N91-21069

57-34
27/3
P-8
ND315753

SIMULATION OF TURBOMACHINERY FLOWS

John J. Adamczyk
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

ABSTRACT

With the interest in jet propulsion at the end of World War II, aerodynamicists were challenged to develop mathematical models which could be used to design turbomachinery components for jets. NASA Lewis engineers and scientists played a major role in meeting this challenge. This paper highlights some of their accomplishments as well as those of others. The paper also addresses several problems which must be solved if jet propulsion technology is to advance.

INTRODUCTION

NASA Lewis Research Center has a long history of developing mathematical models of turbomachinery flows and of constructing numerical algorithms to solve the equations associated with these models. That history began in the late forties and fifties with the modeling of axial, centrifugal, and mixed flow machinery. Stantiz, Wu, Ellis, Hamrick, Hansen, Klapproth, Goldstein, Prain, Costello, Cummings, Sinnette, and Yohner are just a few of the names behind this pioneering work. Both "Axial Compressors" (NASA SP-36, ref. 1) and "Aerodynamics of Turbines and Compressors" (ref. 2) give excellent accounts of Lewis' many contributions. Similar projects were pursued concurrently by academic and industrial researchers, of which Marble (ref. 3) and Smith (ref. 4) are exemplary.

Most of these early works can be divided into two categories, blade-to-blade (cascade plane) and throughflow (meridional plane) analyses. Although the blade-to-blade analyses originated from isolated airfoil theory, throughflow analyses had no direct counterpart in external aerodynamics and were unique, at this time, because they accounted for the effects of vorticity on the development of the flow field. In fact, the external aerodynamic community has only very recently found it necessary to include the effects of vorticity outside of boundary layer regions in the analysis of aircraft components.

Although throughflow analyses were the first to find wide use, it was not until the seventies that these blade-to-blade analyses were developed to a point that designers could use them with confidence. Before they could be directly useful to designers, throughflow analyses had to await the development of digital computers and of blade element correlations which corrected for the effects of viscosity and in-passage shocks.

In 1954 NASA Lewis undertook a comprehensive effort (lasting nearly three years) to correlate the available data on axial flow compressors for the purpose of design. These correlations, which were developed according to blade element theory, corrected for the effects of viscosity on blade element efficiency and for the flow angle exiting the element (ref. 1). A similar effort was undertaken by turbine aerodynamicists at Lewis (ref. 5). At the same time, a number of nongovernment efforts with similar goals yielded correlations for the departure of the flow angle from the mean chamber line. These correlations were used to establish the flow turning produced by a blade element and the pressure distribution along the element surface.

In the late fifties the desire for a higher pressure ratio per stage forced designers to consider transonic blading. This new class of blading necessitated the development of models for estimating the losses created by an in-passage shock. Such a model was developed by Miller and Hartmann (ref. 6). Today updated versions of this model, which account for the oblique inclination of the shock wave relative to the flow direction, can be found in many throughflow analyses. A more recent addition to the list of models used with throughflow analyses introduces the effects of spanwise mixing on the radial distribution of total temperature, total pressure, and angular momentum. These models associate the mixing process with either the development of secondary flows, Adkins and Smith (ref. 7), or turbulent diffusion, Gallimore and Cumpsty (ref. 8). Although these models have been shown to give comparable results, the issue of the controlling mixing process is still being debated.

With the development of blade element correlations, all of the parts were in place to attempt a computer execution of a throughflow simulation. One of the first publications which illustrated the capabilities of this new technology for compressor design was authored by Wright and Novak (ref. 9). This publication was followed by a number of others, of which Crouse et al. (ref. 10) and Hearsey (ref. 11) have resulted in computer codes that are available to the compressor design community. All three of these publications relied on the streamline curvature procedure to solve the throughflow equations.

In the mid and late sixties Katsanis pioneered the development of aerodynamic analysis codes for turbomachinery which used matrix iteration procedures to solve the finite difference form of the fluid flow modeling equations. One of these was a throughflow analysis code, co-written with McNally and known as "Meridional" (ref. 12), which is still widely used by turbomachinery designers. Recently Katsanis added to "Meridional" the ability to analyze more than one blade row, and a publication which outlines this enhancement is forthcoming. Katsanis also succeeded in developing a blade-to-blade analysis code. This code and its successor, TSONIC (ref. 13), was one of the first blade-to-blade analysis codes to find wide acceptance by designers. These codes provided a reliable estimate of the blade surface pressure and Mach number distribution. McNally later developed a boundary layer analysis (ref. 14) which could be used with TSONIC to estimate the blade boundary growth and blade element total pressure loss at design conditions. Today the codes developed by Katsanis and McNally, or similar codes, are routinely used to design subsonic and mildly transonic compressors as well as turbine blade sections.

In the mid seventies Garabedian and Korn (ref. 15) developed a hodograph procedure for designing supercritical, shock-free airfoil sections. The relevance of their work to turbomachinery blade element design was immediately recognized. With the assistance of Korn, Stephens (ref. 16) succeeded in using this procedure to redesign a cascade of airfoils which had previously been designed with the Pratt & Whitney cascade design codes. Back-to-back performance tests clearly showed the performance improvements that could be attained by using this hodograph direct design procedure. These gains came from improved control of the diffusion rate of the suction surface flow. For compressors, designers have been able to duplicate the arbitrary shapes associated with this hodograph design concept by using existing blade element geometry generators. Today, one finds these cascade sections routinely used in many advanced compressor designs. An additional illustration of the progress that has been achieved in blade-to-blade analysis is the recent publication of Davis, Hobbs, and Weingold (ref. 17). These authors succeeded in predicting the total pressure loss of a cascade of airfoils over its useful range of operation to within experimental accuracy. Today, cascade tests are used primarily to validate and calibrate computer codes and in experimental studies of fluid dynamic phenomena that establish empirical relationships utilized in cascade flow simulation.

With the introduction of supercomputers in the late seventies, it became feasible to use three-dimensional codes to analyze a blade row in the final stages of design. The first component to benefit from this capability was the fan. This application was brought about, in large part, by the work of Thompkins and Oliver (ref. 18) and Denton and Singh (ref. 19). Nearly all of the fan designs executed today (ref. 20) are evaluated by using three-dimensional Euler codes with corrections added to account for the effects of viscosity. These codes have demonstrated their ability to accurately predict the structure and strength of the in-passage shock, a capability that has resulted in improved fan performance. However, there remains the issue of accurate loss prediction for both design and off-design operating conditions.

The references cited in this introduction are by no means complete. They represent the author's best attempt to condense the many publications written over the years on the subject of turbomachinery flow modeling and simulation.

The progress made in modeling and simulating turbomachinery flows since the late forties is most impressive. There are, however, many unsettled issues and unsolved problems which remain, and computational fluid dynamics can help resolve them. The rest of this paper outlines some of these problems.

WHERE DO WE GO FROM HERE?

What follows is an attempt to identify turbomachinery-related flow problems whose solutions remain to be found or are less than satisfactory. (These problems were identified by a number of sources.) The solution of these problems would greatly advance design. Their solution is not simply bigger and faster computers. Fluid flow modeling as well as experimental fluid mechanics will have to be combined with numerical simulation to develop solutions to these problems which have a timely impact on the design process. All three of these disciplines are needed, for without each of them, the end result will fall short of meeting the needs of designers.

These problems can be divided into four categories: (1) intra-blade-row flows, (2) inter-blade-row flows, (3) component interactions, and (4) interdisciplinary interactions.

One of the most important intra-blade-row flow problems is the prediction of relative total pressure loss at design and off-design operation. In addressing this problem, we must first assess our ability to predict the loss of both subsonic and transonic compressor and turbine blade rows at design conditions. We need to establish the requirements to attain grid-independent solutions. The deficiencies of the simulation codes need to be established and an engineering assessment made of the merit of correcting them. For off-design operation, the prediction of loss is made more complicated by large regions of flow separation and large radial migrations of flow. In addition, vortex shedding may occur and the endwall flow may become unstable, causing the flow field to be unsteady. The impact of these unsteady flows on the intra- and inter-blade-row flow field has just begun to be examined. These intra-blade-row flow structures appear to establish the stability limits for compressors.

Another important problem whose solution is far from optimal is the design of endwall blade sections. It has been known for some time that the maximum loss for compressors occurs in this region. In addition, recent experimental studies have shown this region to be critical in establishing the stall stability limit. For turbines, the endwall region is the origin of large-scale, secondary flow structures which are known to have a significant impact on blade row performance and life. These secondary flows can also influence the performance and life of the neighboring downstream blade row. To date we have simply acknowledged the existence of the endwall flow region in our designs or employed empirically derived flow control concepts to improve the quality of the flow in this region. We need to establish why some of these concepts worked, while others failed. We need to develop design criteria for the endwall region which are more closely related to the controlling flow physics than those we have today.

The second group of problems is associated with the flow process in multistage machinery induced by blade row interactions. Currently two approaches are being examined to analyze these problems. The first involves simulating the unsteady flow field generated by more than one blade row. The second attempts to simulate the time-averaged flow field within a typical passage of a blade row embedded in a single or multistage configuration. This approach introduces the effects of neighboring blade rows by means of semiempirical models of blade forces, energy sources, and both momentum and energy correlations. Significant progress has been made using both approaches. The unsteady simulations have provided new insight into the complex blade row interactions which occur in transonic machinery, whereas the time-averaged flow simulations have been useful in modeling the complex flow fields within multistage configurations. These two approaches to multi-blade-row simulation can complement each other. The unsteady simulations can provide a data base for calibrating the models used to close the time-averaged flow modeling equations, and the time-averaged simulations can provide answers to many problems which would be prohibitive to solve with an unsteady code.

It would be of great benefit to define and execute numerical simulations which go beyond those that have been accomplished to date. Numerical simulations need to be executed which focus on specific issues or questions pertaining to the flow physics controlling the performance and the life of a blade row. Codes used in these studies need to be carefully documented for their ability to accurately capture the space and time scales associated with these flow processes. The costs of obtaining this information will be substantial, and thus every attempt should be made to document the simulation code. In addition, every attempt should be made to make the results available to as many researchers as feasible.

The third area focuses on problems which result from component interactions. Very often these problems are concerned with system transient behavior during stall or surge or the response of the system to a flow distortion. Because of the diverse range of space and time scales associated with these problems, the only practical means of addressing them is to resolve only those flow structures which have a first order effect on the system response. The direct effects of those structures which are of secondary importance are either neglected or approximated. An example of this approach is the model developed by Moore and Greitzer (ref. 21) for predicting the onset and recovery from surge and rotating stall. This model has reproduced many of the observed responses of compression systems during post stall operation. The equations associated with this model are challenging to solve, and because of the insight gained from them, they should be of interest to the CFD community.

Much more needs to be done in the area of component interaction modeling. Because most models are either quasi one dimensional or two dimensional, they cannot effectively treat problems for which the tangential and radial scales are nearly equal.

Solving the last problem requires the application of two or more disciplines. Problems which fit under this category are flutter and forced vibration of blading, hot section life prediction, and finally, engine simulation and control. We have made significant progress in the last 10 years in predicting when flutter will occur in high speed fans and advanced turboprops. This progress is partly due to new, unsteady aerodynamic models which include the effects of aerodynamic loading and blade sweep. We have developed analyses which examine flutter in the frequency domain and more recently have begun to address the problem in the time domain. Additional work, however, is needed in predicting part speed or stall flutter.

Aerodynamically induced, forced vibrations seem to be a problem which refuses to go away. They can occur whenever the natural frequency of a blade crosses the frequency of the aerodynamic excitation. The resulting blade vibration amplitude is dependent on the excitation level, the aerodynamic damping, the structural damping, and the mode of vibration. Because we generally uncover the problem only after the machine has been built, it is costly to solve. We have begun to address forced vibration with unsteady aerodynamic models which include the effect of blade loading and blade thickness. These models are either based on the asymptotic theory or rapid distortion or are numerical solutions of either the Reynolds-averaged form of the Navier-Stokes equations or Euler equations. Asymptotic models provide the rapid solutions which are very important to designers, whereas numerical simulations help establish the limits of these asymptotic models and address those problems which are beyond their capability. Both approaches should be further developed.

The prediction of the life of a component in the hot section of an engine has been, and continues to be, an area of intense activity at NASA Lewis. The impact of material properties, the stress state of the structure, the environment, and both the gas-side and air-side surface heat transfer on engine hot section life were investigated intensively under the NASA Lewis HOST program. The interested researcher should read reference 22 for more details about this program.

The confluence of the solutions to the problems that were discussed is an engine simulator. Developing a numerical propulsion system simulator (NPSS) is truly a grand challenge, requiring a coordinated, multidisciplinary research effort. NASA Lewis (ref. 23) is currently in the midst of planning such a program whose ultimate goal is a detailed, front-to-back computation of an engine, including its aero-thermal-structural response, the combustion process, and its response both to control inputs and external disturbances. To achieve such a capability, an hierarchy of engine systems, component-specific models, and component subsystem models of varying degrees of fidelity will have to be developed and mathematically "coupled." Managing the computation and data flow within the host computing engine will also be challenging. It is estimated that a saving of 25 to 40 percent in engine development costs and manpower would result from NPSS technology - significant numbers given the cost of engine development. Equally important, NPSS technology will permit us to explore and test new concepts without committing large outlays of funds and manpower. NPSS would screen out those concepts of little or no value, and those which promise to have significant impact could be introduced into production at reduced cost and time.

CONCLUSIONS

The last 40 years have produced significant advances in our understanding, modeling, and simulation of turbomachinery flows. With these advancements has come increased component efficiency, reduced weight, and increased durability. The engineers and scientists at NASA Lewis have played a major role in bringing this about. Today, the challenges we face as turbomachinery aerodynamicists are as demanding as those of the past. The researchers at NASA Lewis in partnership with their colleagues in industry and in universities are addressing many of them. The research program they have put in place emphasizes experimentation, modeling, and simulation. All three of these elements are needed, for without one of them, the end product would fall short of meeting its goal. The confluence of this research will be an engine simulator which will permit a detailed front-to-back computation of an engine. Such a simulator would reduce engine development costs 25 to 40 percent and allow the introduction of new concepts in a timely fashion.

ACKNOWLEDGMENT

The author wishes to express his thanks to Kristine Dugas for her assistance in the draft of this manuscript.

REFERENCES

1. Johnson, I.A.; and Bullock, R.O., eds.: Aerodynamic Design of Axial-Flow Compressors. NASA SP-36, 1965.

2. Hawthorne, W.R.: Aerodynamics of Turbines and Compressors. High Speed Aerodynamics and Jet Propulsion, Vol. X, Princeton University Press, 1964.

3. Marble, F.E.: The Flow of a Perfect Fluid Through an Axial Turbomachine with Prescribed Blade Loading. J. Aeronaut. Sci., vol. 15, no. 8, Aug. 1948, pp. 473-485.

4. Smith, L.H., Jr.: Three-Dimensional Flow in Axial-Flow Turbomachinery, Part 1, Theoretical Determination of Secondary Flow. Rep. I-14, Mechanical Engineering Dept., The Johns Hopkins University, Nov. 1953.

5. Glassman, A.J.: Turbine Design and Application: NASA SP-290, Vols. 1,2,3, 1975.

6. Miller, G.R.; and Hartmann, M.J.: Experimental Shock Configurations and Shock Losses in a Transonic-Compressor Rotor at Design Speed," NACA RM-E58A14b, June 1958.

7. Adkins, G.G., Jr.; and Smith, L.H., Jr.: Spanwise Mixing in Axial-Flow Turbomachines. J. Eng. Power, vol. 104, no. 1, Jan. 1982, pp. 97-110.

8. Gallimore, S.J. and Cumpsty, N.A.: Spanwise Mixing in Multistage Axial Flow Compressors. ASME Paper 86-GT-20, June 1986.

9. Wright, L.C. and Novak, R.A.: Aerodynamic Design and Development of the General Electric CJ805-23 Aft-Fan. ASME Paper 60-WA-270, June 1960.

10. Crouse, J.E.; Janetzke, D.C.; and Schwirian, R.E.: A Computer Program for Computing Compressor Blading from Simulated Circular Arc Elements on Conical Surfaces. NASA TN D-5437, 1969.

11. Hearsey, R.M.: A Computer Program for Axial Compressor Design, Vol. 1, Theory Descriptions, and Users Instructions. AFAPL-TR-73-66-VOL-1, Dayton University, July 1973. (Avail. NTIS, AD-764733.)

12. Katsanis, T.; and McNally, W.D.: Revised FORTRAN Program for Calculating Velocities and Streamlines on the Hub-Shroud Midchannel Stream Surface of an Axial-, Radial-, or Mixed-Flow Turbomachine or Annular Duct, Vol. II - Programmer's Manual. NASA TN D-8431, 1977.

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

14. McNally, W.D.: FORTRAN Program for Calculating Compressible Laminar and Turbulent Boundary Layers in Arbitrary Pressure Gradients. NASA TN D-5681, 1970.

15. Garabedian, P.R. and Korn, D.G.: Numerical Design of Transonic Airfoils. Numerical Solution of Partial Differential Equations-II, B. Hubbard, ed., Academic Press, 1971, pp. 253-271.

16. Stephens, H.E.: Application of Supercritical Airfoil Technology to Compressor Cascades: Comparison of Theoretical and Experimental Results. AIAA Paper 78-1138, July 1978.

17. Davis, R.L.; Hobbs, D.E.; and Weingold, H.D.: Prediction of Compressor Cascade Performance Using a Navier-Stokes Technique. J. Turbomachinery, vol. 110, no. 4, Oct. 1988, pp. 520-531.

18. Thompkins, W.T., Jr.; and Oliver, D.A.: Three-Dimensional Flow Calculation for a Transonic Compressor Rotor. Through-Flow Calculations in Axial Turbomachinery, AGARD-CP-195, AGARD, Paris, France, 1976.

19. Denton, J.D.; and Singh, U.K.: Time Marching Methods for Turbomachinery Flow Calculation. Applications of Numerical Methods to Flow Calculations in Turbomachines, VKI-LEC-SER-1979-7, Von Karman Institute for Fluid Dynamics, Rhode-Saint-Genese, Belgium, 1979. (Avail. NTIS.)

20. Cedar, R.D.; and Holmes, D.G.: The Calculation of the Three-Dimensional Flow Through a Transonic Fan Including the Effect of Blade Surface Boundary Layers, Part-Span Shroud, Engine Splitter and Adjacent Blade Rows. ASME Paper 89-GT-325, 1989.

21. Moore, F.K. and Greitzer, E.M.: A Theory of Post-Stall Transients in Axial Compression Systems: Part I - Development of Equations. J. Eng. Gas Turbines Power, vol. 108, no. 1, Jan. 1986, pp. 68-76.

22. Sokolowski, D.E.: Toward Improved Durability in Advanced Aircraft Engine Hot Sections. NASA TM-4087, 1989.

23. Miller, B.A., et al.: A Perspective on Future Directions in Aerospace Propulsion System Simulation. NASA TM-102038, 1989.