NASA Technical Memorandum 103624
ICOMP-90-23

Navier-Stokes Analysis of Transonic Cascade Flow

A. Arnone
Institute for Computational Mechanics in Propulsion
Lewis Research Center
Cleveland, Ohio

and

M.-S. Liou and L. Povinelli
Lewis Research Center
Cleveland, Ohio

October 1990

NASA

NAVIER-STOKES ANALYSIS OF TRANSONIC CASCADE FLOW

A. Arnone*
Institute for Computational Mechanics in Propulsion
Lewis Research Center
Cleveland, Ohio

and

M.-S. Liou and L. Povinelli
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio

E-5791

INTRODUCTION

In recent years, Computational Fluid Dynamics (CFD) has seen an important evolution which is due to both efforts in the development of numerical models and great improvement in the power of computers. As the need for efficiency and weight reduction has driven designers to investigate the details of the complex flowfields in which each component is expected to operate, it is reasonable to see CFD taking an indispensable role in the development of modern, high performance turbomachinery. In the past few years several two and three-dimensional codes for solving inviscid flows have reached a good level of maturity and are commonly used in turbomachinery applications [1, 2] . Inviscid prediction is often cost effective and gives important basic information under design conditions. On the contrary, this approach does not provide any information about heat transfer and boundary-layer thickening, both of which generally require solutions of Navier-Stokes equations. Moreover, in turbine blades, we generally have to deal with rounded and thick trailing edges. The flow about this region is very complex and dominated by viscous effects which may strongly influence the blade load and the whole flowfield. Although important progress has been made in solving the Navier-Stokes equations (e.g [3, 4, 5, 6, 7]), much work is still neded to achieve robustness, accuracy, and especially in turbulence modelling. In addition, rotor and stator cascades of modern turbomachinery are often characterized by a high turning geometry and/or by strong flow deviations from the axial. As results, generation of meshes capable of picking up the flow details is not as straightfoward and is an item that still needs to be improved.

The aim of this work is to present some recent progress in two-dimensional cascade viscous calculations with particular attention to aspects which are important for the designer , such as accurancy, computational cost and correct prediction of loss coefficients and exit flow angles.

*On leave from Department of Energy Engineering, University of Florence, Florence, Italy. Work funded, in part, under Space Act Agreement C-99066-G.

As for accurancy, a new kind of elliptically-generated C-type grid is adopted. The removal of periodicity on the wake allows the grid to be only slightly distorted even for cascades having a large camber and a high stagger angle. This allows us to pick up details of shock systems with a reasonable number of grid points. In addition , a very low level of artificial dissipation is guaranteed by using Swanson's and Turkel's eigenvalues scaling [9].

The Reynolds-averaged Navier-Stokes equations are efficiently solved using a Runge-Kutta scheme in conjunction with accelerating techniques like multigridding and variable coefficient implicit residual smoothing. The thin-layer assumption is adopted and the two-layer eddy-viscosity model of Baldwin and Lomax is used for the turbulence closure.

The capability of the procedure is shown by comparing computed results to experiments for a typical highly loaded blade. Due to the complexity of transonic cascade flows, we also investigate the convergence in space of the computed solutions. A grid independence study is carried out and discussed by comparing wall pressure distributions and loss coefficient.

With the accelerating strategies, viscous solution can be obtained in less than one minute on a modern supercomputer, such as a Cray Y-MP.

COMPUTATIONAL GRID

It is known that the grid structure must be selected carefully in order to achieve an accurate resolution of complex flow fields. When dealing with the construction of grids for turbomachinery blade passages , aspects such as accurate leading and trailing edge flow solution and description of wakes and shocks are of great importance.

Sheared H-type grids are fairly common in turbomachinery applications. They are easy to generate and to extend to three-dimensional applications. Unfortunately those grids provide poor leading and trailing edge flow reproduction. On the leading edge truncation errors due to the grid distortion introduce extra entropy which is transported downstream on the blade surface.

Those kinds of problems do not exist on O-type grids. However, now the difficulty is in the wake, where grid clustering, while avoiding distortion, is desired. In addition, when the flow is transonic, O-type meshes are not suitable for a fine reproduction of the outlet shock system.

Therefore , if a single grid structure is chosen, the C-type seems to have the best overall capability. The non-periodic C-type grids we used for the present calculations are generated with an elliptic procedure that solves the discretized Poisson equations using a relaxation procedure [10]. Forcing functions, like the one proposed by Steger and Sorenson [11], are used to control the grid spacing and orientation at the wall; while periodicity conditions are imposed on the external part of the mesh. Viscous flow grids are obtained from the inviscid one by embedding lines with the desired spacing distribution.

COMPUTATIONAL METHOD

The unsteady, Reynolds-averaged, thin-layer Navier-Stokes equations are discretized in space using a finite volume approach and a cell-centered scheme [12, 7]. The effect of turbulence is taken into account by using the eddy-viscosity hypothesis and the two-layer mixing length algebraic model of Baldwin and Lomax [13]. Also, the simple transition model suggested in reference 13 is adopted. On the wake, where the grid is not periodic, linear interpolations are used to compute the necessary flow quantities.

In order to ensure stability and to prevent odd-even point decoupling, artificial dissipation terms are added to the governing equations. The Jameson artificial dissipation model used in this paper is a blending of second and fourth differences. In order to minimize the amount of artificial dissipation inside the boundary layer the eigenvalues scaling of Swanson and Turkel [9] has been used to weigh those terms. Smoothing fluxes are computed on the boundary so that no errors in the conservation property are introduced, globally, by the artificial dissipation [2, 9, 14]. Boundary conditions are treated via the theory of characteristics. Total enthalpy, total pressure, and the flow angle are specified at the subsonic-axial inlet while the outgoing Reimann invariant is taken from the interior. At the subsonic-axial outlet, the average value of the static pressure is prescribed and the outgoing Reimann invariant, the total enthalpy, and the component of velocity parallel to the boundary are extrapolated together with the circumferential distribution of the pressure. On the solid wall the momentum equation and the no-slip conditions are used to compute the pressure which is the only variable needed from the cell-centered discretization.

The equations are advanced in time till the steady-state state solution is obtained using an explicit four-stage Runge-Kutta scheme. Good, high-frequency damping properties, which are important for the multigrid process, are obtained from this scheme by performing two evaluations of the dissipative terms at the first and second stages. For economy the contribution of the viscous terms is computed only at the first stage and then frozen for the remaining stages.

Four techniques are employed to improve the computational efficiency: 1) local time-stepping; 2) residual smoothing; 3) multigrid; 4) grid refinement. When only the steady-state solution is of interest, local time-stepping and implicit residual smoothing can be used to improve the robustness and the convergence of the basic scheme. In the present work the variable coefficient formulation of the implicit smoothing [7, 8] is used and the time step is computed locally on the basis of a fixed Courant number (typically 5). However most of the reduction in the computational effort is obtained through a multigrid method. Jameson's Full Approximation Storage (FAS) scheme [15] and a V-type cycle with subiterations are used as multigrid strategies. In addition, a grid refinement procedure is used to provide an efficient initialization of the flow field. This strategy is implemented in conjunction with multigrid to obtain a Full-Multigrid process (FMG) [9].

RESULTS AND DISCUSSION

As an application of the computational procedure which has been briefly described we present a detailed study of a typical highly loaded transonic cascade: the VKI-MIT. For this large camber cascade the inlet flow is experienced about 85 degrees turn and accelerated from Mach .25 up to transonic conditions.

We chose a non periodic C-type 353×33 grid as a reference grid , in which 129 points are located on the suction side of the blade and 65 on the pressure side. The spacing at the wall in the normal like direction is $1. \times 10^{-4}$ times the axial cord which allows an average Y^+ value of about unity at the exit Reynolds number of one million.

A four-decade drop in the residual of the continuity equation is used as convergence criteria and this never required more than 50 sec on NASA Lewis Cray Y-MP.

Figure 1 compares the computed isentropic surface Mach number distribution to experiments. The agreement is very good in the whole range of exit conditions that vary from an isentropic Mach number of 1.02 up to 1.44. Both the strength and the location of the throat shock are correctly predicted by the code. In all three cases the flow has a recirculation at the foot of the shock which is cleary indicated by the smooth behavior in the recompression part of the shock.

Due to the complexity of the flowfield, we believed it is necessary to investigate the convergence in space of the computed solutions. Both fine and coarse grids of respectively 769×65 and 193×25 grid points have been used to study the case with sonic exit. Results are depicted in fig. 2 where the isoentropic surface Mach number for the three grids is compared. The convergence in space is basically achieved since the fine grid solution differs from the reference one just in small details near the shock. All the solutions exhibit separation at the foot of the shock. The computation of the fine grid solution took 280 sec on Cray Y-MP at NASA Lewis.

Figures 3 and 4 compare the exit flow angles and the loss coefficients to experiments. Here we report computed data for all the three grids. The agreement with experiments is very good especially in terms of the loss coefficient which is generally very difficult to predict. The level of the convergence in space of the reference solution also is good in terms of loss coefficient.

Details of the fine grid solutions are given in figs. 5 and 6. Figure 5(a) shows an enlargement of the computational grid near the blade passage region. It is evident that low level of skewness obtained using the non-periodic wake. Density contours for the three flow conditions are given in figs. 5(b)-(d). The complex shock system for this blade is sharply captured in the whole computational domain.

Mach number contours near the leading edge are shown in fig. 6(a) where the very clean resolution achieved with the C-type structure of the grid is evident. The flow is well-behaved and the stagnation point sharply captured. The recirculation at the foot of the shock can be seen through the stream function plot of fig. 6(b), resulting in the thickening in the boundary layer after the shock. As is well known, the flow around rounded trailing edges is very complex and care is taken to generate appropriate grids, as shown in fig. 6(c). In fig 6(d), the flow pattern as well as the two vorticies downstream of the separation on the suction and pressure side of the blade are clearly reproduced.

CONCLUSIONS

A new kind of C-type grid has been introduced, this grid is non-periodic on the wake and allows very good flow predictions even for cascades with high turning and large camber. The central-difference, finite volume, scheme with eigenvalues scaling for artificial dissipation originally developed for external flows has also proven to be accurate and to converge well in space for cascade viscous flows. Good overall prediction can be obtained with the Baldwin-Lomax turbulence model both in terms of pressure distribution and loss coefficient, for the cases studied. With the accelerating strategies, accurate transonic viscous solutions can be obtained in less than one minute on a modern supercomputer.

BIBLIOGRAPHY

[1] R. H. Ni. Multiple-Grid Scheme for Solving the Euler Equations. *AIAA Journal*, 20:1565–1571, June 1981.

[2] D. G. Holmes and S. S. Tong. A Three-Dimensional Euler Solver for Turbomachinery Blade Rows. *Transaction of the ASME*, 107:258–264, 1985.

[3] R. V. Chima. Inviscid and Viscous Flows in Cascades with an Explicit Multiple-Grid Algorithm. *AIAA Journal*, 23:1556–1563, October 1985.

[4] C. Hah and R. J. Selva. Navier-Stokes Analysis of Flow and Heat Transfer inside High-Pressure-Ratio Turbine Blade Rows. Technical Report 90-0343, AIAA, 1990.

[5] Ni R. H. Davis, R. L. and J. E. Carter. Cascade Viscous Flow Analysis Using the Navier-Stokes Equations. Technical Report 86-0033, AIAA, 1986.

[6] Nozaki O. Kikuchi K. Nakahashi, K. and A. Tamura. Navier-Stokes Computations of Two- and Three-Dimensional Cascade Flowfields. *Journal of Propulsion and Power*, 5:320–326, May-June 1989.

[7] A. Arnone and R. C. Swanson. A Navier-Stokes Solver for Cascade Flows,. CR 181682, NASA, July 1988.

[8] L. Martinelli. Calculations of Viscous Flows with a Multigrid Method. Ph.d dissertation, MAE Department, Princeton University, July 1983.

[9] R. C. Swanson and E. Turkel. Artificial Dissipation and Central Difference Schemes for the Euler and Navier-Stokes Equations. Technical Report 87-1107, AIAA, 1987.

[10] R. L. Sorenson. A Computer Program to Generate Two-Dimensional Grids About Airfoils and Other Shapes by the Use of Poisoon's Equation. TM 81198, NASA, May 1980.

[11] L. Steger, J. and L. Sorrenson, R. Automatic Mesh-Point Clustering Near a Boundary in Grid Generation with Elliptic Partial Differential Equations. *Journal of Computational Physics*, 33:405–410, 1979.

[12] Schmidt W. Jameson, A. and E. Turkel. Numerical Solutions of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes. Technical Report AIAA-81-1259, AIAA, 1981.

[13] B. S. Baldwin and H. Lomax. Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows. Technical Report AIAA-78-0257, AIAA, 1978.

[14] A. Arnone and S. S. Stecco. Inviscid Cascade Flow Calculations Using a Multigrid Method. Technical report, ASME Turbo Expo, Toronto, June 1989.

[15] A. Jameson. Transonic Flows Calculations. MAE Report 1651, MAE Department, Princeton University, July 1983.

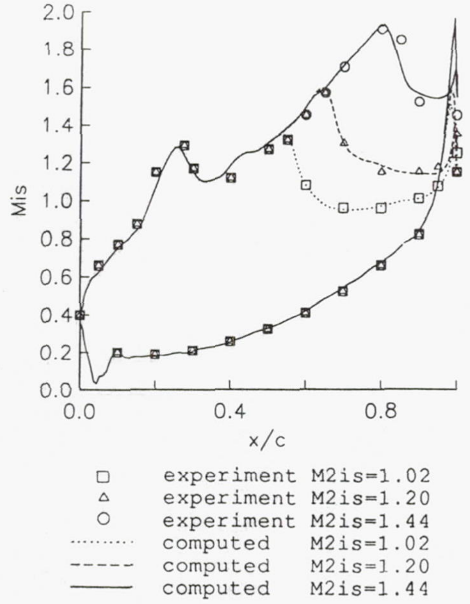


Fig. 1:Isentropic surface Mach number distribution for the VKI-MIT (353x33 grid).

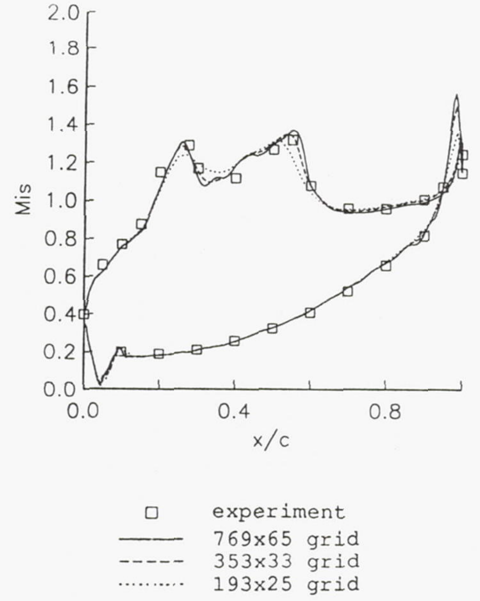


Fig. 2:Isentropic surface Mach number distribution for the VKI-MIT blade (M2is=1.02).

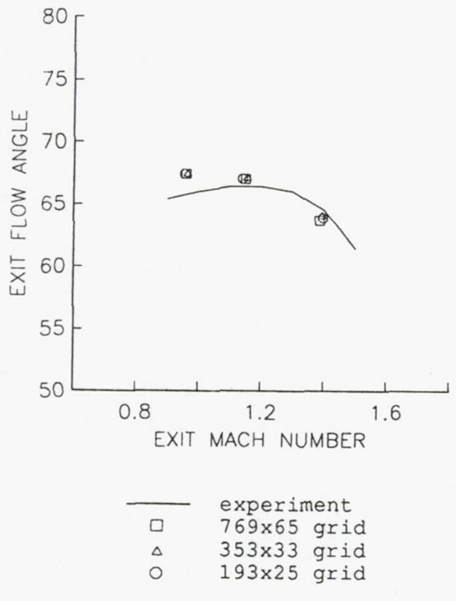


Fig. 3:Exit flow angle as function of exit Mach number for the VKI-MIT blade.

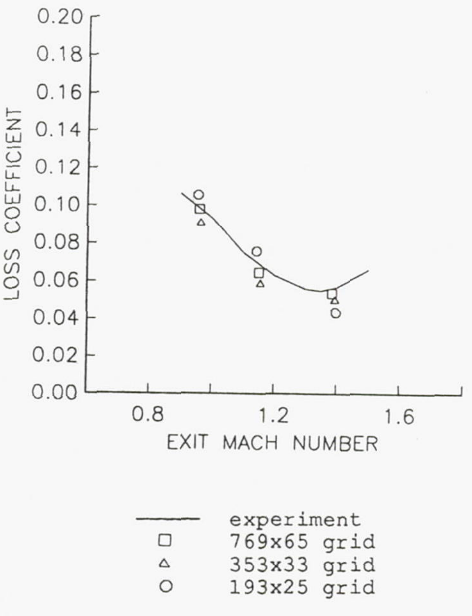
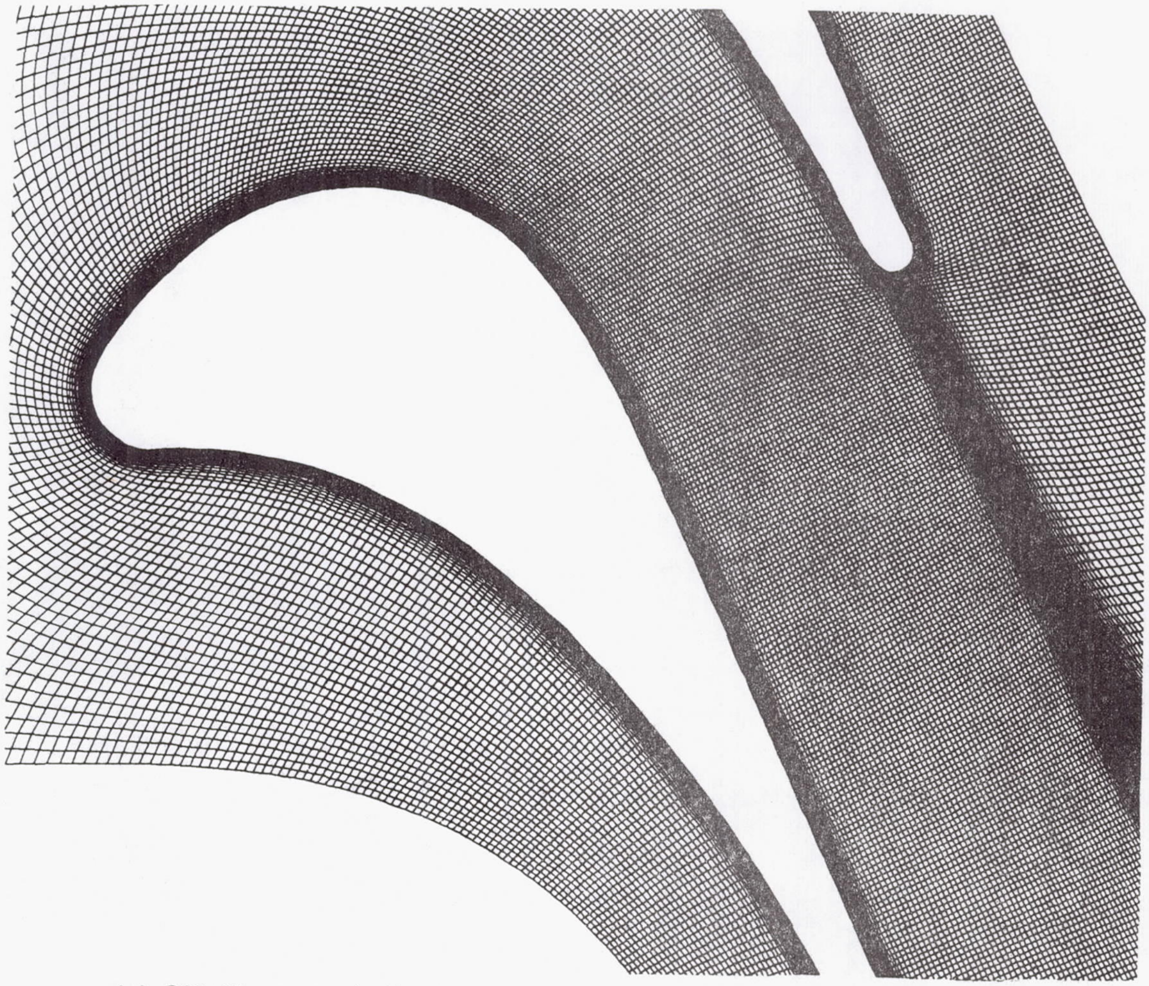
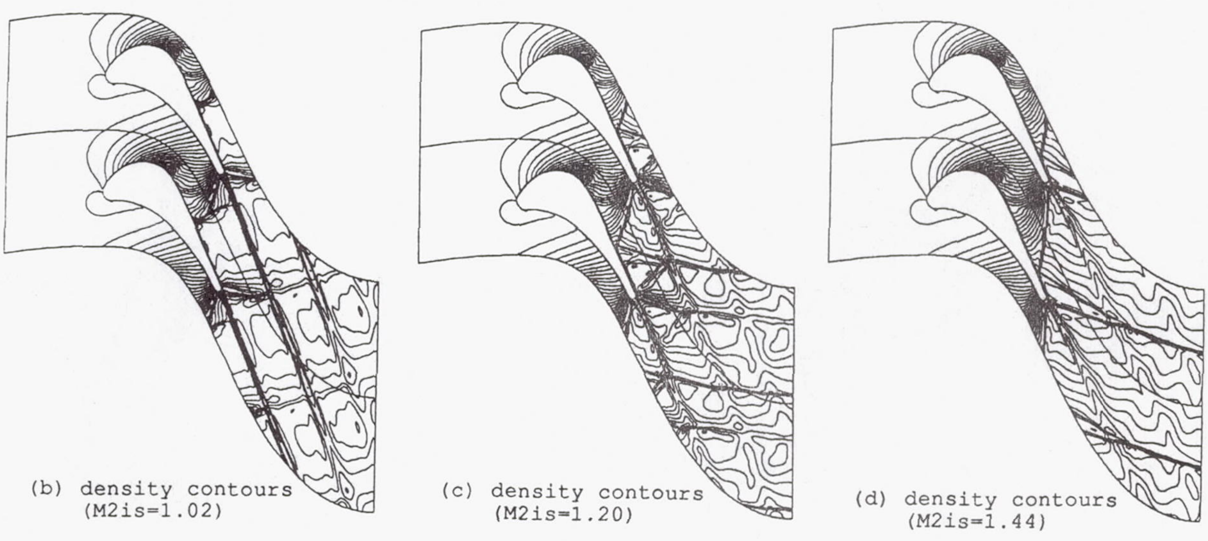


Fig. 4:Loss coefficient as function of exit Mach number for the VKI-MIT blade.



(a) 769x65 non periodic C-type grid for the VKI-MIT blade

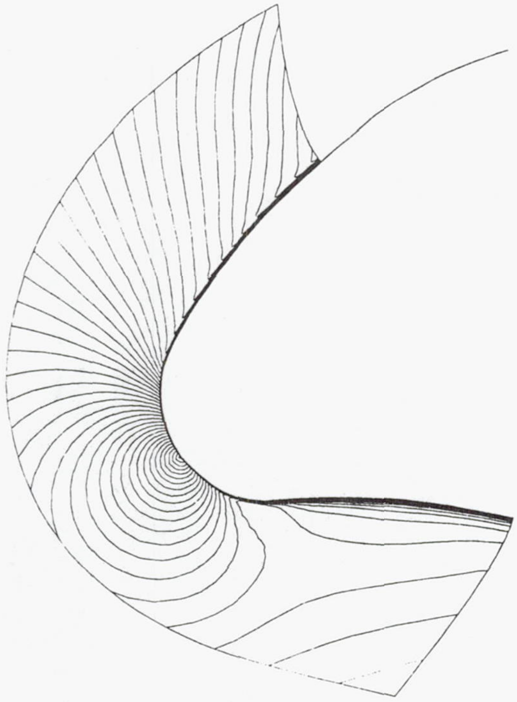


(b) density contours (M2is=1.02)

(c) density contours (M2is=1.20)

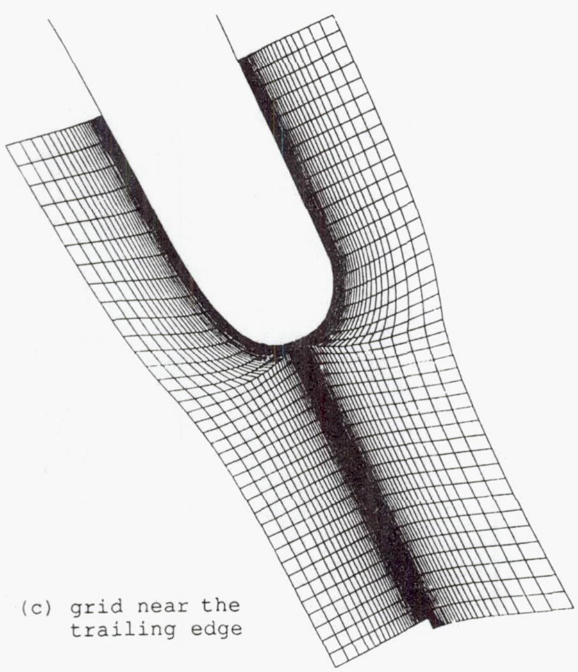
(d) density contours (M2is=1.44)

Fig. 5: Details of the fine grid solution for the VKI-MIT blade.

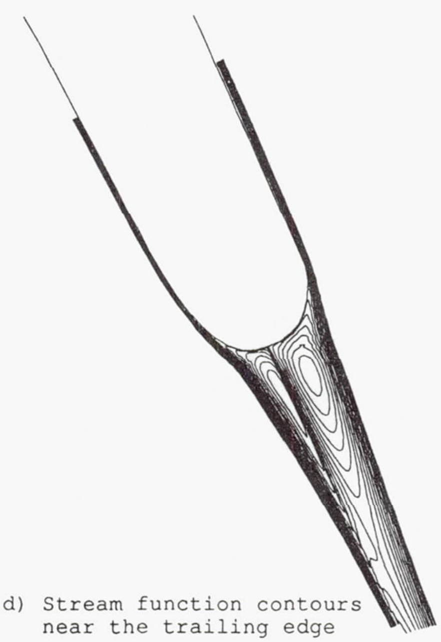


(a) Mach number contours near the leading edge

(b) Stream function contours near the recirculation at the foot of the shock



(c) grid near the trailing edge



(d) Stream function contours near the trailing edge

Fig. 6: Details of the fine grid solution for the VKI-MIT blade (M2is=1.02).

NASA
National Aeronautics and Space Administration

Report Documentation Page

1. Report No. NASA TM-103624 ICOMP-90-23	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle Navier-Stokes Analysis of Transonic Cascade Flow		5. Report Date October 1990
		6. Performing Organization Code
7. Author(s) A. Arnone, M.-S. Liou, and L. Povinelli		8. Performing Organization Report No. E-5791
		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. Arnone, Institute for Computational Mechanics in Propulsion, Lewis Research Center (on leave from Department of Energy Engineering, University of Florence, Florence, Italy); M.-S. Liou and L. Povinelli, Lewis Research Center. Work funded, in part, under Space Act Agreement C-99066-G (Space Act monitor, L. Povinelli).

16. Abstract

A new kind of C-type grid is proposed, this grid is non-periodic on the wake and allows minimum skewness for cascades with high turning and large camber. Reynolds-averaged Navier-Stokes equations are solved on this type of grid using a finite volume discretization and a full multigrid method which uses Runge-Kutta stepping as driving scheme. The Baldwin-Lomax eddy-viscosity model is used for turbulence closure. A detailed numerical study is proposed for a highly loaded transonic blade. A grid independence analysis is presented in terms of pressure distribution, exit flow angles, and loss coefficient. Comparison with experiments clearly demonstrates the capability of the proposed procedure.

17. Key Words (Suggested by Author(s)) Turbomachinery Cascades Multi-grid Navier-Stokes	18. Distribution Statement Unclassified – Unlimited Subject Category 34

19. Security Classif. (of this report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 11	22. Price* A03

NASA FORM 1626 OCT 86

*For sale by the National Technical Information Service, Springfield, Virginia 22161