Simulation of 3-D Viscous Flow Within a Multi-Stage Turbine

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

Mark L. Celestina and Tim A. Beach
Sverdrup Technology, Inc.
NASA Lewis Research Center Group
Cleveland, Ohio

and

Mark Barnett
United Technologies Research Center
Hartford, Connecticut

Prepared for the
34th International Gas Turbine and Aeroengine Congress and Exposition
sponsored by the American Society of Mechanical Engineers
Toronto, Canada, June 4–8, 1989
ABSTRACT

This work outlines a procedure for simulating the flow field within multistage turbomachinery which includes the effects of unsteadiness, compressibility, and viscosity. The associated modeling equations are the average passage equation system which governs the time-averaged flow field within a typical passage of a blade row embedded within a multistage configuration. The results from a simulation of a low aspect ratio stage and one-half turbine will be presented and compared with experimental measurements. It will be shown that the secondary flow field generated by the rotor causes the aerodynamic performance of the downstream vane to be significantly different from that of an isolated blade row.

INTRODUCTION

A goal of computational fluid dynamics for turbomachinery is the prediction of performance parameters and the flow processes which set their values. Achieving this goal for multistage devices in made difficult by the wide range of length and time scales in the associated flow fields. Currently the procedure used in design and off-design analysis is based on a quasi-three-dimensional flow model whose origins can be traced back to the late forties and early fifties, e.g., (Wu, 1952; Smith, 1966). This model requires calculations to be executed on two orthogonal surfaces within a blade row passage of a multistage configuration. One of these surfaces is an axisymmetric surface of revolution whose intersection with a blade row defines a cascade. The flow field relative to this cascade is assumed to be steady in time. In practice, a finite number of such surfaces are chosen to define a series of cascade flows from hub to shroud. The other surface represents a meridional through-flow surface. The flow associated with this surface is an axisymmetric representation of the flow field within the machine. This flow field is also assumed to be steady. The flow fields on both surfaces are coupled and are solved iteratively. The effects of unsteadiness, turbulence, and endwall secondary flows are introduced through empirical correlations.

Although proven to be very useful, this flow model has its limitations. Among these is off-design performance analysis, and the ability to analyze unconventional machinery where extrapolation of the underlying empirical database is required. Other problems arise whenever there are large local variations in the radial velocity component within a blade passage. Such variations can be brought about by in-passage shock waves, separated boundary layers, and endwall secondary flows. It is generally agreed upon that a way of overcoming these shortcomings is the development of a true three-dimensional flow model.

Two three-dimensional flow models have been proposed for the simulation and analysis of multiple blade row flows. The first (Denton, 1979; Adamczyk 1984; NI, 1987) referred to as the average passage flow model in Adamczyk (1986) simulates the time-averaged flow field within a typical passage of the blade row. The second simulates the unsteady deterministic flow field within the machine. Although a number of unsteady simulations of single stage turbine configurations and counterrotating propellers have been reported (Rai, 1987; Whitfield et al., 1987), executing an unsteady simulation of a multistage configuration of practical interest is far beyond the capabilities of today's advanced supercomputers. Furthermore, it is by no means obvious that performance prediction requires such a high degree of flow resolution. However, because unsteady simulations of an idealized configuration may prove to be a useful means of investigating the closure issue associated with time-averaged flow models, this activity should be pursued. In this work it will be shown that the simulation of the time-averaged flow field within multistage machinery is within the capabilities of today's advanced computers and that the average passage flow model gives more insight into the flow phenomena that control the performance of multistage machinery than today's quasi-three-dimensional flow models.

The objective of this paper is to outline a procedure for simulating the time-averaged flow field within a typical passage of a blade row within a multistage machine. This model includes the effects of viscosity and compressibility, and the influence of neighboring blade rows. The mathematical formulation upon which
this model is based has been outlined in Adamczyk (1984). The algorithm used to solve the inviscid form of the governing equations is reported in Celestina, (1986); Adamczyk et al. (1986). The current work outlines a numerical solution procedure for the viscous form of these equations and an acceleration technique to enhance convergence. In addition, a comparison will be made between experimental data recorded during tests of a one and one-half stage, large scale, low speed, axial flow research turbine and simulation prediction. The underlying unsteady flow physics which appears to control the performance of the second vane of this machine will also be discussed.

GOVERNING EQUATION

A complete derivation of the three-dimensional average-passage equation system is presented in Adamczyk (1984). These equations were derived by filtering the Navier Stokes equation in both space and time to remove all information except that associated with the time average flow field within a typical passage of a blade row of a multi-stage configuration. With respect to this blade row, the integral form of these equations can be written

\[ \int \frac{\partial}{\partial t}(\lambda(q))dV + L(\lambda q) = \int \lambda S dV + \sum \alpha_k dV + L_v(\lambda q) \] (1)

The vector \( q \) contains variables density, axial and radial momenta, angular momenta, and total internal energy. \( \lambda \) is the neighboring blade row blockage factor and ranges between zero and unity, unity being the value associated with zero blade thickness. This parameter explicitly introduces the effect of the neighboring blade row blade thickness. The operator \( L(\lambda q) \) balances the mass, axial and radial momenta, angular momentum, and energy through a control volume.

\[ \sum \alpha_k dV \] is a source term due to the cylindrical coordinate system and \( \int \lambda S dV \) contains the body forces, energy sources, momenta, and energy temporal and spatial mixing correlations associated with the neighboring blade rows. A procedure for estimating \( S \) has been outlined in Adamczyk et al. (1986) and is extended here to include the effects of viscosity. The operator \( L_v(\lambda q) \) contains the viscous and heat transfer terms. The vector \( q \) and the operators \( L \) and \( L_v \) are defined as

\[ q = [p, p v_z, p v_r, \rho v^2, \rho v_\theta, \rho v_r^2, \rho v^2 v_z, \rho v^2 v_r, \rho v^2 v_\theta] \] (2)

\[ L = \int [\lambda F dA_z + \lambda G dA_r + \lambda H dA_\theta] \] (3)

and

\[ L_v = \int [\lambda F_v dA_z + \lambda G_v dA_r + \lambda H_v dA_\theta] \] (4)

where

\[ F = [p v_z, pv^2, p v_r, \rho v^2 v_z, \rho v^2 v_r, \rho v^2 v_\theta] \] (5)

\[ G = [p v_r, pv^2 v_r, p v^2, \rho v^2 v_\theta, \rho v^2 v_r, \rho v^2 v_\theta] \] (6)

\[ H_v = [0, v_{12}, v_{13}, v_{14}, v_{15}, v_{16}] \] (7)

\[ F_r = [0, \tau_{rr}, \tau_{rz}, \tau_{r\theta}, \tau_{r\phi}] \] (8)

\[ G_r = [0, \tau_{rz}, \tau_{rr}, \tau_{r\theta}, \tau_{r\phi}] \] (9)

\[ H_y = [0, \tau_{\theta z}, \tau_{\theta r}, \tau_{\theta \phi}, \tau_{\theta \phi}, \tau_{\theta \phi}] \] (10)

\[ \tau_{zz} = 2 \mu \frac{\partial v_z}{\partial z} + \lambda \frac{\partial^2 v_z}{\partial z^2} \] (11)

\[ \tau_{rr} = \tau_{zz} = \mu \left( \frac{\partial v_r}{\partial r} + \frac{\partial v_z}{\partial z} \right) \] (12)

\[ \tau_{r\theta} = \mu \left( \frac{\partial v_r}{\partial \theta} + \frac{\partial v_\theta}{\partial r} \right) \] (13)

\[ \tau_{\theta \phi} = \mu \left( \frac{\partial v_\theta}{\partial \phi} + \frac{\partial v_\phi}{\partial \theta} \right) \] (14)

\[ \tau_{r\phi} = \mu \left( \frac{\partial v_r}{\partial \phi} + \frac{\partial v_\phi}{\partial r} \right) \] (15)

\[ \tau_{\theta \phi} = \mu \left( \frac{\partial v_\theta}{\partial \phi} + \frac{\partial v_\phi}{\partial \theta} \right) \] (16)

\[ q_z = v_z + v_r \tau_{rz} + v_\theta \tau_{r\theta} \] (17)

\[ q_r = v_r + v_r \tau_{rr} + v_\theta \tau_{r\theta} + k \frac{\partial T}{\partial r} \] (18)

\[ q_\theta = v_\theta + v_r \tau_{r\theta} + v_\theta \tau_{r\theta} + k \frac{\partial T}{\partial \theta} \] (19)

\[ k = \begin{bmatrix} 0, & 0, & 0, \rho v^2 + p, & -\tau_{r\phi}, & 0, & 0 \end{bmatrix} \] (20)

In the above equations \( p \) represents the density, \( \vec{v} \) the absolute velocity vector, \( p \) the pressure, and \( T \) the temperature. The differential \( dV \) is the volume of the control volume, \( dA_z, dA_r, dA_\theta \) the differential areas of its sides. From the equation of state, the total internal energy is related to pressure through the equation

\[ e_o = \frac{p}{\rho(T - 1)} + \frac{1}{2} |\vec{v}|^2 \] (21)

and the total enthalpy, \( H \), is related to \( p \) and \( e_o \) by

\[ H = e_o + \frac{p}{\rho} \] (22)

Sutherland's law is used to determine the molecular viscosity coefficient, and Stokes' hypothesis gives \( \lambda_v = -2/3 \mu \). Turbulence is accounted for by adding a turbulent viscosity \( \mu_t \) to the molecular viscosity \( \mu \).
\[ \mu = \mu_0 + \mu_t \]  
(23)

In a similar manner, the molecular thermal conductivity \( k \) is replaced by

\[ k = C_p \left[ \frac{\mu_0}{\rho} + \frac{\mu_t}{\rho} \right] \]  
(24)

where \( C_p \) is the specific heat at constant pressure and \( P_r_0 + P_r_t \) is the laminar and turbulent Prandtl number respectively. The two-layer algebraic model of Baldwin and Lomax (1978) is used to model \( \mu_t \).

All lengths in the above equations are nondimensionalized by a reference length normally taken as the largest blade row diameter. The velocity components are nondimensionalized by a reference speed of sound, \( \rho_0 \) where \( \gamma \) is the ratio of specific heats.

Pressure and density are nondimensionalized by a reference speed of sound, \( \rho_0 \) and are estimated using central differences of the velocity field. The shear stresses and heat flux are evaluated at the surface area and volume is included in the calculations. The construction of this algorithm is as follows. First, the through-flow equations of the axisymmetric component of a variable was a significant fraction of the magnitude of the error associated with the variable itself. It thus seems reasonable to expect a solution algorithm which explicitly reduces the error of the axisymmetric component of a variable. This is accomplished by premultiplying Eq. (1) by the transformation

\[ N_{ABSOLUTE} = N_{RELATIVE} + \Delta t \]  
(25)

where \( N \) is the rotational wheel speed. Introducing Eq. (24) into Eq. (1) transforms \( L \) and \( L_v \) to

\[ L = \int_{\Omega} \left( \frac{\partial F}{\partial v} dA_v + \lambda \frac{\partial F}{\partial r} dA_r + \lambda \frac{\partial F}{\partial \theta} dA_\theta \right) \]  
(26)

and

\[ L_v = \int_{\Omega} \left( \frac{\partial F}{\partial v} dA_v + \lambda \frac{\partial F}{\partial r} dA_r + \lambda \frac{\partial F}{\partial \theta} dA_\theta \right) \]  
(27)

Discretization of the inviscid portion of Eq. (1) including estimates of the surface area and volume is presented in Celestina et al. (1986). The viscous and heat transfer portion of Eq. (1) is discretized by evaluating the shear stress and the heat flux at the center of each face. The shear stresses and heat flux are estimated using central differences of the velocity and the temperature field.

ARTIFICIAL DISSIPATION

To suppress odd-even point decoupling of the solution to the discretized equations, dissipative terms are added to the equations. The operator \( L_v \) is replaced in the discretized form of Eq. (1) with \( D(\lambda q) \)

\[ D(\lambda q) = D_1(q) + L_v(\lambda q) \]  
(28)

where \( D_1(q) \) is the added artificial dissipation operator needed to prevent decoupling of the solution within inviscid regions of the flow. The operator \( D_1(q) \) is patterned after the model developed by Jameson et al. (1981) and is composed of three spatial operators.

\[ D_1(q) = (D_2 + D_3 + D_4)q \]  
(29)

which can be evaluated separately. The dissipation in, for example, the axial direction (and similarly for the others) is expressed as follows:

\[ D_1 = d_1 + 1/2, j, k - d_1 - 1/2, j, k \]  
(30)

where

\[ d_1 + 1/2, j, k = \beta_{1+1/2, j, k}(\Delta_2 q_{1+1/2, j, k}) \]  
(31)

\[ e_{1+1/2, j, k} = \beta_{1+1/2, j, k} \min(v_{1, j, k}; v_{1+1, j, k}; 0.5) \]  
(32)

\[ e_{1+1/2, j, k} = \max(0, \kappa(4) \beta_{1+1/2, j, k} \min(1 + 1/2, j, k) \]  
(33)

and

\[ \kappa(2), k(4) \]  

are constants set at 1/8 and 1/512 respectively. The symbols \( \Delta \) and \( \Delta^2 \) denote the first and second difference operators, while \( \beta \) is the maximum eigenvalue of the Jacobian matrix formed from \( F \).

The coefficient \( v_{1, j, k} \) is defined as

\[ v_{1, j, k} = \frac{P_{1+1, j, k} - P_{1-1, j, k} + P_{1-1, j, k}}{P_{1+1, j, k} + P_{1-1, j, k}} \]  
(34)

and is used primarily to prevent oscillations near stagnation points and shocks.

Vatsa and Wedan (1988) scaled the Jameson artificial dissipation operator by a function of the local Mach number to reduce its effect within viscous regions of the flow. The present work uses the local meridional Mach number \( M \) normalized by a reference upstream meridional Mach number \( M_0 \) to accomplish this task. To prevent this function from increasing the level of dissipation in the inviscid flow regions, the maximum value of this function is taken as unity. To reduce the level of artificial viscosity resulting from highly stretched mesh cells, we also adopted what is referred to by Vatsa and Wedan as individual eigenvalue scaling of the artificial dissipation operator.

SOLUTION PROCEDURE

The discretized form of the equations associated with the averaged passage flow model are solved using a dimensional sequencing algorithm. The motivation for the present algorithm came from observing the evolution of the error history associated with the algorithm reported in Celestina (1986). For many cases the magnitude of the error associated with the axisymmetric component of a variable was a significant fraction of the magnitude of the error associated with the variable itself. It thus seems reasonable to expect a solution algorithm which explicitly reduces the error of the axisymmetric component of the flow field would enhance the convergence of the three-dimensional field. With the body forces assumed known, (i.e., \( S \) assumed known) the algorithm iterates between the three-dimensional flow equations and the corresponding through-flow equations to enhance the rate of convergence to the three-dimensional time asymptotic flow problem. The construction of this algorithm is as follows. First, the through-flow equations compatible with Eq. (1) are derived by summing Eq. (1), as modified according to the discussion in the preceding sections, over the tangential index \( k \). This is accomplished by premultiplying Eq. (1) by the operator

\[ D = \frac{1}{k} \sum_{k=1}^{K} \]  
(35)
where \( K \) is the number of control volumes spanning the pitch. The result may be written as
\[
\frac{d}{dt} \lambda q \, dv + \mathcal{L}(q) - \mathcal{D}(q) = \int \mathcal{L} \lambda \, dv + \mathcal{L} \int S \, dv + \left\{ \mathcal{L} \lambda, \mathcal{D}(q) \right\}
\]
where \( dv, \lambda, q \) are defined as
\[
\lambda = \lambda q \, dv = \lambda q \, dv
\]
while the operators \( \mathcal{L}, \mathcal{L}, \text{and} \mathcal{D} \) are
\[
\mathcal{L} \lambda = \lambda q \, dv = \lambda q \, dv \quad \text{and} \quad \mathcal{D}(q) = \mathcal{L}(q) - \lambda q \, dv
\]
The terms which appear on the right hand side of Eq. (36) are treated as forcing functions and are estimated using the most recent value of the three-dimensional flow variables. Note that the expressions which appear within brackets vanish by construction in regions of the flow where \( q = q \), and upon convergence of Eq. (36) (i.e., \( \partial / \partial t \lambda q \, dv = 0 \)), \( \mathcal{L} \lambda q \, dv = q \). The time asymptotic solution of Eq. (36) is thus identical to the axissymmetric average of the time asymptotic solution to Eq. (1). The steady state solution of Eq. (1) can thus be obtained by cycling between a time-advancing algorithm for Eq. (1) and a similar algorithm for Eq. (36).

The discretized form of Eq. (1) is advanced in time using the four-stage Runge-Kutta algorithm of Jameson et al. (1981). Local time stepping (i.e., constant C.F.L. number) and residual averaging is employed to enhance convergence. Upon completion of a fixed number of temporal relaxation cycles, the three-dimensional flow variables are used to evaluate the right hand side of Eq. (36). This equation is then advanced in time using the same integration procedure as that for the three-dimensional system. After a fixed number of time steps, the value of \( q \) is updated according to the equation
\[
q = \bar{q} + (q - \mathcal{L} \lambda \bar{q})
\]
where the \( q \)'s within the brackets are those that were used to evaluate the right hand side of Eq. (36).

The three-dimensional residual error based on the updated value of \( q \) (i.e., Eq. (43)) is generally found to be largest within the blade passage region. Prior to initiating the next three-dimensional iteration cycle, the residual is reduced by performing a fixed number of three-dimensional iteration cycles (between five and ten iterations) over a local three-dimensional flow region which extends a modest distance upstream and downstream of the blade passage leading and trailing edges. The inlet and exit boundaries of this reduced flow domain are located in a region where the sum of the bracketed terms in Eq. (36) is small. The inlet and exit boundary conditions associated with the reduced flow domain are derived using the updated variables obtained from Eq. (43). Within this region of flow the values of \( q \) obtained from relaxing the reduced flow domain equations replace those obtained from Eq. (43).

The outlined solution procedure is a multigrid algorithm of the form introduced by Brandt (1982). It was specially constructed to reduce the axisymmetric component of the error vector, which at the start of the solution procedure is often large. It also recognizes the spatial relaxation of the three-dimensional flow field to an axisymmetric field away from the blade row of interest. This recognition reduces the computational work required to obtain a solution relative to a more traditional multigrid strategy.

The solution procedure outlined above has been implemented in both a V and W cycle framework. In the V cycle strategy, one proceeds directly from the three-dimensional solver to the through-flow solver to the reduced flow solver before going back to the three-dimensional solver. In the W cycle, one proceeds from the three-dimensional solver to the through-flow solver to the reduced flow solver and then cycles between the through-flow solver and the reduced flow solver before going back to the three-dimensional flow solver. The simulation to be reported was executed using the V cycle strategy. Experience with the W cycle is limited and needs further development. However, a preliminary analysis showed the W cycle strategy to require less computational work than the V cycle to converge to a fixed tolerance level.

When the three-dimensional flow field converges to a predetermined level, the body forces and energy sources required as input to simulations of the remaining blade rows can be estimated using the procedure outlined in Ref. 9. The cycling of information between the simulated blade rows of the multiblade row machine is carried out until the tangential average of each simulated blade row flow field agrees with each other to a predetermined tolerance.

BORDER CONDITIONS

All solid surfaces are modeled as rigid, nonslip, and impermeable. The surfaces are also assumed to be adiabatic. These assumptions imply that the velocity relative to a solid surface is zero and that the temperature gradient normal to the surface is also zero. The pressure at a solid surface is obtained from the normal momentum equation evaluated at the surface. At the inlet, either the mass flow or the total pressure is specified along with the total temperature and the radial and tangential velocity components. The one-dimensional Reimann invariant \( C \) is extrapolated from the interior to the boundary; with the specified flow variables, it defines the incoming pressure, axial velocity component, and temperature. The shear stresses and heat flux at the inlet are also set to zero. At the exit radial equilibrium, with the pressure specified at the hub, is used to establish the radial pressure distribution. The flow quantities \( p, \rho v_x, \rho v_y, \rho v_z \) are extrapolated from the interior.

GRID GENERATION

As discussed in Celestina et al. (1986), the averaged passage equation system requires that a mesh be specified for each blade row of a multistage machine. In addition, the meshes must have a common meridional
mesh in order to eliminate the need for interpolating the body forces and correlations from grid to grid. To capture shear layer and stagnation points properly a fine mesh spacing is required in a direction normal to solid surfaces and in the blade leading and trailing edge regions. A mesh generator which is capable of generating these features is discussed in detail in Mulac (1986).

RESULTS AND DISCUSSION

The simulation executed was the Low-Speed Rotating Rig, at United Technologies Research Center. The Low-Speed Rotating Rig (LSRR) is a stage and a half turbine consisting of an inlet guide vane, a rotor, and a stator. The inlet guide vane contains 22 blades and the rotor and stator both contain 28 blades. The flow coefficient, \( \phi \), is 0.78 and the spacing between blades, \( \delta_y \), is 0.5. The LSRR grid contains 228 axial, 25 radial, and 41 circumferential points. Each blade is divided along the chord with 26 axial points between each blade row, the inlet and exit.

The results to be presented required eleven hours of Cray 2 C.P.U. time. They represent but a small fraction of the information obtained from the simulation. They are intended to illustrate the degree to which one can quantify performance parameters which are of interest to designers, and to reveal qualitative information identifying flow phenomena which may have an impact on performance. These results also reflect the current state of model development. The first series of results shows the predicted pressure distribution on the surface of each blade row of the turbine as a function of axial chord and percent of span height. The span locations measured from the hub are 1.3, 12.5, 50, 87.5 and 98.7 percent, respectively. The experimental measurements taken at these locations are also shown. Experimental data was also available for 25 and 75 percent of span but was not utilized since it provided little additional information relative to the current discussion. The results for the first vane are shown in Fig. 1. The predicted loading level is seen to be in good agreement with the measurements of Driling (1988). The predicted pressure surface pressure distribution is in excellent agreement with the experimental results. For the suction surface, the agreement between measurement and simulation is good for the region forward of minimum pressure. Aft of the peak, the agreement between experiment and simulation deteriorates. This deterioration is believed to be related to viscous effects (i.e., turbulence and transition modeling) whose modeling could be improved. Some exploratory calculations suggest that the boundary layer aft of the suction surface minimum pressure is growing too rapidly and, as a result, of the radial pressure gradient, is being transported towards the hub to an extent greater than that suggested by a flow visualization studies. Improvements in the agreement between simulation and experiment have been obtained by incorporating a simple transition model in which the flow remains laminar forward of the minimum pressure peak and Baldwin-Lomax turbulence model as implemented by Dawes (1986). The current simulation assumes the flow to be fully turbulent from the leading edge of each blade. Work on this problem is continuing. The next figure shows the predicted and measured pressure distribution for the rotor. The predicted loading levels appear to be in good agreement with measurements, with the exception of the hub and tip region. The present simulation does not include a clearance region, which should account for some of the discrepancy in the tip region. The pressure distribution along the pressure surface is quite evident. The pressure distribution along the suction surface is in good agreement with the measurements. At the midspan and at 25 and 75 percent (not shown) of span the predicted pressure distribution along the suction surface is in good agreement with the data. At 1.3 percent and 25 percent of span, the suction surface pressure coefficient is lower than that measured. As a result the loading is lower over the forward portion of the rotor than what has been measured. Although the cause of this discrepancy is unknown at the present time, one could speculate that it may be due to an overestimate of the magnitude and extent of the low momentum fluid exit from the blade row.

The pressure distribution for the last vane is shown in Fig. 3. Once again the loading level is well predicted with the exception of location that is 1.3 percent of span. The underpredicted suction surface pressure coefficient at 1.3 percent of span. The underpredicted suction surface pressure coefficient at 1.3 and 12.5 percent of span suggests that the location of the hub vortex as suggested by the experimental data is in better agreement with measurements. Similarly, the predicted suction surface distribution at midspan agrees well with the experimental distribution. The next figure shows the predicted relative total pressure coefficient forward and aft of the rotor as a function of span. The measured distribution reported in Sharma et al. (1988) is also shown. The magnitude of the predicted exit flow coefficient is lower than that measured; however, the shape of the curve is consistent with the data. The magnitude of the predicted exit flow coefficient is also higher than measured. The influence of the secondary vortices generated within the rotor passage on the exit flow coefficient is more evident in the experimental data than in the simulation result. The data in Sharma et al. (1988) suggests that the secondary vortices exit the rotor at approximately 30 and 70 percent of span. The local minimum in the measured exit flow distribution at 25 and 85 percent of span are a consequence of the velocity field induced by these vortices. The location of the tip vortex as suggested by the experimental data is significantly closer to the blade tip than the location suggested by the simulation. The difference is believed to be due to the tip leakage flow which was not accounted for. It appears that this flow drives the tip secondary vortex inward towards the hub.

The simulation results place the hub secondary flow vortex at 25 percent of span at the exit of the rotor, which is in good agreement with measurements. This vortex, however, appears to be more diffuse than measured, which may account for the lack of a local minimum in the exit flow relative total pressure coefficient at 25 percent of span.

The deviation from the design intent of the rotor relative exit flow angle is shown in Fig. 5. This angle is plotted as a function of span, with the solid curve representing the simulated data and the open symbols the measured data from Sharma et al. 1988. The agreement between the two is reasonable with the exception of the tip region, where the result of neglecting the tip leakage flow is quite evident. The noticeable overturning of the flow near the endwalls and the subsequent subsequence in the region of midspan caused by the secondary vortices is clearly seen in both the
SUMMARY AND CONCLUSION

Given the early state of the average passage model development, the results presented in this report are very encouraging. The amount of empirical information used in the stage and one-half turbine simulation is considerably less than that required to achieve comparable results using today's quasi threedimension flow models. The average passage flow model also appears to be able to capture the physical features of the secondary flow field generated within a multistage turbine configuration. It appears that an accurately modeled of the time-averaged vorticity field produced by the unsteady secondary vorticity exiting a blade row is required to establish the performance of the downstream blade row. The outlined closure procedure, unlike some others which have been suggested, insures that the unsteady vorticity field exiting a blade row is consistent with the exiting time-averaged vorticity field. Hence, no spurious vorticity is produced as a result of coupling one blade row to another.

The simulation of the stage and one-half turbine has shown the complex nature of the endwall flow in a low aspect ratio turbine. In the rotor, the secondary vortices generated within the endwalls affect the flow at midspan. They cause the endwall fluid to deposit on the suction surface. This transport of low momentum fluid leads to significant spanwise mixing across the axi-symmetric stream surface. It is doubtful that an endwall boundary layer model could be used to predict this phenomenon.

There are numerous research activities that need to be pursued to further the development of the average passage flow model. Other configurations need to be simulated, including multistage compressors and high-speed machinery. Grid generators need to be developed which are compatible with the average passage model that can cluster grid points in regions of high flow gradients that occur near and away from solid surfaces. Algorithm improvements need to be made to speed convergence. The sensitivity of design parameters to turbulence modeling for the average passage model must be established. The authors wish to express their gratitude to Professor E.M. Greitzer of the Massachusetts Institute of Technology and to Dr. O.P. Sharma of Pratt & Whitney Aircraft for their many useful suggestions during the course of this work. The authors would also like to thank Dr. R.P. Dring of the United Technologies Research Center for providing the experimental pressure distribution data and Dr. L.S. Langston of the University of Connecticut for his flow visualization results.

ACKNOWLEDGEMENTS

The authors wish to express their gratitude to Professor E.M. Greitzer of the Massachusetts Institute of Technology and to Dr. O.P. Sharma of Pratt & Whitney Aircraft for their many useful suggestions during the course of this work. The authors would also like to thank Dr. R.P. Dring of the United Technologies Research Center for providing the experimental pressure distribution data and Dr. L.S. Langston of the University of Connecticut for his flow visualization results.

REFERENCES


FIGURE 2. - ROTOR PRESSURE DISTRIBUTIONS.

FIGURE 3. - SECOND BLADE PRESSURE DISTRIBUTIONS.
CRISSAL DATA IS OF POOR QUALITY

Figure 4. - Rotor relative total pressure coefficient versus span.

Figure 5. - Rotor relative exit flow angle versus span.
Figure 6. - Second blade total pressure coefficient versus span.

Figure 7. - Second stator flow angle distribution.
FIGURE B. - SECOND BLADE PRESSURE SURFACE LIMITING STREAMLINE PATTERN.
Abstract

This work outlines a procedure for simulating the flow field within multistage turbomachinery which includes the effects of unsteadiness, compressibility, and viscosity. The associated modeling equations are the average passage equation system which governs the time-averaged flow field within a typical passage of a blade row embedded within a multistage configuration. The results from a simulation of a low aspect ratio stage and one-half turbine will be presented and compared with experimental measurements. It will be shown that the secondary flow field generated by the rotor causes the aerodynamic performance of the downstream vane to be significantly different from that of an isolated blade row.