The Role of Computational Fluid Dynamics in Aeropropulsion Ground Testing

J M Barton*
Sverdrup Technology, Inc , Arnold Air Force Station, Tennessee

The function of computational fluid dynamics (CFD) in an engine test organization is described and compared with the use of CFD in a design environment Both the test facility operator and the aircraft manufacturer have a common goal for CFD—to reduce the cost of the ultimate development of an aircraft component or system The aircraft manufacturer utilizes CFD to eliminate as much testing as possible in the process of hardware development, whereas the test facility operator relies on CFD to increase facility productivity and to improve the test data quality. Examples are presented to demonstrate how CFD is used in test feasibility studies, in pretest planning calculations, and in facility technology development Several key areas are addressed that must undergo further developments to expand the applicability of CFD methods

I Introduction

A TEST environment presents some rather unique challenges in the development and application of CFD tools Several articles have appeared in the past five years describing the benefits of CFD in the aerospace industry;[1 5] however little has been written concerning the impact CFD can have in a test facility Whitfield et al [6] noted the lack of attention paid in the literature and presented several examples of the "marriage" of wind tunnels and computers Later, Swafford described (unpublished work) a variety of applications of computational methods in support of wind tunnel tests at AEDC As mentioned in Refs 1 6, testing costs are rapidly increasing, primarily because of increasing labor rates and electrical power rates, and an increasingly larger amount of test time is required to develop and evaluate new aircraft systems because of an inadequate test data base Yet, computational costs are rapidly diminishing, since larger, faster computers and more efficient numerical algorithms are available. The trends in testing and computational costs have thus provided a strong impetus for implementing CFD in test and design environments

Although a test facility operator and an aircraft manufacturer share a common objective for CFD—to reduce the cost of the ultimate development of an aircraft component or system—a test facility operator is not in the business of hardware design, in contrast to the aircraft manufacturer In the aircraft industry, CFD is being incorporated into the design process with three major objectives in mind[5]: 1) to minimize parametric model scale testing; 2) to reduce risks by permitting more configurations to be evaluated numerically than could be evaluated experimentally, and reduce risks involved in scaling model results to full scale; and 3) to remove the constraint of limitations in the existing data base The goal is to eliminate as much testing as possible. Because the facilities at AEDC exist exclusively for testing, CFD is not employed to proscribe testing In contrast to hardware design and aerodynamic testing, where configurations are evaluated at model scale, engine testing involves full scale systems Similarly, CFD applied to external aerodynamics focuses on the geometry of the flight vehicle in an assumed infinite fluid, whereas CFD applied to engine testing is concerned with evaluating the effects of the test cell flow on the test article The major objectives of CFD here are to increase the test facility productivity, to improve the test data quality, and to shed light on anomalous test results The application of CFD in propulsion testing is rather new, but rapid progress has been made because of the extensive experience for external flows.

The remainder of this paper has a primary objective of outlining the role CFD plays in a test organization, with specific emphasis on the operation of the Engine Test Facility (ETF) at the AEDC A secondary objective is to list generic problem areas that must be addressed to provide continued growth in the problem solving capability of CFD Section II describes how computational methods are employed in propulsion testing, and Sec. III presents several problems that have been addressed. Finally, Sect IV highlights some work in progress and discusses a possible future direction of the CFD development effort in the ETF

II Approach

The Engine Test Facility, an aeropropulsion ground test facility, consists of 12 major operational altitude test cells and several adaptable research cells used for development and evaluation testing of propulsion systems for advanced aircraft, missiles, satellites, and space vehicles The ETF has the capability of testing air breathing engines with thrust ratings up to 50,000 lb and to simulate flight conditions ranging from sea level to altitudes up to 80,000 ft (Ref 7)

Computational methods are applied in several aspects of the testing, for example, in test feasibility studies, in test planning, execution and posttest analysis, and in test technique and instrumentation development The objectives of increased productivity and improved data quality are realized in several ways through CFD For example, in test planning, CFD has been used to determine how well a forebody simulator could reproduce the actual forebody flow as "seen" by an inlet CFD was used in two other cases to evaluate the flow quality (nonuniformities) ahead of a turbofan engine in a test cell Details of these and other applications are discussed in Sec III.

As stated in Ref 5, the design environment requires the consideration of hundreds of configurations This large number alone dictates that any CFD tool used must be inexpensive on a per case basis, in terms of both computer and manpower resources The requirements of fast, efficient,

Presented as Paper 83 0149 at the AIAA 21st Aerospace Sciences Meeting Reno Nev , Jan 10 13 1983; received Feb 11, 1983; revision received April 18 1984 Copyright © American Institute of Aeronautics and Astronautics, Inc 1984 All rights reserved

*Lead Engineer; presently Supervisor CFD Branch, Sverdrup Technology Inc , LeRC Group, Middleburg Heights Ohio Member AIAA

easy-to-use codes have motivated design-oriented industries, especially aircraft manufacturers, to employ inviscid methods (frequently coupled with boundary-layer codes) and parabolized Navier-Stokes (space-marching) methods instead of the more expensive time-dependent or steady Navier-Stokes codes.[1,5,8] In many instances, the more approximate analyses are satisfactory or can be modified to extend their range of usefulness. In addition, the collection of codes constitutes a library or a menu of codes applicable in a zonal modeling approach.[5,9] The necessity of maintaining and validating many computer programs, establishing coupling strategies among the codes, developing convergent iterative schemes for a coupled solution, and validating each new combination of codes are constraints that are accepted in a zonal approach.

In propulsion testing, repeat calculations are rarely performed. Each application is unique and requires specific tailoring of the code(s) involved; lead times are typically short (a few weeks). In most problems addressed, the full Navier-Stokes or Euler equations are required, and parabolized or partially parabolic schemes are inadequate. (A few problems are amendable to solution using a coupled Euler/boundary-layer code.) In solving a complete flowfield with one code, research versions of codes are usually used. Many three-dimensional (3-D) flows are as yet intractable, in which case a simplified approach must be taken.

III. Applications

Four applications of CFD to testing will be presented in this section. Two cases are concerned with determining a priori whether the test cell environment will reproduce the desired free-flight conditions. One case involves examining flow nonuniformities produced by disturbances upstream of the test article. The final case is aimed at facility hardware maintenance and entails using CFD tools to provide empirical input for compressor analysis models. Experimental results do not exist for all applications, but where measurements are available they are compared with the computations.

Forebody Simulator Evaluation: Subsonic Flow

In many tests, it is not possible or practical to install a complete vehicle in the test cell. It is often necessary to incorporate a modified configuration, with the requirement that the test data not be impacted. The size of the forebody/inlet combination associated with a missile propulsion system to be tested precluded consideration of the entire configuration. The focus of the test was in the area of the inlet, so the forebody was replaced by a forebody simulator, i.e., hardware that produces approximately the same flowfield as the actual forebody as measured at the inlet. Figure 1 illustrates the forebody/inlet geometry for free flight and in a test cell. A 3-D time-dependent Euler code[10,11] was employed for the calculations. The equations were solved using a finite volume form of MacCormack's unsplit explicit method. The mesh was generated in two-dimensional (2-D) parallel planes, from the inflow to the outflow boundary, using a Poisson equation approach (AEDC unpublished work).

Two forebody configurations were considered, a baseline and an improved model; and cases were run for the actual vehicle in free flight and for test cell installations involving the forebody simulators. Figure 2 shows a cross section of the mesh through the forebody and inlet, for the test cell case, indicating that the pitch plane was assumed as a symmetry plane. In Fig. 3, computed static pressures are plotted from the free-flight and test-cell cases for a free-stream Mach number of 0.6 at zero angle of attack. The results differ appreciably over the front half of the body because of significant geometry differences, but are quite close over the rear half. Before these computations were made, a wind tunnel test had been conducted; these measurements exhibit good agreement with the free-flight calculations. Measurements from a recent AEDC test are also plotted.

Figure 4 compares computed test cell and free-flight static pressure profiles in front of the inlet, and the agreement is again quite good. (All test cell computations shown are for the improved forebody simulator.) The comparisons verify that the test cell accurately reproduced free-flight conditions and that an improved forebody simulator improved the quality of the test. The results were obtained using CFD for substantially less cost than would be required to test both configurations.

Forebody Simulator Evaluation: Supersonic Flow

The emphasis again concerned using a modified forebody in the test to reproduce the desired flowfield at the inlet. The evaluation was performed by computing the flow over a 3-caliber tangent ogive for various freestream Mach numbers and attitudes. The development of the velocity profiles was then examined to determine where the profiles ceased changing along the body. The assertion was made that the inlet could be moved forward to the position where the

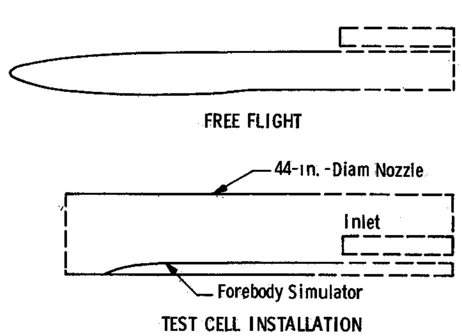


Fig. 1 Schematic of forebody/inlet.

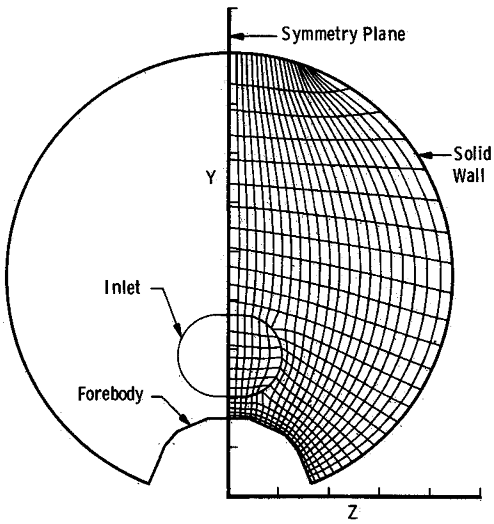


Fig. 2 Cross section of grid.

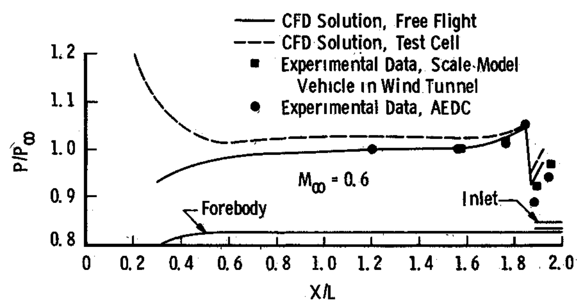


Fig. 3 Static pressure along forebody top surface.

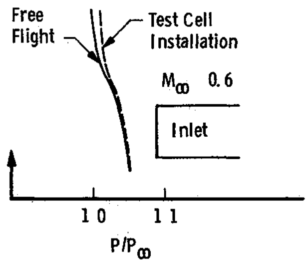


Fig 4 Static pressure profile ahead of inlet

profiles ceased changing, and the length of the forebody could be shortened for the test

The calculations were performed using a 3 D time-dependent "thin-layer" Navier-Stokes code developed by Pulliam and Steger [12] The equations were solved using approximate factorization, and turbulence closure was effected with a two-layer eddy viscosity model The computational mesh was generated in parallel planes using the elliptic mesh generator of Fernquist and Steger [13] Figure 5a illustrates the configuration Figures 5b, 5c, and 5d compare computed and measured[14] pressure coefficients along the body at various circumferential locations for three angles of attack at $M_\infty = 1\ 6$ The windward side ($\phi = 180$ deg) produces the best agreement; and the worst agreement occurs on the leeward side ($\phi = 0$), where significant flow separation can occur The disagreement on the leeward side was not a concern, since, in the present studies, the windward quadrants (90 deg $\leq \phi \leq 270$ deg) where propulsion system inlets are generally located were of primary interest Figures 6a, 6b, and 6c present profiles of the axial and normal velocity components (nondimensionalized by the freestream sound speed) in the wind ward plane for three angles of attack. Observe that the axial velocity component (which dominates the total velocity) is similar from $x/R_0 = 12$ to $X/R_0 = 20$, where R_0 is the body radius The inlet could be moved to the front of the similarity region and a shortened forebody used for the range of attitudes considered. This general conclusion also holds for higher Mach numbers [15] The computational results thus eliminated the need for parametric testing

Inlet Duct Flow Quality

In a proposed engine test, it was necessary to inject liquid air into the cell flow upstream of the engine to achieve the required low temperature (Figure 7 illustrates the arrangement) Venturis were used for metering mass flow, and the number of venturis open depended on the test requirements In this instance only, the outer ring was opened, thus producing an annular flow pattern Two concerns arose: 1) the flow at the liquid-air injection station would be so distorted that an unacceptable temperature distribution would result, and 2) a separated region upstream of the liquid-air injection station would produce nonuniformities in the inlet duct

The flow was computed using an axisymmetric incompressible, steady Navier-Stokes code[16 18] in conjunction with a k-ϵ turbulence model (The liquid air injection nozzles were ignored in the computations) The mesh was generated using the elliptic mesh generator of Middlecoff and Thomas [19] Two cases were considered: 1) the outer ring of venturis open and 2) all venturis open Figure 8 shows streamlines traced upstream from near the compressor face to determine their origin (for the case of only the outer venturis open) The streamlines indicate that the initially nonuniform flow is uniform at the liquid-air injection station and in the inlet duct The case of all venturis open produced similarly "clean" flow at the engine inlet station Notice in Fig 8, just downstream of the inflow boundry, that the streamlines deflected away from the wall indicate a separation bubble

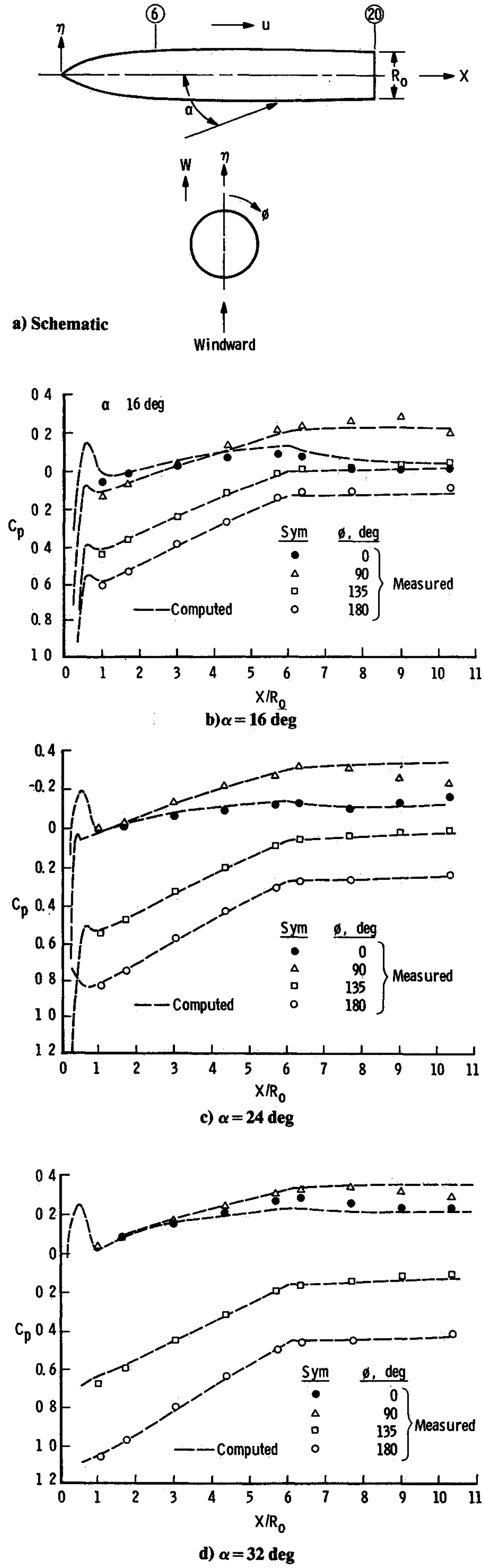


Fig 5 Pressure coefficient for 3 caliber tangent ogive; $M_\infty = 1\ 6$

The detailed results show that the maximum thickness of the separation corresponds to less than 2% of the flow area, and no distortion is produced downstream The results demonstrate that the flow in the vicinity of the liquid-air injection nozzles is sufficiently uniform to produce adequate tem perature conditioning, and the inlet duct flow is undistorted Subsequent tests and temperature measurements confirmed these conclusions

Unsteady Cascade Flow

A variety of turbomachinery is involved in conducting a propulsion test, for example, the test article as well as the equipment used to supply air to and exhaust gases from a test cell. Proper maintenance of the machinery and operation of the test hardware requires a priori knowledge of the effects of the test on turbomachinery, particularly the effects of transients For example, will a sudden ingestion of hot gases stall a compressor? Empirical models are employed to study the transients, and the empirical inputs must come from experiments or from flowfield calculations. An effort was undertaken to compute the unsteady flow through a compressor cascade Information on the effects of the unsteadiness on compressor stage characteristics are then derivable from the time-dependent calculations for incorporation into advanced compressor analysis models [20]

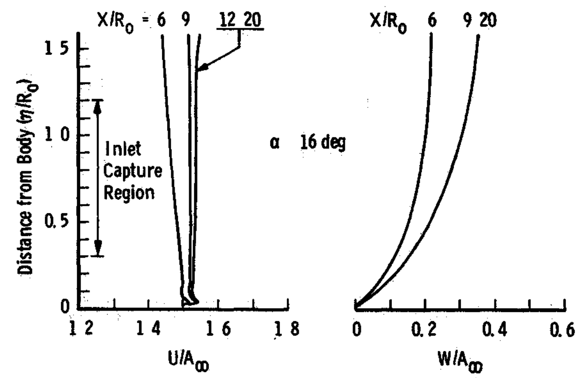
The geometry chosen was a 2-D cascade of double circular arc airfoils The basic assumptions were that the flow was two dimensional, inviscid, and periodic in the blade-to-blade direction both upstream and downstream The flow was computed using the Euler code of Refs 10 and 11, and an H mesh was generated algebraically

Computations were first performed for a steady $M=0$ 7 flow and an angle of attack of 2 45 deg Figure 9 compares computed and meaured[21] surface Mach numbers for the pressure and suction surfaces, and the agreement is good The computed lift coefficient C_L is 0 193, and the value calculated from the measurements is 0 204, a difference of 5% An unsteady case was then run at $M=0.7$, the source of the unsteadiness being a periodic variation of the incoming flowfield The flow variation produced an angle-of-attack variation ranging from 0 45 to 4 45 deg A reduced frequency $\bar{\omega}$, of 0 5 was simulated, where

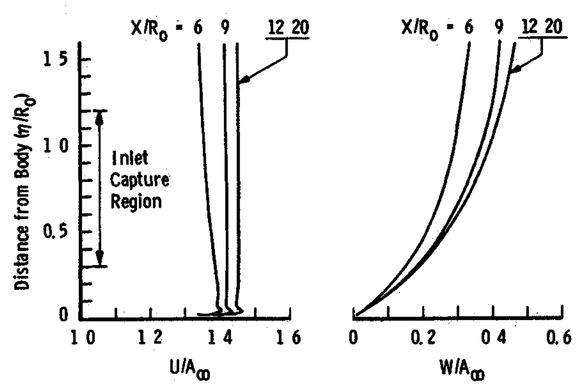
$$\bar{\omega}=2\pi\omega c/q_\infty$$

with ω being the frequency of inflow disturbance, c the blade chord, and q_∞ the freestream total velocity

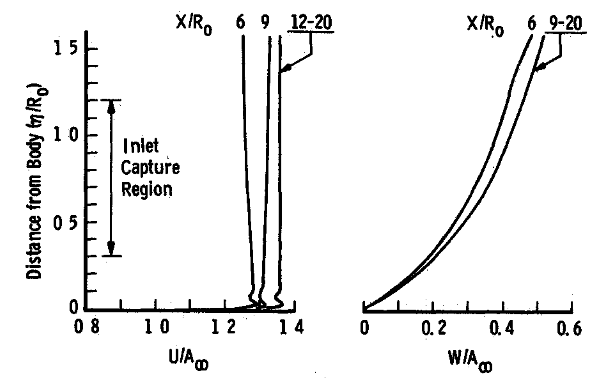
Figure 10 presents the lift coefficient as a function of time where, after the initial transient, a periodicity is displayed.



a) $\alpha = 16$ deg.



b) $\alpha = 24$ deg



c) $\alpha = 32$ deg

Fig 6 Windward side velocity profiles; $M_\infty = 1.6$

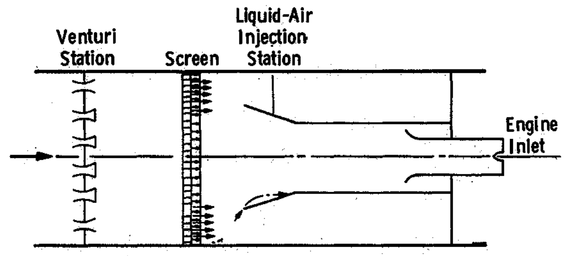


Fig 7 Schematic of liquid air injection system

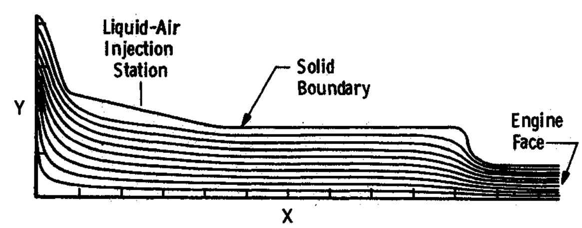


Fig 8 Streamlines traced upstream from engine face

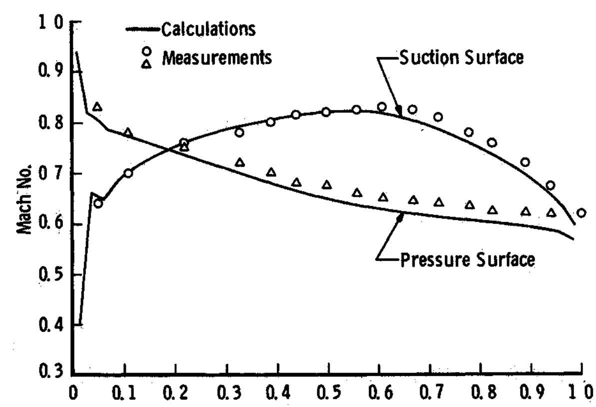


Fig 9 Surface Mach number distribution for a double circular arc (DCA) airfoil cascade

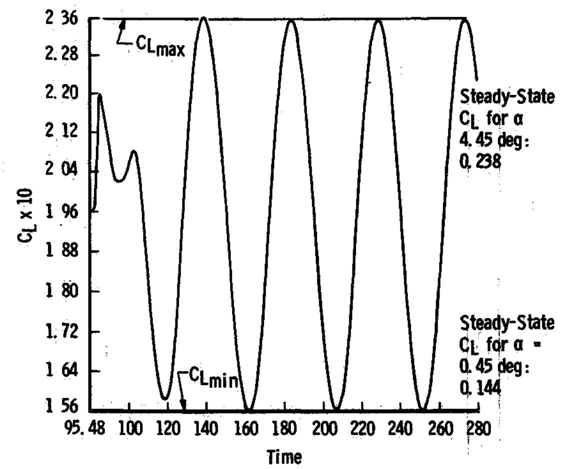


Fig. 10 Unsteady behavior of C_L for periodic inflow conditions ($\bar{\omega}=0.5$)

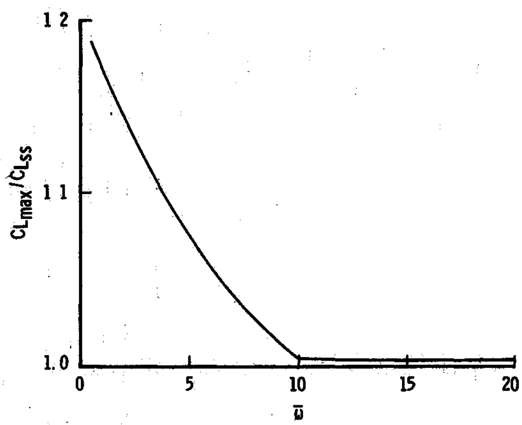


Fig. 11 Normalized $C_{L_{max}}$ as a function of reduced frequency.

Two further steady-state calculations were made for $\alpha=0$ 45 and 4.45 deg and produced lift coefficients of 0 144 and 0.238, respectively. Figure 10 demonstrates that the unsteady calculations produce minimum and maximum values of C_L within the envelope of the steady values at $\alpha=0$ 45 and 4 45 deg A range of reduced frequencies was then considered, and the results are plotted in Fig 11 as the normalized $C_{L_{max}}$ as a function of reduced frequency The normalizing factor is the steady-state value of C_L at $\alpha=2$ 45 deg The unsteady results are in qualitative agreement with previous calculations [22,23] Trends, such as the one depicted in Fig 11, will be derived from the flowfield analyses and incorporated into compressor analysis models.

IV. Conclusions and Future Work

The examples presented demonstrate that CFD is already manifesting a positive influence in the operation of a propulsion test facility, and the future potential is great Expanded incorporation of CFD in testing requires effort in two fundamental areas: 1) the ability to model 3-D flows (larger and faster computers, improved algorithms, better understanding of flow phenomena, etc) and 2) the inclusion of CFD support in the formative stages of test planning The former requires technological advances, and the latter requires changes in attitudes and in traditional approaches to test planning Paynter[5] discusses the analogous problems in the aerospace industry and states that "experience indicates that the use of CFD for design must cost an order of magnitude less than an equivalent test based approach if it is to be accepted by industry as an alternative to parametric testing " A comparable estimated cost savings cannot be offered that would motivate test planners to include CFD in the test effort from the outset Experience at AEDC teaches that repsonse time quite often is as important as cost

Several authors[5 8 24 25] have recently addressed various areas in which future improvements must occur to permit practical computing of three-dimensional flows, for example, improved turbulence modeling, more flexible grid generation, and even new computer language constructions In addition, there is an ongoing need for quality, detailed flowfield data, which becomes increasingly problematic in 3-D because there are few generic 3-D flows (in contrast to the widely studied 2-D flows, such as jets, wakes, mixing layers, boundary layers, etc) All these areas are of key importance to the future of CFD

An area that gets little in-depth attention is the accuracy of calculations (not merely the formal order of accuracy of a difference scheme) Most publications contain comparisons of calculations with measurements, yet this is inadequate, even misleading, at times For example, in Ref. 26, calculations with a Navier-Stokes code were made for a turbulent shock/boundary-layer interaction, and predictions of the skin friction compared favorably with measurements When examined in detail, however, the computed velocity gradients were found to be half the correct values, and the effective viscosity was a factor of 2 too large, thus producing a fortuitously correct skin friction Similar anomalies have been discussed by Messina [27] Some authors attempt to identify mesh-related errors by performing mesh-refinement studies, yet in 3-D this is seldom possible The upshot is that the errors being committed in the calculations must be known before the solutions can be used with confidence; and comparisons of predictions with measurements is a necessary but not sufficient step. Some further suggestions on error assessment are described by Forester [28]

The work currently in progress in the ETF, or to be addressed in the near future, represents problems of increased complexity over those discussed herein. One effort involves assessing the impact of the test-cell environment on performance measurements of high-bypass-ratio turbofan engines. Another concerns the injection of water droplets in an airstream and the formation of ice on a test article downstream. A third problem entails evaluating forebody simulators for freejet testing. The CFD technology developments underway in the ETF to provide the necessary problem-solving capability are as follows: 1) a 3-D Navier-Stokes code using the new MacCormack method;[29 30] 2) a 3-D time-dependent, compressible boundary-layer code for use with the Euler code[31]; 3) a code to treat multiphase flows, including phase change but omitting chemical reactions; 4) direct generation of 3-D body-fitted grids using algebraic and Poisson equation methods; 5) grids that are a function of the evolving flowfield solution; 6) extension to 3-D of the cascade calculations, including viscous and rotational effects; and 7) development of a method for computing cell effects in a turboprop engine test With the expanded CFD capability, the impact on testing will increase, with an attendant increase in the test facility productivity and improvement in the test data quality

Acknowledgments

The research reported herein was performed at the Arnold Engineering Development Center, Air Force Systems Command Work and analysis for this research were done by personnel of Sverdrup Technology, Inc /AEDC Group, operating contractor of the AEDC propulson test facilities Particular gratitude is expressed to W C. Armstrong and W J Phares of the Computational Fluid Dynamics Group, R J. Matz of the Fluid Mechanics Group, and R R. Jones III of the Engine Technology Group The Air Force Project Manager was K L Kushman, AEDC/DOT

References

[1]Smelt R , The Role of Wind Tunnels in Future Aircraft Development " *11th Congress of the ICAS* Vol 1 Sept 1980 pp 1 10.

[2]Chapman, D R 'Computational Aerodynamics Development and Outlook ' *AIAA Journal* Vol 17, Dec 1979 pp 1293 1313

[3]Tjonneland, E and Birch S F ' Development and Validation of Computational Flow Analyses for Design of Inlets and Nozzles ' Fifth Symposium on Airbreathing Engines London England Feb 1981 pp 14 1 14 12

[4]Graves R. A Jr , 'Computational Fluid Dynamics: The Coming Revolution, *Astronautics and Aeronautics* Vol 20 London England March 1982, pp 20 28

[5]Paynter G C , ' CFD Technology for Propulsion Installation Design—Forecast for the 80's ' ASME 27th International Gas Turbine Conference April 1982 pp 1 8

[6]Whitfield, J D , Pate S R Kimzey W F and Whitfield D L The Role of Computers in Aerodynamic Testing ' *Computers and Fluids* Vol 8 No 1 1980, pp 71 99

[7]AEDC *Test Facilities Handbook* (11th Edition) April 1981

[8]Chapman, D R 'Trends and Pacing Items in Computational Aerodynamics *Lecture Notes in Physics* Vol 141 Spinger Verlag New York 1981 pp 1 11.

[9]Barton J M et al 'The Computation of Complex Jet and Nozzle Flows JANNAF 12th Plume Technology Meeting CPIA Pub 332, Dec 1980, pp 57 84

[10]Jacocks J L and Kneile, K R , Computation of Three Dimensional Time-Dependent Flow Using the Euler Equations " AEDC TR 80 49 (AD A102463) July 1981

[11]Whitfield D L Swafford T W and Jacocks J L. ' Calculations of Turbulent Boundary Layers with Separation Reattachment and Viscous/Inviscid Interaction ' *AIAA Journal* Vol. 19, Oct 1981, pp 1315 1322

[12]Pulliam T H and Steger J L , On Implicit Finite Difference Simulations of Three Dimensional Flow *AIAA Journal* Vol 18 Feb 1980 pp 159 167

[13]Fernquist A R and Steger J L A General Two Dimensional Curvilinear Grid Generation Program with Applications to Cascade Geometries Flow Simulations Inc Mountain View Calif , FSI Report 80 02 Aug 1980

[14]Landrum E J Wind Tunnel Pressure Data at Mach Numbers from 1 6 to 4 63 for a Series of Bodies of Revolution at Angles of Attack from − 4 to 60 deg " NASA TMX 3558 Oct 1977

[15]Matz R J , Theoretical Studies of Flow About Pitched Slender Bodies and Related Freejet Test Techniques AEDC TR 82 21 March 1983

[16]Chien, J C Numerical Analysis of Turbulent Separated Subsonic Diffuser Flows " AEDC TR 76 159 (AD A036005) Feb 1977

[17]Chien, J C Numerical Computation of Subsonic Conical Diffuser Flows with Nonuniform Turbulent Inlet Conditions AEDC TR 77 78 (AD A044948) Sept 1977

[18]Chien, J C Vorticity Stream Function Formulation of Compressible and Incompressible Turbulent Internal Flows AEDC TR 79 13 (AD × A070043) May 1979

[19]Middlecoff J F and Thomas, P D Direct Control of the Grid Point Distribution in Meshes Generated by Elliptic Equations ' *AIAA Journal* Vol. 18 June 1980 pp 652 656

[20]Davis M W Jr A Stage by Stage Dual Spool Compression System Modeling Technique ASME Paper 82 6T 189 April 1982

[21]Anon Transonic Flows in Turbomachinery, Vol 2: Sample Calculations Compressor Tests, ' von Karman Institute Lecture Series 59 Rhode St Genese Belgium May 1973

[22]Reddy, K C A Potential Flow Analysis of Oscillatory and Distorted Flows Through A Cascade, Appendix III in AFAPL-TR 70 51 July 1970

[23]Schorr B and Reddy K C Inviscid Flow Through Cascades in Oscillatory and Distorted Flow *AIAA Journal* Vol 9 Oct 1971 pp. 2043 2050

[24]Lomax H Some Prospects for the Future of Computational Fluid Dynamics ' *AIAA Journal* Vol 20 Aug 1982 pp 1033 1043

[25]Hall M G 'Computational Fluid Dynamics—A Revolutionary Force in Aerodynamics ' AIAA Paper 81 1014 June 1981

[26]Arieli R and Murphy J D Pseudo Direct Solution to the Boundary Layer Equations for Separated Flows ' *AIAA Journal* Vol 18 Aug 1980 pp 883 891

[27]Messina N A ' A Computational Investigation of Shock Waves, Laminar Boundary Layers and Their Mutual Interaction Ph D Thesis Princeton University N J 1977

[28]Forester C K Advantages of Multi grid Methods for Cer tifying the Accuracy of PDE Modeling *Multigrid Methods* NASA CP 2202 Oct 1981

[29]MacCormack R W A Numerical Method for Solving the Equations of Compressible Viscous Flow *AIAA Journal* Vol 20 Sept 1982 pp 1275 1281

[30]MacCormack R W Numerical Solution of the Equations of Compressible Viscous Flow " *Transonic Shock and Multidimen sional Flows: Advances in Scientific Computing* Academic Press New York 1982 pp 161 179

[31]Swafford T W , Three Dimensional, Time Dependent Compressible Turbulent Integral Boundary Layer Equations in General Curvilinear Coordinates and Their Numerical Solution " Ph D Thesis Mississippi State University Mississippi State Miss Aug 1983