Computational Fluid Dynamics for Internal Flows

Introduction

ABOUT a year ago, Gordon Oates asked me to organize a special issue of the *Journal of Propulsion and Power* that would be devoted to the application of computational fluid dynamics (CFD) to internal flows. We both shared an interest in highlighting the unique aspects of this problem to the CFD and propulsion communities. The aerodynamic problems associated with air-breathing propulsion systems differ from their external aerodynamic counterparts in three principal ways:

1) They usually involve complicated geometries.

2) There is a close coupling of aerodynamic components involving three-dimensional unsteady and viscous effects.

3) There is a significant energy exchange from either chemical reactions or mechanical work.

Whereas these aerodynamic complexities would seem to discourage the growth of CFD technology for propulsion systems, they have instead forced the propulsion community to improve its modeling of the internal environment by including more realistic simulations of the geometry and flow physics.

The long-range goal of the CFD community should be the development of computer programs that can be used to perform aerodynamic designs, to aid understanding of fluid flow phenomena, and to provide an inexpensive and available supplement for experimental testing. CFD methods today can simulate propulsion system flowfields about complex geometries with relatively simple physics or about relatively simple geometries with more complex physics. The geometric complexity of gas turbine engine components is manifested both in single-stage blade rows and in multistage interacting blade rows. Computational grids must be devised to be body-conforming, and mesh distributions must be set for accuracy, efficiency, and ease in applying boundary conditions. Nonsteady and rotational in-flow conditions, as well as blade-to-blade periodicity, introduce additional complexity to the formulation of an analysis procedure. It should be noted that pressure variations at a point on the surface due to unsteady effects can be larger than the steady flow variations over the blade surface. On the other hand, the flowfield in a supersonic combustion ramjet propulsion system (scramjet) and the flow in a gas turbine combustor are dominated by transitional and separated viscous flows undergoing nonequilibrium fuel/air combustion and air chemistry processes producing significant radiative and convective heating.

The storage limitations of current computers and practical cost issues have driven CFD codes to zonal solution methodologies, treating local regions of the flow with different equation sets and associated solution algorithms. The classical aerodynamic example of this method is the analysis of airfoil loading using a boundary-layer analysis in the near-wall region coupled to an inviscid analysis. The complexity of the internal flow problem, with its close-coupled and complicated aerodynamic elements, is difficult enough to analyze independently, much less the interaction of these elements. If the interaction effects of adjacent components are dominant, even the use of the zonal approach may not be possible.

Background

During the last twenty-five years, computers have enabled engineers to construct and use codes that solve the governing fluid dynamic equations for increasingly more complex physical and geometric applications. In the late 1960s, CFD codes were developed for low-speed applications over extremely complex geometries and for high-speed applications over relatively simple geometries. In the former case, the effects of compressibility were neglected and the governing equations linearized. CFD codes, based on the work of Martensen[1] and Hess,[2] became known as either singularity superposition or panel methods and have been extensively applied to cascade flows[3,4] and to duct flows, as well as to external aerodynamic configurations. In the latter case, the analysis of the "blunt-body" problem and the study of reentry aerodynamics provided the catalyst for the development of time asymptotic solutions of MacCormack[5] and others.

With the advent of high-speed computers, more sophisticated treatments of the governing equations began to emerge. In external aerodynamic applications, the effects of compressibility were included in full potential methods,[6] with subsequent extensions to Euler methods[7] for wake tracking and rotational effects and to Navier-Stokes methods for viscous dissipation and separation effects. The presence of strong shock waves in turbomachinery applications, however, necessitated the development of blade-to-blade Euler methods for internal flow codes as early as 1971.[8-10] These methods worked but were extremely inefficient, used approximate non-body-conforming H meshes, and were of limited accuracy. Similarly, the significant effects of geometrically induced flowfield three-dimensionality were formulated by Wu[11] as early as 1952. He defined two planar stream surfaces and solved the Euler equations on them. Implementation of this method was delayed until the late 1960s when the development of computer capabilities permitted Katsanis[12] and others to develop an accurate and efficient solution of Wu's formulation. This method, while still in use, relies heavily on empiricism to correct the procedure for the viscous effects of blockage and flow deviation. A more detailed history of the development of CFD codes for turbomachinery and duct flow applications has been previously documented by McNally and Sockol.[13] Subsequent improvements to internal flow CFD codes have been driven largely by techniques developed for external flows, e.g., improved computational grids, multigrid acceleration, etc. A good example of this transfer of technology is presented in this issue of the Journal, where Rai uses patched and overlaid grids to analyze rotor/stator interaction.

Unresolved Problems

In general, CFD codes used by the propulsion community have achieved a relative degree of maturity. Two-dimensional analyses of stationary or single-stage gas turbine engine components are regularly and reliably used in conjunction with the design of new propulsion systems. The state-of-the-art CFD codes for simulating nonsteady aerodynamic flowfields, multistage flow interactions, and flows with significant chemical/viscous/pressure wave interaction are far less mature. Recent advances, however, including the works cited in this Journal, suggest that sufficient progress can be made by focusing on these complex problems. These capabilities represent pacing technologies for the analysis of gas turbine and scramjet combustors and for the analysis of a complete turbine or compressor.

Application of Codes

A key element that must parallel the improvement of geometric and physical modeling in CFD codes is the *Effec-*

tiveness of the code for general use by the technical (non-CFD) community. Miranda[14] defines effectiveness as

$$\text{Effectiveness} = \text{Quality} * \text{Acceptance}$$

where *Quality* can and should be demonstrated through a series of validation calculations against accepted data bases (experimental, analytical, etc.). *Acceptance* is defined in terms of those often used criteria: user-friendiness, robustness, and affordability. Although Miranda's formula is given as a linear relationship, experience has indicated that the acceptance factor has a much stronger influence outside the CFD community.

This issue of the Journal presents a cross-sectional view of current CFD research and applications as related to both gas turbine and scramjet propulsion systems. The topics covered include a survey paper, as well as several papers describing numerical methodology development, code validation calculations, and applications to current design problems. These papers identify the need for further improvement in numerical and physical modeling of internal flow applications. Demonstration calculations are presented in each paper, but only limited comparisons with experimental data are provided for validating and calibrating the codes. Finally, these papers identify the need for developing procedures for effectively and efficiently applying CFD codes in the propulsion design environment.

Acknowledgment

A final comment should be made about my friend Gordon Oates, who initiated this issue and who had been a driving force for this Journal, for the AIAA, for his students, and for the entire aerospace community. His untimely death represents a real loss to our community. He will be remembered for his foresight, intelligence, and technical contributions. More importantly, however, he will be remembered for his warmth, friendship, and character. His presence will be sorely missed.

Dr. Thomas J. Barber
United Technologies Research Center
East Hartford, Connecticut

References

[1]Martensen, E., "Berechnung der Druckverteilung an Gitterprofilen in Ebener Potentialstromung mit einer Fredholmschen Integralgleichung," *Arch. Rat. Mech. and Analysis,* Vol. 3, 1959, pp. 235–270.

[2]Hess, J.L. and Smith, A.M.O., "Calculation of Potential Flow About Arbitrary Bodies," *Progress in Aeronautical Sciences,* Vol. 8, Pergamon Press, New York, 1966, pp. 1-138.

[3]Geising, J.P., "Extension of the Douglas Neumann Program to Problems of Lifting Infinite Cascades," Douglas Aircraft Co. Rept. LB-31653, 1964.

[4]Weingold, H.D., "Two Dimensional Potential Flow Program," Pratt & Whitney Aircraft Rept. TDM-2033, 1967.

[5]MacCormack, R.W., "The Effects of Viscosity in Hypervelocity Impact Cratering," AIAA Paper 69-354, 1969.

[6]Jameson, A., "Transonic Potential Flow Calculations in Conservation Form," AIAA 2nd Computational Fluid Dynamics Conference, Hartford, CT, 1975, p. 148.

[7]Jameson, A., Schmidt, W., and Turkel, E., "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time Stepping Schemes," AIAA Paper 81-1259, 1981.

[8]Gopalakrishnan, S. and Bozzola, R.A., "Numerical Techniques for the Calculation of Transonic Flow in Turbomachinery Cascades," ASME Paper 71-GT-42, 1971.

[9]McDonald, P.W., "The Computation of Transonic Flow Through Two-Dimensional Gas Turbine Cascades," ASME Paper 71-GT-89, 1971.

[10]Denton, J.D., "A Time Marching Method for 2 and 3 Dimensional Blade to Blade Flows," Aeronautical Research Council R&M 3775, 1975.

[11]Wu, C.H., "A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial, Radial and Mixed Flow Type," NACA TN 2604, 1952.

[12]Katsanis, T., "FORTRAN Program for Calculating Transonic Velocities on a Blade-to-Blade Stream Surface of a Turbomachine," NASA TN D-5427, 1969.

[13]McNally, W.D. and Sockol, P.M., "Computational Methods for Internal Flows With Emphasis on Turbomachinery," NASA TM 82764, 1981.

[14]Miranda, L.R., "A Perspective of Computational Aerodynamics from the Viewpoint of Airplane Design Applications," AIAA Paper 82-0018, 1982.