NASA Technical Memorandum 104447
AIAA-91-2010

The Design/Analysis of Flows Through Turbomachinery
A Viscous/Inviscid Approach

D.P. Miller
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

and

D.R. Reddy
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park,, Ohio

NASA
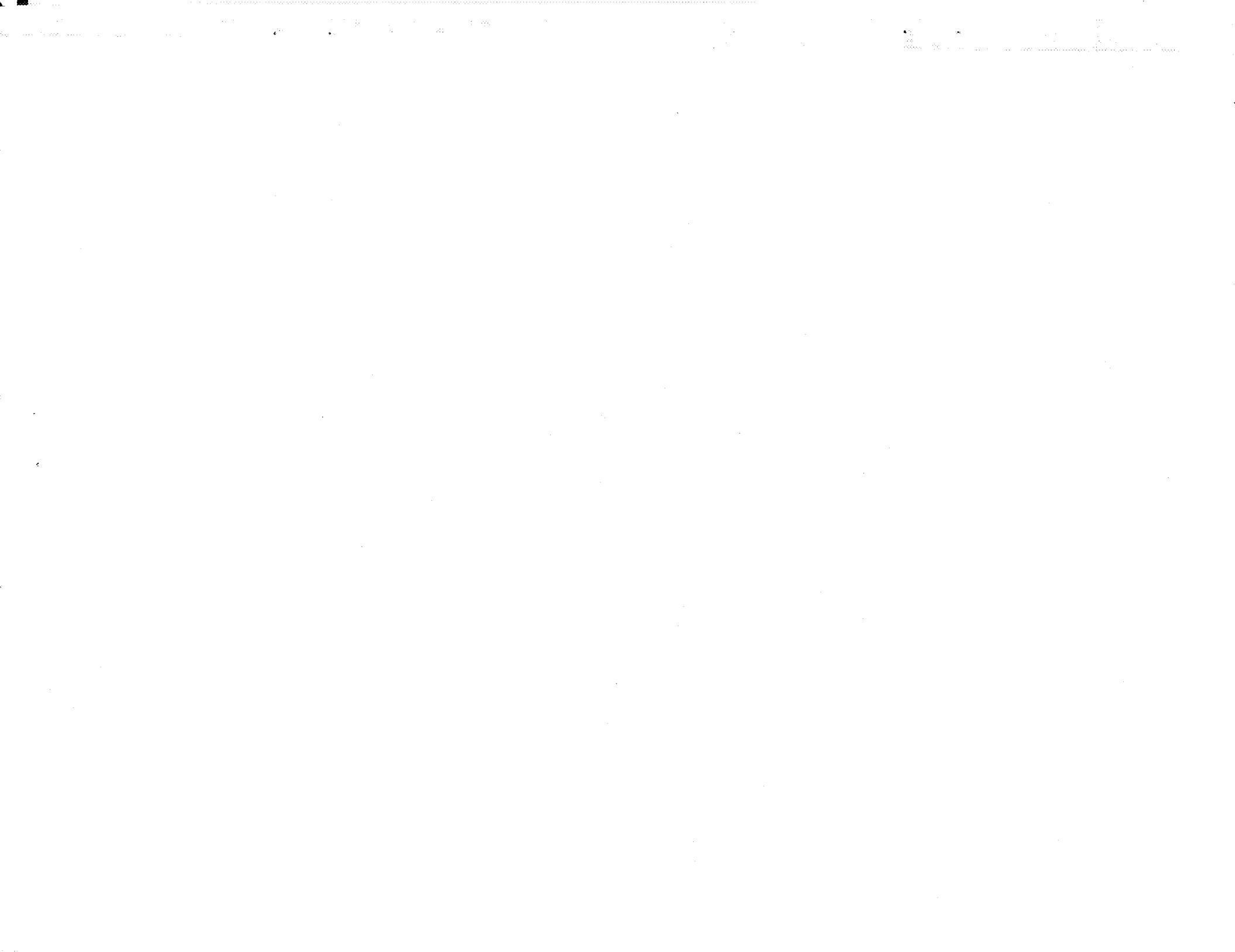
THE DESIGN/ANALYSIS OF FLOWS THROUGH TURBOMACHINERY
A VISCOUS/INVISCID APPROACH

D.P. Miller
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

D.R. Reddy*
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park, Ohio 44142

Abstract

A New Design/Analysis system for the flows through turbomachinery is currently being developed for studying turbomachinery problems with an axisymmetric viscous/inviscid "average-passage" throughflow code. The advantage of this approach, compared to streamline curvature codes, is that the solutions obtained simulate some of the unsteadiness, compressibility and viscous effects of a multistage turbomachine. The Design/Analysis system consists of three elemental parts, the axisymmetric block grid generator, the blade surface element code, and the axisymmetric flow code. Each element of the system will be discussed and the flow solutions for three axisymmetric geometries will be shown compared to experimental data where available. The computations are shown to be in very good agreement with test data for SR7 Spinner Body and Transonic Boattail Geometry obtained in the wind tunnels at NASA Lewis Research Center. While the VIADAC Rotor 67 Fan results were compared to PARC2D calculated results and shown to be in very good agreement.

Introduction

Early, the design methodology used for multistage turbomachinery has relied heavily on the streamline curvature approach [1-7] for computing the flows through modern gas turbine engines. These systems have their foundations in the early discussions of Wu [1] and Marble [2] on the three-dimensional flows in turbomachinery. Even though these methods have proven to be quite successful, the current approach has been to model the flows with Euler and Navier-Stokes formulations. This approach has been taken in the development of a new design/analysis system for flows through turbomachinery at NASA Lewis Research Center.

Smith [3], Novak [4], and Jennions and Stow [6,7] have all described in detail, the advantage of using a quasi-three-dimensional streamline curvature approximation to design. These formulations are based upon Wu's [1] work with S_1 and S_2 streamsurfaces. The throughflow calculations are done on the S_2 streamsurface, while the blade-to-blade computations are done on the S_1 streamsurface. In general, the axisymmetric computations are calculated with some built in empirical data base to account for secondary flows and spanwise mixing effects [8-12]. This methodology has been quite successful in computing the performance and design features of the blades for turbomachinery.

More recently, Ni [13] and Holmes [14] have described methods of computing the 3-D inviscid flows for isolated blade rows in turbomachinery, and Hah [15] and Chima [16] have developed methods of computing 3-D viscous flows for single-blade-row

turbomachinery problems. One limitation to all of these methods is that they are primarily used for analysis of single-blade-rows and not applicable to the design of turbomachinery.

Dawes [17], and Denton [18] have proposed various methods of calculating the 3-D flows in multi-blade-row turbomachinery by passing tangentially average flow quantities between blade rows in the computation. In Denton's method, the flow field effects of the adjacent blade row are axisymmetrically averaged at an axial station approximately mid-way between the blade rows. The flow calculations neglects any unsteady effects which might arise from the adjacent blade rows. The approach by Dawes is very similar to that of Denton, however, Dawes allows each blade row to be calculated either fully 3D, or axisymmetrically with forces, loss and deviation either obtained from a correlation or computed from a 3D solution.

Adamczyk's "average-passage" approach [19], describes a model in which the 3D computations are performed on each individual blade row with the results from adjacent blade rows being represented as force terms. 3D solutions obtained from individual blade rows are averaged circumferentially to obtain the force terms needed to perform the 3D calculations on the other blade rows. This method models the deterministic unsteadiness of the flow field through a multi-stage turbomachine in a time-averaged sense.

This "average-passage" approach is being used in the development of the new design/analysis flow solver. The flow code being developed at NASA Lewis Research Center is similar to the traditional quasi-3D design codes. The flow solver is axisymmetric and can be run inviscidly with assumed or calculated blockages, or with the viscous terms computed. The blade forces for each blade row are computed from blade-to-blade solutions, correlated data or force model, or from a full 3D solution. Blade-to-blade computations can be obtained with existing codes or with the blade-to-blade solver provided for the design system. This method represents an approximation to the full 3D computations. With the proper modeling of the spanwise mixing, secondary flow effects, and blade force terms, this model should closely represent 3D flows through multi-stage turbomachinery without actually having to compute the full 3D solutions.

The Design/Analysis System

The design and analysis codes which are currently under development at NASA Lewis Research Center have been separated into three distinct elements. These areas consist of the turbomachinery interactive grid generator energy distributor

restart code, TIGGERC, the interactive blade element geometry generator, IBEGG, and the viscous/inviscid multi-blade-row average passage flow solver, VIADAC. The block diagram, shown in figure 1, provides a flow chart as to how information is shared between codes.
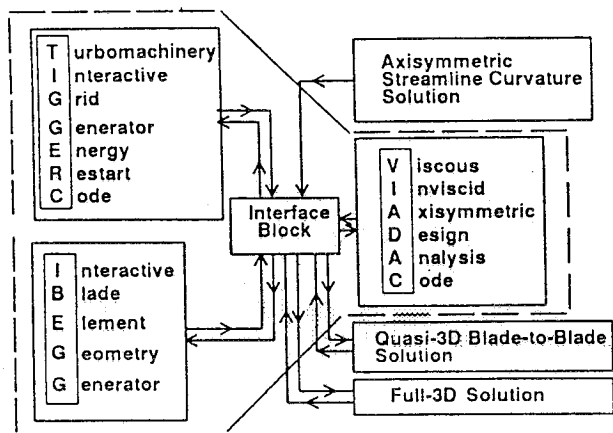

Figure 1. Flow Chart for NASA LERC Turbomachinery Design/Analysis System

The large blocked areas represent the activities currently being undertaken at Lewis Research Center. The other areas, such as the axisymmetric streamline curvature solution, quasi-3-D blade-to-blade solution and full 3-D solution, have already been developed as separate flow solvers. A quasi-3D blade-to-blade solver is being provided to compute the blade forces if no other method of obtaining these forces is available. The codes are modular and the design/analysis system can be used with existing codes.

Turbomachinery Interactive Grid Generator Energy Restart Code (TIGGERC)

One of the key elements in obtaining the axisymmetric flow solution is the grid generation. An interactive block grid generator, called TIGGERC, has been developed for any type of turbomachinery problem. The grid generator creates an axisymmetric multi-block grid for any duct , single row or multi-blade-row turbomachinery problem.

TIGGERC is fully interactive and can be used to modify flowpath coordinates and grid packing. The code can be run on the Silicon Graphics 4D/GT Personal Iris or any other Silicon Graphics Workstation. The input coordinates can be entered interactively or by an input data set. TIGGERC is mouse driven and several menu features have been added to make the system as general as possible. For example, Bezier curves are available to modify block boundary coordinates. Grids are generated using a hyperbolic tangent or algebraic distribution of points on the boundaries and the interior points of each block are distributed using a transfinite interpolation. Primarily, TIGGERC can generate a Blocked H-Grid for the axisymmetric code like the examples of a small multi-stage engine and NASA Rotor 67 Fan stage, shown in Figure 2. However, The grid generator can also be used to generate C-Grids, I-Grids or O-grids, as shown in Figure 3.

The code is general enough to handle any arbitrary geometry for 2D grids.

TIGGERC can be used to sequentially generate several different grids on the same geometry. This feature provides the ability to generate inviscid or viscous grids, repack the grid, and add more points along the surface, all without restarting the code. TIGGERC is to be linked to the Interactive Blade Element Geometry Generator by the leading and trailing edge locations of the blade, blade stacking, number of blades and blockage. A more detailed discussion on the menu system used by TIGGERC and the internal workings of the grid generator will be published later this year.

Interactive Blade Element Geometry Generator (IBEGG)

The second key element for the design system is the interactive blade element geometry generator, IBEGG. This code is a crucial part of any design/analysis system. The blade element code computes the upper and lower blade surface geometry, stacks the blade along a prescribed stacking axis, provides choke margin information for compressors, and allows the chord, solidity, thickness distributions, blade angle distribution or blade surface distributions to be altered interactively by the designer.
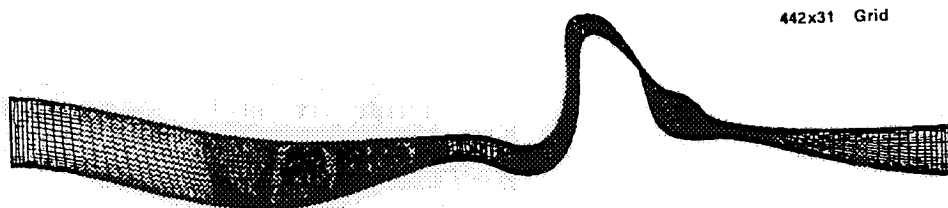
IBEGG will be developed to allow complete generality in turbomachinery blade geometry specification. This provides a means to analyze and design all types of blades, forward and aft swept fan blades, axial and radial compressors, and axial and radial turbines. These features will be incorporated into the design/analysis system currently being developed at NASA Lewis.

Either B-Spline data or Non-Uniform Rational B-Spline (NURBS) data will be used to generate surface coordinates for each blade row. This will allow the data to be compatible with CAD/CAM systems. Instead of storing many blade coordinates, it is hoped that the above methodology will provide easier access to blade coordinates for manufacturing and as well as for finite element modeling for structural and dynamic analysis. Since IBEGG is currently in the development stage, formal discussion of the code and the specific methodology of the internal workings of the code will be provided in a subsequent report.

Interface Block

In the middle of figure 1, there is a block which serves as an interface between elements. This block ties together the grid, geometry, and flow solvers. As shown in figure 4, there are various components common to all the major block elements. First, there is the geometry describing the turbomachinery problem one is interested in solving. The geometry consists of blade and flowpath information essential to the interactive grid generator, the viscous/inviscid axisymmetric design/analysis code, quasi-3-D blade-to-blade calculations, and the full 3-D solutions. This geometry is also important to the interactive blade element package, because once the blades have been created and the data stored, the blades can easily be reconstructed for further analysis.

A Simulated Small Axial-Centrifugal Engine
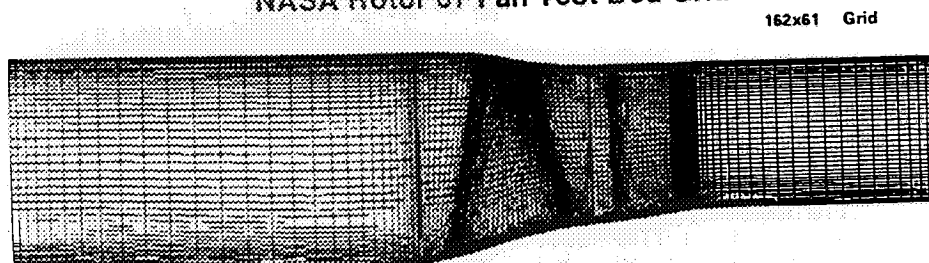
442x31   Grid

NASA Rotor 67 Fan Test Bed Grid

162x61   Grid

Figure 2. Examples of TIGGERC H-Grid Mesh Generation

O-Grid

I-Grid

Simulated Turbine Vane

NACA 64-410   C-Grid
Series Airfoil
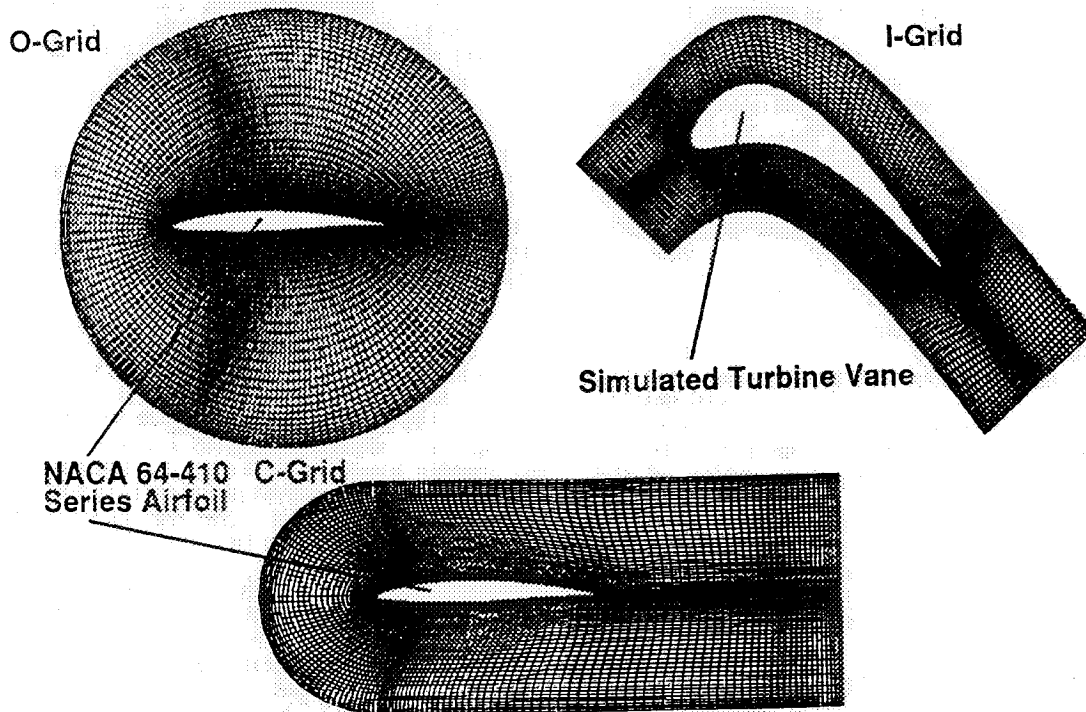
Figure 3. Examples of O-Grid, I-Grid and C-Grid Meshes

3

Interface Block

Geometry
Grid File
Flow File
Blade Forces
Blockage
  o Blade
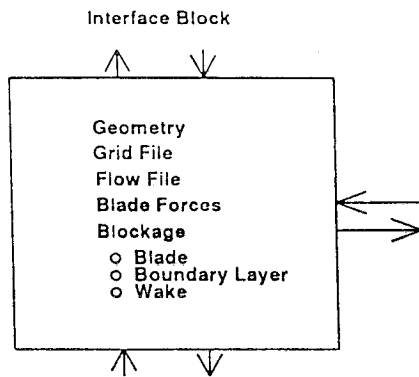 o Boundary Layer
  o Wake

Figure 4. Interfacing Block for LERC Turbomachinery
Design/Analysis System

The grid and flow file information is essential to all flow solvers. The grid file can be generated by the interactive grid code or independently from existing grid codes. The flow file connects all the elements together. If an axisymmetric streamline curvature solution has been computed, this can be used to find the flow field from the VIADAC computation or as a starting guess for the 3-D flow solvers. The 3-D solution can be averaged axisymmetrically and used for the VIADAC computation. This is an important feature of this design/analysis system which allows closure between the full 3D and axisymmetric approximation to the flow field. The grid file and flow file are written in a binary format compatible with the IRIS and Cray Computers in a PLOT3D readable format, and can be submitted directly to each computer without pre- and post-processing these files.

Another important feature of the interface block is the "Blade Forces". The "Blade Forces" as described by Adamczyk [19], are generally obtained from the 3-D solution, however, these forces can be obtained independently from quasi-3-D solutions or the forces can be modeled. These forces are important in order to compute the work for a single or multi-blade-row turbomachinery problem with the design/analysis code.

Finally, there are blade blockages, boundary layer blockages and blockage due to wakes or secondary flows. It should be pointed out that the boundary layers are computed if the axisymmetric flow code is run viscously. Wakes and secondary flow effects are to be modeled. These effects may also be included in the inviscid operation of the code in the form of a blockage. However, they are not essential to the operation of the code while running inviscidly. The blade blockage is always used while running the axisymmetric computation in the multi-blade-row or single-blade-row operation if a blade is present in the computation.

Viscous/Inviscid Axisymmetric Design/Analysis Code (VIADAC)

The third element of the design system is the flow code itself. VIADAC, the viscous/inviscid axisymmetric design/analysis code, is formulated as an "average-passage" axisymmetric multi-blade-row flow solver. The code uses a discretized cell

centered control volume approach with a four-stage Runge-Kutta Scheme. Residual smoothing has been added which allows a larger Courant number and accelerate the convergence. Second and fourth difference artificial dissipation terms are used to stabilize the solution. The flow code incorporates a Baldwin-Lomax algebraic turbulence model for boundary layer computations. Wall functions are automatically used when the grid stretching into the boundary layer falls outside the laminar sublayer. "Blade Forces" are used to simulate the effects of blade rows acting on the flow field. The spanwise mixing and secondary flow effects are currently being added to the flow solver.

The flow solver can be operated independently of the system being developed at NASA Lewis Research Center. Although the current approach to the design of turbomachines can perform the viscous flow computation, the design system being developed will have the ability to use a streamline curvature calculation as a starting guess and run without the shear terms being computed if desired.

Currently the axisymmetric part of VIADAC is operational, but the blade force and blockage terms have yet to be added. The modeling of the spanwise mixing and secondary flow effects are still being investigated. The axisymmetric code has been tested on three different configurations and the results will be examined below. A publication describing the details of the system of equations and other modeling in the flow solver will be published in the future.

Results and Discussion

Several test cases were run to validate the VIADAC code. These cases are representative of typical axial turbomachinery duct and axisymmetric wind tunnel body problems. Flow solutions were obtained from the NASA Rotor 67 Fan Test Program [20], herein referred to as Rotor 67 Fan Test Bed. Rotor 67 Fan was tested at NASA Lewis Research Center as a single stage compressor with laser anemometry data being acquired. There has been a great deal of valuable test data taken on Rotor 67 Fan and are available to compare with to the solutions obtained by VIADAC. Eventually this involves including the force terms to simulate the multi-blade-row operation of the average-passage axisymmetric code.

The SR7 Spinner axisymmetric body was a configuration which was used during a propfan test in the wind tunnels at NASA Lewis Research Center. Experimental data were available on the axisymmetric SR7 spinner geometry from an earlier test without the propfan blades attached to the spinner. The SR7 spinner has some interesting flow features along the spinner afterbody which test the axisymmetric code capabilities along with the grid generation.

A transonic boattail body, tested by Shrewsbury [21], used to study the performance of air-breathing propulsive systems of supersonic aircraft operating at subsonic cruise conditions has provided an opportunity to study shock-boundary layer interaction with VIADAC.

VIADAC was operated without blade forces for all the cases to be shown in this paper. First, the axisymmetric grid for each case was generated using TIGGERC. In Figure 5, the grids for the SR7 spinner
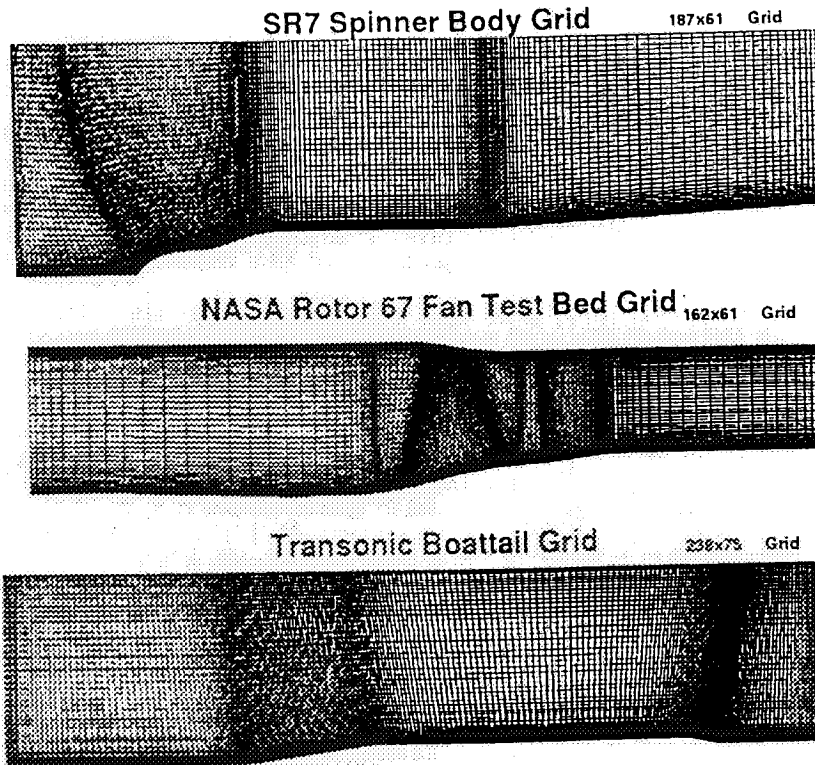
4

Figure 5. SR7, Rotor 67 Fan and Transonic Boattail Grid Geometries

body, Rotor 67 Fan and Transonic Boattail are shown in blocks generated by TIGERC. Notice that all the grids have different number of grid nodes. The grid packing was varied from case to case to refine the boundary geometry where important flow features were likely to occur. The grid for the Rotor 67 Fan was blocked to incorporate the actual measurement planes upstream and downstream of the rotor. The grid for the SR7 Spinner body was packed around the spinner nose where large gradients in the flow field are expected and at the downstream end of the spinner where there is a step in the boundary. The grid for the boattail model is packed on the afterbody, to allow accurate calculation of the transonic drag rise that occurs in this region.

The computed surface static pressures and isentropic mach numbers for the SR7 spinner body are shown in comparison to the test data, in Figures 6 and 7. These results were computed assuming a free stream boundary condition along the upper grid line. All the results are presented in nondimensional form. The test data was acquired downstream of the spinner portion of the body. Static Pressure taps were located behind the spinner along the afterbody. The afterbody becomes cylindrical from z/L=0.1101 to z/L=0.4540 and conical from z/L=0.4540 to z/L=1.00. The surface geometry of the nose of the spinner and part of the nacelle were smoothed to ensure convergence of the flow solver. The actual spinner geometry has a very small step along the hub which was later incorporated into the grid. The solution obtained for that geometry will be discussed later. The test data indicated a sharp drop with a sharp rise in the static pressure just aft of the spinner.
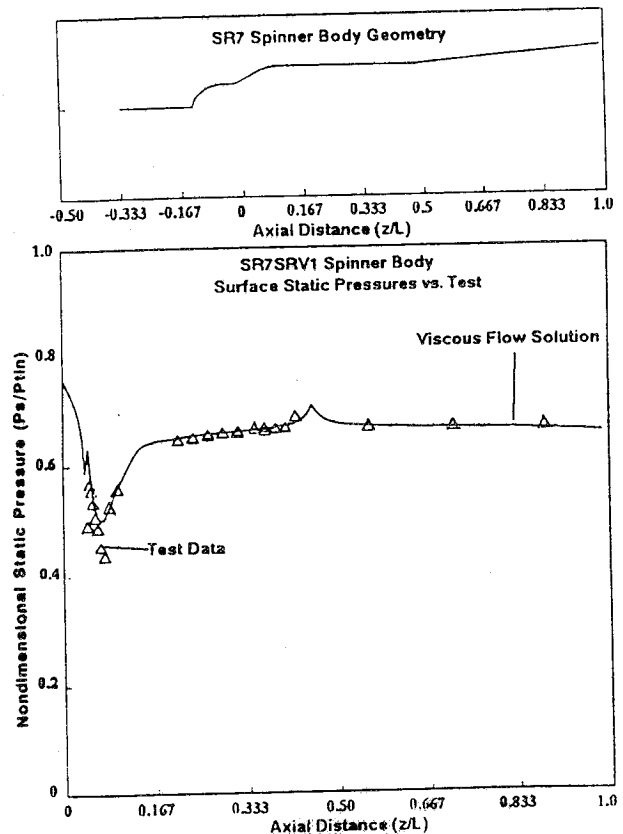

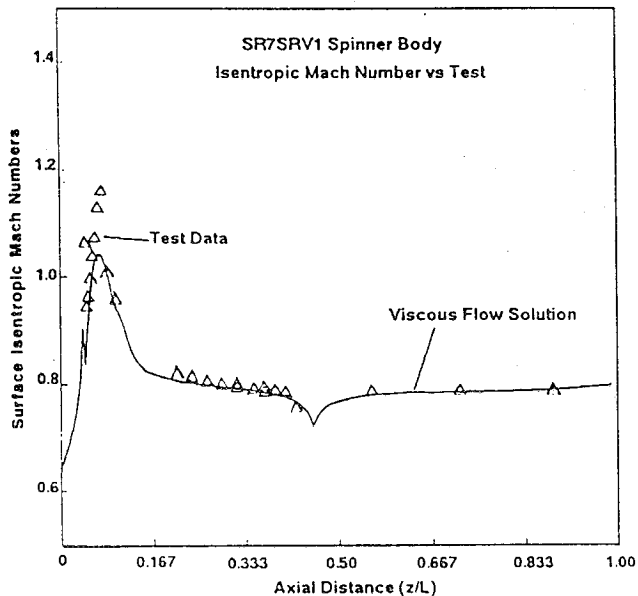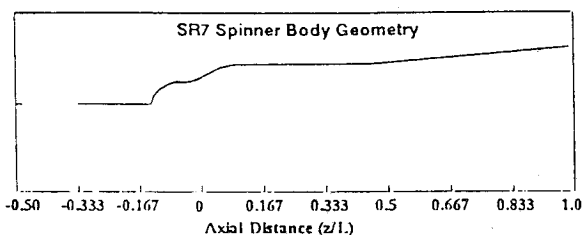Figure 6. Predicted Surface Static Pressure for SR7 Spinner Body

5

SR7 Spinner Body Geometry

Axial Distance (z/L)

SR7SRV1 Spinner Body
Isentropic Mach Number vs Test

Surface Isentropic Mach Numbers

Test Data

Viscous Flow Solution

Axial Distance (z/L)

Figure 7. Predicted Surface Insentropic Mach Numbers for
SR7 Spinner Body

This is due to a sonic bubble along the surface of the afterbody as shown by the isentropic Mach number plot in figure 7. The computed results show a weaker shock at this point than that indicated by the data, however, the results closely match the data downstream and even where the geometry changes from a cylindrical to conic section, around $z/L = 0.4540$, where a small rise in the static pressure is detected. The reason there is a weaker shock in the computation is due to smoothing of the boundary step ahead of the shock region. The sonic bubble can clearly be seen along the hub surface in the static pressure and Mach number contours shown in figure 8.

As mentioned earlier, the step for the SR7 spinner was eventually added to the flowpath and the solution computed. Particle traces shown in figure 9 along the wall near the step at $z/L=0.0425$ show a recirculation region and reattachment of the flow aft of the step. Another separated region existed upstream on the spinner body around $z/L=-0.030$. This region was near the portion of the spinner where the blades would be attached. The small separation is evident without the blades on the spinner. Once the blades have been added to the spinner, the blockage would increase, probably eliminating the separation.

Rotor 67 Fan was analyzed because of the large experimental data base for both an isolated fan rotor and a full stage. Since no experimental data existed on the "duct only" operation, a second analysis was run on the duct using the same grid. For this purpose, PARC2D [22] was used to perform a second set of computations for comparison with results obtained from VIADAC. PARC2D is a complete Navier-Stokes (2-D) flow solver in which the viscous terms can be selectively calculated with a thin-

Static Pressure Distribution for SR7 Spinner Body
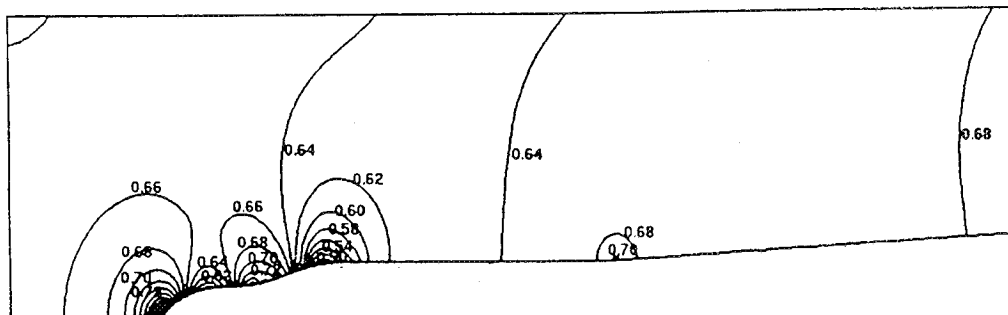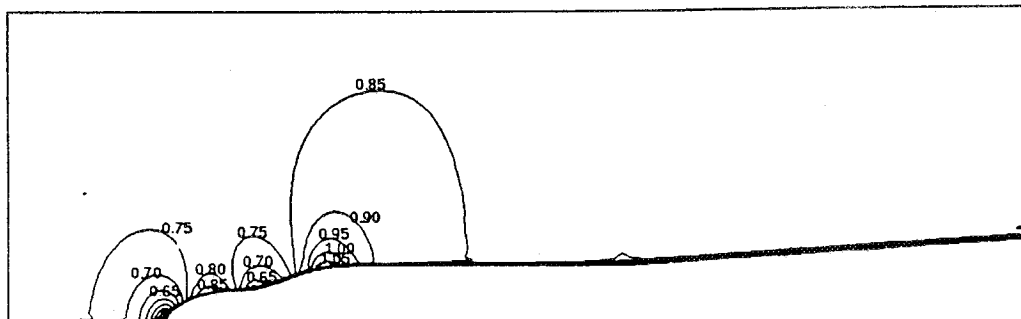
Mach Number Distribution for SR7 Spinner Body

Figure 8. Static Pressure and Mach Number Distributions
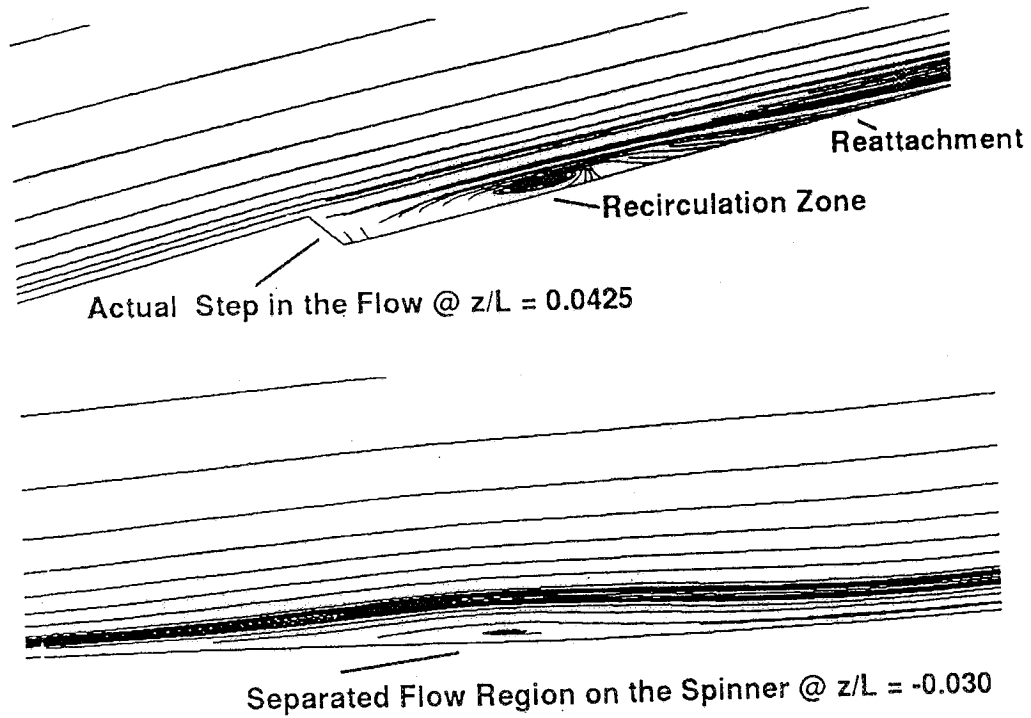For SR7 Spinner Body

Reattachment

Recirculation Zone

Actual Step in the Flow @ z/L = 0.0425

Separated Flow Region on the Spinner @ z/L = -0.030

Figure 9. Particle Traces Along the Endwall for the SR7 Spinner Body

Mach Number Distribution From VIADAC
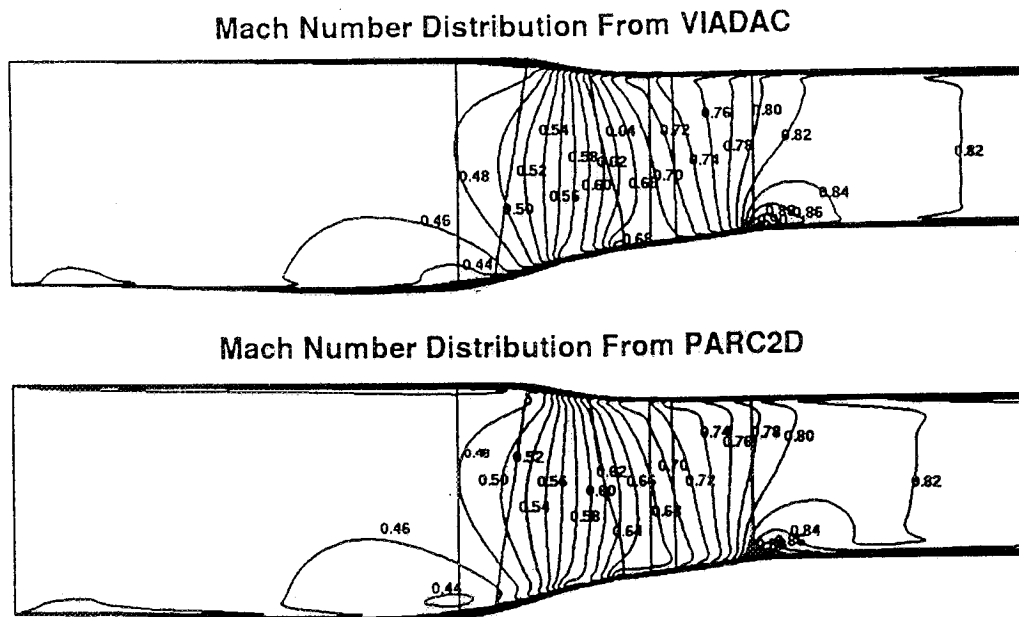

Mach Number Distribution From PARC2D

Figure 10. Mach Number Distributions From VIADAC and PARC2D
For Rotor 67 Fan Test Bed (Duct Only)

layer simulation. The code uses a Beam and Warming approximate factorization algorithm. The algorithm is an implicit central difference scheme with a implicit artificial dissipation with a Jameson-style second-order term for improved shock capturing. A comparison of the Mach number contours for the "duct only" are shown in figure 10, the results are in very good agreement. The surface static pressure distributions are shown in figure 11. The results
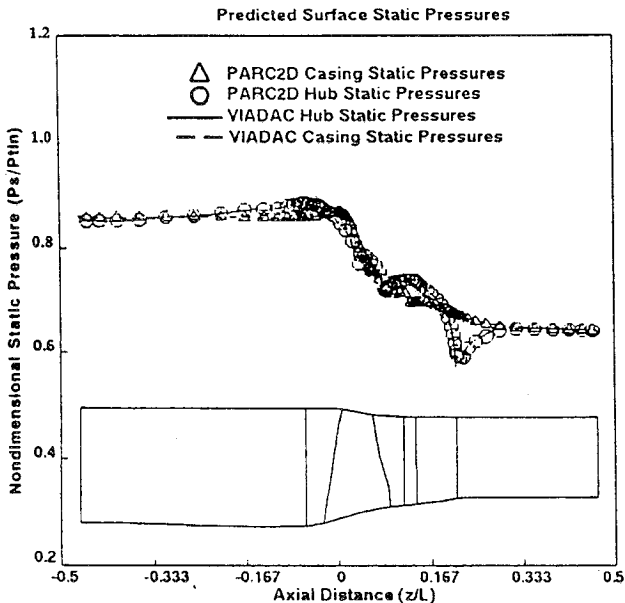
Figure 11. Predicted Surface Static Pressures From VIADAC and PARC2D for Rotor 67 Fan Duct
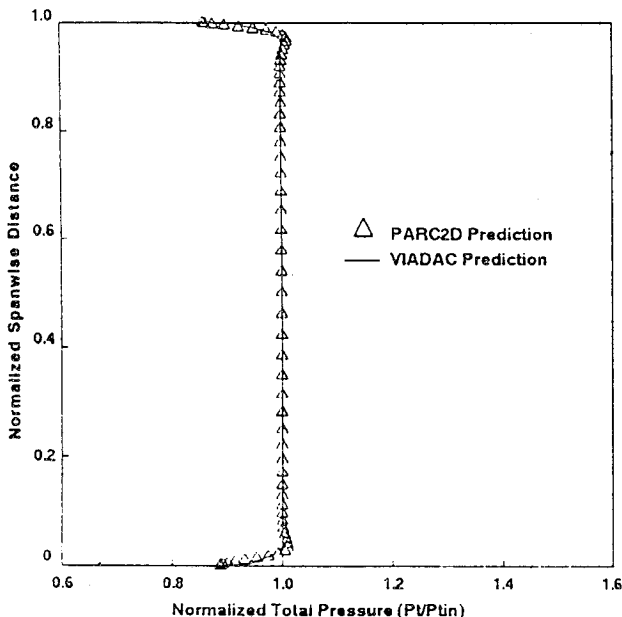
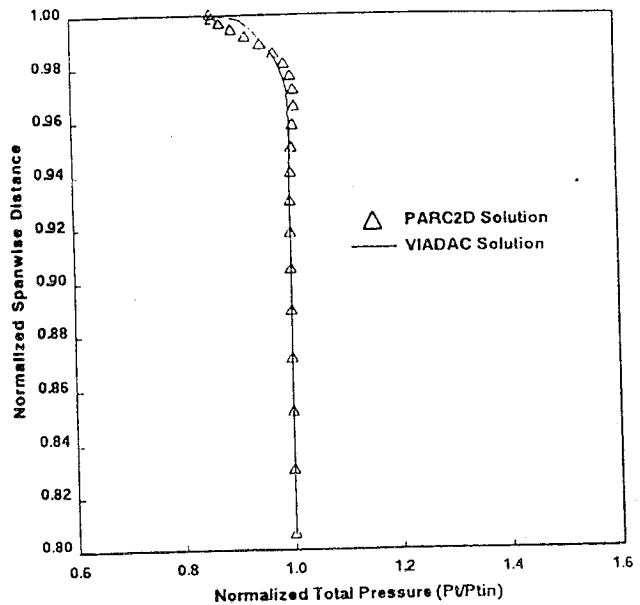Figure 12a. Calculated Radial Total Pressures from VIADAC and PARC2D @ z/L = -0.061 for Rotor 67 Fan Duct

Figure 12b. Boundary Layer Comparison of Total Pressure from VIADAC and PARC2D @ z/L = -0.061 for Rotor 67 Fan Duct

from the two codes are identical. Total pressure profiles at z/L=-0.061 just upstream of the leading edge of the rotor shown in figure 12a are in fairly good agreement. As shown in Figure 12b, PARC2D produces a very small overshoot in the total pressure near the edge of the boundary layer while no overshoot in the distribution is detected from the VIADAC computation. The artificial dissipation is turned off with the present code by locating the edge of the boundary layer in the computation, while PARC2D distributes the artificial dissipation to the walls. VIADAC used approximately 300 cpu sec on the Cray YMP and converged in about 3000 cycles. To completely eliminate the total pressure overshoot in PARC2D, PARC2D would require about 10,000 or more cycles to ensure proper convergence had been achieved. Eventually, this case will be run with VIADAC including the blade forces for the rotor only, then the stage.

Finally, solutions were obtained for an axisymmetric boattail geometry model [21], which had a boattail angle of 15 degrees and R/D=1.0, at three different Mach numbers to investigate the transonic drag rise and shock formation on the afterbody. The computed pressure coefficients for the three Mach numbers , Mach = 0.56, 0.70, and 0.90, are compared with wind tunnel data in figure 13 and are in very good agreement. The flow accelerates onto the afterbody at z/L=0.9 and forms a weak shock at a free stream Mach number of 0.90. This is more evident in the Mach contours shown in figure 14. The weak shock forms around z/L=0.94 and perhaps a very small separation region forms near the exhaust region around z/L=1.00. It is not clear from the test data as to whether this separation is large, small, or nonexistent, however, the code was able to predict the shock location fairly well.

Transonic Boattail Geometry

Axial Location (z/L)
-0.393   0   0.393   0.787   1.180

Predicted Coefficient of Pressures
Axisymmetric Transonic Boattail

Experimental Data	Predicted Data
O Mach = 0.56	—— Mach = 0.56
△ Mach = 0.70	- - - Mach = 0.70
◇ Mach = 0.90	— — Mach = 0.90

Coefficient of Pressure (Cp)

Axial Location (z/L)

Figure 13. Calculated Surface Pressures from VIADAC for a Transonic Boattail @ M∞ 0.56, 0.70 and 0.90

A new design/analysis system for viscous/inviscid flows through turbomachinery is currently being added to the flow codes at NASA Lewis Research Center. Grid generation is always a crucial part in any analysis using a Navier-Stokes flow code. TIGGERC is able to grid sophisticated geometry shapes and versatile enough to create the grid density and packing necessary for computing complex flows. This is a crucial step in providing a significant capability in solving the flows in turbomachinery. As demonstrated, the ability to grid real geometries, such as the step in the spinner hub surface, is important in computing real physics in the flow field. The ability to interactively regrid and rerun the flow solver with short turn-around times provides a capability not available in the past.

Solutions obtained from the axisymmetric viscous flow solver, VIADAC, have shown to be in good agreement with experimental data and solutions obtained from PARC2D. The SR7 spinner and transonic boattail examples established the ability of the codes to solve and analyze complex geometries. The ability of the code to predict separation and shock boundary layer interaction is critical, since the code is to be run in all types of regimes.
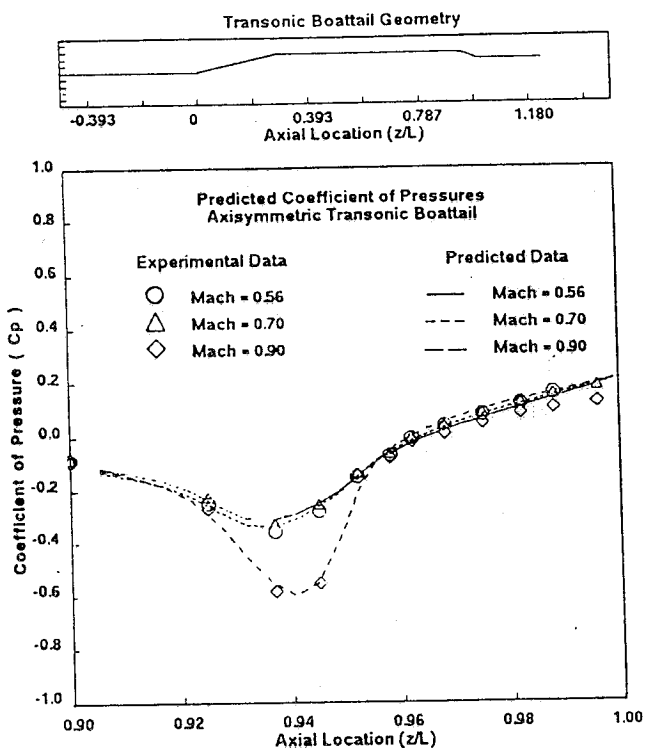

Mach Contours @ M∞ = 0.56

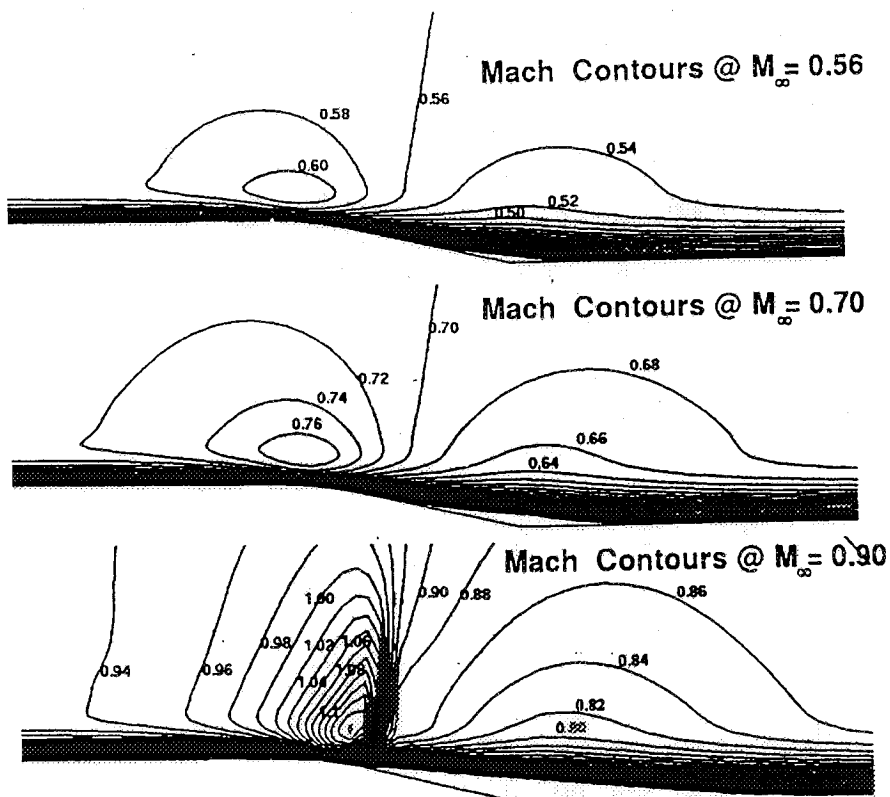Mach Contours @ M∞ = 0.70

Mach Contours @ M∞ = 0.90

Figure 14. Computed Mach Number Contours for the Transonic Boattail AfterBody

The comparisons between VIADAC and PARC2D were shown to be in good agreement. The solutions obtained for Rotor 67 fan are only a starting point in the development of VIADAC. Currently, blade forces for the rotor are being added to the solution. Future efforts will include adding a more sophisticated model for spanwise mixing and turbulent diffusion for each blade row.

The grid generator is complete and has already proved to be a valuable tool in investigating real physics. The flow solver has provided excellent agreement to "duct only" flow fields and is currently being upgraded to include the forces necessary to compute the flows in multi-blade-row turbomachinery problems. Future efforts will include the completion of the surface geometry generator for the design of the blades.

Acknowledgements

References

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

[2] Marble, F. E., "Three-Dimensional Flow in Turbomachines," Princeton Series on High Speed Aerodynamics and Jet Propulsion, Princeton U. Press, Vol. 10, Sec. C, 1964, pp. 83-166.

[3] Smith, L. H., Jr., "The Radial Equilibrium Equation of Turbomachinery," ASME Journal of Engineering for Power, Jan. 1966, pp. 1-12.

[4] Novak, R. A., "Streamline Curvature Computing Procedures for Fluid-Flow Problems," ASME Journal of Engineering for Power, October 1966, pp. 1-13.

[5] Wennerstrom, A. J., "On the Treatment of Body Forces in the Radial Equilibrium Equation of Turbomachinery," ARL Report No. 75-0052, April 1974, pp. 351-367.

[6] Jennions, I. K., and Stow, P., "A Quasi-Three-Dimensional Turbomachine Blade Design System: Part 1. Throughflow Analysis," ASME Paper No. 84-GT-26, presented at the 29th Int. Gas Turbine Conference, Amsterdam, 1984.

[7] Jennions, I. K., and Stow, P., "A Quasi-Three-Dimensional Turbomachine Blade Design System: Part 2. Computerized System," ASME Paper No. 84-GT-27, presented at the 29th Int. Gas Turbine Conference, Amsterdam, 1984.

[8] Serovy, G. K., "Recent Progress in Aerodynamic Design of Axial Flow Compressors in the United States," ASME Paper No. 66-GT-95, 1966, pp.1-11.

[9] Adkins, G. G., Jr., and Smith, L. H., Jr., "Spanwise Mixing in Axial-Flow Turbomachines," ASME Journal of Engineering for Power, January 1982, Vol. 104, pp. 97-110.

[10] De Ruyck, J., and Hirsch, Ch., "A Radial Mixing Computation Method," ASME Paper No. 88-GT-68, Presented at the Gas Turbine and Aeroengine Congress, Amsterdam, June 1988, pp. 1-14.

[11] Gallimore, S. J., and Cumpsty, N. A., "Spanwise Mixing in Multistage Axial Flow Compressors: Part I-Experimental Investigation," ASME Journal of Turbomachinery, Vol. 108, July 1986, pp. 2-9.

[12] Gallimore, S. J., "Spanwise Mixing in Multistage Axial Flow Compressors: Part II-Throughflow Calculations Including Mixing," ASME Journal of Turbomachinery, Vol. 108, July 1986, pp. 10-16.

[13] Ni, R. H., "A Multiple-Grid Scheme for Solving Euler Equations," AIAA Journal, Vol. 20, No. 11, November 1982, pp. 1565-1571.

[14] Holmes, D. G., and Tong, S. S., "A Three-Dimensional Euler Solver for Turbomachinery Blade Rows," Journal of Engineering for Power, 1984, pp. 1-7.

[15] Hah, C., "A Navier-Stokes Analysis of Three-Dimensional Turbulent Flows Inside Turbine Blade Rows at Design and Off-Design Conditions," Journal of Engineering for Power, Vol. 106, April 1984, pp. 421-429.

[16] Chima, R. V., and Yokota, J. W., "Numerical Analysis of Three-Dimensional Viscous Internal Flows," AIAA Journal, Vol. 28, No. 5, May 1990, pp. 789-806.

[17] Dawes, W. N., "Towards Improved Throughflow Capability: The Use of 3D Viscous Flow Solvers in a Multistage Environment," ASME Paper No. 90-GT-18, Presented at the Gas Turbine Congress, Brussels, June 1990, pp. 1-10.

[18] Denton, J. D., "The Calculation of Three Dimensional Viscous Flow Through Multistage Turbomachines," ASME Paper No. 90-GT-19, Presented at the Gas Turbine Congress, Brussels, June 1990, pp. 1-10.

[19] Adamczyk, J. J., Celestina, M. L., Beach, T. A., and Barnett, M., "Simulation of 3-D Viscous Flow Within a Multi-Stage Turbine," NASA TM-101376, Prepared for the 34th Int. Gas Turbine and Aeroengine Congress, Toronto, June 1989, pp. 1-11.

[20] Strazisar, A. J., Wood, J. R., Hathaway, M. D., and Suder, K. L., "Laser Anemometer Measurements in a Transonic Axial-Flow Fan Rotor,", NASA Technical Paper No. 2879, November 1989.

[21] Shrewsbury, G. D., "Effect of Boattail Juncture Shape on Pressure Coefficient of Isolated Afterbodies. NASA TM x-1517, 1968.

[22] Cooper, G. K. , "The PARC Code: Theory and Usage," AEDC-TR-87-27, October 1987, pp. 1-123.

NASA

National Aeronautics and
Space Administration

Report Documentation Page

1. Report No. NASA TM-104447 AIAA-91-2010	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle The Design/Analysis of Flows Through Turbomachinery A Viscous/Inviscid Approach		5. Report Date
		6. Performing Organization Code
7. Author(s) D.P. Miller and D.R. Reddy		8. Performing Organization Report No. E-6288
		10. Work Unit No. 505-62-10
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191		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, D.C. 20546-0001		14. Sponsoring Agency Code

16. Abstract

A New Design/Analysis system for the flows through turbomachinery is currently being developed for studying turbomachinery problems with an axisymmetric viscous/inviscid "average-passage" throughflow code. The advantage of this approach, compared to streamline curvature codes, is that the solutions obtained simulate some of the unsteadiness, compressibility and viscous effects of a multistage turbomachine. The Design/Analysis system consists of three elemental parts, the axisymmetric block grid generator, the blade surface element code, and the axisymmetric flow code. Each element of the system will be discussed and the flow solutions for three axisymmetric geometries will be shown compared to experimental data where available. The computations are shown to be in very good agreement with test data for SR7 Spinner Body and Transonic Boattail Geometry obtained in the wind tunnels at NASA Lewis Research Center. While the VIADAC Rotor 67 Fan results were compared to PARC2D calculated results and shown to be in a very good agreement.

17. Key Words (Suggested by Author(s)) Turbomachinery Design/Analysis system Viscous Inviscid	18. Distribution Statement Unclassified - Unlimited Subject Category 07		
19. Security Classif. (of the report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 12	22. Price* A03

NASA FORM 1626 OCT 86   *For sale by the National Technical Information Service, Springfield, Virginia 22161

National Aeronautics and
Space Administration

Lewis Research Center
Cleveland, Ohio 44135

Official Business
Penalty for Private Use $300

FOURTH CLASS MAIL

ADDRESS CORRECTION REQUESTED

NASA