A THREE-DIMENSIONAL SHOCK LOSS MODEL APPLIED TO AN AFT-SWEPT, TRANSONIC COMPRESSOR ROTOR

Steven L. Puterbaugh
William W. Copenhaver
Wright Laboratory
Wright-Patterson Air Force Base, Ohio

Chunill Hah
NASA Lewis Research Center
Cleveland, Ohio

Arthur J. Wennerstrom
Engineering Consultant
Hot Springs Village, Arkansas

ABSTRACT

An analysis of the effectiveness of a three-dimensional shock loss model used in transonic compressor rotor design is presented. The model was used during the design of an aft-swept, transonic compressor rotor. The demonstrated performance of the swept rotor, in combination with numerical results, is used to determine the strengths and weaknesses of the model. The numerical results were obtained from a fully three-dimensional Navier-Stokes solver. The shock loss model was developed to account for the benefit gained with three-dimensional shock sweep. Comparisons with the experimental and numerical results demonstrated that shock loss reductions predicted by the model due to the swept shock induced by the swept leading edge of the rotor were exceeded. However, near the tip the loss model under-predicts the loss because the shock geometry assumed by the model remains swept in this region while the numerical results show a more normal shock orientation.

The design methods and the demonstrated performance of the swept rotor is also presented. Comparisons are made between the design intent and measured performance parameters. The aft-swept rotor was designed using an inviscid axisymmetric streamline curvature design system utilizing arbitrary airfoil blading geometry. The design goal specific flow rate was 214.7 kg/sec/m² (43.98 lbm/sec/ft²), the design pressure ratio goal was 2.042, and the predicted design point efficiency was 94.0. The rotor tip speed was 457.2 m/sec (1500 fps). The design flow rate was achieved while the pressure ratio fell short by 0.07. Efficiency was 3 points below prediction, though at a very high 91 percent. At this operating condition the stall margin was 11 percent.

INTRODUCTION

High performance aircraft gas turbine engines must employ compression systems that provide maximum performance at minimum weight. This requirement, in addition to maintainability and fabrication issues, has led to the development of compressors that maximize pressure ratio per stage while maintaining high efficiency. In response to these timeless requirements, state-of-the-art gas turbines have employed the transonic compressor stage for many years. The transonic stage is defined as one which includes a rotor that operates at supersonic tip speeds with some portion of the span operating at subsonic relative blade speed.

The shock generated by this type of rotor increases diffusion in the relative frame and, hence, does work on the fluid by turning the flow in the absolute frame. However, along with the benefit, the shock also presents challenges that have been the subject of research and development programs for decades. These include reducing the magnitude of loss generated by the shock system, managing the shock-boundary layer interaction at the suction surface, and providing proper area relationships within the passage to achieve the desired flow and to allow the shock to start.

This study seeks to evaluate a model which was utilized in an attempt to reduce shock loss. The model describes the salient features of the three-dimensional shock structure which is present in transonic compressor rotors. The availability of such a model allows the compressor designer to exploit the physical characteristics of the three-dimensional shock surface to enhance compressor performance. Specifically, the loss due to the shock may be reduced by inducing a shock surface which is oblique to the relative flow. This is done by sweeping the leading edge of the rotor away from the direction of rotation. Since the Mach number normal to the shock surface sets the strength of the shock, the more the shock is swept relative to the incoming flow, the weaker the shock and its attendant loss.

The experimental hardware which is the subject of the current work was also the subject of a paper by Hah and...
The emphasis of the earlier paper was to demonstrate a numerical technique applied to an advanced, swept, transonic rotor and to investigate the shape of the shock surface. The intent of the current work is to 1) evaluate the shock model which was first used during the subject rotor design, 2) provide details of the design and the design process, and 3) provide experimental measurements and test analysis results.

BACKGROUND
The concept of using backward sweep to reduce shock losses began as a result of some observations made by Prince (1980) about research fans that demonstrated higher efficiency than expected. He proposed the idea that a swept shock surface could be responsible for the increase. Each of the rotors reviewed by Prince incorporated a modest amount of sweep that was an incidental result of design procedures that sought to optimize other parameters, such as mid-span solidity. The logical next step was to intentionally design a swept rotor in order to determine how far this concept might be extended.

However, additional information was required to guide the many choices which impact the leading edge profile, or the sweep, of the rotor. A model was therefore developed by Wennerstrom and Puterbaugh (1984) which relate shock loss to the 3-dimensional shape of the shock surface. The model extended the well known 2-dimensional shock loss model of Miller, Lewis, and Hartmann (1961) to include the effect of the spanwise sweep of the shock surface. The new approach was to compute the angle of the three-dimensional shock surface relative to the local flow vectors on each stream surface. The shock surface was assumed to be formed by normal (when viewed in the cascade plane), attached shocks on each stream surface. The modeling approach allowed the resulting assumed shock surface to be swept all the way to the casing. It was recognized at the time that it did not satisfy the correct boundary conditions at the outer case.

After implementing the 3-dimensional shock loss model into the design system, a rotor was designed that utilized sweep to minimize losses generated by the shock. The intent was to evaluate the overall effect of blade sweep on compressor performance. Therefore the swept rotor was designed as a derivative of a rotor having a more conventional leading edge profile as a baseline. The "baseline" rotor, the sweep of which was small and unintentional, was one of the subjects of Prince's study. The swept rotor was tested in a single stage configuration during a research program conducted by Wright Laboratory's Compressor Aero Research Lab at Wright-Patterson Air Force Base. Though the stator influences rotor performance to various degrees, it is felt that the results presented here can be generalized to stages incorporating well designed stators.

DESIGN APPROACH
A detailed description of the approach and tools used for the research compressor design was given by Law and Wadia (1993). A brief description is given below. Typically a two phase approach is used where a preliminary design is followed by a detailed design. The preliminary design defines fundamental geometric characteristics such as annulus shape, blade speed, solidity, etc. The detailed design phase takes these parameters along with fully defined aerodynamic information, obtains a flow field solution using the streamline curvature computational scheme, and generates the airfoil shape in the form of blade surface coordinates. In the subject effort, the detailed design phase was used exclusively since the new design was derived from the "baseline" rotor design.

The optimization criteria used during detailed design was to obtain a chordwise circumferentially averaged static pressure distribution which rises linearly from the leading edge to about 3/4 chord and then smoothly transitions to zero slope. "Arbitrary" airfoils were used rather than a specific family of airfoils, such as double-circular-arc. The airfoils were developed from relative flow angles and assumed incidence and deviation angle distributions at different chordwise computing stations. This approach to airfoil definition gives the designer the flexibility to closely achieve an optimum static pressure distribution without constraints on blade shape. The validity of this approach has been demonstrated through the generation of several very successful designs (Wennerstrom, 1984; Law and Wadia, 1993; Law and Wennerstrom, 1987).

The rotor design point performance predictions along with the pertinent geometric parameters are shown in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure Ratio</td>
<td>2.042</td>
</tr>
<tr>
<td>Adiabatic Efficiency</td>
<td>94.0</td>
</tr>
<tr>
<td>Corrected Specific Flow</td>
<td>214.7</td>
</tr>
<tr>
<td>Corrected Tip Speed</td>
<td>457.2</td>
</tr>
<tr>
<td>Tip Inlet Relative Mach Number</td>
<td>1.65</td>
</tr>
<tr>
<td>Mean Rotor Aspect Ratio</td>
<td>1.26</td>
</tr>
<tr>
<td>Rotor Inlet Hub/Tip Ratio</td>
<td>0.312</td>
</tr>
<tr>
<td>Rotor Diameter</td>
<td>43.18</td>
</tr>
<tr>
<td>Number of Blades</td>
<td>20</td>
</tr>
</tbody>
</table>

Table 1. Rotor design point predictions and geometric parameters.

The application of this type of rotor to an engine would be as a fan stage or as an inlet stage to a turbojet. High flow per frontal area required by these applications is achieved by the combination of the high specific flow and the low hub/tip ratio. A cut-away view of the test stage is shown in Figure 1.
The rotor is cantilevered with support provided by 4 struts located 2 stator chord lengths downstream of the stator trailing edge. The rotors used in the test rig are the integral blade and disk, or "blisk", type. The blisk was made of 6Al4V titanium.

The aerodynamic parameter changes were guided by earlier experience with rotors of similar geometry. The spanwise work and deviation angle distribution obtained from earlier experiments were used in the swept rotor design. However, the work distribution internal to the blade row was modified to achieve the optimum static pressure distribution according to the criteria described earlier. The incidence angles were also determined from earlier design experience. In order to achieve the design flow rate, the spanwise incidence angle distribution (relative to the suction surface) was set at 2 degrees inboard of 66 percent span and smoothly transitioning to 3.5 degrees at the tip. The estimate of relative total pressure loss was obtained from a combination of a diffusion factor/loss parameter correlation (Figure 2) and the 3-dimensional shock loss model (Wennerstrom and Puterbaugh, 1984). The aerodynamic blockage distribution was established from earlier design experience.

![Figure 2. Diffusion factor/loss parameter correlation.](image)

Previous experience had revealed that low aspect ratio, high tip speed rotor designs undergo a significant 3-dimensional deformation of the manufactured geometry from rest to design speed. Specifically, the camber line is deformed in addition to the familiar untwist of the airfoil. Therefore a 3-dimensional mechanical design procedure was used to account for the complex deformation. An iterative technique was used where the 3-dimensional displacements obtained from the structural analysis were subtracted from the design intent to create the cold geometry. The structural analysis was rerun with new cold geometry and the resulting geometry was compared with the design intent. If the differences exceeded a tolerance of 0.0762 mm (0.003 in.), then new cold-hot displacements were computed and another iteration was begun.

**Incorporation of Blade Sweep**

The intent of the use of sweep was to decrease the shock loss generated in the rotor. Therefore, it was most important to introduce sweep in increasing amounts beginning just outboard of the sonic radius. Figure 3 shows the spanwise distribution of aerodynamic sweep angles that was obtained after optimizing the leading edge profile for minimum shock loss. Two distributions of angles are shown, labeled leading edge sweep and suction surface sweep. Both angles are computed as the difference between the local air angle and a normal to the local shock surface. The leading edge sweep angles are computed at the intersection of the leading edge of the blade and the shock surface. The suction surface sweep angles are computed at the intersection of the shock surface and the suction surface of the adjacent blade. Both distributions reach peaks between 85 and 90 percent span. The meridional view of the leading edge does not change shape in this region. The decrease outboard of this location is due to changes in streamline slope and blade stagger. Figure 3 also includes a distribution of the inlet relative Mach number as a reference. The sonic radius occurs at about 35 percent span.

![Figure 3. Inlet relative Mach number and sweep angles.](image)

The structural challenges associated with the swept rotor design concept required that the mechanical design be considered simultaneously with the aerodynamic design. Therefore an iteration between the blade design procedure and a simple finite element analysis was used to keep stresses within allowable limits. The steady state stress limit was set to allow for a minimum of 103.4 MPa (15 ksi) dynamic stress margin.

The initial attempt to introduce more sweep was to add mid-span chord symmetrically so that section centroids would remain approximately on the stacking axis. This resulted in exceptionally high blade lean angles, especially near the hub, and very high stress levels. The second approach was to hold the trailing edge to its original baseline configuration in the meridional plane and to examine the impact on stress of the new leading edge plus non-radially stacked centroids. This offered an improvement but still produced unacceptably high stresses. The third and successful approach was to again hold the trailing edge to its original baseline configuration and, in addition, to introduce a reflex curvature in the meridional view of the leading edge bringing it from its new swept profile in the tip region back tangent to the original baseline configuration where the leading edge intersects the hub platform. This resulted in lean angles at the platform comparable to the baseline and acceptable stress levels in spite of the off-axis centroids. The resulting leading edge profile is evident in Figure 1.
TEST FACILITY, INSTRUMENTATION, AND DATA REDUCTION

The facility and instrumentation was described in detail by Law and Wadia (1993). The facility has a closed-loop airflow path and is powered by a 1.49 Mw (2000 hp) electric motor. Facility flow rate is measured using a calibrated venturi. The test article instrumentation consisted of over 150 steady state pressure measurements, over 130 steady state temperature measurements, and 10 high frequency response static pressure measurements over the rotor tip. Radial clearance between the rotor tip and the casing was measured on-line by spark discharge type clearance probes. A clearance of 0.43 mm (0.017 in.), or 0.44 percent of chord, was measured at design speed. The measurement uncertainty of individual measurements along with computed quantities for a 95 percent confidence interval are given in Table 2.

| Mass Flow | ±0.5% Reading |
| Pressure Ratio | ±0.1% Reading |
| Efficiency | ±1.1 Points |
| Pressure | ±208.8 Pa (0.03 psi) |
| Temperature | ±0.78° C @ 10° C, ±1.17° C @ 121.1° C |
| Tip Clearance | ±0.051 mm (0.002 in.) |

Table 2. Measurement uncertainty for instrumentation used in the current study.

The data presented in this paper is for rotor performance. The rotor exit conditions, in terms of spanwise distributions of total pressure and total temperature, were determined using stator leading edge instrumentation. These distributions were input directly to a streamline curvature type computer program to generate the data match for subsequent analysis. The particular computer program used in the analysis did not simulate effects due to mixing or secondary flow.

The stator leading edge instrumentation consisted of 9 total pressure and 9 total temperature vane-mounted probes. The probes were located at even increments in the spanwise direction on the vane. A total of 4 vanes were instrumented, chosen at approximately even circumferential spacing. The pressure probes were mounted on 2 different vanes at alternating spanwise locations on each vane. The temperature probes were mounted similarly.

The performance analysis is based on stator leading edge results to minimize the impact of the stator on the conclusions drawn about the rotor losses. The focus of this work is on shock sweep and a loss model developed to predict advantages of swept shocks. Therefore it was important to reduce the uncertainty regarding shock loss that would be added if stage exit results were used. Although stator leading edge measurements add their own level of uncertainty because they are sensing a highly unsteady flow field, they provide a clearer view of the shock losses than interpretations made from stage discharge measurements.

NUMERICAL METHOD

The numerical method described here was not available during the design phase of this rotor and therefore did not influence the design. The numerical method is due to Hah (1987). This approach utilizes a fully 3-dimensional, steady state formulation of the Reynolds-averaged Navier Stokes equations. The equations are solved using a finite-volume approach. The overall numerical scheme is second order accurate in the physical domain.

The system of equations is closed using a k-ε type turbulence model to provide the Reynolds stresses. A low-Reynolds number correction was employed to extend the validity of the model to the viscous sub-layer. This approach allows the numerical scheme to be utilized in situations where strong secondary flow and flow separation is present. The constants in the turbulence transport equations are the same as those used in many earlier computations of the flow field in transonic fans and compressors (Hah and Puterbaugh, 1992; Copenhagen, et al., 1993; Copenhagen, et al., 1994). At the inflow plane, total pressure, total temperature, and flow angle are defined. At the outflow plane, static pressure is specified at a point and asymptotic conditions are imposed on the remaining dependent variables. Non-reflective boundary conditions are imposed at both the inflow and outflow boundaries. Periodicity is enforced at the periodic boundaries and no-slip is enforced at solid walls. For the current study, a total of 347,300 nodes make up the blade-centered, 1-type grid. Fifty nodes are used in the pitchwise direction, 46 nodes in the spanwise direction, and 151 nodes in the streamwise direction. The blade chord has 95 nodes distributed from leading edge to trailing edge. In order to resolve the clearance flow, 7 nodes are distributed from suction surface to pressure surface with 6 nodes in the clearance gap. The clearance gap was set to the value measured during the experiment, 0.43 mm (0.017 in.). The design, hot-running blade shape was modeled by the grid.

RESULTS

Overall Results

The rotor performance map is shown in Figure 4.
The stall line was fully characterized without encountering stall flutter. The minimum back pressure condition capable of being run in the facility did not allow operation at a fully choked condition. Choke flutter was not encountered in the region tested. Full speedlines were obtained from 40 to 100 percent design speed. Peak efficiency on speedlines below 100 percent speed was not determined due to the facility back pressure limit. The very high design speed efficiency peaked just under 92 percent, whereas pressure ratio peaked at about 2.05. The design system prediction is indicated on the figure by the filled circle symbol. The design flow rate was exceeded slightly but the demonstrated peak efficiency was about 2 points below the predicted value. While this difference is significant from an engine performance point of view, it must be kept in mind that the predicted absolute level is very high.

The maximum stall margin demonstrated was nearly 11 percent at an efficiency of 91 percent and a pressure ratio of 1.97. The facility back pressure limit restricted the maximum demonstrated stall margin to this value. Since a stall margin of 11 percent is closer to acceptable levels for military designs, this operating condition was chosen for detailed analysis and comparisons with design predictions.

The spanwise distributions of total pressure ratio and adiabatic efficiency as compared to design intent are shown in Figures 5 and 6, respectively. These and subsequent plots show comparisons between predicted and demonstrated quantities along with results from the numerical analysis. The numerical results will be discussed later.

Pressure was under-predicted in the inboard 30 percent and over-predicted outboard. Recall that the sonic radius is about 35 percent span, therefore the under-prediction reflects the performance of the subsonic blade sections. The perceived under-prediction near the hub is reflected in the unreasonably high efficiencies as seen in Figure 6. This condition is believed to be produced by strong secondary flow within the suction surface boundary layer and blade wake which was not accounted for when analyzing the measurements. This secondary flow is not unusual in a highly loaded configuration with high hub ramp angle and is very evident in the numerical results (not shown here). In the mid-span, the efficiency is somewhat under-predicted. However in the tip region the efficiency is significantly over-predicted. Hah and Wennerstrom (1991) demonstrated graphically that the shock surface transitions to a normal orientation relative to the casing wall as it approaches the tip.

Figures 7 and 8 show comparisons of spanwise distributions of incidence and deviation angles. Note that this plot shows incidence relative to the camber line whereas the comments in the Design Approach section referred to incidence relative to the suction surface.
This requires that the relative flow angles agree as well. It then and demonstrated deviation angles agree fairly well (Figure 9). In this region the two distributions agree fairly well. This is also in the region of highest solidity and longest chord.

Moving outboard of 30 percent span the demonstrated total temperature ratio first is less than design and then, at about 85 percent span, crosses the design distribution and is much greater than design at the tip. Consider the mid-span region in more detail, between 45 and 70 percent span. In this region, the design and demonstrated deviation angles agree fairly well (Figure 9). This requires that the relative flow angles agree as well. It then follows that the absolute flow angles, and turning, agree but the work done does not agree. The only way for the work to be over-predicted at the same relative flow angle is for the diffusion of the flow to be reduced while maintaining the same flow direction in the relative frame. A potential cause for this condition is the presence of a shock that is weaker (produces less diffusion), than what was expected.

**Numerical solution validation.** The numerical method described previously has been the subject of several prior studies of rotors with similar loading and relative Mach number levels (Häh and Puterbaugh, 1992; Copenhaver, et. al., 1993; Copenhaver, et. al., 1994). In all cases where the rotor performance is concerned the comparisons were very favorable. Therefore, this method was considered to be an excellent tool, in conjunction with the experimental results, for use in evaluating the performance of the swept shock model.

For the case currently under study, the validity of the numerical results will be evaluated in three ways. First, the integrated performance obtained from the numerical solution in terms of pressure ratio and efficiency versus corrected flow rate will be compared with the experimental results. Second, various spanwise distributions will be compared to experimental results. The comparisons were done at the maximum stall margin operating point for convenience of presentation on Figures 5, 6, and 9. Finally, the tip shock structure is compared at peak efficiency to lend credence to the evaluation of the shock structure geometry assumed by the model.

Computations were carried out for 6 different operating points at 100 percent speed. The exit pressure and temperature results were area averaged to obtain integrated performance characteristics for comparison with the measured data. Experimental integrated performance values were obtained from the measurements of fixed frame probes in a non-steady rotor exit flow field. There is no obvious approach to sampling numerical results in a similar manner. Therefore, area averaging was selected simply for convenience in determining integrated quantities.

The integrated performance obtained from numerical results is compared with the measured experimental results in Figure 10. The general shape of the characteristics agree very well. The numerical results give a lower flow rate and higher peak pressure ratio than that seen with the measurements. In both flow and pressure ratio the agreement is within 2 percent. The agreement in peak efficiency is within 1 percent.

The spanwise distributions of quantities were computed from the numerical solution by area averaging the circumferential distributions at various spanwise radii. A comparison of total pressure ratio is shown in Figure 5. Agreement between the numerical solution and experiment is quite good in terms of the shape of the distribution. The numerical results under-predict near the hub and over-predict near the tip, however. Efficiency is compared in Figure 6. The agreement between the numerical solution and experimental results is quite evident. Finally, total temperature is compared in Figure 9. The largest discrepancy appears at the hub, but the overall character of the experimental distribution is reflected in the numerical results, i.e. a reduction of work in mid-span and an increase of work near the tip as compared to the design distribution.
The analysis in the remainder of the paper will center around the peak efficiency operating point in order to more fairly evaluate the shock loss model’s accuracy in the light of assumptions made during development. The assumed shock orientation in the cascade plane, i.e. an attached, normal shock, corresponds to that typically found at the tip when operating at peak efficiency.

The measurements were made with high response pressure transducers positioned on the rotor casing at equal axial increments of 6.35 mm (0.25 in.). Details of the measurement method is described by Sellin, et al. (1993). The shock locations as measured and computed at the blade tip agree very well. In both cases the shock stands off the leading a small distance and is oriented normal to the flow in the cascade plane. In addition the absolute values of the static pressure downstream of the passage shock agree.

The integrated performance and detailed comparisons shown in Figures 5, 6, 9, 10 and 11 serve to establish the credibility of the numerical results for use in extended analysis of the swept shock loss model. On the basis of the above comparisons and the prior studies cited, the computation was deemed to be an excellent tool for evaluating the swept shock loss model in more detail. Comparisons will now be made between the modeled shock structure and the computed structure along with the losses these structures produce.

Assessment of Swept Shock Model

An integral factor in the design of the aft-swept rotor was the use of the three-dimensional shock loss model. This model provided criteria for tailoring the leading edge profile to minimize shock losses. The model’s strengths and weaknesses must be assessed before improvements can be made. However, a detailed evaluation from test data is difficult because of the inability to directly measure shock loss. Losses result from many mechanisms and measurements only provide the ability to compute an overall loss. Furthermore, spanwise transport within the flow field via secondary flow and mixing masks the location of loss production. This makes it difficult to assign loss generation to a specific location and, therefore, a specific mechanism. Therefore the approach taken to analyze the model will be to compare detailed information derived from numerical results with design information, the credibility of the numerical solution for the subject rotor having already been established.

The three-dimensional shock loss model’s assumed shock geometry is compared to the actual shock geometry (as computed numerically) in Figure 12 by comparing shock sweep angles at the suction surface.

Figure 10. Performance comparisons at 100 percent speed.

Figure 11 compares static pressure contours in the cascade plane obtained from the numerical solution with experimental measurements at the peak efficiency operating point.

Figure 11. Tip static pressure distribution comparisons at design speed, peak efficiency operating point.

Figure 12. Shock sweep angles at the shock/suction surface intersection at peak efficiency operating point.
The design sweep angles were obtained directly from the compressor design computer program which uses the three-dimensional shock loss model. The method used to compute the sweep angles from the numerical solution was similar to the design method once the shock surface and the local velocity vector were defined. The shock surface was determined by locating the sonic line on each stream surface near the suction surface. The local velocity vector was computed from the solution such that the vector component normal to the stream surface was zero. This was done by transforming the Cartesian vector to an intrinsic coordinate system relative to the stream surface, setting the normal velocity component to zero, and then transforming back to the Cartesian system. This approach was taken to provide a similar basis of comparison between the throughflow-based design information and numerical results. In the mid-span region, where the loss was slightly over-predicted (Figure 6), the design suction surface sweep angle is significantly less than the angle obtained from the numerical results. This implies that the shock surface is swept back by a greater amount than was assumed during the design. Recall that the model assumed an attached, normal shock (when viewed in the cascade plane) from the tip all the way to the sonic radius. Several papers published since the completion of the model and design illustrate that, at least at some locations inboard of the tip region, the shock is actually oblique in the cascade plane even when the shock at the tip is in a normal orientation (Pierzga and Wood, 1985; Strazisar, 1985; Bloch, et al., 1995). This is an area of improvement necessary for the three-dimensional shock model.

The most important area for improvement is near the tip region. This was a known weakness of the model. Figure 12 clearly shows the disagreement in shock shape near the tip, (normalized span approaching 1.0). The sweep from the numerical results approach zero whereas the design values remain at about 40 degrees. Further study is required to determine where, in the spanwise sense, the modeled shock surface should transition to a more normal orientation near the tip. The decrease in shock sweep results in a stronger shock, and therefore increased loss. This effect is seen in the spanwise distribution of efficiency in Figure 6. It must be kept in mind that the increase in losses in the tip region are also due in part to the losses associated with the clearance flow.

The numerical results were further analyzed to shed some light on the fraction of the overall loss which can be attributed to shocks in a transonic compressor rotor. Using a method due to Schreiber (1987), an analysis of the blade wake was used to provide the required information. It is recognized that this is an approximate method that may be more applicable to cascades since secondary flows which exit in the rotating compressor environment can distort the interpretation. However, this approach can at least give an approximate means for attempting the very difficult task of determining losses due to shocks. The viscous loss is assumed to be contained within the peak of the circumferential loss distribution whereas the shock loss was assumed to be contained within the "base" of the distribution as diagrammed in Figure 13. The method is not usable at the endwalls, however. At the hub, the nonzero "base" loss cannot be attributed to shocks since the blade section operates in the subsonic regime. The false shock loss contribution seen there is presumed to be a result of secondary flow and the hub boundary layer, though it was not investigated fully in this work. The shock loss component is arbitrarily set to zero in this region. At the tip, the clearance flow and its interaction with the shock invalidates the method.

Figure 14 shows the spanwise distribution of the fraction of overall loss which is attributable to the shock system from design data and the numerical results. The fraction of overall loss was chosen as the value for comparison in order to normalize the difference in integrated efficiency between the measured and numerical results.

CONCLUSIONS

The goal of this study was to evaluate the three-dimensional shock loss model which was used during the rotor's design and
examine the performance of a highly swept transonic compressor rotor relative to its design intent. Experimental and numerical results provided the means to conduct the investigation. The following conclusions with regard to the design and performance of the swept rotor can be drawn.

1. A rotor utilizing a high degree of leading edge sweep at very high tip speeds can be designed for safe operation. Design optimization utilizing an iterative aerodynamic and structural analysis approach allowed for adequate structural margins for safe operation of an aggressive blading geometry.

2. The design flow rate goal was achieved, but the demonstrated pressure ratio and efficiency fell short of design predictions. At the point selected for analysis, pressure ratio was about 3 percent low and efficiency was about 3 points low relative to the prediction. The peak demonstrated efficiency fell about 2 points short of the predicted efficiency. It must be kept in mind that the efficiency prediction was extremely high. The efficiency prediction was established in large part by the swept shock model which is the main focus of this study.

3. Measured spanwise work distributions indicate that the design fell short of work in the mid-span region and was high near the tip. The following conclusions can be drawn from the experimental and numerical evaluations of the swept shock loss model:

4. In the tip region modifications are required in the swept shock model to accommodate the change in shock sweep. The shock sweep angle reduces significantly toward the tip, resulting in a significant reduction in obliquity at the casing.

5. The mid-span shock sweep angle is under-predicted with the swept shock model. The model requires modification to reflect the oblique orientation of the shock in the cascade plane. The model currently assumes a normal shock in the cascade plane.

ACKNOWLEDGMENTS

The authors would like to acknowledge the Compressor Aero Research Lab staff for their excellent support in obtaining the data presented here. We also thank the management at Wright Lab and NASA for allowing us to present this work.

REFERENCES


