Printed in U.S.A.

71-GT-89

an ASME publication

$2.00 PER COPY
$1.00 TO ASME MEMBERS

The Computation of Transonic Flow Through Two-Dimensional Gas Turbine Cascades

P. W. McDONALD

Assistant Project Engineer,
Scientific Analysis Section,
Technical & Research Organization,
Pratt & Whitney Aircraft,
East Hartford, Conn.

Steady transonic flow through two-dimensional gas turbine cascades is efficiently predicted using a time-dependent formulation of the equations of motion. An integral representation of the equations has been used in which subsonic and supersonic regions of the flow field receive identical treatment. Mild shock structures are permitted to develop naturally without prior knowledge of their exact strength or position. Although the solutions yield a complete definition of the flow field, the primary aim is to produce airfoil surface pressure distributions for the design of aerodynamically efficient turbine blade contours. In order to demonstrate the accuracy of this method, computed airfoil pressure distributions have been compared to experimental results.

Contributed by the Gas Turbine Division of The American Society of Mechanical Engineers for presentation at the Gas Turbine Conference & Products Show, Houston, Texas, March 28-April 1, 1971. Manuscript received at ASME Headquarters, January 11, 1971.

Copies will be available until January 1, 1972.

THE AMERICAN SOCIETY OF MECHANICAL ENGINEERS, UNITED ENGINEERING CENTER, 345 EAST 47th STREET, NEW YORK, N.Y. 10017

The Computation of Transonic Flow Through Two-Dimensional Gas Turbine Cascades

P. W. McDONALD

INTRODUCTION

The development of aerodynamically efficient, highly loaded gas turbine engines requires the rapid and accurate prediction of the flow through turbine cascades well into the transonic flow regime. Here the normal cascade design problems become more critical: Slight contour variations significantly affect pressure distribution and may induce shocks. Furthermore, the high camber may readily lead to separation. In contrast to fully subsonic cascades, a smooth geometry does not guarantee a smooth pressure distribution. The designer, therefore, is faced with the problems of evaluating various transonic airfoil contours to arrive at nonseparating flows with a minimum amount of shock losses.

The gas turbine must be designed for endurance as well as performance. To enjoy the efficiency of high-temperature cycles, many turbines must eject cooling air through the surface of the airfoils. The cooling system should be carefully designed to avoid overheating and to reduce the thermal stress on the blades. Since the air ejection is a function of the external conditions, the prediction of the airfoil surface pressure distribution can be related to the endurance, as well as the aerodynamic efficiency of the gas turbine.

Numerous computational procedures have been developed based on the Lax-Wendroff procedures[1] which predict transonic passage flow. The most accurate (and sophisticated) methods follow the procedure of Burstein[2] by solving the time-dependent partial differential equations in divergence-free form. These methods have, however, two significant limitations which restrict their applicability in the case of high-turning turbine cascades:

1 Their development has been in the direction of high accuracy with little priority given to computational speed and efficiency. Satisfactory comparison with alternate analyses or with experimental data has required extremely fine

---

[1] Lax, P. D., and Wendroff, B., "Difference Schemes with High Order Accuracy for Solving Hyperbolic Equations," NYO Report 9759, Courant Institute of Mathematical Sciences, New York University, New York, July 1962.

[2] Burstein, S. Z., "Numerical Calculations of Multidimensional Shocked Flows," AIAA Journal, Vol. 2, No. 12, Dec. 1964, pp. 2111-2117.

NOMENCLATURE

- b = axial chord
- D = normalized density
- F = normalized flux of axial momentum in axial direction
- G = normalized flux of tangential momentum in tangential direction
- k = specific heat ratio
- M = Mach number
- p = static pressure
- P = normalized static pressure
- s = boundary of control volume
- S = normalized boundary of the finite area element
- t = time
- u = axial velocity
- U = normalized axial momentum and axial mass flux (per unit area)
- v = tangential velocity
- V = normalized tangential momentum and tangential mass flux (per unit area)
- X = normalized axial coordinate
- Y = normalized tangential coordinate
- Z = normalized flux of tangential momentum in the axial direction or axial momentum in the tangential direction
- β = flow angle measured from the plane of the cascade
- Δa = finite area
- ΔA = normalized finite area
- ΔT = normalized time increment
- ρ = density

Subscripts

- 1 = upstream station
- 2 = downstream station

mesh spacing and, therefore, long computing times.

2 The high turning angles encountered in high-performance turbines make an orthogonal grid difficult to use and, due to the typical sharp boundary curvatures, increase the danger of computational instability.

FINITE AREA METHOD

The foregoing problems have been overcome to a significant extent by the proposed "finite area method" which is a numerical representation of the transient conservation equations in integral form. The transient formulation is retained since it remains hyperbolic in nature for both subsonic and supersonic flows. Starting with an initially "guessed at" flow distribution in the cascade (when possible, the results of a similar design), the equation system is relaxed over finite time intervals during which adjacent area elements of the field exchange mass and momentum.

If the two-dimensional turbine cascade problem is approximated by assuming the isentropic flow of a perfect gas, the three integral conservation equations of mass and momentum, together with the isentropic relation, fully define the problem. These are:

$$\frac{\partial \rho}{\partial t} = - \lim_{\Delta a \to 0} \left\{ \frac{1}{\Delta a} \oint_s (i \left[\rho u\right] + j \left[\rho v\right]) \cdot \hat{n} ds \right\}, \quad (1)$$

$$\frac{\partial \rho u}{\partial t} = - \lim_{\Delta a \to 0} \left\{ \frac{1}{\Delta a} \oint_s (i \left[p + \rho v^2\right] + j \left[\rho uv\right]) \cdot \hat{n} ds \right\}, \quad (2)$$

$$\frac{\partial \rho v}{\partial t} = - \lim_{\Delta a \to 0} \left\{ \frac{1}{\Delta a} \oint_s (i \left[\rho uv\right] + j \left[p + \rho v^2\right]) \cdot \hat{n} ds \right\}, \quad (3)$$

where

$$p/\rho^k = \text{constant} \quad (4)$$

The isentropic relation is retained, since shocks normally encountered in a cascade are weak and the results show only small deviations which, according to this study, do not lead to computational instabilities once the integral formulation is adopted.

These equations are then normalized by introducing the upstream properties as reference conditions and by scaling with regard to the axial chord length (b) such that

$$D = \rho/\rho_1, \qquad X = x/b,$$

$$U = \rho u/\rho_1 u_1, \qquad Y = y/b,$$

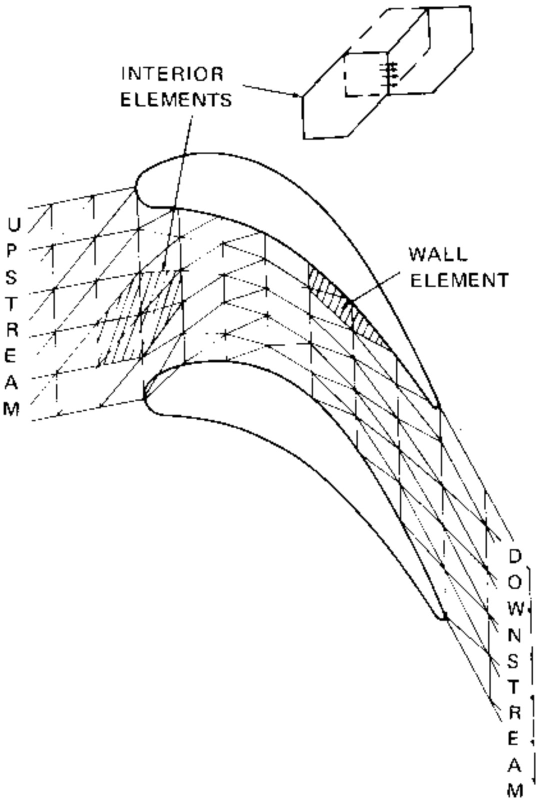

Fig. 1 Cascade system with finite area mesh. (coarse mesh is for illustrative purposes only)

$$V = \rho v/\rho_1 v_1, \qquad \Delta A = \Delta a/b^2,$$

$$P = p/p_1, \qquad \text{and} \quad T = (t/b)\left(\frac{p_1}{\rho_1}\right)^{1/2}$$

This leads to the normalized equations

$$\frac{\partial D}{\partial T} = - \lim_{\Delta A \to 0} \left\{ \frac{1}{\Delta A} \oint_S (iU + jV) \cdot \hat{n} dS \right\}, \quad (5)$$

$$\frac{\partial U}{\partial T} = - \lim_{\Delta A \to 0} \left\{ \frac{1}{\Delta A} \oint_S (i \left[P + U^2/D\right] + j \left[UV/D\right]) \cdot \hat{n} dS \right\}, \quad (6)$$

$$\frac{\partial V}{\partial T} = - \lim_{\Delta A \to 0} \left\{ \frac{1}{\Delta A} \oint_S (i \left[UV/D\right] + j \left[P + V^2/D\right]) \cdot \hat{n} dS \right\}, \quad (7)$$

with the isentropic relation, $P = D^k$. (8)

These conservation equations are applied to finite area elements where each interior plant is surrounded by six sides as shown in Fig. 1. The overlapping regions are the basic control elements for the computational procedure. The numerical representation of these equations can be described in three steps:

1 The density and momentum distributions,

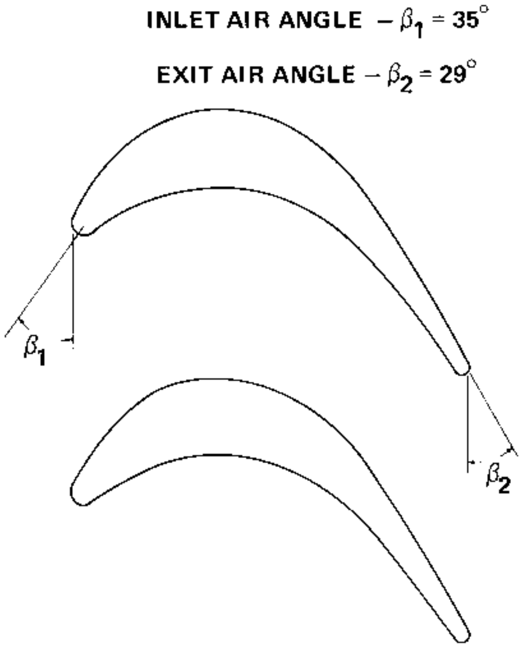

Fig. 2(a) Turbine airfoil geometry

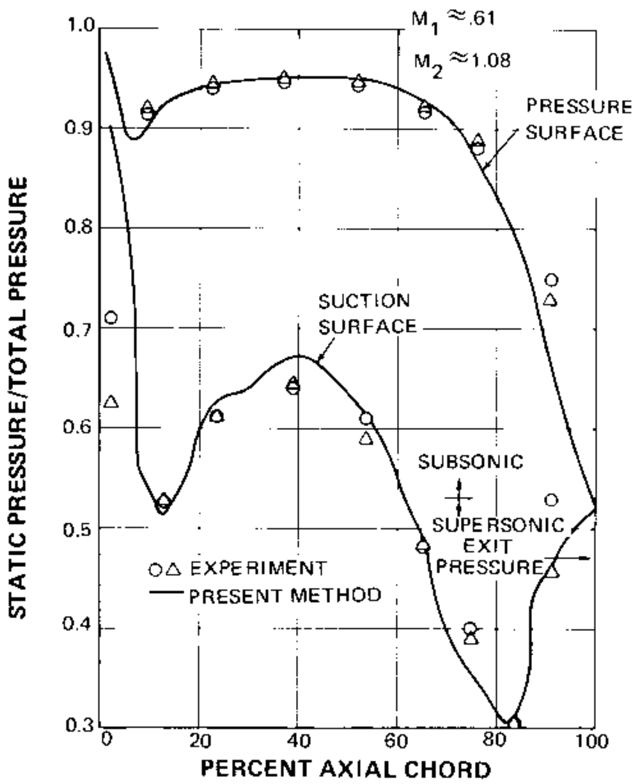

Fig. 2(b) Airfoil surface pressure distribution

D(X, Y, T), U(X, Y, T), and V(X, Y, T) are known at time (T). It is, therefore, possible to compute the momentum flux terms at each nodal point in the mesh with the equations.

$$F = P + U^2/D, \tag{9}$$

$$G = P + V^2/D, \tag{10}$$

and

$$Z = UV/D. \tag{11}$$

2 The second step is to compute the transport rate of mass and momentum across each line segment in the mesh. This is done assuming a linear variation of the mass and momentum flux between nodal points. Consider, for example, adjacent points, j and j+1, where $\Delta X_j = X_{j+1} - X_j$ and $\Delta Y_j = Y_{j+1} - Y_j$. Then the transport rate of momentum across this line segment is given by the expression, $1/2 \left[(F_{j+1} + F_j)\Delta Y_j - (Z_{j+1} + Z_j)\Delta X_j \right]$. Similar expressions are computed for the mass and tangential momentum transport rates.

3 The density and momentum distributions, D(X, Y, T+ΔT), U(X, Y, T+ΔT), and V(X, Y, T+ΔT), at time T+ΔT can then be computed from the net transport of mass and momentum into each control element. In the numerical representation, the integral terms of equations (5), (6), and (7) are replaced by a summation of the mass and momentum transports across the sides of each element. The numerical equations then become

$$D^{T+\Delta T} = \tilde{D}^T + \frac{\Delta T}{\Delta A} \sum_{j=1}^{6} \frac{1}{2} \left| (U_{j+1}+U_j)\Delta Y_j - (V_{j+1}+V_j)\Delta X_j \right|, \tag{12}$$

$$U^{T+\Delta T} = \tilde{U}^T + \frac{\Delta T}{\Delta A} \sum_{j=1}^{6} \frac{1}{2} \left| (F_{j+1}+F_j)\Delta Y_j - (Z_{j+1}+Z_j)\Delta X_j \right|, \tag{13}$$

and

$$V^{T+\Delta T} = \tilde{V}^T + \frac{\Delta T}{\Delta A} \sum_{j=1}^{6} \frac{1}{2} \left| (Z_{j+1}+Z_j)\Delta Y_j - (G_{j+1}+G_j)\Delta X_j \right| \tag{14}$$

In equation (12), $\tilde{D}^T$ implies a mild spatial smoothing of D(X, Y, T) at time, T. The momentum variables, U and V, are also smoothed. This is analogous to the artificial damping procedures often employed in other transient computational methods. The damping is automatically adjusted to provide a proper balance between accuracy and required stability and is decreased as the steady-state solution is approached, thus making the residual damping error negligible.

As seen in Fig. 1, the transport of mass or momentum across each line segment is applicable to two adjacent elements. This assures that the transport exiting from one element is precisely equal to that entering the adjacent element and, therefore, guarantees enforcement of the conserva-

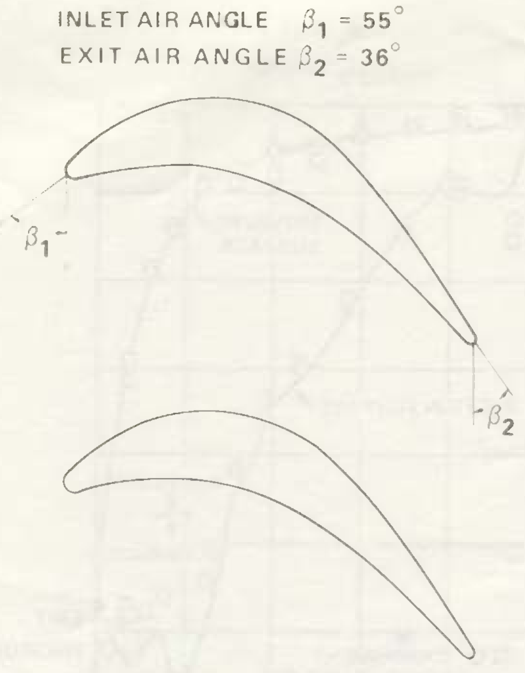

Fig. 3(a) Turbine airfoil geometry

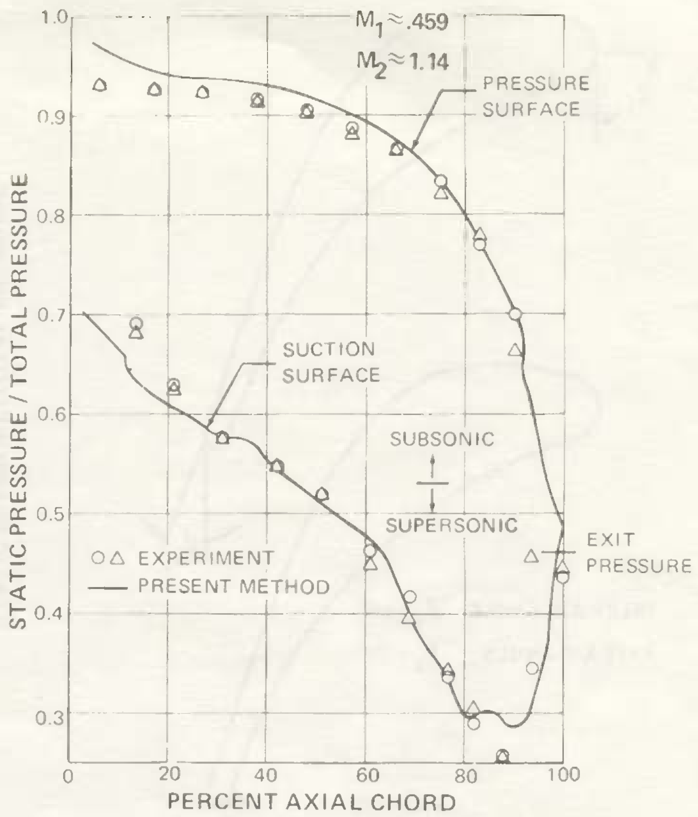

Fig. 3(b) Airfoil surface pressure distribution

tion laws even with a relatively coarse mesh. It is this aspect which distinguishes the finite area method from the transient finite difference procedures and allows the desired accuracy to be obtained with a coarser mesh.

BOUNDARY CONDITIONS

In the computation of flow through turbine cascades, four types of boundary conditions must be specified. On the solid boundaries, the transient variation of mass and momentum is obtained by summing their transport into a five-sided element as shown in Fig. 1. Across the solid wall line segments, there is no mass transport, and the momentum transport depends only on the wall static pressure.

The periodic boundaries (upstream and downstream of the blades) are treated with the same equations that are applied to the interior points. Information is combined from the upper and lower boundary to construct a six-sided element about each point. The computations on the periodic boundaries require no additional assumptions or external information.

At the downstream boundary, the density (or pressure) is assumed constant and uniform. The axial and tangential momentum are computed using a five-sided control element. The downstream air angle is, therefore, obtained as part of the solution. The error, which is incurred by assuming a uniform downstream pressure, has a negligible effect on the predicted flow field when this boundary is placed at least one chord length downstream of the trailing edge.

At the upstream boundary, the pressure is assumed uniform and constant. The inlet air angle is specified, and the momentum variables are continually adjusted until the average axial pressure gradient approaches zero. This condition implies the proper balance between the cascade static pressure ratio and the inlet Mach number. This approach to the upstream boundary conditions accounts for the fact that the turbine cascade may be choked. Whether or not this occurs, the upstream Mach number automatically adjusts to the appropriate value.

NUMERICAL RESULTS

To evaluate the effectiveness of the finite area method, computed results were compared to test data obtained with experimental airfoils in a two-dimensional, nonrotating cascade. The following cases were selected to demonstrate the accuracy of the method in indicating potentially undesirable features of transonic turbine designs.

The first comparison to experimental cascade data is shown in Fig. 2. Here the pressure distribution has been computed for a transonic airfoil with a relatively high turning angle of 115

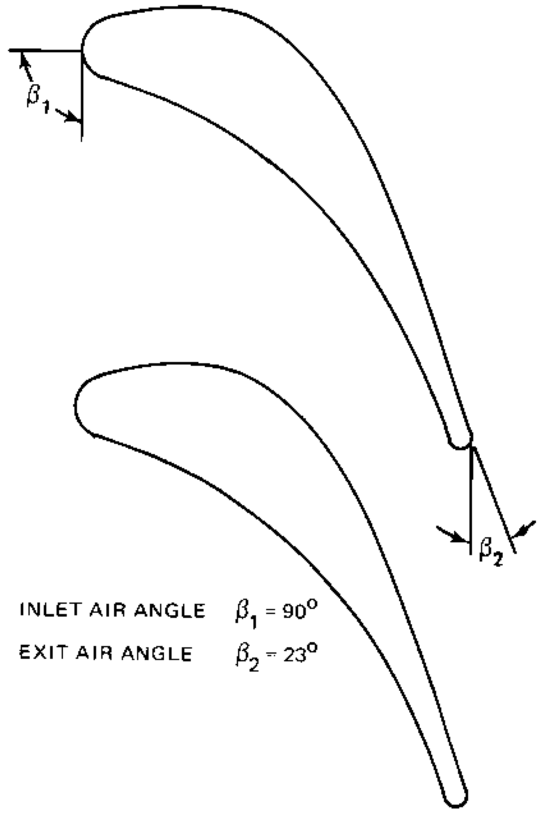

Fig. 4(a) Turbine airfoil geometry

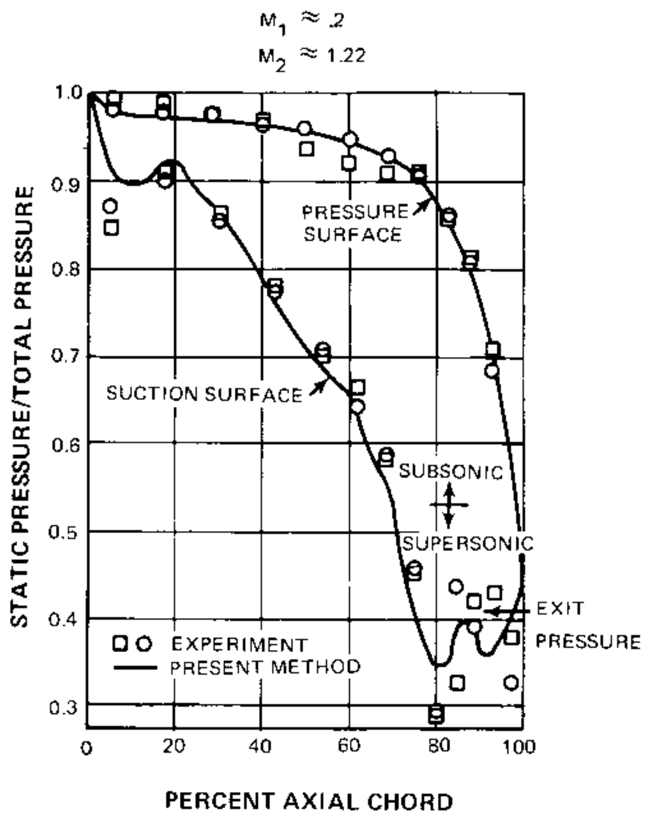

Fig. 4(b) Airfoil surface pressure distribution

deg. Excellent agreement between prediction and experiment can be noted. The analysis correctly predicts two undesirable aspects of this blade: The rapid expansion and recompression on the forward section of the suction surface is caused by leading edge blockage. This problem is frequently encountered in the design of a highly loaded turbine. The second problem is the overexpansion near the 80 percent chord location which results in a complex shock structure and possible boundary-layer separation.

In Fig. 3, the pressure distribution is predicted on a very highly loaded turbine airfoil. High lifting designs are necessary to reduce the number of airfoils and thereby minimize the weight of the gas turbine. However, in an actual engine design, the load would be redistributed to reduce the overexpansion near the suction surface trailing edge, since the pressure rise occurring in this area is the most common cause of boundary-layer separation. Although this problem is also found in subsonic airfoils, the recompression is usually more extreme in transonic designs. Again, it may be noted that such potential shortcomings can be determined by the numerical computation of the transonic flow field.

In the operation of a high-temperature turbine, the airfoils are often protected by the ejection of cooling air. The proper mass flow can be obtained if the designer knows the static pressure at the exit of each orifice. Fig. 4 shows a rapid expansion on both the suction and pressure surfaces which is typical of a first-stage turbine vane. This wide variation in the static pressure adds to the complexity of the cooling system design. However, the agreement shown between prediction and experiment demonstrates that the mass flow of cooling air can be determined using numerically predicted static pressure distributions. This is especially significant, since engine cooling studies begin in the preliminary design stage when experimental pressure distributions are not yet available.

The analytical approach to cascade design, which was motivated by the attempt to reduce the need for experimental data, has been validated for a number of design problems. In addition, the results were obtained rapidly using a mesh of 975 finite area elements for which the computation time was approximately 7 min. on a Univac 1108. This can be compared to an estimated minimum of 30 min. which would be required in solving the same problem with transient finite difference formulations.

CONCLUSION

An accurate and rapid computational method has been developed which permits the prediction of pressure distributions in two-dimensional transonic cascades. Adverse effects, which might be caused by leading edge blockage, overexpansion, shock structures, and boundary-layer separation, can be predicted and often avoided. The method can, therefore, eliminate costly and time-consuming experimentation in the early stages of an engine design.

ACKNOWLEDGMENT

The author wishes to express his gratitude to Fred Landis for his valuable advice in the preparation of this paper.