on body-conforming grids. The zonal modeling, or multi-block, concept appears especially attractive for complex configurations. This is one of the most rapidly growing areas of CFD, and there have been several meetings in recent years devoted exclusively to grid generation. The recent surveys by Thompson [28,30] are particularly noteworthy, and another major international conference on grid generation will be held in Miami in December 1988.

For computational efficiency, algebraic and hyperbolic schemes seem better suited to the classes of unsteady problems that require regenerating the grid at each time step. High efficiency is necessary to exploit the concept of adapting the grid to some feature of the solution, such as clustering grid points in regions of large gradients, so as to obtain the maximum accuracy with the least number of grid points. As noted earlier in connection with Figure (4), dynamic solution-adaptive grids have been successfully used in two dimensions, and comparable capability will be essential for computing three-dimensional vortical wakes accurately. We believe that the grid point requirements, and hence the CPU times, can be reduced by a factor of two or more by dynamic grid adaption. However, this will require a significant and specialized effort.

C. Turbulence Models

The simulation of the dynamics of turbulence remains the foremost challenge in fluid mechanics today, and turbulence modeling is probably the weakest link in the chain of computational aerodynamics technology. For conventional helicopters, retreating-blade stall is probably the practical aerodynamic problem most impacted by turbulence modeling, while for tilt-rotor configurations, accurate predictions of the separated flow associated with rotor-wing interactions in hover are next to impossible today.

Because of the limitations of the computational power available in the foreseeable future, turbulence modeling throughout the aeronautics community has taken the approach of single-point closure of the Reynolds-averaged Navier-Stokes equations, and no single turbulence model exists that can be applied to a wide variety of flows. An overview of the range of possible turbulence models, ranging from essentially no modeling at all, to the hypothetical full simulaton of turbulence, is given in Table 2. As no universal turbulence

Table 2. Summary of Turbulence Models

Model	Physical Generality	Numerical Compatibility	Remarks
Viscous wedge	Very low	Very high	Shock-B.L. interaction
Integral B.L.	Low	High	Very good when highly tuned
Eddy viscosity			
Zero equation	Low	High	Needs more tuning
.			
.			
.			
2-equation	Medium	Low to high	$\Delta W_{GT} \approx 20\%$, $\Delta\tau$ = ?
Reynolds stress equations	High	Low	3-D separation?
Large eddy simulation	Very high	Low (?)	Guidelines for above models
Complete simulation	Complete	nth generation supercomputers	

Are the more complex models any better?

model exists, most contemporary researchers are focusing their attention on creating a catalog of models based on fundamental building-block experiments. Most of these models are being carefully tested computationally to determine their capabilities and limitations.

As shown in Table 1 and Figure (19), enormous computer resources are required to solve time-dependent problems with finite-difference simulations of the Reynolds-averaged Navier-Stokes equations, even with simple turbulence models. Furthermore, even those solutions that have been published for steady flows have used grids whose fine spacing is limited to the single direction nearly normal to the body, and hence fall within the spirit of the thin-layer approximation. This resulting computational process qualitatively simulates separated flows and flows with large-scale unsteady behaviors, but the accuracy of such simulations is still controversial.

At the Ames Research Center we are well aware of their limitations and we are always looking for something better, but we have only used the zero-equation eddy-viscosity model of Baldwin and Lomax [8] and the thin-layer approximation for rotorcraft problems up to now. Holst [10] and Kaynak and Flores [20] showed a few examples in which improved turbulence models had some influence on the shock wave location on airfoils and wings, but our transonic airfoil studies [6] did not indicate a serious deficiency in the Baldwin-Lomax model for rotorcraft applications below stall. The main concern that we have uncovered so far is in the details of the tip separation and early stages of tip-vortex formation. As discussed in Section II in connection with Figure (10), we not have been able to reproduce quantitatively the measured local suction peaks outboard of $Y/B \approx 0.96$.

In the larger arena of applied aerodynamic computations for fixed-wing aircraft, however, more sophisticated models are beginning to be used that are "tuned" in conjunction with the numerical procedure for a specific class of flow problems. As a result, validations by means of experimental comparisons are mandatory, and confidence in the absolute values of the numerical predictions remains low for flows with significant separation, including stall, of course.

In principle, the more general models in Table 2 should cover a wider range of flows with less "tuning," but helicopter engineers will probably remain suspicious of the idea that "bigger is better." Nevertheless, in some rotorcraft problems which involve three-dimensional flow separation, the two-equation eddy-viscosity models may turn out to be the best compromise between simplicity and generality. Some of the models in this category lead to stiff equations, and this raises again the problem of restrictive values of $\Delta\tau$ in Equation 1.

From this brief overview, it is clear that turbulence modeling will remain a primary pacing item in computational aerodynamics over the next decade, for both fixed-wing and rotorcraft applications.

D. Modeling Vortical Flows

Whereas the treatment of shock waves in transonic flow was a major focal point for computational aerodynamics in the 1970's, compressible flow fields with embedded regions of concentrated vorticity are gaining prominence today. Again, the key role is being played by the fixed-wing community. Some interest comes from adverse effects of trailing wing-tip vortices, but the vortical flows shed from sharp leading edges of tactical aircraft at high angles of attack are the principal driver.

The helicoidal vortical wakes of rotor blades have a much larger influence in hover and at low forward speeds than do the trailing vortices of fixed-wing aircraft. These wakes are complex in geometry and structure, and treating them accurately and efficiently is one of the greatest challenges in helicopter aerodynamics today. Possible special treatments include 1) coupling some form of wake modeling with the finite-difference computations, as in Reference (5), 2) three-dimensional extensions of the prescribed-vortex or split-potential methods discussed in Section II, as in References (25), (31), and (32), 3) adapting a refined computational grid to the concentrated vortical regions as they are being computed, 4) developing new vortex-preserving schemes that reduce the inherent numerical dissipation in current codes which rely on vortex capturing, as in Reference (13), or 5) combinations of these.

The flow fields of rotors are inherently vortical and three-dimensional, and three-dimensional computations are inherently expensive. Furthermore, the results of the numerical simulations will generally be no better than the predictions of the vortical wake structure. Therefore, it is extremely important to establish what minimum level of complexity in the governing equations will suffice and to determine the most expeditious way either to model or to capture vortices within the computational domains. This topic is one of the most fruitful and opportune areas for creative research, and it seems destined to be an extremely active one for the next few years.

VI. SUMMARY AND CONCLUSIONS

The rotorcraft industry appears to be entering a new era in which computational fluid dynamics will play an increasingly important role in the design and analysis of advanced aircraft. A major effort is under way at the Ames Research Center to hasten this process and to capitalize on the arrival of future generations of supercomputers.

Airfoil codes are now available that can be used to complement wind tunnel testing. The accuracy of the numerical simulations is comparable to that of airfoil data, but the entire flow field structure can now be readily examined. The cost for the calculation of each condition is probably greater than that of an individual test point, but the geometry of the airfoil can be changed much more easily in the computer than in the wind tunnel.

The major focus at present is on obtaining more accurate and less costly three-dimensional solutions. Although impressive progress has been made, a finite-difference simulation of the complete flow field about a helicopter in forward flight is not currently feasible. The principal CFD limitations are the computer speeds and memory capacities, algorithm and solution methods, grid generation, turbulence models, and vortex modeling. Other important limitations are the inadequate structural and aerodynamic coupling, and a shortage of engineers and scientists who are skilled in both CFD and helicopter aerodynamics and dynamics. Nevertheless, the potential benefits of CFD to the rotorcraft industry are even larger than those that have already accrued to the fixed-wing industry, and the ever-increasing applications of this new tool must continue to be encouraged.

VII. ACKNOWLEDGEMENTS

This paper draws heavily from the research accomplishments of my coworkers, J.D. Baeder, C.L. Chen, E.P. Duque, and G.R. Srinivasan, and their ideas and contributions are gratefully acknowledged. Also, the stimulating discussions and suggestions of M.L. Green, T.L. Holst, J.L. Steger, and R.C. Strawn are deeply appreciated.

VIII. REFERENCES

1. Johnson, W., Helicopter Theory, Princeton University Press, NJ, 1980.

2. Harris, F.D., "Rotary Wing Aerodynamics - Historical Perspective and Important Issues," American Helicopter Society Specialists' Meeting on Aerodynamics and Aeroacoustics, Ft. Worth, Texas, Feb. 1987.

3. McCroskey, W.J., and Baeder, J.D., "Some Recent Advances in Computational Aerodynamics for Helicopter Applications," International Symposium on Computational Fluid Dynamics, Tokyo, Japan, Sept. 1985; also NASA TM 86777, Oct. 1985.

4. Johnson, W., "Recent Developments in Rotary-Wing Aerodynamic Theory, "AIAA Journal, Vol. 24, (8), Aug. 1986.

5. Caradonna, F.X., and Tung, C., "A Review of Current Finite-Difference Codes," American Helicopter Society 42nd Annual Forum, Washington, D.C., June 1986.

6. McCroskey, W.J., Baeder, J.D., and Bridgeman, J.O., "Calculation of Helicopter Airfoil Characteristics for High Tip-Speed Applications," Journal of the American Helicopter Society, Vol. 31, (2), Apr. 1986.

7. Pulliam, T.H. and Steger, J.L., "Recent Improvements in Efficiency, Accuracy, and Convergence for Implicit Approximate Factorization Algorithms," AIAA Paper 85-0360, Reno, Nevada, Jan. 1985.

8. Baldwin, B.S., and Lomax, H., "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257, Huntsville, Ala., Jan. 1978.

9. Holst, T.L., "Viscous Transonic Airfoil Workshop - Compendium of Results," AIAA Paper 87-1460, Honolulu, Hawaii, June 1987.

10. McCroskey, W.J., and Goorjian, P.M., "Interactions of Airfoils with Gusts and Concentrated Vortices in Unsteady Transonic Flow," AIAA Paper 83-1691, Danvers, Mass., July 1983.

11. Srinivasan, G.R., McCroskey, W.J., and Baeder, J.D., "Aerodynamics of Two-Dimensional Blade-Vortex Interaction," AIAA Journal, Vol. 24 (10), Oct. 1986.

12. Baeder, J.D., "Computation of Non-Linear Acoustics in Two-Dimensional Blade-Vortex Interactions," 13th European Rotorcraft Forum, Arles, France, Sept. 1987.

13. Rai, M.M., "Navier-Stokes Simulations of Blade-Vortex Interaction Using High-Order Accurate Upwind Schemes," AIAA Paper 87-0543, Reno, Nev., Jan. 1987.

14. Srinivasan, G.R., and McCroskey, W.J., "Numerical Simulations of Airfoil-Vortex Interactions," Vertica, Vol. 11 (1/2), Jan. 1987.

15. Baeder, J.D., Srinivasan, G.R., and McCroskey, W.J., "Acoustic Propagation using Computational Fluid Dynamics," American Helicopter Society 42nd Annual Forum, Washington, D.C., June 1986.

16. Srinivasan, G.R., McCroskey, W.J., Baeder, J.D., and Edwards, T.A., "Numerical Simulation of Tip Vortices of Wings in Subsonic and Transonic Flows," AIAA Paper 86-1095, Atlanta, Ga., May 1986.

17. Srinivasan, G.R., and McCroskey, W.J., "Navier-Stokes Calculations of Hovering Rotor Flowfields," AIAA Paper 87-2629, Monterey, Cal., Aug. 1987.

18. Mansour, N.N., "Numerical Simulation of the Tip Vortex off a Low-Aspect Ratio Wing at Transonic Speed," AIAA Journal, Vol. 23 (8), Aug. 1985.

19. Kaynak, U., Holst, T., Cantwell, B.J., and Sorenson, R.L., "Numerical Simulation of Transonic Separated Flows over Low-Aspect Ratio Wings," Journal of Aircraft, Vol. 24 (8), Aug. 1987.

20. Kaynak, U., and Flores, J., "Advances in the Computation of Transonic Separated Flows over Finite Wings," AIAA Paper 87-1195, Honolulu, Hawaii, June 1987.

21. Chen, C.L., McCroskey, W.J., and Ying, S.X., "Euler Solution of Multiblade Rotor Flow," 13th European Rotorcraft Forum, Arles, France, Sept. 1987.

22. Chen, C.L., and McCroskey, W.J., "Numerical Simulation of Helicopter Multi-Bladed Rotor Flow," AIAA Paper 88-0046, Reno, Nev., Jan. 1988.

23. Schmitz, F.H., and Yu, Y.H., "Transonic Rotor Noise - Theoretical and Experimental Comparison," Vertica, Vol. 5 (1), Jan. 1981.

24. Agarwal, R.K., and Deese, J.E., "Euler Calculations for Flowfield of a Helicopter Rotor in Hover," Journal of Aircraft, Vol. 24 (4), Apr. 1987.

25. Strawn, R.C., and Caradonna, F.X., "Conservative Full Potential Model for Unsteady Transonic Rotor Flows," AIAA Journal, Vol. 25 (2), Feb. 1987.

26. Rai, M.M., "Unsteady Three-Dimensional Navier-Stokes Simulations of Turbine Rotor-Stator Interaction," AIAA Paper 87-2058, San Diego, Cal., June 1987.

27. Dougherty, F.C., Benek, J.A., and Steger, J.L., "On Applications of Chimera Schemes to Store Separation," NASA TM 88193, Oct. 1985.

28. Thompson, J.F., "Grid Generation Techniques in Computational Fluid Dynamics," AIAA Journal, Vol. 22 (11), Nov. 1984.

29. McCroskey, W.J., Kutler, P., and Bridgeman, J.O., "Status and Prospects for Computational Aerodynamics for Unsteady Transonic Viscous Flows," Paper No. 9, AGARD CP-374, Toulouse, France, Sept. 1984.

30. Thompson, J.F., "Review of the State of the Art of Adaptive Grid Generation," AIAA Paper 84-1606, Snowmass, Col., June 1984.

31. Steinhoff, J., "A Vortex Embedding Method for Free Wake Analysis of Helicopter Rotor Blades in Hover," 13th European Rotorcraft Forum, Arles, France, Sept. 1987.

32. Roberts, T.W., and Murman, E.M., "Euler Solutions for the Flow Around a Hovering Helicopter Rotor," AIAA Paper 86-1784CP, San Diego, Cal., June 1986.

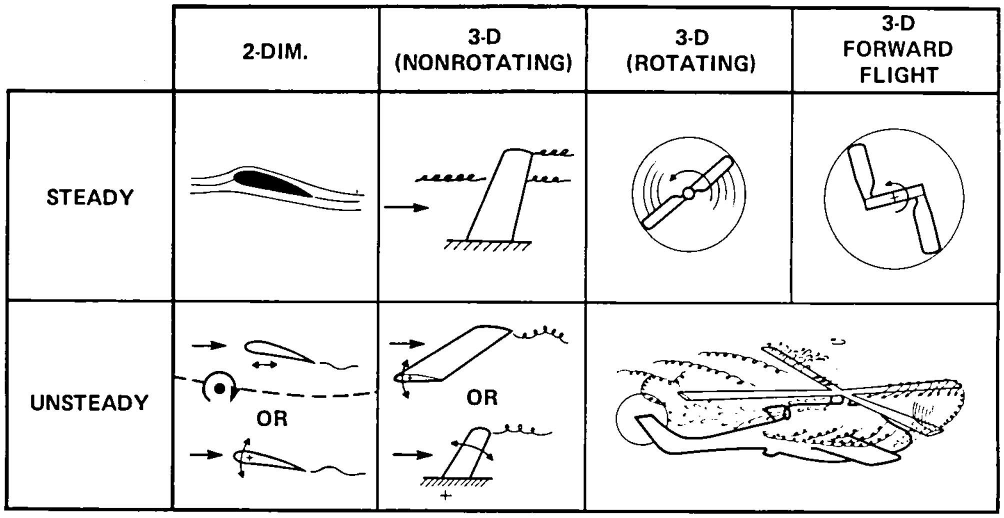


Figure 1. Development of rotor blade aerodynamics from simpler cases

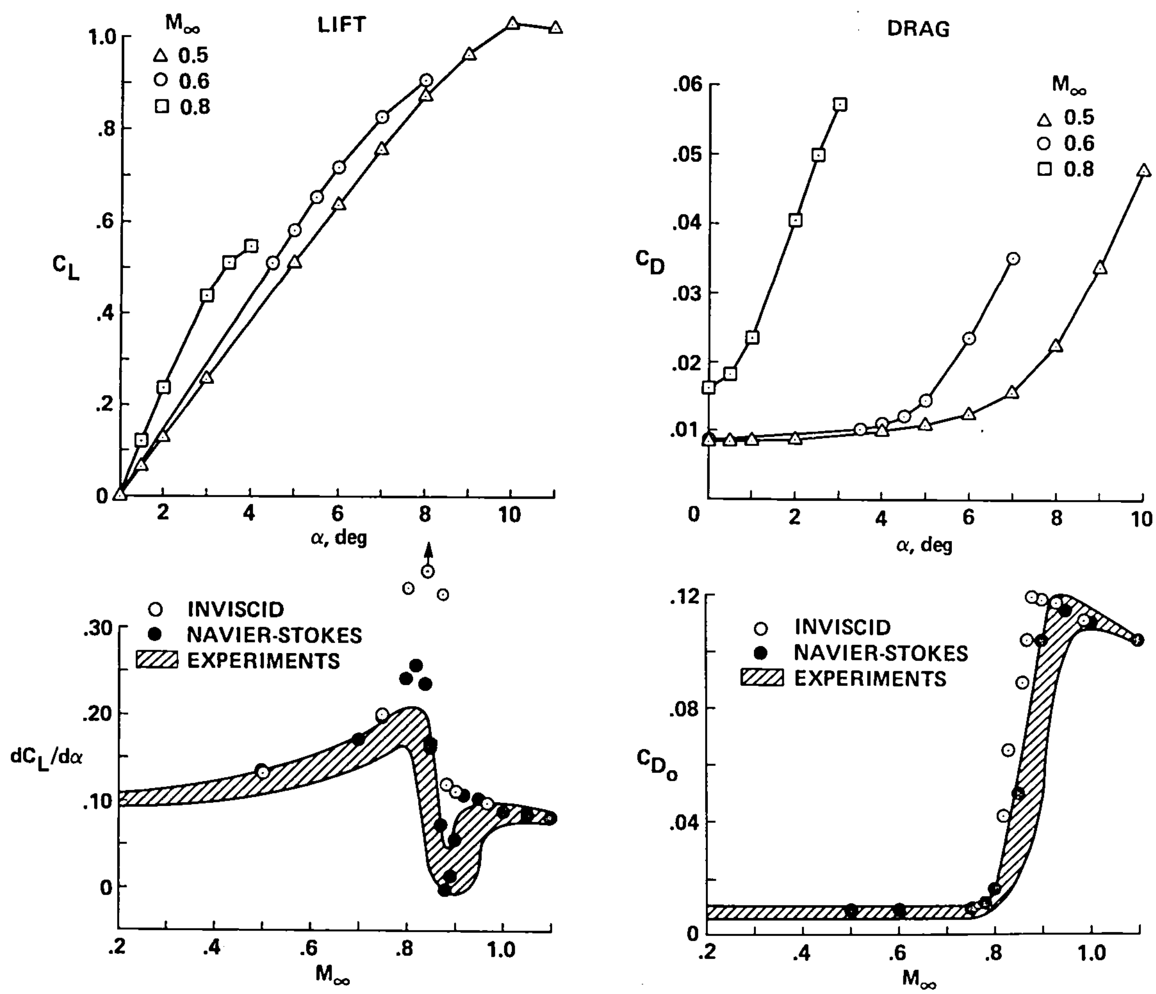


Figure 2. Calculated NACA 0012 airfoil characteristics, Re = 6 × 10^6

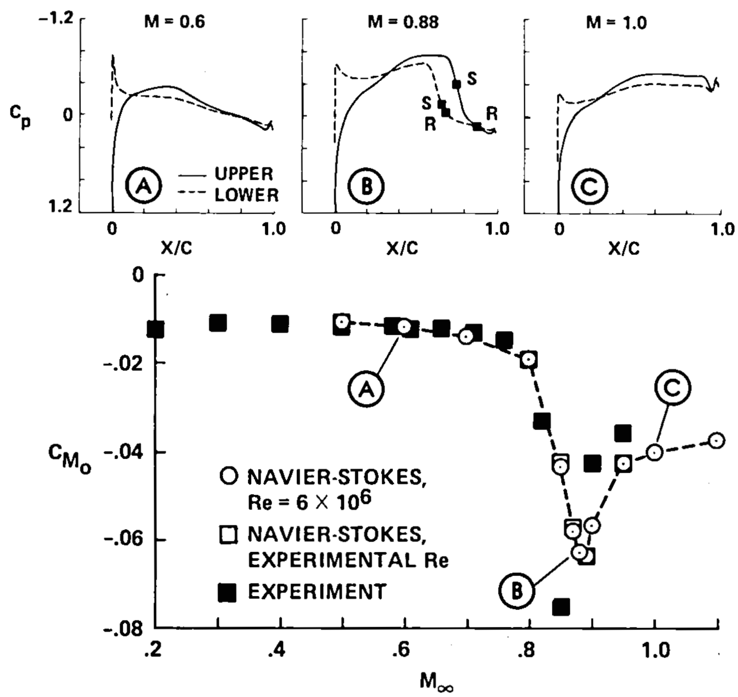


Figure 3. Pitching moment vs. Mach number for the VR-8 airfoil at $C_L = 0$

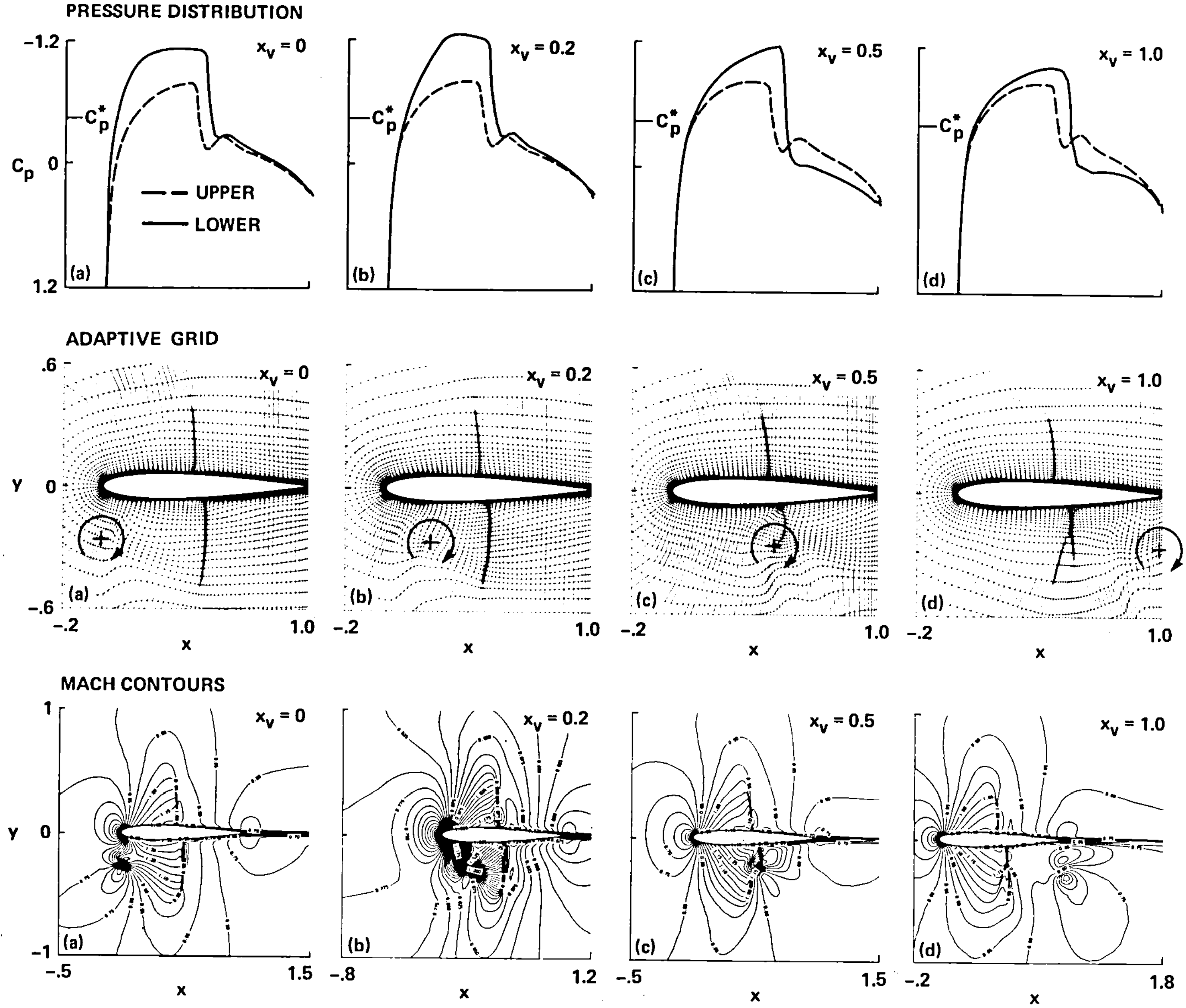


Figure 4. Airfoil-vortex interactions for NACA 0012 airfoil at M = 0.80 (a) instantaneous pressure distributions, (b) solution-adaptive grid (c) instantaneous pressure contours

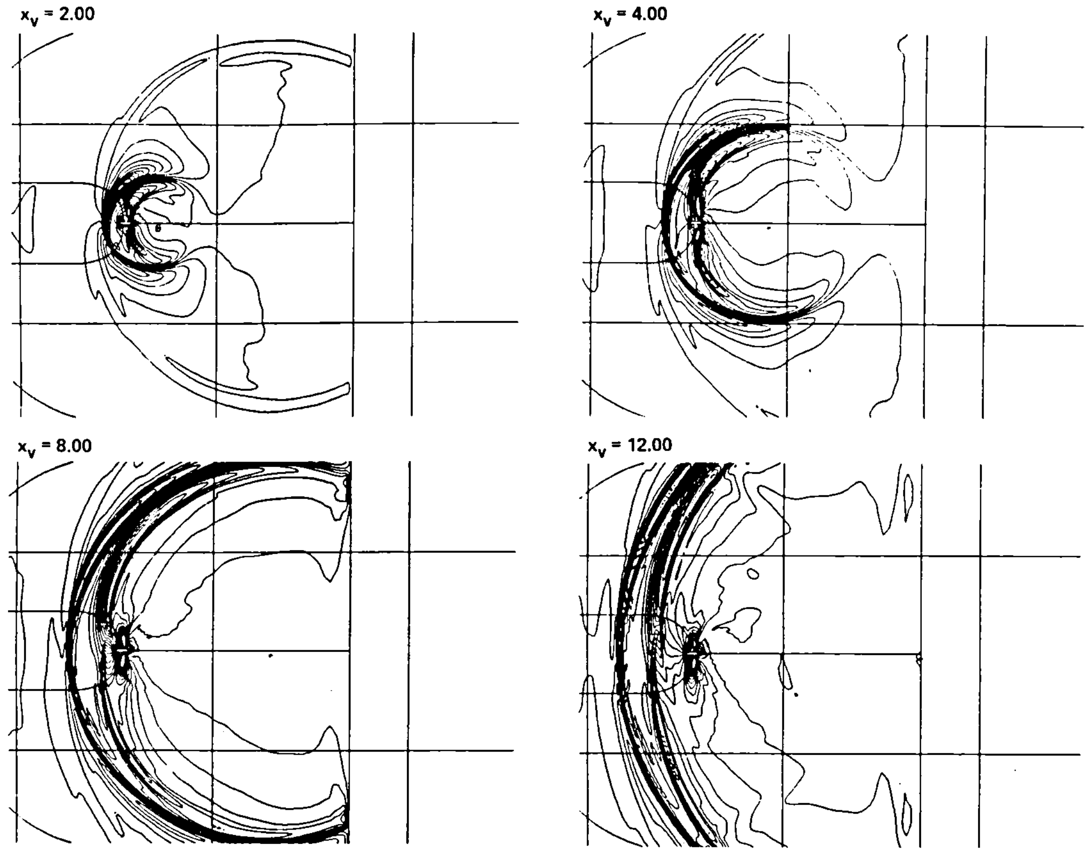


Figure 5. Disturbance-pressure contours for an airfoil-vortex interaction

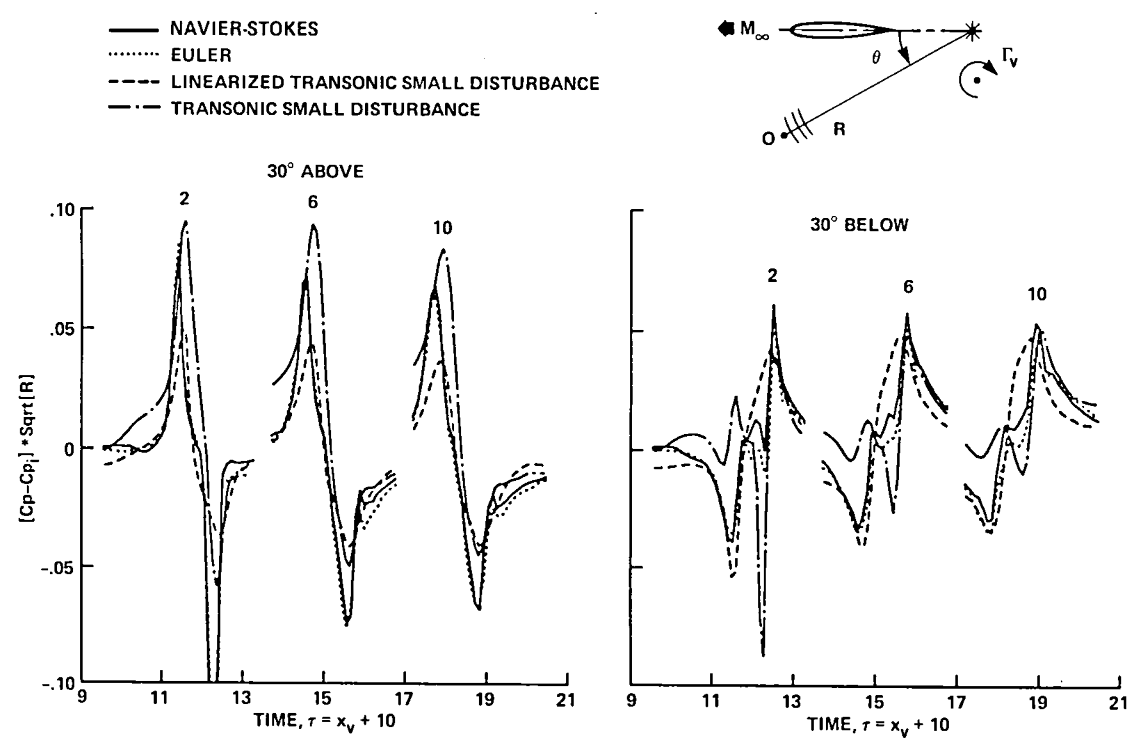


Figure 6. Time-histories of scaled acoustic disturbances by various computational methods

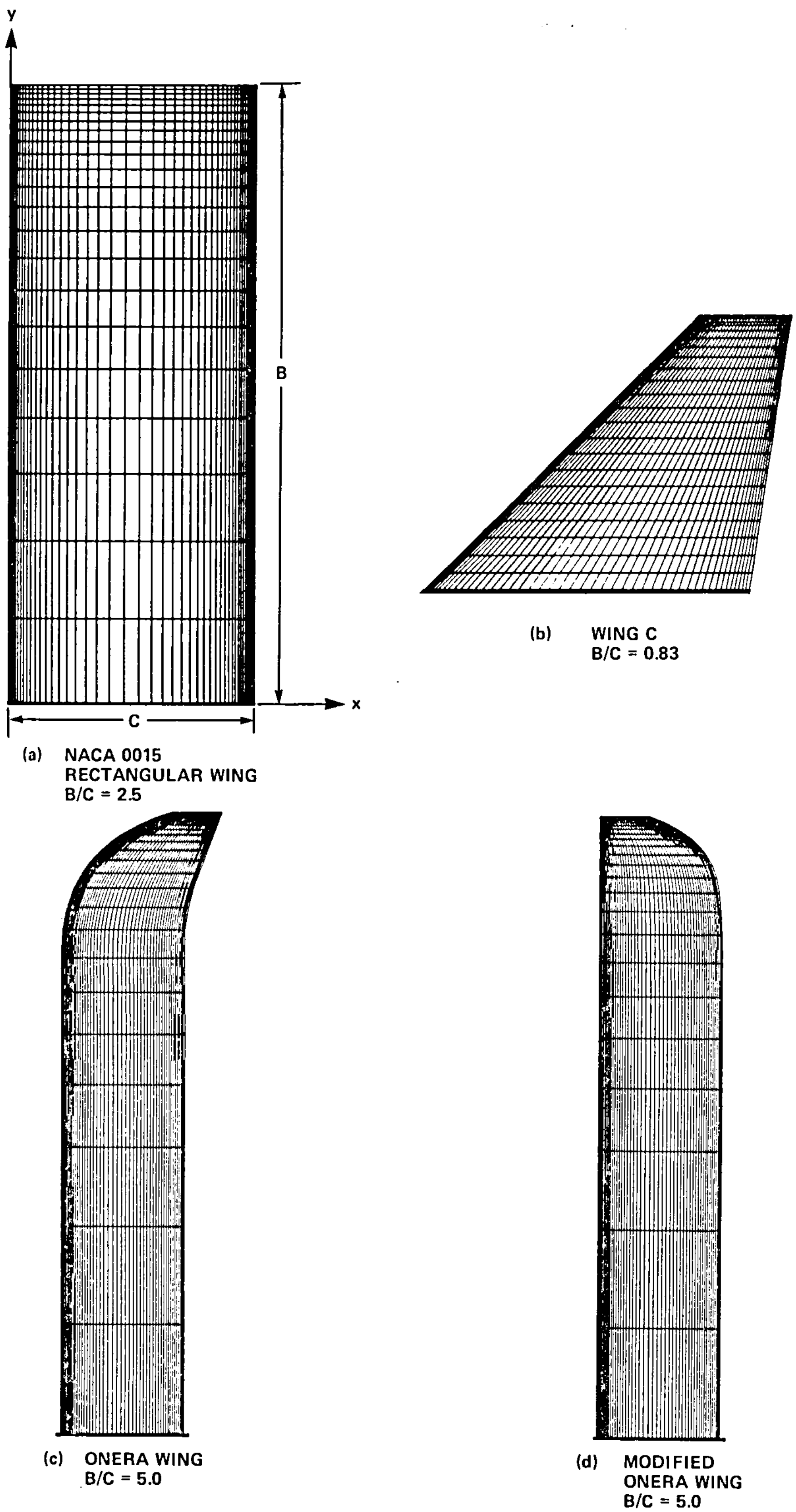


Figure 7. Planform and surface grids of four wings for tip vortex simulations

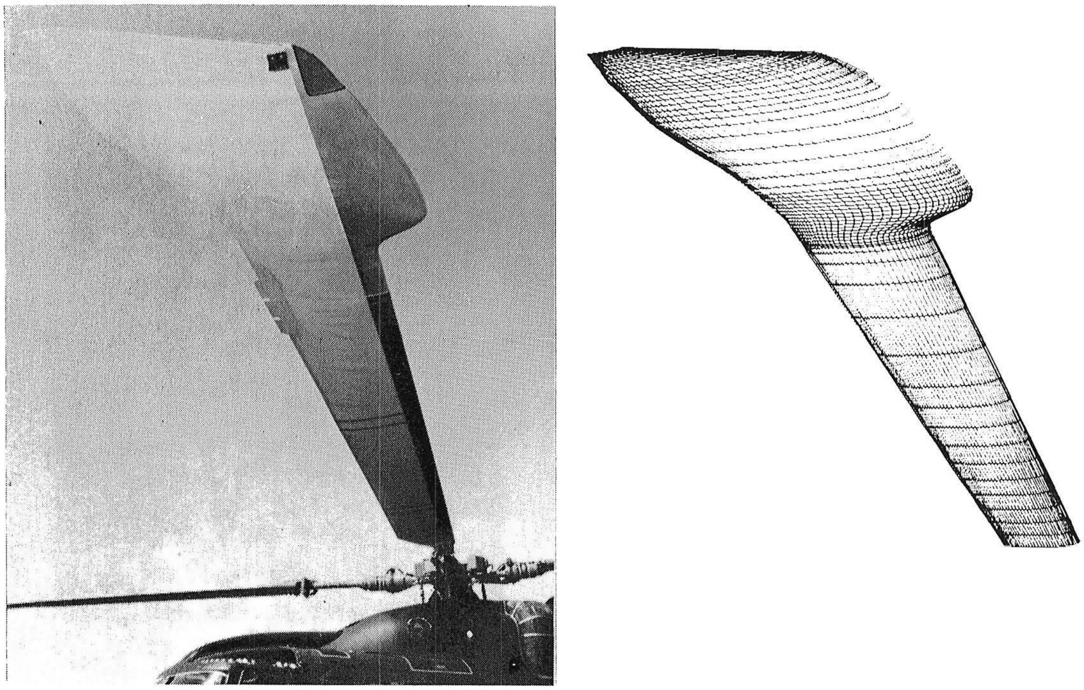


Figure 8. Planform and surface grid for the BERP blade tip

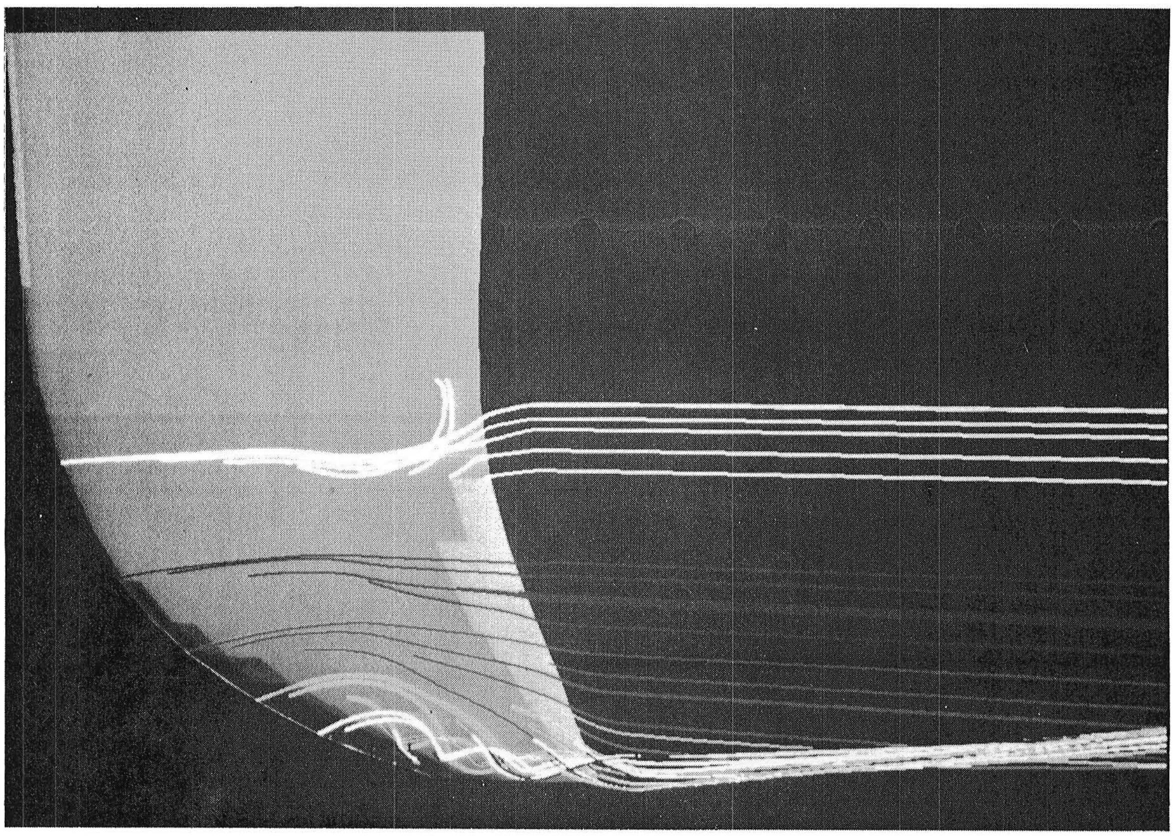


Figure 9. Streamlines and tip vortex of the ONERA swept-tip blade at $M = 0.85$ and $\alpha = 5°$

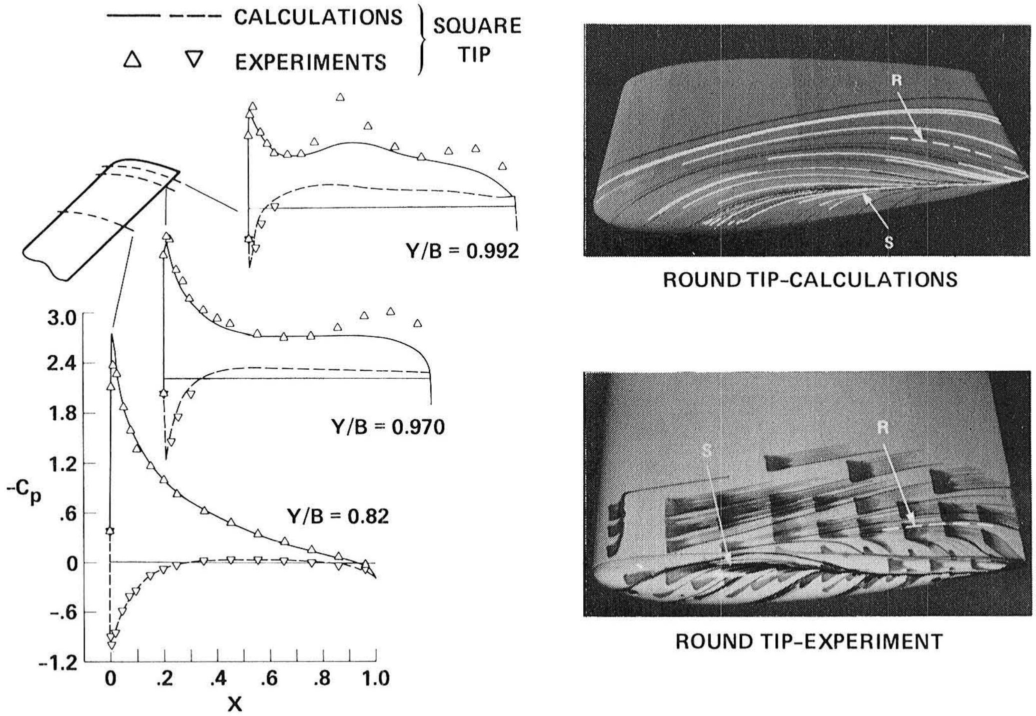


Figure 10. Surface pressure distributions and streamline patterns on an NACA 0015 wing at $M = 0.17$ and $\alpha = 12°$

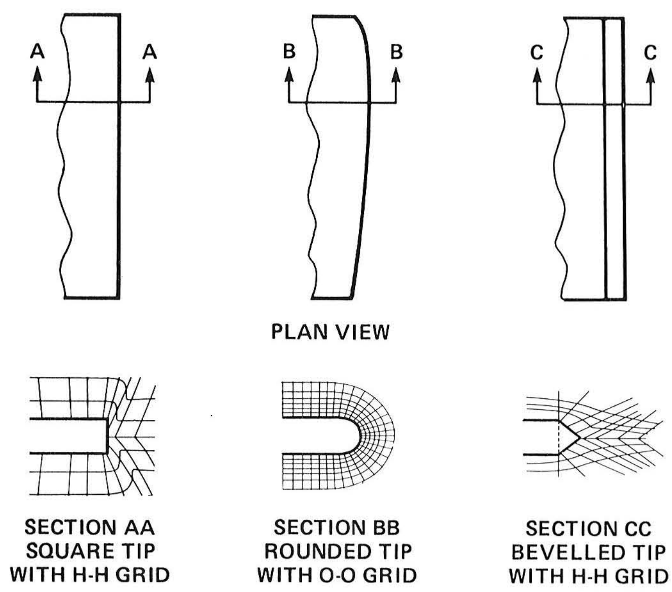


Figure 11. Schematic of tip caps and grid topologies for tip-vortex studies

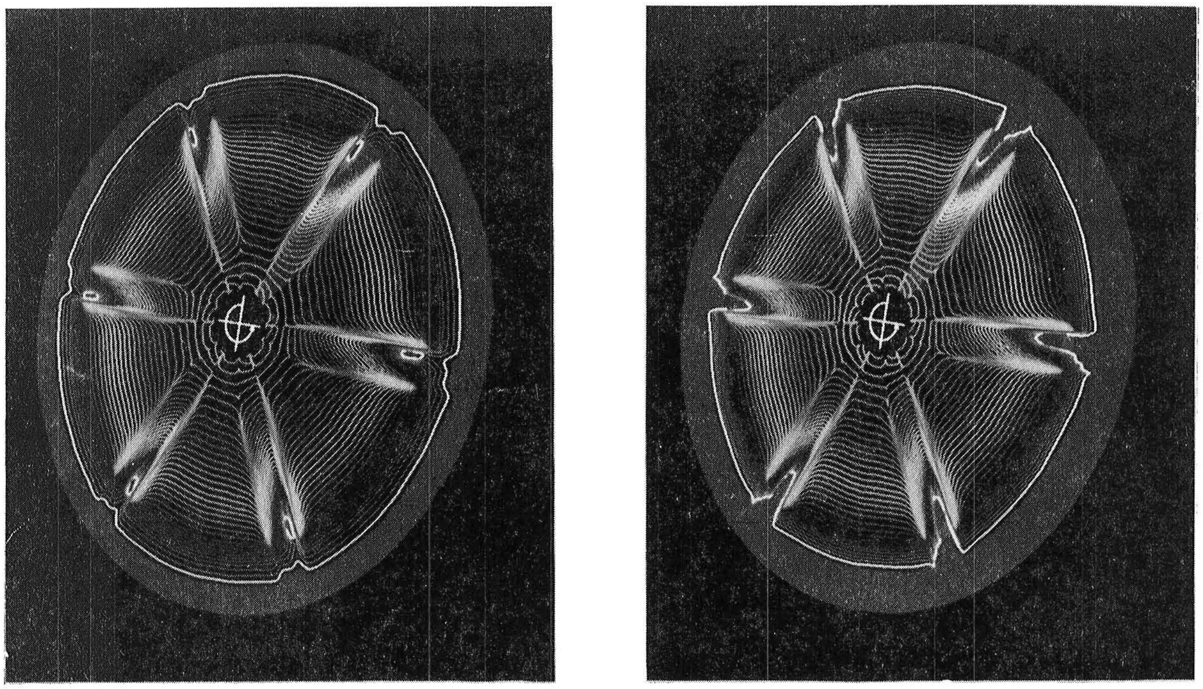


Figure 12. Mach-number contours on a nonlifting rotor. (a) $M_T = 0.85$ (b) $M_T = 0.90$

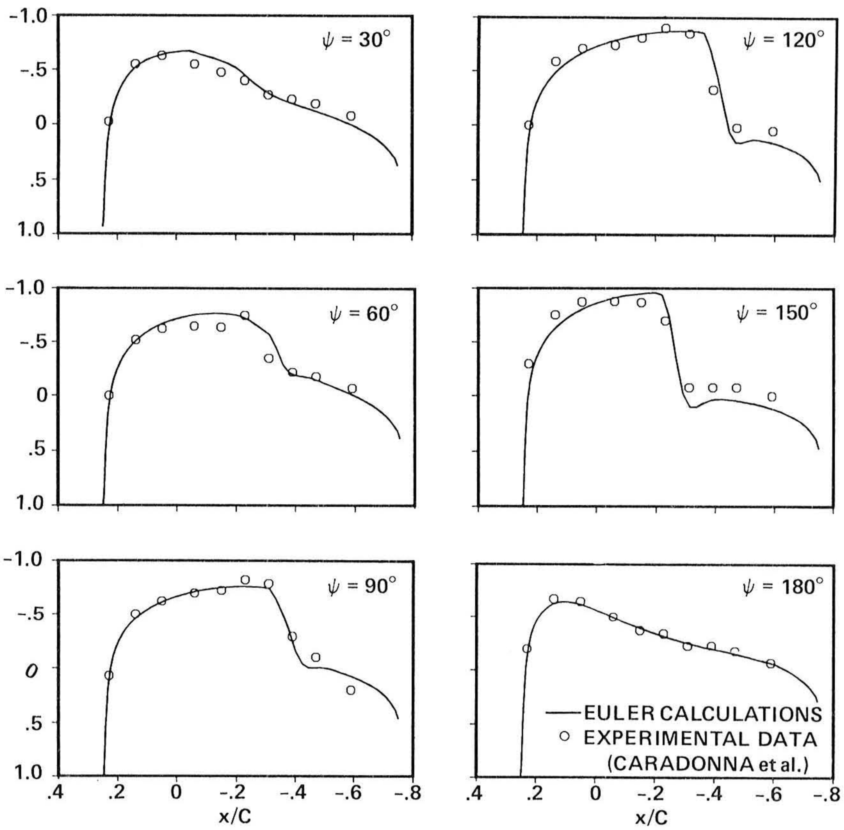
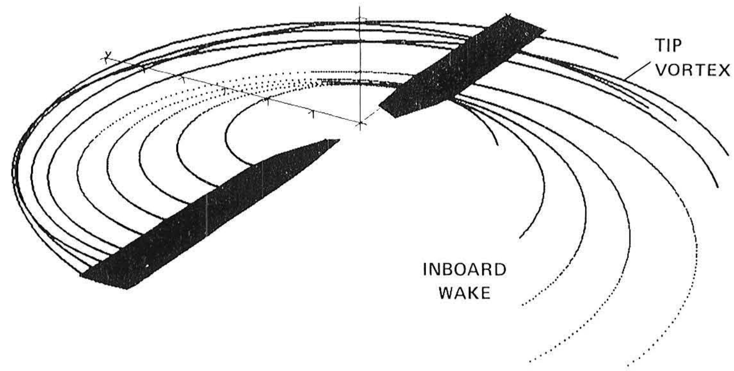


Figure 13. Surface pressure distributions on a two-blade nonlifting rotor in forward flight; $M_T = 0.80$, $\mu = 0.20$, AR = 7



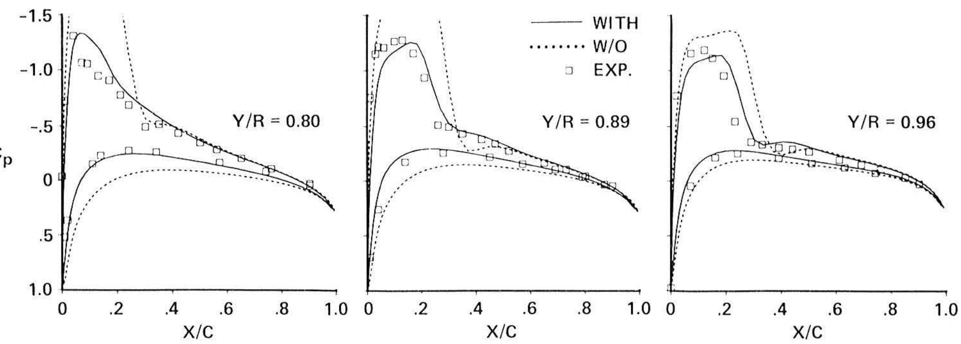
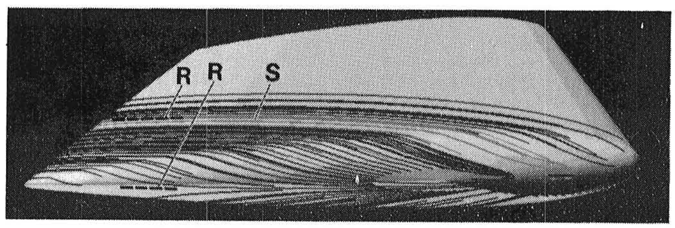


Figure 14. Streamlines and pressure distributions on a two-blade rotor in hover; Euler solution, M_T = 0.794, θ = 8°, AR = 6



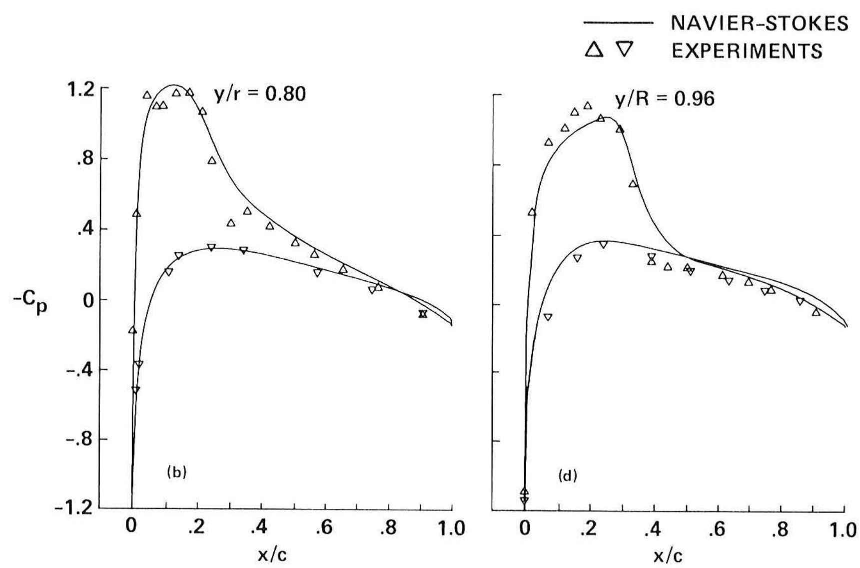


Figure 15. Surface streamlines and pressure distributions on a two-blade rotor in hover; Navier-Stokes solution, M_T = 0.877, θ = 8°, AR = 6

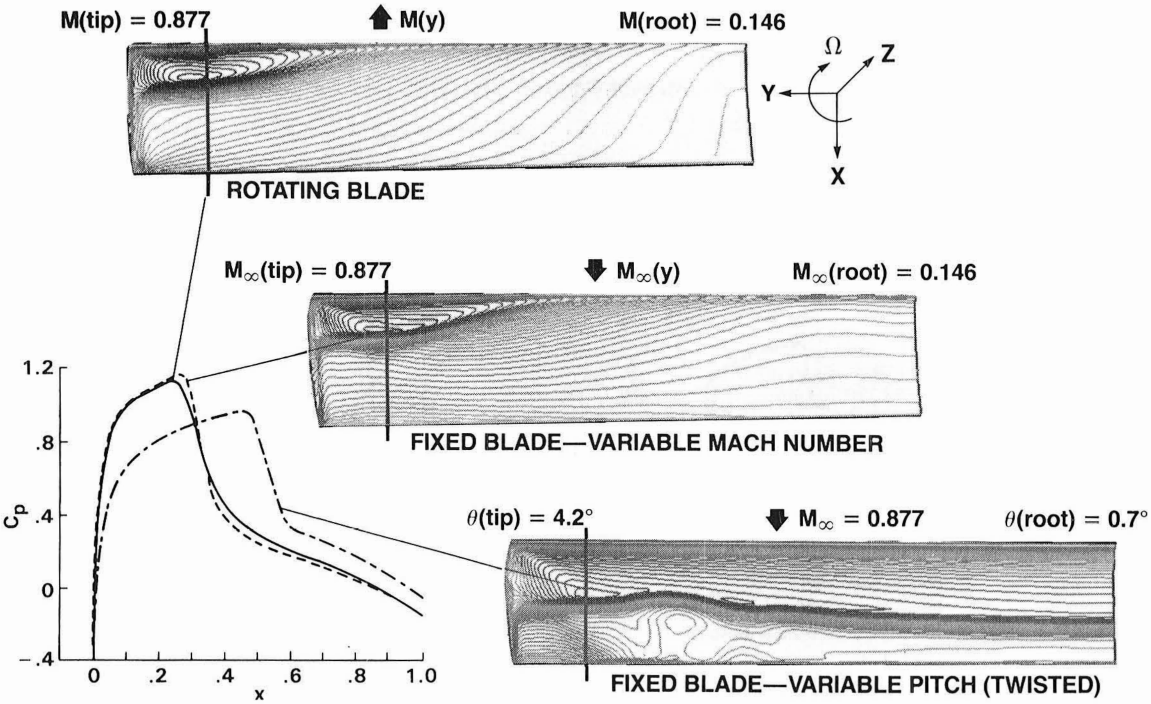


Figure 16. Rotating and nonrotating simulations of a rotor in hover; Navier-Stokes solution, $M_T = 0.877$, $\theta = 8°$, AR = 6

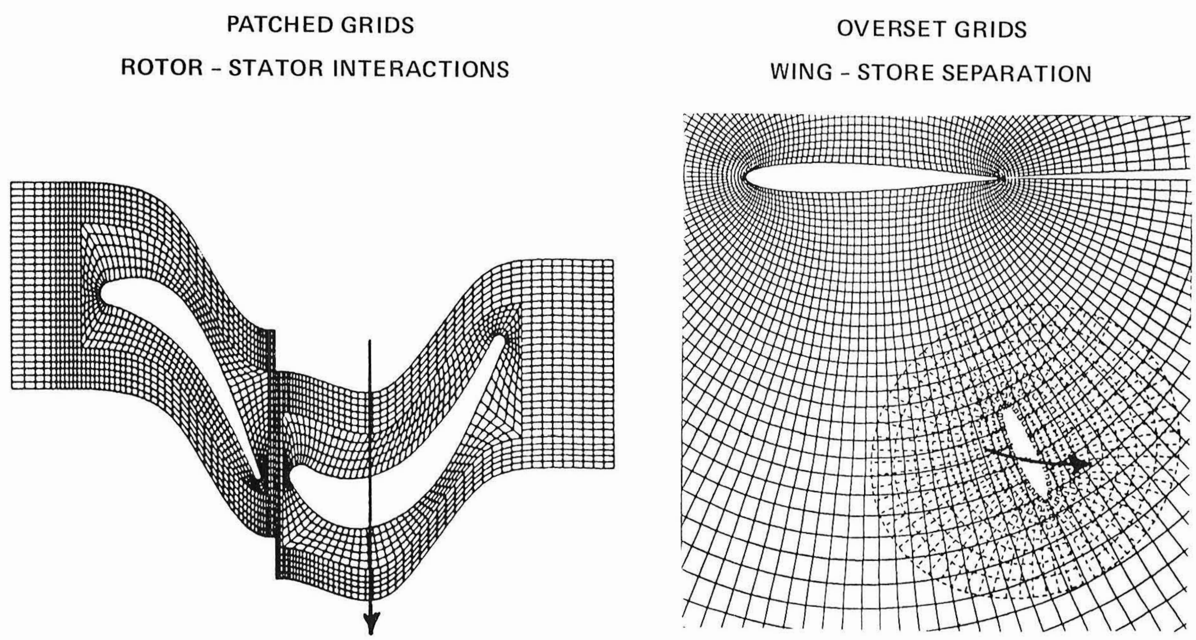


Figure 17. Example of grid interfaces in relative motion

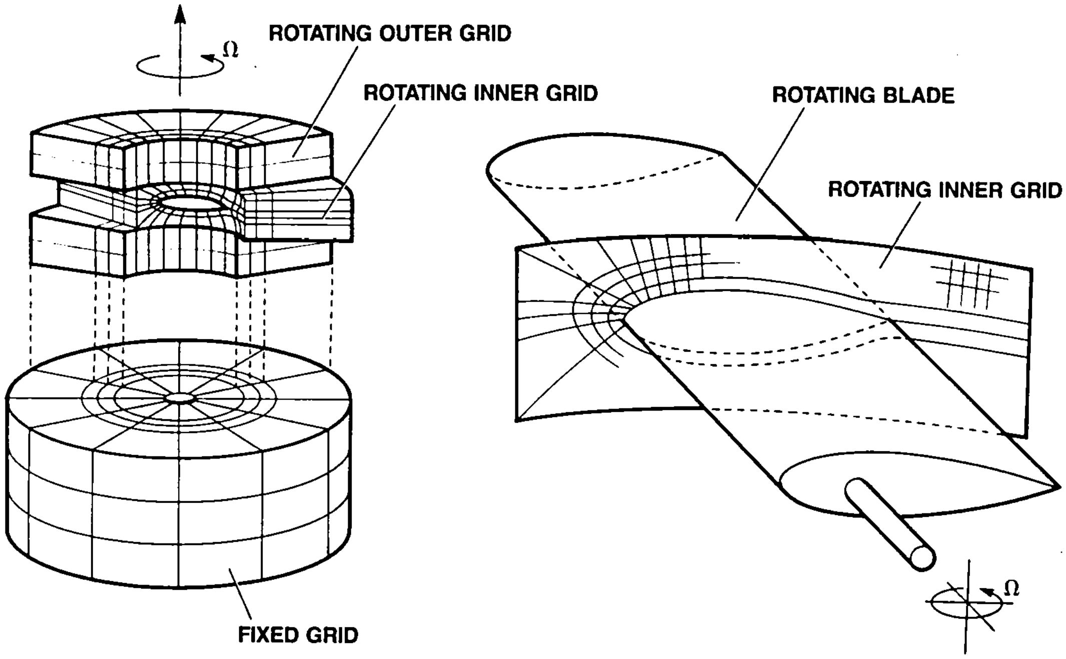


Figure 18. Representative grid topology for rotor-body combinations

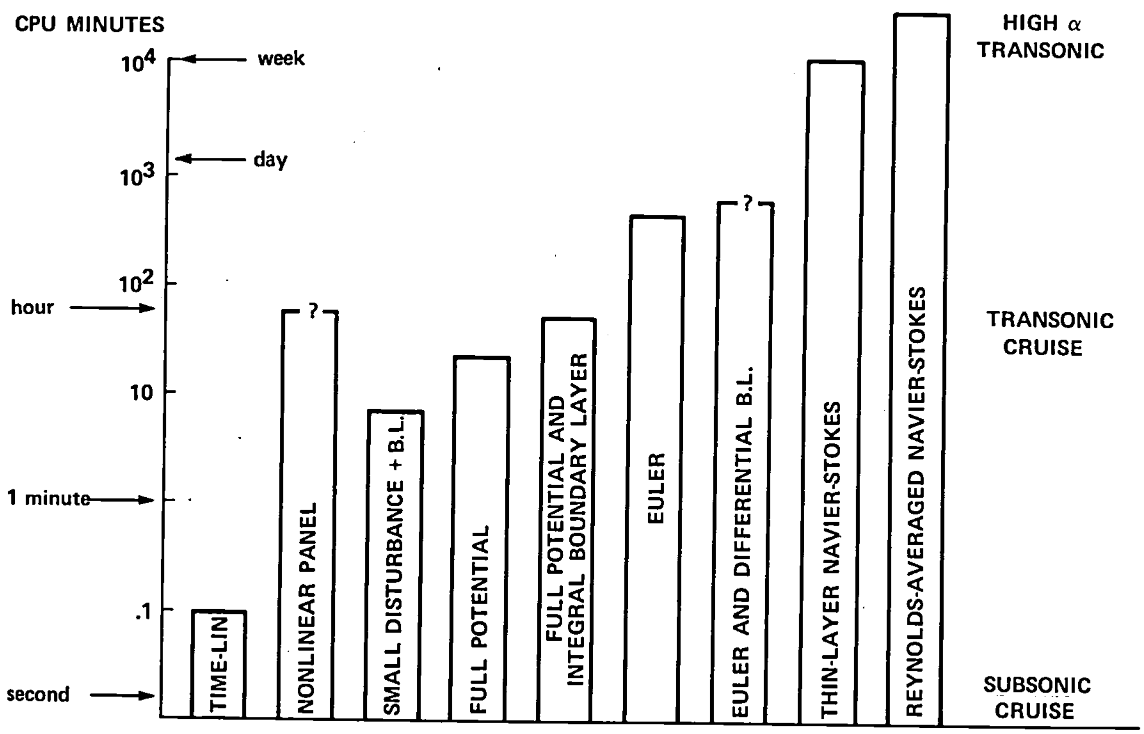


Figure 19. Estimated solution times for an oscillating wing using different CFD methods

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM-100066 USAAVSCOM TR 88-A-001	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle: Some Rotorcraft Applications of Computational Fluid Dynamics		5. Report Date: March 1988
		6. Performing Organization Code
7. Author(s): W. J. McCroskey		8. Performing Organization Report No.: A-88094
		10. Work Unit No.: 992-21-01
9. Performing Organization Name and Address: Ames Research Center and Aeroflightdynamics Directorate, U.S. Army Aviation Research and and Technology Activity, Ames Research Center Moffett Field, CA 94035-1099		11. Contract or Grant No.
		13. Type of Report and Period Covered: Technical Memorandum
12. Sponsoring Agency Name and Address: National Aeronautics and Space Administration Washington, DC 20546-0001 and U.S. Army Aviation Systems Command, St. Louis, MO 63120-1798		14. Sponsoring Agency Code

15. Supplementary Notes

Point of Contact: W. J. McCroskey, Ames Research Center, MS 258-1
Moffett Field, CA 94035 (415) 694-6428 or FTS 464-6428
Paper presented at the Second International Conference on Basic Rotorcraft Research in College Park, Maryland, in February 1988.

16. Abstract

The growing application of computational aerodynamics to nonlinear rotorcraft problems is outlined, with particular emphasis on the development of new methods based on the Euler and thin-layer Navier-Stokes equations. Rotor airfoil characteristics can now be calculated accurately over a wide range of transonic flow conditions. However, unsteady three-dimensional viscous codes remain in the research stage, and a numerical simulation of the complete flow field about a helicopter in forward flight is not currently feasible. Nevertheless, impressive progress is being made in preparation for future supercomputers that will enable meaningful calculations to be made for arbitrary rotorcraft configurations.

17. Key Words (Suggested by Author(s))	18. Distribution Statement
Rotorcraft aerodynamics Navier-Stokes applications	Unclassified-Unlimited Subject Category - 02

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of pages	22. Price
Unclassified	Unclassified	24	A02