

NASA Technical Memorandum 101944

# Transonic Viscous Flow Calculations for a Turbine Cascade With a Two Equation Turbulence Model

(NASA-TM-101944) TRANSONIC VISCOUS FLOW  
CALCULATIONS FOR A TURBINE CASCADE WITH A  
TWO EQUATION TURBULENCE MODEL (NASA Lewis  
Research Center) 16 p CSCL 21E

N89-22607

G3/07 Unclass  
0204419

A.A. Boretti  
*Universita' di Firenze*  
*Florence, Italy*

*and Lewis Research Center*  
*Cleveland, Ohio*

April 1989

**NASA**

TRANSONIC VISCOUS FLOW CALCULATIONS FOR A TURBINE CASCADE  
WITH A TWO EQUATION TURBULENCE MODEL

A.A. Boretti\*

Dipartimento di Energetica, Universita' di Firenze,  
350139 Florence, Italy

and NASA Lewis Research Center  
Cleveland, Ohio 44135 U.S.A.

SUMMARY

The paper presents a numerical method for the study of steady, transonic, turbulent, viscous flow through plane turbine cascades. The governing equations are written in Favre-averaged form and closed with a first order model. The turbulent quantities are expressed according to a two-equation  $K-\epsilon$  model where low Reynolds number and compressibility effects are included. The solution is obtained by using a pseudo-unsteady method with improved perturbation propagation properties. The equations are discretized in space by using a finite volume formulation. An explicit multistage dissipative Runge-Kutta algorithm is then used to advance the flow equations in the pseudo-time. First results of calculations compare fairly well with experimental data.

INTRODUCTION

As the efficiency levels of turbine engines continue to increase, the accurate prediction of blade performances becomes increasingly critical in the development and design process. Although numerical methods to solve steady, transonic, turbulent, viscous flows have been developed, efforts to apply these methods to the calculation of performance of turbine blades have so far proved somewhat unsatisfactory. This is mainly due to the failure of present mathematical models to consistently simulate the complex phenomena inherent in these flows. Furthermore, additional problems arise due to the presence of numerical viscosity in the solution algorithm, sometimes of the same order of magnitude of the physical one.

Previous calculations have been performed by using the simplifying assumption of local equilibrium conditions and then evaluating the turbulent viscosity coefficient via a generalized mixing length formula (ref. 1). These calculations have proved to lead to quite accurate predictions of blade pressure distributions at design point conditions, but in an analysis of blade performances, while qualitative behavior of loss generation has been correctly predicted, predictions of quantitative behavior must still be further improved. Furthermore, in performing calculations of separated flows, only qualitative results can be obtained by retaining the assumption of local equilibrium conditions, as clearly represented by backward facing step flow calculations.

---

\*University visitor.

The work presented here is focused on developing low-Reynolds number and compressible  $K-\epsilon$  turbulence models for solving the problem of correctly predicting blade performances. The method includes low Reynolds number terms, so that the equations are valid all over the laminar, transition, and turbulent region (ref. 2). Furthermore, the method includes a density gradient term to better simulate variable density effects.

The solution is obtained by using a pseudo-unsteady method with improved perturbation propagation properties (ref. 3). The equations are discretized in space by using a finite volume formulation. An explicit multi-stage dissipative Runge-Kutta algorithm is then used to advance the flow equations in the pseudo-time. Multi-stage schemes for the numerical solution of ordinary differential equations are usually designed to give a high order of accuracy, but in a pseudo-unsteady solution these schemes are selected only for their properties of stability and damping. The enhanced stability of a four-stage scheme allows one to considerably reduce the numerical viscosity of the dissipative terms.

#### SYMBOL LIST

C	constant
CFL	Courant number
D	diffusion vector
E	total specific energy
e	specific internal energy
F	flux tensor
f	unknown vector
f	function
H	total enthalpy
H <sub>p</sub>	optimization matrix
I	identity matrix
j	index of the spatial discretization
K	turbulence kinetic energy
k	index of the multistage algorithm
L	length scale of turbulent motions
M	Mach number

m index of the temporal discretization  
N unit outward normal  
NV updating rate  
P production of turbulence kinetic energy  
Pr Prandtl number  
p pressure  
Q work due to turbulence  
q heat flux vector  
R low Reynolds number term  
Re Reynolds number  
S source vector  
Sc Schmidt number  
T residual  
t time  
tu turbulence intensity  
V velocity  
v volume  
x axial coordinate  
Y tangential coordinate  
 $\alpha$  flow angle  
 $\beta$  perturbation speed  
 $\gamma$  specific heat ratio  
 $\delta r$  characteristic volume dimension  
 $\delta t$  time step  
 $\delta \Sigma$  surface area  
 $\varepsilon$  turbulence kinetic energy dissipation rate  
 $\zeta$  kinetic energy loss coefficient

$\theta$  multistage scheme coefficient  
 $\kappa$  thermal conductivity  
 $\mu$  viscosity coefficient  
 $\rho$  density  
 $\Sigma$  boundary of the fixed volume  
 $\tau$  viscous stress tensor  
 $\Omega$  artificial viscosity coefficient

Subscripts

*i* inviscid  
*is* isentropic  
*l* laminar  
*t* turbulent  
*v* viscous  
*0* total  
*1* inlet  
*2* outlet

GOVERNING EQUATIONS

The unknown vector  $f$  is the solution of a system of conservation equations. This system is written Favre-averaged, dimensionless, vector, integral form as follows

$$\int_{\Sigma} \int N \cdot (F_i + F_v) d\Sigma = \int \int \int_v S dv$$

The basic dependent variables are density, velocity, and energy. Their conservation equations read as follows

$$f = \begin{pmatrix} \rho \\ \rho \cdot V \\ \rho \cdot E \end{pmatrix}$$

$$F_i = \begin{pmatrix} \rho \cdot V \\ \rho \cdot V V + p \cdot I \\ \rho \cdot E \cdot V + p \cdot I \cdot V \end{pmatrix}$$

$$F_V = \begin{pmatrix} 0 \\ -\tau \\ -\tau \cdot V + q \end{pmatrix}$$

$$S = \begin{pmatrix} 0 \\ 0 \\ \rho \cdot \varepsilon - P + Q \end{pmatrix}$$

where from the equation of state of a perfect gas

$$P = (\gamma - 1) \cdot \rho \cdot e$$

with

$$e = \left( E - V^2/2 \right) = \left( H - V^2/2 \right) / \gamma$$

The stress tensor  $\tau$  is the sum of a laminar and a turbulent part, where the latter is expressed according to a classical eddy viscosity concept, i.e.

$$\tau = -2/3 \cdot (\mu/Re \cdot \text{div}(V) + \rho \cdot K) \cdot I + 2 \cdot \mu/Re \cdot (\text{def}(V))$$

where

$$\mu = \mu_l + \mu_t$$

Similarly, the heat flux vector is given by

$$q = -\gamma \cdot k \cdot \text{grad}(e) / (Re \cdot Pr)$$

$$k = k_l + k_t$$

The turbulent viscosity coefficient is expressed according with the Prandtl-Kolmogorov formulation

$$\mu_t = C_\mu \cdot f_\mu \cdot \rho \cdot Re \cdot K^2 / \varepsilon$$

while

$$k \approx \mu$$

The turbulence variables are the turbulence kinetic energy and its dissipation rate. Their conservation equations are written in the following low Reynolds number and compressible form

$$f = \begin{pmatrix} \rho \cdot K \\ \rho \cdot \varepsilon \end{pmatrix}$$

$$F_i = \begin{pmatrix} \rho \cdot V \cdot K \\ \rho \cdot V \cdot \epsilon \end{pmatrix}$$

$$F_v = \begin{pmatrix} D_K \\ D_\epsilon \end{pmatrix}$$

$$S = \begin{pmatrix} P - Q - \rho \cdot \epsilon + R_K \\ (P - Q) \cdot C_{\epsilon 1} \cdot f_{\epsilon 1} \cdot \epsilon / K - C_{\epsilon 2} \cdot f_{\epsilon 2} \cdot \rho \cdot \epsilon^2 / K + R_\epsilon \end{pmatrix}$$

where the production of turbulent energy from the mean flow energy  $P$  and the work due to turbulence  $Q$  are given by

$$P = \mu_t \cdot (\text{def}(V) : \text{def}(V)) / \text{Re}$$

$$Q = C_\rho \cdot \mu_t \cdot (\text{grad}(\rho) \cdot \text{grad}(\rho) / \rho^2) / \text{Re}$$

the low Reynolds number functions are given by (ref. 2)

$$f_\mu = \exp(-3.4 / (1. + \text{Re}_t / 50.))$$

$$f_{\epsilon 1} = 1.0$$

$$f_{\epsilon 2} = 1.0 - 0.33 \cdot \exp(-\text{Re}_t^2)$$

$$R_\epsilon = -2. \cdot \mu_\varrho \cdot (\text{grade}(\epsilon))^2 / \text{Re}$$

$$R_K = -2. \cdot \mu_\varrho \cdot (\text{grad}(K))^2 / \text{Re}$$

where

$$\text{Re}_t = \rho \cdot K^2 \cdot \text{Re} / (\mu_\varrho \cdot \epsilon)$$

and the model constants are assumed as

$$C_{\epsilon 1} = 1.43$$

$$C_{\epsilon 2} = 1.92$$

$$Sc_\epsilon = 1.3$$

$$Sc_K = 1.0$$

$$C_\mu = 0.09$$

$$C_\rho = 1.0$$

The above two equation model does not take into account the preferential damping of velocity fluctuations in the direction normal to the wall, but it is quite general and it is useful in laminar, transition, and turbulent regions. Furthermore, the model adopts the assumptions and approximations which are normally used for constant density flows, by retaining the gradient diffusion model to be rewritten in the density weighted form without any

explicit account being taken of density fluctuations. However, the introduction of the compressibility term  $Q$  allows a partial consideration of variable density effects.

Along the inflow boundary, the total pressure, total density, flow angle, inlet turbulence level, and length scale of the turbulent motions are specified, while the Mach number is extrapolated from the interior. Along the outflow boundary, the static pressure is specified, and all the other variables are extrapolated. Along the solid boundaries, the no slip condition requires the vanishing of velocity, turbulence kinetic energy, and turbulence kinetic energy dissipation rate, the latter intended to be the modified quantity used in the conservation equations. Furthermore, for adiabatic flows, the specific internal energy gradient normal to the wall is set equal to zero. The density is finally extrapolated by the interior.

### NUMERICAL SOLUTION

The proposed equations are solved in two-dimensional geometries by using a pseudo unsteady method with a finite volume, dissipative, explicit discretization. The solution of the steady equations is obtained as the asymptotic solution of the following artificial unsteady equations

$$\int_V \int \int \left( \frac{\partial f}{\partial t} \right) \cdot H_p^{-1} dv + \int_{\Sigma} \int N \cdot (F_i + F_v) d\Sigma = \int_V \int \int S dv$$

These unsteady equations are generally constructed in order to obtain the better convergence rate, obviously providing that the steady state solution is not altered.

From the identity between the convergence process and the elimination process of the initial perturbations to the steady solution, the convergence parameters are determined in order to improve the perturbation propagation or damping. We use (ref. 3)

$$H_p = \delta r / \beta_{\max}$$

$$\begin{vmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ f_1 \cdot v^2/2 & -f_1 \cdot v_x & -f_1 \cdot v_y & f_2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{vmatrix}$$

where

$$f_1 = \min(M^2 - 1., 0)$$



$$f_2 = \max(f_1 + 1., C_p)$$

with

$$M = V \cdot (\gamma \cdot P/\rho)^{-1/2}$$

$C_p$  is a small positive number, and

$$\beta_{\max} = \min(\beta_1, \beta_2)$$

$$\beta_1 = V \cdot \{(1. + M^2)/2. + [((1. - M^2)/2.)^2 + 1.]^{1/2}\}$$

$$\beta_2 = V \cdot (1. + 1./M)$$

These convergence parameters produce an improved ratio for subsonic flows between the speeds of the fastest and slowest perturbation (ref. 3), and therefore result in an improved propagation.

These equations are then discretized in space by using a finite volume discretization. The mesh is nonorthogonal and curvilinear, conforming to the boundaries of the domain, with lines intersecting at arbitrary angles, properly refined where high gradients are expected to occur. The discretization nodes, located at the intersection of these lines, are the centers of hexagonal control volumes, obtained by connecting the six surrounding nodes. A sample computational domain and the hexagonal control volume are shown in figure 1.

The discretized equations are written as follows (refs. 2 and 4)

$$\frac{\partial f}{\partial t} = H_p \cdot \left\{ - \left[ \sum_{j=1}^6 (F_{l,j} + F_{v,j}) \cdot N_j \cdot \delta \Sigma_j \right] / v + S \right\}$$

where the subscript  $j$  refers to every face of the finite volume. The discretization is second order accurate on smoothly varied meshes.

The equations are finally discretized in time by using an explicit, dissipative discretization. Let the previous equation be rewritten with the addition of a dissipative term as follows

$$\frac{\partial f}{\partial t} = T(f) + D(f)$$

where  $T$  represents the residual and  $D$  is the dissipative term. An explicit  $k$ -stage Runge-Kutta algorithm, based on the work of Jameson (ref. 6), may be written as follows

$$f^{(0)} = f^m$$

$$f^{(1)} = f^{(0)} - \theta_1 \cdot \delta t \cdot [T(f^{(0)}) + D(f^{(0)})]$$

$$f^{(k-1)} = f^{(0)} - \theta_{k-1} \cdot \delta t \cdot [T(f^{(k-2)}) + D(f^{(k-2)})]$$

$$f^{(k)} = f^{(0)} - \delta t \cdot [T(f^{(k-1)}) + D(f^{(k-1)})]$$

$$f^{m+1} = f^{(k)}$$

A four-stages scheme, with the standard coefficients

$$\theta_1 = 1/4$$

$$\theta_2 = 1/3$$

$$\theta_3 = 1/2$$

has a Courant limit CFL = 2.8 (refs. 6 and 7). If the dissipative part is evaluated only once, the scheme has a slightly reduced Courant limit CFL = 2.6 (ref. 6), but it is computationally more efficient.

In order to further improve the computational efficiency, the dissipation is corrected and the viscous terms are evaluated only at fixed iterations. Then the k-stage scheme can be conveniently rewritten as follows

$$f^{(0)} = f^m$$

$$f^{(1)} = f^{(0)} - \theta_1 \cdot \delta t \cdot [T_i(f^{(0)}) + T_V(f^{m*}) + D(f^{(0)}) - \Omega \cdot D(f^{m*})]$$

$$f^{(k-1)} = f^{(0)} - \theta_{k-1} \cdot \delta t \cdot [T_i(f^{(k-2)}) + T_V(f^{m*}) + D(f^{(0)}) - \Omega \cdot D(f^{m*})]$$

$$f^{(k)} = f^{(0)} - \delta t \cdot [T(f^{(k-1)}) + T_V(f^{m*}) + D(f^{(0)}) - \Omega \cdot D(f^{m*})]$$

$$f^{m+1} = f^{(k)}$$

The Courant limit remains substantially unaltered.

The dissipative terms are given as follows (ref. 2)

$$D^m = 1/(6 \cdot \delta t) \cdot \sum_{j=1}^6 (f_j^m - f^m)$$

and

$$\Omega = C_{\Omega 1} \cdot \left| 1 - C_{\Omega 2} \cdot 1/6 \cdot \sum_{j=1}^6 (\rho_j^m - \rho^m) \right|$$

The subscript j refers now to every surrounding node involved in the finite volume approximation, and the superscript m refers to local time m · δt. The terms referring to time m\* · δt are updated only at specific

iterations and assumed constant between two updating. The updating rate is taken equal to  $NV$  iterations, where  $NV$  is equal to 25, as a result of a numerical optimization.

The dissipative term  $D$  is an approximation of second order differences. Without correction, the artificial viscosity coefficient is of the order of  $\delta r^2/\delta t$ , and the scheme is only first order accurate. When the viscosity is corrected and  $\Omega = 1 - O(\delta r)$ , the viscosity has a coefficient of the order of  $\delta r^3/\delta t$ , and the scheme is second order accurate.

$C_{\Omega 1}$  and  $C_{\Omega 2}$  are vectors of coefficients, i.e., the viscosity coefficients are dependent on this particular equation, for an improved accuracy/convergence ratio (ref. 2).

The four-stage Runge-Kutta algorithm adopted here replaces an Euler algorithm, i.e., a one-stage Runge-Kutta algorithm (refs. 1 to 3). Due to the enhanced stability, the total amount of the numerical viscosity is considerably reduced. This is the main advantage of the present time integration, since the total CPU time remains substantially unaltered, the reduction in the number of time steps due to convergence being balanced by the increase in the computational time required by every time step. If we assign two work units to the evaluation of the residual (there are both viscous and inviscid contributions) and one work unit to the evaluation of the dissipative term and we define the efficiency of the scheme as the permitted CFL number divided by the number of work units, the one-stage scheme has an efficiency of about  $2.6/(5 + 2/NV)$ , i.e. about the same efficiency.

The time step is evaluated according to the classical CFL stability limit all over the computational domain. It is taken slightly smaller than the local CFL number in order to take into account the neglected stability limit due to the viscous terms.

Due to the efficient pseudo unsteady solution, the method appears to be rather fast, while the explicit finite volume discretization allows ease of understanding and computer programming.

## RESULTS

The turbulence model has been previously applied to the computation of backward facing step flows, in order to test the prediction capability of the low Reynolds number formulation when dealing with strongly recirculating flows (ref. 2). The computed length of the recirculation region generally compares fairly well with the experimental one. For a nearly incompressible flow, a Reynolds number (based on step height and inlet flow conditions) of 42 000 and an expansion ratio of 0.66, the predicted reattachment length is 7.50 against a measured value of 7.33 (ref. 2). Further calculations performed by modifying the Reynolds number, the inlet turbulence level, or the inlet length scale of the turbulence have shown the expected behavior.

Sample calculations are performed here on a transonic turbine profile. The blade tested is a mean section of a gas rotor blade. The blade cascade is

shown in figure 2. The blade coordinates and experimental velocity distributions are given in (ref. 5), while the experimental kinetic energy loss coefficient distributions are given in (ref. 8).

Calculations were performed on a very fine computational grid, made up of 141 pitchwise lines, 91 between leading and trailing edge, and 61 pseudo streamlines. About  $20 \times 10^3$  iterations are performed before convergence. The total CPU time on a VAX 8800 is less than 60 h.

The inlet total temperature is  $T_{01} \sim 280$  K. The inlet total pressure varies more or less linearly with the outlet isentropic Mach number, the latter defined as follows

$$M_{2, is} = \{2/(\gamma - 1) \cdot [P_2/P_{01}]^{(\gamma-1)/\gamma} - 1\}^{1/2}$$

and  $p_{01} \sim 1.5$  bar at  $M_{2, is} \sim 0.8$ ,  $p_{01} \sim 2.75$  bar at  $M_{2, is} \sim 1.4$ . The inlet Reynolds number, based on inlet conditions and blade chord, carries linearly with the outlet isentropic Mach number in the same range, It is  $Re_1 \sim 300\ 000$  at  $M_{2, is} \sim 1.4$ . Calculations were performed for an inlet flow angle and an outlet isentropic Mach number given as

$$\alpha_1 = 24^\circ \quad M_{2, is} = 1.10, 1.42$$

In the comparison of predicted and measured /5/ blade isentropic Mach number distributions at  $M_{2, is} \sim 1.42$  in figure 3, agreement appears to be very satisfactory.

In a comparison of predicted and measured /8/ outlet kinetic energy loss coefficients, defined as

$$\zeta = [(p_{01}/p_{02})^{(\gamma-1)/\gamma} - 1] / [1/2 \cdot (\gamma-1) \cdot M_{2, is}^2]$$

where  $P_{02}$  is a mass averaged value over a blade pitch, the predicted values are  $\zeta \sim 0.047$  at  $M_{2, is} \sim 1.10$ ,  $\zeta \sim 0.032$  at  $M_{2, is} \sim 1.42$ , while the measured values are  $\zeta \sim 0.048$  at  $M_{2, is} \sim 1.10$ ,  $\zeta \sim 0.030$  at  $M_{2, is} \sim 1.42$ , thus resulting in an experimental uncertainty in the predicted values very close to the experimental one.

The good accuracy is due to the use of a relatively refined mathematical model and of a numerical integration with high spatial resolution and reduced numerical viscosity.

If only the blade isentropic Mach number distributions has to be predicted, a simpler mathematical model and a less refined numerical integration can be used. If the two equation turbulence model is replaced by a mixing length model, the total enthalpy is assumed to be constant, the time integration is performed with a simple one step scheme, and if only a 65 by 19 computational grid is used, the isentropic Mach number distributions are predicted within an engineering accuracy, as represented in figures 4, 5, 9, and 10. In /9/ the length scale of the turbulent motions is given by using a classical expression, derived from the formulation proposed by Nikuradse for pipe flows and from the Van Driest damping factor, In /10/, it is simply taken proportional to the size of the spatial discretization.

In the prediction of the outlet kinetic energy loss coefficient, these simplified solutions allow neither a qualitative nor a quantitative agreement. If a finer grid is used, the loss can be reduced, but the qualitative behavior cannot be properly reproduced. In calculations performed with a 79 by 39 computational grid/1/, the predicted values are  $\zeta \sim 0.054$  at  $M_{2is} \sim 1.10$ ,  $\zeta \sim 0.055$  at  $M_{2is} \sim 1.42$ , while the measured values are  $\zeta \sim 0.048$  at  $M_{2is} \sim 1.10$ ,  $\zeta \sim 0.030$  at  $M_{2is} \sim 1.42$ .

## CONCLUSIONS

The paper has presented a method for the solution of the Navier Stokes equations in transonic cascade flow fields.

The work has been focused on the development of a low-Reynolds number and compressible K- $\epsilon$  turbulence model. The turbulence model includes low Reynolds number terms, so that the equations are valid all over the laminar, transition, and turbulent regions. Furthermore, the model includes a density gradient term to better simulate variable density effects.

The use of a four-stage Runge-Kutta algorithm allows one to significantly reduce the numerical viscosity due to the dissipative terms, thus leading to a better accuracy.

First results of calculations compare favorably with experimental data. Even if uncertainties still remain, due to the limit of the numerical solution algorithm (influence of grid refinement, numerical viscosity, . . .) the proposed method appears to be adequate for the study of steady transonic flows in turbine cascades.

## REFERENCES

1. Boretti, A.A.; and Martelli, F.: Improvements in Gas Turbine Blade Loss Prediction Methods. Paper presented at the 1988 ASME COGEN-TURBO, Montreux, Switzerland, Aug.-Sept. 1988.
2. Boretti, A.A.; and Martelli, F.: Accuracy and Efficiency of Time Marching Approach for Combustor Modelling. Validation of Computational Fluid Dynamics, AGARD, Paris, France, May 1988, AGARD CP- , to be published.
3. Boretti, A.A.; and Martelli, F.G.: Application of a Fast Pseudo Unsteady Method to Steady Compressible Turbulent Reacting Flows. First National Fluid Dynamics Congress, Part 2, AIAA, 1988, pp. 1436-1442.
4. Arts, A.: Cascade Flow Calculations Using a Finite Volume Method. Numerical Methods for Flows in Turbomachinery Bladings, Vol. 1, VKI-LS-1982-05-VOL-1, Brussels, Belgium, 1982.
5. Sieverding, C.: Base Pressure Problem in Transonic Turbine Cascades. Transonic Flows in Axial Turbomachinery, Vol. 2, VKI-LS-84-VOL-2, Brussels, Belgium, 1976.

6. Jameson, A., et al.: Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes. AIAA Paper 81-1259, June 1981.
7. Jorgenson, P.C.E.; and Chima, R.E.: An Explicit Runge-Kutta Method for Unsteady Rotor-Stator Interaction. AIAA Paper 88-0049, Jan. 1988 (NASA TM-100787).
8. Boretti, A.A.; and Martelli, F.: A Simple Procedure to Compute Losses in Transonic Turbine Cascades. ASME Paper 85-GT-21, Mar. 1985.
9. Boretti, A.A.; and Martelli, F.: Pseudo Three-Dimensional Transonic Flow in Turbine Bladings. AIAA Paper 87-2148, June 1987.
10. Boretti, A.A.; and Martelli, F.: Calcolo del Flusso Viscoso in Schiere di Turbina. Proceedings of the 42nd Congresso ATI, Genoa, Italy, Sept. 1987.

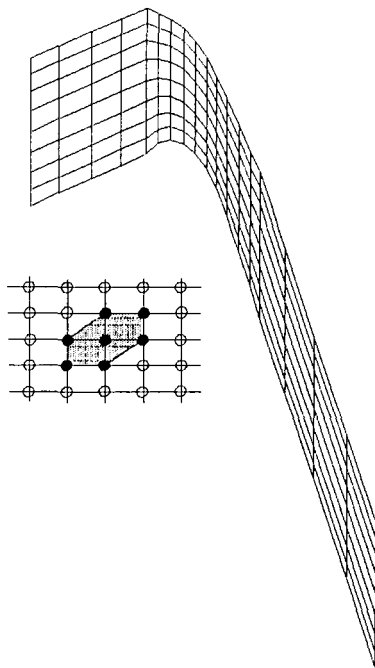


FIGURE 1. - COMPUTATIONAL DOMAIN AND HEXAGONAL CONTROL VOLUME.

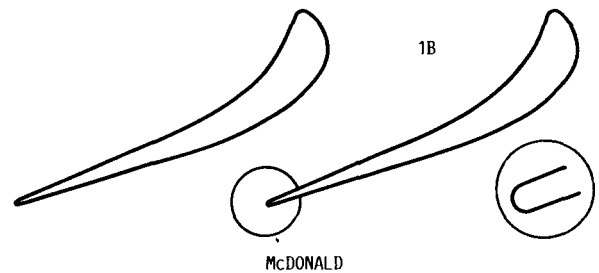


FIGURE 2. - BLADE CASCADE TESTED.

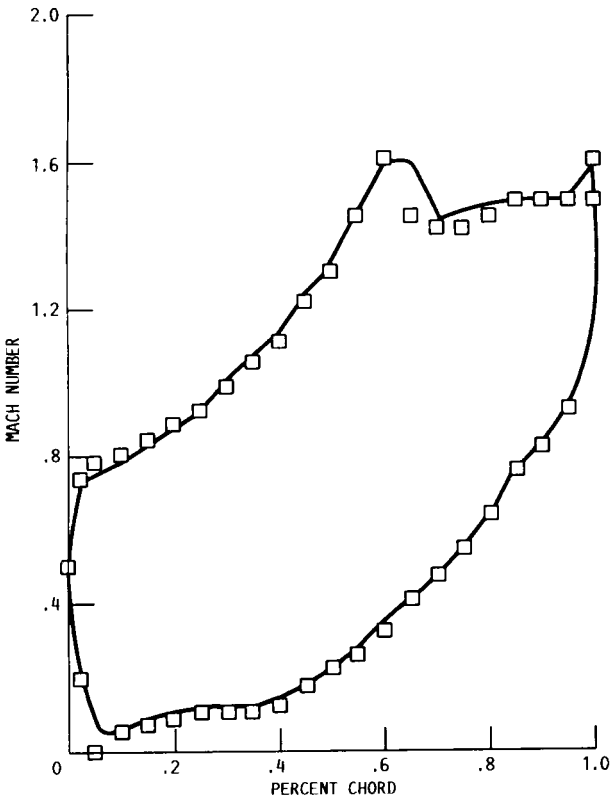


FIGURE 3. - COMPARISON OF PREDICTED AND MEASURED BLADE VELOCITY DISTRIBUTIONS.

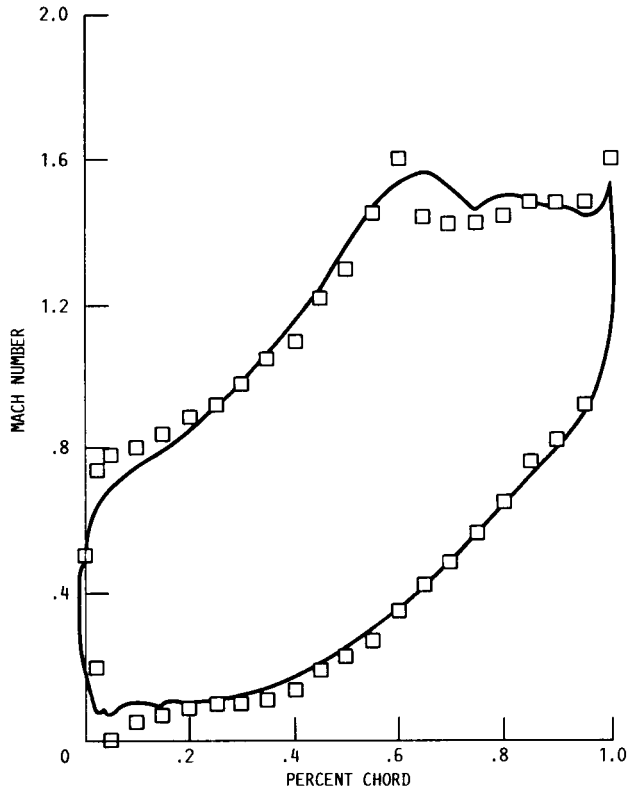


FIGURE 4. - COMPARISON OF PREDICTED AND MEASURED BLADE VELOCITY DISTRIBUTIONS.

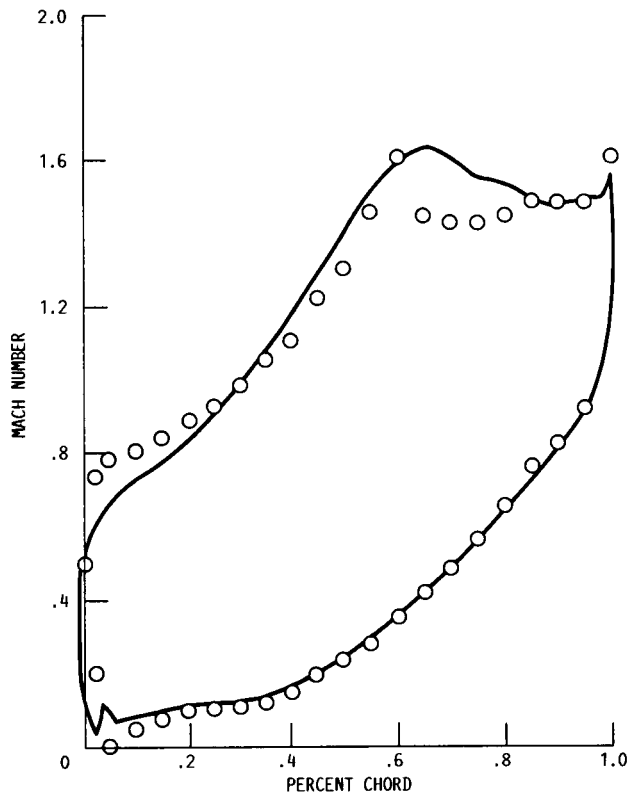


FIGURE 5. - COMPARISON OF PREDICTED AND MEASURED BLADE VELOCITY DISTRIBUTIONS.

1. Report No. NASA TM-101944		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle Transonic Viscous Flow Calculations for a Turbine Cascade With a Two Equation Turbulence Model				5. Report Date April 1989	
				6. Performing Organization Code	
7. Author(s) A.A. Boretti				8. Performing Organization Report No. E-4634	
				10. Work Unit No. 505-62-21	
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	
15. Supplementary Notes A.A. Boretti, Dipartimento di Energetica, Universita' di Firenze, Via S. Marta, 3 50139 Florence, Italy; university visitor at NASA Lewis Research Center. Technical Monitor: Louis A. Povinelli.					
16. Abstract The paper presents a numerical method for the study of steady, transonic, turbulent viscous flow through plane turbine cascades. The governing equations are written in Favre-averaged form and closed with a first order model. The turbulent quantities are expressed according to a two-equation k-ε model where low Reynolds number and compressibility effects are included. The solution is obtained by using a pseudo-unsteady method with improved perturbation propagation properties. The equations are discretized in space by using a finite volume formulation. An explicit multistage dissipative Runge-Kutta algorithm is then used to advance the flow equations in the pseudo-time. First results of calculations compare fairly well with experimental data.					
17. Key Words (Suggested by Author(s)) CFD Numerical methods Turbomachinery Cascade flow			18. Distribution Statement Unclassified - Unlimited Subject Category 07		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No of pages 16	22. Price* A03