SECONDARY FLOW IN CASCADES

A. Hamed

University of Cincinnati
Cincinnati, Ohio 45221

June 1984

Final Scientific Report
July 1, 1980 Through December 31, 1983

Grant AFOSR-80-0242

Prepared for

Directorate of Aerospace Sciences
Air Force Office of Scientific Research
Boiling Air Force Base
Washington, D.C. 20332

Approved for Public Release Distribution Unlimited

84 11 20 014
This report covers research work which was conducted at the Univ. of Cincinnati over a period of three years to investigate the secondary flow phenomena in internal flow fields analytically and experimentally. The work was sponsored by the Air Force Office of Scientific Research and its main objective was to develop computational procedures for internal three-dimensional rotational flow fields and to obtain experimental measurements.
Sheet 3 of the three velocity components of the rotational flow in a curved duct with shear inlet velocity profile.

(cont'd on p7)
FORWARD

This technical report covers an analytical and experimental investigation of secondary flow in curved passages for period January 1, 1980 through December 31, 1983. The research was sponsored by the Air Force Office of Scientific Research under Grant No. 80-3006.

The project was monitored by Dr. James Wilson and Captain Michael Johnson, Program Managers, Directorate of Aerospace Research, Air Force Office of Scientific Research, United States Air Force, Washington Air Force Base, DC.

This final report supersedes all the previous interim reports and includes all the technical papers both published and internal preparation, which were written under this research.
ACKNOWLEDGEMENT

The principal investigator wishes to express sincere appreciation to the program managers Dr. J. Wilson and Capt. N. Francis for their continued interest, help to obtain equipment and valuable suggestions during the course of this work. With the help of this grant we have been able to develop our Laser Doppler Velocimeter and data acquisition systems for measuring complex internal flow fields.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acknowledgments</td>
<td>1</td>
</tr>
<tr>
<td>Professional Personnel</td>
<td>2</td>
</tr>
<tr>
<td>Advanced Degrees Awarded</td>
<td>2</td>
</tr>
<tr>
<td>Publications Generated Under This Research Grant</td>
<td>3</td>
</tr>
<tr>
<td>Research Objectives</td>
<td>4</td>
</tr>
<tr>
<td>Accomplished Work</td>
<td>4</td>
</tr>
<tr>
<td>Summary of Significant Results</td>
<td>7</td>
</tr>
<tr>
<td>Technical Applications</td>
<td>8</td>
</tr>
<tr>
<td>Appendix 1 - Inviscid Solution for the Secondary Flow in Curved Ducts</td>
<td>9</td>
</tr>
<tr>
<td>Appendix 2 - The Elliptic Solution of the Secondary Flow Problem</td>
<td>18</td>
</tr>
<tr>
<td>Appendix 3 - Three-Dimensional Rotational Compressible Flow Solution in Variable Area Channels</td>
<td>25</td>
</tr>
<tr>
<td>Appendix 4 - Internal Three-Dimensional Viscous Flow Solution Using the Streamlike Function</td>
<td>37</td>
</tr>
<tr>
<td>Appendix 5 - The Elliptic Solution of 3-D Internal Viscous Flow Using the Streamlike Function</td>
<td>48</td>
</tr>
<tr>
<td>Appendix 6 - LDV Measurements of Three-Dimensional Flow Development in a Curved Rectangular Duct With Inlet Shear Profile</td>
<td>66</td>
</tr>
</tbody>
</table>
ABSTRACT

Measurements of the three velocity components were reported for the flow in a curved rectangular duct with an opposing velocity profile. The results provide the needed information to develop analytical models of internal flows with strong secondary velocities. In addition, new analytical models have developed for predicting three dimensional rotational flows in curved passages, with inlet total pressure distortion. The agreement between the computed results and the experimental measurements suggest that these solutions model the important three dimensional flow field characteristics.
PROFESSIONAL PERSONNEL

A. Kased - Professor
W. Teshkoff - Professor
S. Abdallah - Graduate Research Assistant
C. AbouGhantous - Graduate Research Assistant
C. Cheng - Graduate Research Assistant
M. Malak - Graduate Research Assistant
J. Cupito - Mechanic

ADVANCED DEGREES AWARDED

M.S. Degree to M. Malak, Thesis Title:
"An Investigation Of The Trailing Edge Condition
In The Finite Element Solution Of Inviscid Flow
In Cascade And Single Airfoil".

Ph.D. Degree to S. Abdallah, Dissertation Title:
"An Inviscid Solution For The Secondary Flow In
Curved Passages".
PUBLICATIONS GENERATED UNDER THIS RESEARCH GRANT


RESEARCH OBJECTIVES

The main objective of this research work was to investigate the secondary flow phenomena analytically and experimentally. The experimental work was conducted to obtain measurements for the three velocity components using Laser Doppler Velocimeter (LDV) in a curved duct with inlet shear velocity profile. The purpose of the analytical work was to develop a formulation and a numerical procedure for the solution of internal three dimensional rotational flow fields.

ACCOMPLISHED WORK

Since all six technical papers, which were generated under AFOSR sponsorship are attached in this report, only the important contributions will be summarized here.

The first phase of the analytical work was aimed at developing a numerical computational procedure for inviscid incompressible rotational flow using a marching technique. The governing equations for the through flow velocity and vorticity components and for the streamlike functions in the cross-sectional planes were derived from the conservation of mass and momentum. The numerical results were obtained for the rotational flow field in a curved rectangular duct with constant cross sectional area and curvature, and compared with available experimental data. This work was published in the AIAA Journal, Vol. 19, No. 8, 1981, pp. 993-999 (Appendix 1).
The next step in the analytical work was to develop an elliptic numerical solution for internal inviscid rotational flows. The equation of conservation of mass for three dimensional flows was identically satisfied through the definition of three two-dimensional streamlike functions on sets of orthogonal surfaces. An iterative procedure was developed for the numerical solution of the governing equations for the through flow vorticity, total pressure and streamlike functions. The results of the numerical flow computations were compared with the experimental data and with the results of other analytical studies. This work was presented as ASME Paper No. 82-GT-242 at the 27th International Gas Turbine Conference in London, England, April 1982, and later published in the Journal of Engineering for Power, Vol. 105, 1983, pp. 530-535 (Appendix 2). The analysis was then generalized to compressible flows in curved ducts with variable cross-sectional area, using orthogonal curvilinear coordinates. The results of the numerical computations were compared with the experimental measurements in Stanitz duct. This work was presented as AIAA Paper Number 83-0259 at the AIAA 21st Aerospace Sciences Meeting at Reno, Nevada, January 10-13, 1983, (Appendix 3), and is accepted for publication in the AIAA Journal.

The final phase of the analytical study was to determine the suitability of the streamlike function vorticity formulation for obtaining elliptic solutions for three dimensional viscous flows. The results of the computations were presented for the three dimensional viscous flow in a square duct.
The computed results were found in agreement with the experimentally measured through flow velocity profiles. In addition the viscous elliptic influence was reflected in the computed axial and cross velocity components upstream of the duct inlet. This work was presented as AIAA Paper No. 84-1633, at the AIAA 17th Fluid Dynamics, Plasma Dynamics and Lasers Conference at Snowmass, Colorado June 25-27, 1984 (Appendix 4). It will also appear in Recent Advances in Numerical Methods in Fluids Volume III on Computational Methods in Viscous Flows, Pineridge Press, 1984, (Appendix 5).

The experimental work was conducted to obtain measurements of the three velocity components of the flow in a curved rectangular duct, using Laser Doppler Velocimeter. A nearly linear inlet shear velocity profile was produced using a grid of parallel wires with variable spacing, resulting in secondary velocities as high as 25% of the mean inlet velocity. This work was presented as AIAA Paper No. 84-1601 at the AIAA 17th Fluid Dynamics, Plasma Dynamics and Lasers Conference at Snowmass, Colorado, June 25-27, 1984 (Appendix 6).
SUMMARY OF SIGNIFICANT RESULTS

Two formulations were developed for modeling inviscid three-dimensional rotational flow fields in curved passages. The first, for a very efficient marching solution with hyperbolic equations governing the development of through flow velocity and vorticity profiles along the duct. In the second formulation, a new approach was developed to satisfy the conservation of mass in three dimensional flows by defining two dimensional streamlike functions on fixed orthogonal surfaces in the flow field. The first formulation leads to a faster numerical solution in which the influence of the pressure field on the three dimensional flow can be sensed upstream only through the curvilinear coordinate system. The second formulation on the other hand is more complex since it models the elliptic influence of the three dimensional pressure field. Computer time savings are realized through the two dimensionality of the equations for the streamlike functions and their Dirichlet boundary conditions. Agreement between the computed results and the experimental data was very good in both cases.

The streamlike function vorticity formulation was also tested for its ability to model viscous flow effects and their elliptic influence in a square duct. The viscous elliptic solution predicted the influence of the duct on the flow field upstream of the inlet. In addition, the computed results were in very good agreement with the experimentally measured through
The velocity profiles inside the duct. 

Recently, the measurements of the three velocity components, were conducted in a curved rectangular duct with an inlet shear layer profile. The experiment was designed such that the measurements can be used to validate both viscous and inviscid codes for internal three dimensional flow fields with strong downstream vortex and high secondary flow velocities due to inlet total pressure distortions. A grid of wires with variable spacing used to produce a nearly linear inlet velocity profile. Some secondary velocities of magnitudes up to 25% of the mean inlet velocity, were measured in the curved duct.

TECHNICAL APPLICATIONS

The experimental data for the three velocity components in curved ducts, obtained in this research work, provide detailed description of the secondary flow pattern. This information can be used in the development of appropriate flow models for the mathematical analysis of internal flow. In addition, the presented analytical work can be used in both inviscid and viscous three dimensional flow computations to model the various secondary flow generating mechanisms in turbomachines.
Appendix 1

Inviscid Solution for the Secondary Flow in Curved Ducts
AIAA 80-1116R

Turbulent Solution for the Secondary Flow in Curved Ducts

S. Abdallah and A. Hamed
Inviscid Solution for the Secondary Flow in Curved Ducts

S. Abdulrahim* and A. Hamad†
University of Cincinnati, Cincinnati, Ohio

This paper presents an analytical formulation and a numerical solution for three-dimensional rotational flow in curved ducts. The solution is based on calculating the through flow vorticity and velocity components from the momentum equations, using a matching technique. The secondary velocities are determined from the equation of continuity and vorticity equations. These equations are transformed through the use of the similar functions formulation into a single Poisson equation with Dirichlet boundary conditions. A numerical solution is obtained for the flow with large inlet velocity gradient in a 90-deg curved duct with constant radius of curvature. The results are presented and compared with the existing analytical solutions.

Introduction

A VORTEX is known to develop in cascades and bends through the turning of flowing fluid, thus introducing inertial velocity profiles. The transverse velocities, associated with the vortex, have a significant effect on the losses and flow turning angles. The secondary flow theory was developed by Hawthorne¹ to explore these phenomena. He developed expressions for the change in the streamwise vorticity component for steady inviscid, incompressible flow. Using his theory, Hawthorne¹ also derived expressions for the transverse secondary circulation, the trailing sheet, and the trailing filament circulations at turbomachinery cascade exit. Following Hawthorne, other investigators derived generalised vorticity equations to include effects that are not in Hawthorne’s expression. A. O. Smith² derived an expression for the streamwise vorticity in moving axes, while Laksmananraj and Horlock³ gave a general vorticity equation, valid for compressible, stratified, and viscous flow. The application of Hawthorne’s equations for streamwise vorticity calculations requires prior knowledge of the resulting flowfield. Consequently, some approximations usually have been involved in the application of Hawthorne’s equation for the streamwise vorticity calculation.

A first-order estimate of the streamwise vorticity generation in cascades and bends can be obtained for small perturbations using Squire and White’s formula.⁴ A related streamfunction expression is usually used to calculate the associated transverse velocities and turning angles. These secondary velocities are superimposed on the primary two-dimensional flowfield at the cascade exit. L. H. Smith⁵ defined the secondary vorticity as the difference between the actual streamwise vorticity and the primary vorticity that would exist if there were an infinite number of blades. Horlock⁶ demonstrated that the three-dimensional flows resulting from the superposition of the primary and secondary flows, according to this definition, is the same as that obtained from the traditional secondary flow theory.

The purpose of the present investigation is to study the secondary flow phenomena associated with the distributed secondary vorticity. The rotational flow in curved ducts is considered to investigate this phenomenon without the addition of transverse secondary vorticity. The boundary conditions were not compatible with this scheme in which a duplicate variable is used for the forward velocity, and the continuity equation is integrated twice. The authors rectified this situation by deriving two integral boundary conditions for the duplicate forward velocities from the two continuity equations after dropping the source term. Fagan⁷ on the other hand used Wu’s technique to⁸ on two families of stream surfaces. On these surfaces the inviscid flow governing equations are combined to obtain Poisson type equations. The stream surfaces in this case are skewed and warped because of the streamwise vorticity. This analysis was used successfully in cases of small disturbances, but Fagan reported difficulties in high rotational flow, because of the warpage of the stream surfaces which was more than 90 deg. Corkscrew coordinates that rotate at a specified rate were introduced in Ref. 11 to resolve this problem.

This paper presents a new analytical formulation and an efficient numerical technique for the solution of the rotational flow in curved ducts. Through cross differentiation, the pressure terms are cancelled from the
Incompressible flows in a curved duct are considered. The Navier-Stokes equations are solved numerically by a finite difference method. The continuity, momentum, and vorticity equations are solved simultaneously. The rigid body rotation to the curvilinear coordinates is eliminated from Eqs. 1-3 using cross differentiation. The resulting equations that are solved for $\xi$ and $\eta$ are

$$\frac{\partial^2 \xi}{\partial r^2} + \frac{\partial^2 \xi}{\partial \theta^2} + \frac{\partial^2 \xi}{\partial z^2} = \frac{\partial}{\partial r} \left[ \frac{1}{r} \frac{\partial \xi}{\partial r} \right]$$

where $\xi$ is the $\theta$ component of the vorticity defined as

$$\xi = \frac{\partial w}{\partial r} - \frac{\partial v}{\partial z}$$

The reader is referred to the Appendix for the derivation of Eqs. (6) and (7).

Initial and Boundary Conditions

Referring to Fig. 1, the following initial and boundary conditions are used

$$u(r,0,z) = u_0$$

$$\xi(r,0,z) = 0$$

$$w(R_r,0,z) = 0$$

$$w(R_r,0,z) = 0$$

Equations (6-7) with the boundary conditions Eqs. (8-11) form a closed system that is solved for the variables $u$, $v$, $w$, and $\xi$.

Method of Solution

The streamlike function formulation is used for the simultaneous solution of Eqs. (4) and (5) with the boundary conditions Eqs. (10) and (11). The method of solving Eqs. (4) and (5) for the velocity components $u$ and $w$ will be briefly outlined here. More details about this method can be found in Ref. 14. Equations (4) and (5) are first rewritten in the following form:

$$\frac{\partial}{\partial r} (ru) + \frac{\partial}{\partial z} (rw) = -\frac{\partial v}{\partial \theta}$$

and

$$\frac{\partial}{\partial z} (u) - \frac{\partial}{\partial r} (w) = \xi$$
The boundary conditions, Eqs. (16) and (11) are rewritten in terms of the streamlike function \( \chi \) as follows:

\[
\chi(r, z) = C \quad (17a)
\]

\[
\chi(r, z) = \int_0^r \frac{\partial \psi}{\partial r} \, dr + C \quad (17b)
\]

and

\[
\chi(r, \theta) = \chi(r, 0) = C \quad (18)
\]

Equation (16) is solved using the S.O.R. method, with the boundary conditions Eqs. (17) and (18). In the numerical solution the integrals in Eqs. (16), (16), and (17b) are evaluated using the trapezoidal rule.

Lax’s scheme is used to solve the hyperbolic Eqs. (6) and (7) with initial conditions Eqs. (8) and (9). A forward-difference quotient is used to advance the solution in \( \theta \) direction and the derivatives with respect to \( r \) and \( z \) are approximated using central finite-difference formulas. Two important integral relations must be satisfied by the solution to this system of equations (6), (7), and (16). The first integral relation represents the global condition for the conservation of mass:

\[
\iint_A \psi \, dr \, dz = Q \quad (19)
\]

where \( A \) is the duct cross-sectional area and \( Q \) is the volume flux rate. Due to truncation error, the numerical solution for \( \psi \) may not satisfy condition Eq. (19) exactly; consequently, a uniform correction is introduced in the source term representing the right-hand side of Eq. (12) to satisfy Eq. (19).
The second integral relation equates the circulation at each cross-sectional plane with the flux of the through flow vorticity $\varepsilon$. 

$$ \oint \varepsilon \, dr \, dz = \int V \times dS $$

(20)

where $V_{x}$ represents the velocity tangent to the contour $S$ containing the area $A$. The following boundary conditions for $\varepsilon$ determine a unique solution for Eq. (9) and satisfy the condition Eq. (20).

$$ \varepsilon = \int \frac{\delta^2 x}{r^2} + \int \frac{\delta^2 x}{r} \int_{n} \left(-\frac{\partial u}{\partial \theta}\right) \, dr \text{ at } z = 0, H $$

(21)
The solution to Eq. (7) on the duct solid boundaries is obtained using the two-dimensional form of Lax's scheme. A stable linear extrapolation procedure is used to calculate the boundary conditions for \( u \) along the duct corners.

**Results and Discussion**

The numerical solution of the governing equations is obtained in a 90-deg turning duct with rectangular cross section. The experimental data of Ref. 15 provides flow measurements at the 90-, 60-, and 30-deg turning angles of a rectangular duct with a 0.379-m (15 in.) mean radius and a 0.125 x 0.25 m (5 x 10 in.) cross section. The duct geometry and dimensions are shown in Fig. 1. Some of the experimental data of Ref. 15 is obtained for a flowfield with substantial inlet velocity variations. It can be very useful in the assessment of the present model. The same data was used for comparison with the numerical results in Refs. 10, 11, and 16. The experimentally measured inlet velocity profile of Ref. 15 is reproduced in Fig. 2. In this figure, the values labeling the velocity contours are in meters per second. It can be seen from this figure that the variations in the profile are greater in the low velocity regions compared to the nearly uniform velocity at the centerline. In addition, it may be observed that the variation in the \( r \) direction is not too significant. The inlet velocity profile of Fig. 2 was found too noisy to be used directly in the numerical solution. Therefore, the experimental data is simulated with a parabolic variation in the \( r \) direction to obtain the numerical results. Due to the symmetry of the inlet profile the computations are carried out only in the lower half of the duct. The results in Ref. 10 were also presented in the lower half, while Refs. 11 and 16 presented their results and comparison with the experimental data in the upper half of the duct. The governing equations were nondimensionalized before the numerical solution, so that all results, except the velocity contours, are presented in nondimensional form. The duct inner radius \( R_i \) and the maximum flow velocity at inlet \( V_{in} \) were used in the normalization. The numerical computations were carried out in double precision on the AMDahl 470. The results presented here were generated using an 11 x 11 x 45 grid.

The results are presented in the form of velocity and secondary vorticity contours, and vectors showing the magnitude and direction of the secondary velocities. The velocity contours at the 30-, 60- and 90-deg turning angles are shown in Figs. 3, 5, and 7, respectively. It can be seen from these figures that the rotation of the velocity contours, which were parallel and horizontal at inlet, is very significant in the
The computed velocity contours at the 30-, 60-, and 90-deg cross sections are shown with the experimental data of Ref. 15 and the inviscid flow analysis of Ref. 11 in Figs. 9-11 for comparison. It can be seen from these figures that the present analysis predicts the rotation of the velocity contours accurately at 30- and 60-deg turning angles, and in the high velocity region at the 90-deg turning angle. The agreement between the analysis and the experimental data is very significant in the high velocity regions, where the viscous effects are not dominant. In addition, the present analysis predicts the experimentally measured low velocity regions at the inner wall as can be seen at the inner corner in Fig. 9 and the centerline of the cross section in Fig. 10. The translation of this low velocity region from the corner at the 90-deg cross section to the centerline at the 60-deg cross section is not predicted by the analysis of Ref. 11.

The computer time used in the solution of the present study is significantly less than the computer time in Refs. 10, 11, and 16. Pagano, using the total of six stream surfaces with 260 nodal points per surface, reported a CPU time of 1500-2100 s on the IBM 370/155. Stuart and Hetherington reported a CPU time of 840 s on the IBM 360/65 using a 9 x 15 grid. Roccob used an 8 x 8 x 16 grid to obtain his solution, and reported a CPU time of 350 s on CDC 7600. In the present analysis, the CPU time was 20 s on AMDHAL 470 V/6-II using an 11 x 11 x 45 grid in the numerical solution.

Conclusion

The present analysis of internal rotational flows leads to a very efficient numerical scheme for predicting the secondary flow phenomena. The analysis is applied to the rotational flows in a 90-deg bend with rectangular cross section. From the comparison of the computed results with the experimental data, it can be concluded that the physics of the secondary flow problem are well represented in the analysis. The present formulation can be adapted with some modifications to variable area ducts and turbomachinery passages.

Appendix

Derivation of Eq. (6)

Differentiating Eq. (1), with respect to r and Eq. (3) with respect to θ and subtracting, one obtains

\[
\frac{\partial \phi}{\partial r} + \frac{\partial \psi}{\partial z} + \frac{\partial \omega}{\partial r} z - \frac{\partial \omega}{\partial z} r = \frac{i}{r} \left( \frac{\partial v}{\partial r} - \frac{\partial u}{\partial z} \right) - \frac{\partial \omega}{\partial z} \frac{1}{r} \left( \frac{\partial \theta}{\partial r} - \frac{\partial \phi}{\partial z} \right)
\]

\[-\frac{\partial}{\partial r} \left( \frac{\partial \omega}{\partial r} - \frac{\partial \psi}{\partial z} \right) + \frac{\partial}{\partial z} \left( \frac{\partial \omega}{\partial r} - \frac{\partial \psi}{\partial z} \right) = 0
\]

(A1)

the divergence of the vorticity vector is identically equal to zero. This can be expressed in terms of u, v, and w and ξ as follows:

\[
\frac{\partial}{\partial r} \left( \frac{i}{r} \left( \frac{\partial v}{\partial r} - \frac{\partial u}{\partial z} \right) - \frac{1}{r} \left( \frac{\partial \theta}{\partial r} - \frac{\partial \phi}{\partial z} \right) \right) + \frac{\partial}{\partial z} \left( \frac{i}{r} \left( \frac{\partial v}{\partial r} - \frac{\partial u}{\partial z} \right) - \frac{1}{r} \left( \frac{\partial \theta}{\partial r} - \frac{\partial \phi}{\partial z} \right) \right) = 0
\]

(A2)
SOLUTION FOR THE SECONDARY FLOW IN CURVED DUCTS

Substituting Eqs. (A8) and (A9) into Eq. (A7) and simplifying, one obtains

\[ u \frac{\partial u}{\partial r} + v \frac{\partial u}{\partial z} + w \frac{\partial u}{\partial z} = \left[ u \left( \frac{\partial u}{\partial r} + \frac{\partial v}{\partial z} \right) + \frac{\partial u}{\partial z} \right] \]

\[ \left[ u \left( \frac{\partial v}{\partial r} - \frac{\partial u}{\partial z} \right) - v \left( \frac{\partial u}{\partial r} + \frac{\partial v}{\partial z} \right) \right] \left( \frac{v}{\partial z} + \frac{w}{\partial r} \right) \]

\[ + \left[ \left( 1 - \frac{1}{r^2} \right) \right] \left( \frac{w}{r} - \frac{v}{\partial z} \right) \left( \frac{v}{\partial z} + \frac{w}{\partial r} \right) \]

\[ \times \frac{\partial}{\partial z} \left( \frac{v}{\partial z} + \frac{w}{\partial r} \right) \]

(A10)

Acknowledgment

This research was sponsored under AFOSR Contract 80-0242.

References


The Elliptic Solution of the Secondary Flow Problem

This paper presents the elliptic solution of the inviscid incompressible secondary flow in curved passages. The three-dimensional flow field is synthesized between 3 sets of orthogonal nonstream surfaces. The two-dimensional flow field on each set of surfaces is considered to be resulting from a source/sink distribution. The distribution and strength of these sources are dependent on the variation in the flow properties normal to the surfaces. The dependent variables in this formulation are the velocity components, the total pressure, and the main flow vorticity component. The governing equations in terms of these dependent variables are solved on each family of surfaces using the streamlike function formulation. A new mechanism is implemented to exchange information between the solutions on the three family surfaces, resulting in a unique solution. In addition, the boundary conditions for the resulting system of equations are carefully chosen to insure the existence and uniqueness of the solution. The numerical results obtained for the rotational inviscid flow in a curved duct are discussed and compared with the available experimental data.

Introduction

Three-dimensional flow calculations in turbomachines constitute a complex mathematical problem. The fact that the axial velocity components change greatly in passing through a turbomachine limits the two-dimensional approximations to a few special cases. A major contribution in this field is Wu's applied theory [1], for the inviscid flow calculations. In this theory the flow field is determined from two-dimensional solutions, which are obtained on two intersecting families of constant streamlines with variable thickness. The governing equations are satisfied on the mean surfaces of these streamlines, which are referred to as the S₁ (blade-to-blade) and the S₂ (meridional) surfaces. The correct solution of one family of surfaces requires some data from the other and, consequently, an iterative process between the solutions on these two families of surfaces is involved. Many investigators have applied Wu's theory to obtain solutions on the S₁ or S₂ surfaces. Two techniques have been used in these studies; namely, the matrix method and the streamlines curvature method.

March [2], Katsanis [3, 4], Smith [5], Boozman and Ellingwood [6], and Blaser [7], used the matrix method to obtain solutions on the S₁ and S₂ surfaces. Katsanis developed computer programs for a meridional solution [3] and also for solution on a blade-to-blade surface of revolution [6] with a tube thickness proportional to the blade height. In [3] and [6], a representative mean averaged S₂ stream surface is used, and the S₁ surfaces are generated by rotating the streamlines in the S₂ surfaces about the axis of revolution.

The streamlines curvature method has been used by Wilkinson [8] to obtain blade-to-blade solutions. The same method has also been used by Novak and Hearshy [9] and by Katsanis [10, 11] to obtain meridional and blade-to-blade solutions. The assumptions and approximations used for the stream surface shape and stream filament thickness distribution in the S₁ and S₂ solution using this method are mainly similar to those discussed previously in connection with the matrix method.

Several problems are encountered in computing a synthesized three-dimensional turbomachine flow field from the solutions on the S₁ and S₂ surfaces. These problems are related to the exchanged information between the two solutions, concerning the stream surface shape and the stream filament thickness. Novak and Hearshy [9] pointed out that the S₂ filament thickness, as calculated from the blade-to-blade solution, is not constant upstream and downstream of the blade row. This is in contradiction with the requirement that the flow must be treated as axisymmetric in these regions. They also discussed the effect of the lean of the S₂ mean stream surface extending upstream and downstream of the blades, where the net angular momentum changes must be zero. Stuart and Hetherington [12] tried to use a technique similar to Wu's in their solution for rotational flow in a 90-degree bend, by synthesizing the three-dimensional flow field through the iterations between two-dimensional solutions. They were unable to reach convergence in the iterative numerical solution and had to abandon this approach. They speculated that the information conveyed between the two solutions was not sufficient to produce convergence.

The assumption that the S₁ stream surface is a body of revolution was common in all applications of Wu's theory in turbomachines [2-12]. This assumption is valid if the flow is irrotational. In general, the S₁ surface is twisted and skewed due to the presence of the secondary velocities. These
Appendix 2

The Elliptic Solution of the Secondary Flow Problem
transverse velocities exist due to the streamwise vorticity which is generated by the turning of a nonuniform flow with a vorticity component in the curvature plane [13, 14]. The nonuniformities of inlet velocities in turbomachines are associated with the hub and the tip boundary layers. Stream surface warpage pose additional mathematical difficulties in the solution of the rotational flow. Fagan [15] could not obtain a solution for highly rotational flow in curved ducts using Wm's approach. He resolved the problems encountered when the stream surfaces warpage approaches 90 deg through using a corkscREW coordinate system that operates at a specified angular rate.

In reference [16], Abdallah and Hamed developed an effective method for the solution of three-dimensional rotational flow in curved ducts, in which the throughput flow velocity and vorticity components were computed using a marching technique in the main flow direction. This led to a very efficient numerical solution whose results compared favorably with the experimental data for duct flows. However, because of the marching technique used in computing the through flow velocity, the influence of the downstream conditions on the flow characteristics is not modeled. This effect, although not significant in duct flow, may be quite important in turbomachinery applications. Barber and Langston [17] contrasted the blade row and duct flow problems and discussed the importance of the elliptic solution to the flow in blade rows.

This investigation represents an elliptic solution for the internal nonviscous incompressible rotational flow in curved passages. The streamlike function method, which was developed by the authors [18] for the efficient numerical solution of the continuity and rotationality equations is implemented in the present problem formulation. The dependent variables in this formulation are the three streamlike functions, the total pressure, and the throughflow vorticity component. The equations of motion are satisfied on three arbitrary sets of orthogonal surfaces. On these surfaces, two-dimensional boundary equations are derived, for the streamlike functions, with source terms representing the flow three-dimensionality. The source terms are dependent upon the variation of the flow properties in the direction normal to these surfaces. Because of the dependency of the solution on each set of surfaces on the solutions for the remaining two sets of surfaces, an iterative process is involved in the solution. The three flow velocity components are determined by the source terms and the three streamlike function derivatives. The total pressure and throughflow vorticity are computed from Bernoulli and Helmholtz equations, respectively.

The initial and boundary conditions for a closed system of equations are carefully chosen to ensure the existence and uniqueness of the solution. The no-flux condition at the solid boundaries leads to Dirichlet boundary conditions for the streamlike functions. Downstream, the derivatives of the flow properties in the throughflow direction is set to zero. A Poisson type equation with Neumann boundary conditions is derived and solved for the static pressure at the inlet plane, which is then used together with the inlet velocity profile to determine the inlet total pressure profile.

Numerical results are obtained for the case of rotational flow in a curved duct with rectangular cross sections. The results are discussed and compared with the experimental data.

Mathematical Formulation
For simplicity and to be able to compare with existing experimental results in ducts [19], the cylindrical polar coordinates are used in the following presentation. The basic equations for nonviscous incompressible flow are expressed in terms of the three velocity components, the total pressure and the throughflow vorticity components as follows:

Conservation of mass
\[
\frac{1}{r} \frac{\partial}{\partial r} (r u) + \frac{\partial v}{\partial \theta} + \frac{\partial w}{\partial z} = 0
\]  

Conservation of momentum in \( r \)-direction
\[
u \left[ \frac{\partial u}{\partial r} + \frac{u}{r} - \frac{1}{r} \frac{\partial v}{\partial \theta} \right] - \omega \xi = \frac{\partial P}{\partial r}
\]

Conservation of momentum in \( z \)-direction
\[
u \left[ \frac{\partial w}{\partial z} + \frac{w}{r} - \frac{\partial u}{\partial \theta} \right] = \frac{\partial P}{\partial z}
\]

where \( P \) is the total pressure divided by the density, and \((u,v,w)\) are the three velocity components in \((r,\theta,z)\)-directions. The throughflow vorticity component, \( \xi \), can be written in terms of the cross velocities, \( u \) and \( w \), as follows
\[
\xi = \frac{\partial u}{\partial z} - \frac{\partial w}{\partial r}
\]

Bernoulli's Equation.
Bernoulli's equation is used instead of the momentum equation in the \( \theta \)-direction.
\[
u \frac{\partial u}{\partial r} + \frac{v}{r} \frac{\partial P}{\partial \theta} + \frac{w}{r} \frac{\partial P}{\partial z} = 0
\]

Helmholtz Equation.
\[
u \frac{\partial u}{\partial r} + \frac{\xi}{r} \frac{\partial v}{\partial \theta} + \frac{\xi}{\partial z} + \frac{1}{r} (u \xi - v \eta)
\]

Nomenclature

\( A \) = duct cross-sectional area
\( C \) = contour enclosing \( A \)
\( dC \) = incremental distance along \( C \)
\( n \) = outward normal to the contour \( C \)
\( P \) = total pressure divided by the density
\( P_s \) = inlet static pressure divided by the density
\( (r, \theta, z) \) = cylindrical polar coordinates
\( r_1, z_1 \) = arbitrary integration reference points on the \( r \) and \( z \)-coordinates, respectively
\( S_1, S_2 \) = blade-to-blade and meridional stream surfaces, respectively

Subscripts

\( h \) = horizontal surfaces
\( i \) = inlet conditions
\( v \) = vertical surfaces
\( c \) = cross-sectional surfaces

Journal of Engineering for Power
20 JULY 1983, Vol. 105 / 331
The governing equations (1-4) are satisfied on three families of orthogonal surfaces, \( \alpha, \beta \) and \( \gamma \), as shown in Fig. 1. The three set of orthogonal equations on these surfaces provide the three velocity components, \( u, v, \) and \( w \), whereas the hyperbolic equations (5) and (6) are solved for \( P \) and \( \xi \), respectively.

The streamlines function, formulated by the authors [18] is utilized in the present problem for the purpose of obtaining an economical numerical solution to equations (1-4). Three separate streamline functions, \( \chi_0, \chi_1, \) and \( \chi_2 \), are defined for the three set of orthogonal surfaces shown in Fig. 1, to identically satisfy the principal of conservation of mass, given by equation (4). The streamlines function, \( \chi_0 \), is defined on the horizontal surfaces as follows

\[
\frac{1}{r} \frac{\partial \chi_0}{\partial \theta} = u_0 + \frac{1}{r} \int_{\gamma_1}^r \frac{\partial w_0}{\partial \zeta} \, d\zeta
\]

(7a)

and

\[
\frac{\partial \chi_0}{\partial \theta} = -v_0
\]

(7b)

where \( \gamma_1 \) is the radial coordinate of an arbitrary integration surface and the subsripts \( h \) and \( v \) refer to the solutions on the horizontal and vertical surfaces, respectively.

The streamlines function, \( \chi_1 \), is defined on the vertical surfaces as follows

\[
\frac{1}{r} \frac{\partial \chi_1}{\partial \theta} = w_1 + \frac{1}{r} \int_{\gamma_1}^r \frac{\partial u_1}{\partial \zeta} \, d\zeta
\]

(8a)

and

\[
\frac{\partial \chi_1}{\partial \theta} = v_1
\]

(8b)

where \( \gamma_1 \) is the axial coordinate of an arbitrary integration surface.

The streamlines function, \( \chi_2 \), is defined on the cross planes as follows

\[
\frac{\partial \chi_2}{\partial \theta} = -u_2 + \frac{1}{r} \int_{\gamma_1}^r \frac{\partial v_2}{\partial \zeta} \, d\zeta
\]

(9a)

and

\[
\frac{\partial \chi_2}{\partial \theta} = v_2
\]

(9b)

where the subscript \( c \) refers to the solution on the cross-sectional surfaces.

The governing equations for these stream functions are obtained by substituting equations (7a,b) into equation (3), and equations (8a,b) into equation (4), leading to the following equations

\[
\frac{\partial^2 \chi_0}{\partial \theta^2} + \frac{1}{r} \frac{\partial \chi_0}{\partial r} + \frac{1}{r^2} \frac{\partial^2 \chi_0}{\partial \zeta^2} = \frac{1}{r^2} \left[ \frac{\partial^2 \varphi}{\partial \theta^2} + \frac{\partial \varphi}{\partial \theta} \frac{\partial \varphi}{\partial \zeta} \right]
\]

(10)

\[
\frac{\partial^2 \chi_1}{\partial \theta^2} + \frac{1}{r} \frac{\partial \chi_1}{\partial r} + \frac{1}{r^2} \frac{\partial^2 \chi_1}{\partial \zeta^2} = \frac{1}{r^2} \left[ \frac{\partial^2 \psi}{\partial \theta^2} + \frac{\partial \psi}{\partial \theta} \frac{\partial \psi}{\partial \zeta} \right]
\]

(11)

The elliptic equations (10), (11), and (12) are solved on the horizontal, vertical, and cross-sectional surfaces for \( \chi_0, \chi_1, \) and \( \chi_2 \), respectively. The solution on one set of surfaces is influenced by the solutions on the other two sets of surfaces through the source terms. Consequently, an iterative process between the three families of surfaces is involved. The three-dimensional solution is obtained by adding the computed velocity components in each of the solutions as follows

\[
u = u_0 + u_c
\]

(13a)

\[
v = u_1 + v_c
\]

(13b)

\[
w = w_2 + w_c
\]

(13c)

It can be easily shown that the velocity components are determined by equation (13) represent a unique solution.

Boundary Conditions. The inlet conditions for \( \xi \) and \( P \) are given by

\[
\xi = 0
\]

(14)

\[
P = P_{\xi} + \frac{1}{2} \varphi^2
\]

(15)

where \( P_{\xi} \) is the static pressure at the duct inlet cross section is computed from the numerical solution of the following equation

\[
\frac{\partial^3 P_{\xi}}{\partial r^3} + \frac{1}{r} \frac{\partial P_{\xi}}{\partial r} + \frac{\partial^2 P_{\xi}}{\partial \zeta^2} = \sigma
\]

(16)

where

\[
\sigma = \frac{\partial}{\partial r} \left[ \frac{-\frac{1}{r} \frac{\partial \psi}{\partial \theta} + \frac{\partial \psi}{\partial \theta} \frac{\partial \psi}{\partial \zeta}}{r} \right] + \frac{\partial}{\partial \zeta} \left[ \frac{\partial \varphi}{\partial \theta} \frac{\partial \psi}{\partial \zeta} \right]
\]

(17a)

With the following boundary conditions

\[
\frac{\partial P_{\xi}}{\partial r} = 0, \quad \text{at} \quad r = R_1, R_2
\]

(17a)
The incompressible and unsteady solution of the boundary conditions (17a) and (17b), requires the introduction of the following condition

$$ \int_A \sigma \nu r \, dA = \int_C \frac{\partial p}{\partial n} \, dC $$

(18)

where $A$ is the duct inlet cross-sectional area, $\sigma$ is the induced stress on the surface, $C$, excluding the area, $A$, and $dC$ is the circumferential element along $C$. The derivation of Poisson's equation is based on identifying the continuity equation at the duct vertex, and its solution is unique within an arbitrary context.

In addition, Bernoulli and Helmholtz equations take the following forms at the duct boundaries

$$ \frac{\partial p}{\partial r} + \frac{\nu}{r} \frac{\partial p}{\partial \theta} = 0 \quad \text{at} \quad r = R_i, R_o $$

(19a)

$$ \frac{\partial p}{\partial r} + \frac{\nu}{r} \frac{\partial p}{\partial \theta} = 0 \quad \text{at} \quad \theta = 0, H $$

(19b)

$$ \frac{\partial x_0}{\partial r} + \frac{\partial x_0}{\partial \theta} = \frac{1}{r} \frac{\partial x_0}{\partial \theta} + \frac{1}{r} \frac{\partial x_0}{\partial \theta} $$

(20a)

$$ \frac{\partial x_0}{\partial r} + \frac{\partial x_0}{\partial \theta} = \frac{1}{r} \frac{\partial x_0}{\partial \theta} + \frac{1}{r} \frac{\partial x_0}{\partial \theta} $$

(20b)

The corner values for $F$ and $\xi$ are calculated using a stable extrapolation procedure, similar to the method used in [16].

The boundary conditions for the streamlines functions. The following inlet and exit flow conditions for the flow velocities as well as the no-flux condition at the duct boundaries, are used to determine the boundary conditions for the three streamlines functions

At inlet

$$ x_0 = 0, \quad \eta = 0, \quad \varphi = 0 $$

(21a)

$$ v = 0 $$

(21b)

$$ w = 0 $$

(21c)

At $r = R_i, R_o$

$$ x_0 = 0 $$

(22a)

$$ \eta = 0 $$

(22b)

$$ \varphi = 0 $$

(22c)

At $\theta = 0, H$

$$ x_0 = 0 $$

(23a)

$$ \eta = 0 $$

(23b)

$$ \varphi = 0 $$

(23c)

At exit

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(24a)

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(24b)

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(24c)

In addition, the following condition is used to uniquely determine $x_0$, $x_0$, and $x_0$:

$$ \frac{\partial x_0}{\partial r} + \frac{\partial x_0}{\partial \theta} + \frac{\partial x_0}{\partial \theta} = 0 $$

(25)

Equations (21-26) are used together with equations (7-9) and equations (13), to obtain the following boundary conditions for $x_0$, $x_0$, and $x_0$. The integration reference surfaces for the integrals in equations (7-9) are chosen here to be represented by $r_1 = R_i$ and $\epsilon_i = 0$, respectively.

At inlet

$$ x_0 = 0 $$

(26a)

$$ x_0 = - \int_{R_i} v_1 \, dr $$

(26b)

$$ \frac{1}{r} \frac{\partial x_2}{\partial \theta} = \frac{1}{r} \int_{R_i} v_1 \, dr $$

(26c)

At $r = R_i$

$$ x_0 = 0 $$

(27a)

$$ x_0 = 0 $$

(27b)

$$ \frac{\partial x_0}{\partial \theta} + \frac{x_0}{r} = 0 $$

(27c)

At $r = R_o$

$$ x_0 = - \int_{R_o} v_1 \, dr $$

(28a)

$$ x_0 = 0 $$

(28b)

$$ \frac{1}{r} \int_{R_o} v_1 \, dr $$

(28c)

At $\theta = 0, H$

$$ x_0 = 0 $$

(29a)

$$ x_0 = 0 $$

(29b)

$$ \frac{\partial x_0}{\partial \theta} = 0 $$

(29c)

At exit

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(30a)

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(30b)

$$ \frac{1}{r} \frac{\partial x_0}{\partial \theta} = 0 $$

(30c)

The governing equations (5), (6), (10), (11) and (12) with the conditions given by equations (14), (15), and (26-30) form a closed system which is solved for the variables $x_0$, $x_0$, $x_0$, $\xi$, and $P$.

Results and Discussion

The results of the computations of the secondary flow in curved ducts caused by total pressure inlet distortion are presented. The iterative solution procedure is based on the use of successive over relaxation method for the solution of the three streamline function equations, and Lax's [20] matching scheme for the total pressure and through flow vorticity equations. The results are presented and compared with the experimental measurements of Joy [19] for the flow in a curved duct with constant curvature and rectangular cross section. Joy [19] obtained flow measurements in a curved rectangular duct of $0.125 \times 0.25\text{-m}$ (5 $\times$ 10-in.) cross section, $0.375\text{-m}$ (15-in.) mean radius, and 90 deg turning angle. A large velocity gradient was produced in the experiment using screens placed before the curved portion of the duct, which resulted in a nearly symmetrical velocity profile at inlet to the bend. The velocity contours in the lower half of the duct are shown in Fig. 2. The computations were carried out in the lower half of the duct to take advantage of the symmetry. The
results of the computations are compared with Joy's flow measurements at 30, 60, and 90 deg turning angles. The magnitudes of the flow velocities in Figs. 2-5 are normalized with respect to the maximum inlet velocity. It can be seen from these figures that the computed results are in good agreement with the experimental measurements at the 30 and 90 deg turning angles. There is a lack of agreement however with the experimental results at the 60 deg turning angle inner wall near the duct centerline. Other investigators [12, 15] associated with the same experimental results speculated closure boundary layer separation there and attributed the lack of agreement to it. The analysis otherwise predicts the measured contour rotation caused by the secondary flow development. The computed static pressure contours at 0, 30, 60 and 90 deg turning angles are presented in Figs. 6. These contours show that the static pressure is not constant over the cross sections even at zero duct turning angle. The static pressure gradients in the radial direction is necessary to balance the centrifugal forces. Joy [19] did not obtain static pressure measurements that could be compared with the present results; however, static pressure contours similar to those of Fig. 6a, were measured by Brun [20] in a curved duct at zero turning angles. The effect of the secondary flow development on the pressure distribution is demonstrated in these figures by the variation in the shape of the contours with the duct turning angle. The computations were carried out using a uniform (0 x 9 x 31) grid in the r, z-, and 0-directions, respectively. The convergence of the iterative procedure was very fast, as shown in Fig. 7, which presents the average of the absolute values of the error in the calculation of x0, the through velocity component from the x0 solution. The exchange of information from the solutions on the vertical and cross-sectional surfaces to this surface does not start until the second iteration, which leads to maximum error that consequently decreases very rapidly. The solution was obtained using double precision with 50 outer iterations and 120 iterations in the solution of the differential equations for x0, z1, and z2 on all 99 surfaces and required 4.5 min on AMB-64. The authors have not attempted to optimize the CPU time through changing the number of iterations in the solution of Poisson's equations with the outer iteration cycles, or to reduce the CPU time through the use of noniterative methods [21, 22] in the code. At present, direct Poisson solvers codes are being studied for incorporation into the numerical solution. This is expected to lead to considerable CPU time savings when it is combined with the streamlike function formulation and its corresponding Dirichlet boundary conditions.

Conclusions

It can be concluded that the present analysis can predict the inviscid secondary flow development caused by inlet total pressure distortion and the results of the computations compare with the experimental measurements. Through the elliptic solution, the influence of the downstream conditions
on the flow is included. The solution is very efficient due to the use of the streamline function in the formulation. In addition, convergence is very fast because of the interaction mechanism between the three solutions on the three sets of orthogonal surfaces.

Acknowledgment

This research was sponsored by the U.S. Air Force Office of Scientific Research under contract No. 80-0242.

References

Appendix 3

Three Dimensional Rotational Compressible
Flow Solution in Variable Area Channels
AIAA 21st Aerospace Sciences Meeting

January 10-13, 1983/Reno, Nevada
Three-Dimensional Rotational Compressible Flow Solution in Variable Area Channels

A. Hamed and C. Liu
Department of Aerospace Engineering and Applied Mechanics
University of Cincinnati
Cincinnati, Ohio 45221

Abstract

This paper presents the analytical formulation and numerical solution of compressible inviscid rotational flow in a variable area duct. The three-dimensional flow field is synthesized from the streamline function solution on three sets of orthogonal surfaces in the passage. This leads to a very economical elliptic solution that is adaptable to turbine engineering applications since it simulates the downstream conditions and channel area variation. The computed results for compressible rotational flow with shear inlet velocity in variable area curved ducts are presented to determine the influence of the area variation and the effects of compressibility on the three-dimensional flow field.

Nomenclature

- $h$: total enthalpy
- $h_1, h_2, h_3$: metric coefficients
- $p$: total pressure
- $p$: static pressure
- $S$: entropy
- $T$: total temperature
- $V_1, V_2, V_3$: velocity components
- $x_1, x_2, x_3$: orthogonal curvilinear coordinates
- $\rho$: density
- $\alpha$: source term
- $\chi$: streamlike function
- $\Omega_1, \Omega_2, \Omega_3$: vorticity components

Subscripts

- $c$: cross sectional
- $h$: horizontal
- $i$: inlet
- $\max$: maximum
- $r$: reference point
- $v$: vertical

Introduction

Recent developments of computational methods for three-dimensional flows in turbomachine blade passages and curved ducts include both viscous and inviscid flow models. The parabolic methods were first developed for the solution of internal viscous flows. These methods are based on the assumption of the small influence of the downstream pressure field on the upstream flow conditions, which leads to parabolic governing equations. The use of marching techniques leads to a very efficient solution but it also limits its application to flow in ducts with mild curvature. Later partially parabolic methods were developed in which the diffusion of mass, momentum and energy in the streamwise direction were still neglected but the elliptic influence was transmitted upstream through the pressure field.

Numerical methods have also been developed for the solution of inviscid flows. The approaches used in the numerical solution are considerably different depending on the flow model and the problem statement. Earlier quasi-three-dimensional methods consisted of an interactive procedure between two-dimensional flow solutions on blade to blade and hub to tip stream surfaces. These efficient two-dimensional solvers allowed arbitrary superimposed loss models but for most of the solutions, the blade to blade stream surface was taken as a surface of revolution. The attempts to extend these methods to rotational flow were not successful because the stream surfaces become twisted and warped. It was found that the iterative procedure between the two solutions does not converge in this case through the exchanged information during iterations in the form of stream surface shape and stream filament thickness. Time-dependent techniques have been developed for the solution of three-dimensional rotational flows and were used in flow field calculations. Aside from the time-dependent technique, two other methods were developed for the solution of internal
Incompressible rotational flows. Lambor and Mazenc developed a very simple model for the analysis of elliptic flows by splitting the velocity vector into irrotational and rotational parts. They used a simple function to describe the rotational part, for this special class of problems in which the vortical and entropy gradients are orthogonal to the vorticity vector.

Abdallah and Yenney presented a method for the elliptic solution of three dimensional rotational incompressible inviscid flow in which the velocity vector is synthesized from two dimensional solutions of three streamlike functions on three sets of orthogonal surfaces. Aside from incompressibility, no additional assumptions were imposed that might limit the analysis to a special class of problems. The present work represents an extension of that analysis to compressible flow fields with generalized channel geometries using orthogonal curvilinear body fitted coordinates. The problem formulation leads to a very efficient solution method. The governing equations for the streamlike functions consist of Poisson's equations with Dirichlet boundary conditions while the governing equations for tangential pressure, total enthalpy and streamwise vorticity are hyperbolic. The convergence of this three-dimensional solution is very fast and does not suffer from the problems encountered with the traditional quasi-three-dimensional methods in the presence of streamwise vorticity.

Analysis

The governing equations used in the numerical solution of the compressible three dimensional inviscid rotational internal flow are derived from the basic equations of conservation of mass, momentum and energy in orthogonal curvilinear body fitted coordinates. In the analysis, hyperbolic equations are derived for the through flow vorticity, the total enthalpy, while elliptic equations are derived for three sets of streamlike functions which are defined on fixed orthogonal surfaces. The details of the analysis for incompressible flow can be found in reference 15. In the following derivations, the same approach is applied to compressible flow using orthogonal curvilinear coordinates.

The Streamlike Function Formulation

The equation of conservation of mass for compressible flow in curvilinear coordinates is given by

\[
\frac{\partial}{\partial x_1} (h_2 x_1 p V_1) + \frac{\partial}{\partial x_2} (h_2 x_2 p V_2) + \frac{\partial}{\partial x_3} (h_2 x_3 p V_3) = 0
\]  

(1)

where \( V_1, V_2 \) and \( V_3 \) are the velocity components in the directions of the \( x_1, x_2, x_3 \) coordinates, respectively, \( p \) is the flow density and \( h_1, h_2, h_3 \) are the metric coefficients of the orthogonal curvilinear coordinates.

The equation of conservation of mass is identically satisfied through the definition of three streamlike functions \( x_{1,} x_{2,} \) and \( x_{3,} \). The streamlike function \( x_{1,} \) is defined such that its derivatives on the surface \( x_2 = \text{constant} \) are related to the three velocity components \( V_{1,} V_{2,} \) and \( V_{3,} \) as follows:

\[
\frac{2}{h_2 h_3} (h_2 x_{1,}) = h_1 h_3 p V_{1,} x_2 + \int \frac{2}{h_2 h_3} (h_1 h_3 p V_{3,}) dx_2
\]  

(2a)

and

\[
\frac{2}{h_2 h_3} (h_2 x_{1,}) = -h_2 h_3 p V_{1,} x_1
\]  

(2b)

The following definition of the second streamlike function \( x_{2,} \) is given in terms of its derivatives on the surface \( x_2 = \text{constant} \):

\[
\frac{2}{h_2 h_3} (h_2 x_{2,}) = -h_1 h_2 h_3 p V_{2,} x_3 + \int \frac{2}{h_2 h_3} (h_1 h_2 h_3 p V_{3,}) dx_3
\]  

(3a)

and

\[
\frac{2}{h_2 h_3} (h_2 x_{2,}) = h_2 h_3 p V_{2,} x_1
\]  

(3b)

The third streamlike function \( x_{3,} \) is similarly defined as follows:

\[
\frac{2}{h_2 h_3} (h_2 x_{3,}) = -h_1 h_2 h_3 p V_{3,} x_2 + \int \frac{2}{h_2 h_3} (h_1 h_2 h_3 p V_{2,}) dx_2
\]  

(4a)

and

\[
\frac{2}{h_2 h_3} (h_2 x_{3,}) = h_1 h_2 h_3 p V_{3,} x_1
\]  

(4b)

The integrals on the right hand side of equations (2) through (4) represent source terms which are dependent upon the flow crossing the three sets of fixed orthogonal surfaces. The superposition of the velocity components in the above streamlike function definitions gives the three gas velocity components \( V_{1,} V_{2,} \) and \( V_{3,} \), which identically satisfy the conservation of mass equation (1).
In the above equations $\Omega_1$, $\Omega_2$ and $\Omega_3$ are three vorticity components which can be expressed in terms of the velocity derivatives from the definition of the vorticity vector (equation 8).

The elliptic governing equations for the streamlike functions $x_1^*$, $x_2^*$ and $x_3^*$ can be obtained from the substitution of equations (2) through (5) into the $x_2$ and $x_3$ components of equation (9) and the $x_1$ component of equation (8) respectively.

\[
\begin{align*}
\frac{\partial}{\partial x_1} (\frac{h_3}{h_1 h_2} \frac{\partial x_2^*}{\partial x_1}) + \frac{\partial}{\partial x_2} (\frac{h_1}{h_2} \frac{\partial x_2^*}{\partial x_2}) + \frac{\partial}{\partial x_3} (\frac{h_1}{h_2} \frac{\partial x_2^*}{\partial x_3}) &= h_1 h_2 \sigma_h \\
\frac{\partial}{\partial x_1} (\frac{h_3}{h_1 h_2} \frac{\partial x_3^*}{\partial x_1}) + \frac{\partial}{\partial x_2} (\frac{h_1}{h_2} \frac{\partial x_3^*}{\partial x_2}) + \frac{\partial}{\partial x_3} (\frac{h_1}{h_2} \frac{\partial x_3^*}{\partial x_3}) &= h_1 h_3 \sigma_v \\
\frac{\partial}{\partial x_1} (\frac{h_3}{h_1 h_2} \frac{\partial x_1^*}{\partial x_1}) + \frac{\partial}{\partial x_2} (\frac{h_1}{h_2} \frac{\partial x_1^*}{\partial x_2}) + \frac{\partial}{\partial x_3} (\frac{h_1}{h_2} \frac{\partial x_3^*}{\partial x_3}) &= \sigma_c
\end{align*}
\]

where

\[
\sigma_h = -\rho \left[ \frac{V_1^2}{2h_1} \frac{\partial h_1}{\partial x_1} + \frac{V_2^2}{h_1 h_3} \frac{\partial h_2}{\partial x_2} \right] - \rho \frac{V_3 \Omega_1}{h_1 h_3} + \frac{V_2^2}{h_1 h_3} \frac{\partial h_2}{\partial x_2}
\]

\[
\sigma_v = -\rho \left[ \frac{V_1^2}{2h_1} \frac{\partial h_1}{\partial x_1} + \frac{V_2^2}{h_1 h_3} \frac{\partial h_2}{\partial x_2} - \frac{V_3 \Omega_1}{h_1 h_3} \right] + \frac{V_1}{h_1 h_3} \int \frac{h_3 x_2}{x_3} \left( \frac{\partial h_2 \Omega_2}{\partial x_2} \right) dx_2
\]

and

\[
\sigma_c = \rho \left[ \frac{V_1}{2h_1} \frac{\partial (h_1 \Omega_1)}{\partial x_1} + \frac{V_2}{h_1 h_3} \frac{\partial h_2}{\partial x_2} + \frac{V_3 \Omega_1}{h_1 h_3} \right] - \frac{V_1}{h_1 h_3} \int \frac{h_3 x_2}{x_3} \left( \frac{\partial h_2}{\partial x_2} \right) dx_2
\]

It is clear that the above equation also represents the $x_1$ component of Helmholtz equation which can be written in the following form for inviscid compressible flow.

\[
(\theta \cdot \mathbf{A}) = \rho \left( \frac{\partial}{\partial t} + \mathbf{v} \cdot \nabla \right) \mathbf{v} - \frac{1}{2} \mathbf{v} (\nabla (\rho / \rho))
\]
\[ v_0 = \frac{\partial h}{\partial x} = \frac{1}{2} \left( \frac{\partial h^2}{\partial x^2} + \frac{\partial h_1}{\partial x^2} \right) \]

\[ + \frac{1}{2} \left( \frac{\partial h^3}{\partial x^2} + \frac{\partial h_1}{\partial x^2} \right) \]

\[ + \frac{1}{2} \left( \frac{\partial h^2}{\partial x^2} + \frac{\partial h_1}{\partial x^2} \right) \]

\[ + h_1 \left( \frac{\partial h}{\partial x} + \frac{\partial h_1}{\partial x} \right) \]

\[ + h_2 \left( \frac{\partial h}{\partial x} + \frac{\partial h_2}{\partial x} \right) \]

\[ + h_3 \left( \frac{\partial h}{\partial x} + \frac{\partial h_3}{\partial x} \right) \]

**Solution Procedure**

The analytical formulation results in two-dimensional Poisson's equations for the streamlike functions. These equations are obtained from satisfying the equations of motion on three sets of orthogonal surfaces represented by constant values of the parameter \( x_1, x_2, x_3 \) as shown in Fig. 1. The source term in the resulting equations are dependent upon the variation of the flow properties and on the flow in the direction normal to these surfaces. An iterative procedure is used in the numerical solution because of the dependency of the source term in each set of streamlike function equations on the solutions obtained for the remaining two sets.

The iterative procedure for the numerical computations consists of the use of a marching technique \(^{16}\) in the solution of equations (7), (10) and (11) along the through flow direction \( x_3 \), and a successive over relaxation method for the solution of the two-dimensional elliptic equations (13), (14) and (15) for the streamlike functions. The flow density, which is allowed to lag one iteration of the numerical computations is determined from the local total pressure, total stagnation enthalpy, and from the flow velocity:

\[ \rho = \frac{P}{RT} \left( 1 - \frac{\gamma^2}{2} \right) \]

The boundary conditions for the streamlike functions are carefully chosen to ensure the uniqueness of the solution, \(^{18}\) Dirichlet boundary conditions are determined for the streamlike functions from the requirement of zero flux at the duct boundaries. \(^{17}\) The \( x_1 \) derivatives of the flow velocities are set equal to zero at the duct exit, while the Dirichlet boundary conditions for the streamlike functions at \( x_1 = 0 \), are expressed in terms of the entering flow properties. The total pressure, total temperature and through flow vorticity profiles are required at the duct inlet to start the marching solution. More details about the numerical procedure can be found in reference 16.

**Results and Discussion**

The results of the numerical computations are presented in an accelerating rectangular elbow and compared with existing experimental measurements. \(^{19}\) The duct was designed by Stanitz, using inviscid incompressible two-dimensional analysis to avoid boundary layer separation. This experimental data was also used for comparison with results of the numerical analysis by other investigators. \(^{15,21}\)

The experimental measurements were obtained for different inlet total pressure profiles to investigate the effects of secondary flow. The inlet total pressure profiles were generated using perforated plates of different heights as spacers. Figure 2 shows the duct geometry and the computational grid in the \( x_1 \) and \( x_2 \) directions. Due to symmetry, the flow computations were performed in the lower half of the duct using a \((9 \times 13 \times 53)\) grid in the \( x_1, x_2 \), and \( x_3 \) directions, respectively. Figure 3 shows the inlet velocity profiles which were used in the numerical calculations with no variation in the \( x_1 \) direction. The experimental measurements for 2.5 inch spacers were obtained half way between the pressure and suction surfaces are shown in the same figure. The results of the computations are presented in two-dimensional form in Figs. 4 through 8. The flow velocities are normalized with respect to the maximum inlet velocity, \( V_{\text{max}} \), while the pressures are normalized with respect to the critical pressure, which corresponds to a tank gauge pressure of 20 inches water in the experimental measurements.

The orthogonal curvilinear body fitted coordinates for the flow computations were generated numerically using the code developed by Davis \(^{20}\) which is based on the Schwartz-Kristoffel transformation. The values of the orthogonal coordinate in the transformed plane were between 0 and 1 in the \( x_1 \) direction and 0 and 6.791 in the \( x_2 \) direction. The computational grid is uniform in the \( x_1 \) direction where half of the duct height is equal to 1.875 times the duct exit width. The convergence of the numerical solution was fast with CPU time of 3.5 minutes for 25 outer iterations on AMDAHL 370. The inner iterations did not exceed 2 in all the iterative solutions for \( x_1, x_2 \), and \( x_3 \) solutions on all 77 surfaces.

The results are presented first for the computed static pressure coefficient distribution over the duct curved walls and are compared with the experimental measurements. Complete spanwise static pressure distributions over the pressure and suction surfaces of the elbow were reported by Stanitz et al \(^{19}\) for the flow with nominal exit Mach number of 0.26.

Figures 4 and 5 show the computed static pressure coefficient distribution over the duct curved boundaries at two
The analysis is general and applicable to flow fields with both total pressure and total temperature gradients. The results of the computations are presented for the flow in an accelerating rectangular elbow with shear inlet velocity profile and 90° turning angle. The analysis predicts the secondary flow development and the computed results are in agreement with the experimental measurements in the regions where the viscous dissipation effects are not significant.

**References**


Fig. 1. Schematic of the curved ducts showing the three sets of orthogonal surfaces and the corresponding streamlike functions.
Fig. 2. Duct Geometry and Computational Grid.

Fig. 3. Inlet Velocity Profile.
Fig. 4. Pressure Distribution Near End Wall ($N_3 = 3$).

Fig. 5. Static Pressure Distribution Near Mid-Span ($N_3 = 11$).
Fig. 6. Normalized Total Pressure Contours in the Cross Sectional Surfaces.
Fig. 7. Normalised Through Flow Vorticity Contours in the Cross Sectional Surfaces.
Appendix 4

Internal Three-Dimensional Viscous Flow Solution Using the Streamlike Function
AIAA 17th Fluid Dynamics, Plasma Dynamics, and Lasers Conference
June 25-27, 1984/Snowmass, Colorado
A new method for the numerical solution of the parabolized Navier-Stokes equations in primitive variables for incompressible, compressible, subsonic, supersonic, laminar, and turbulent internal flows is presented. The only elliptic influence which is accounted for in the parabolized solution is that due to the potential pressure field. Partially parabolized solution procedures were subsequently developed for the numerical solution of some complex three dimensional flow fields in which pressure is the dominant transmitter of influences in the upstream direction. These flows are still characterized by the absence of recirculation in the primary flow direction and by high Reynolds numbers so that the viscous diffusion and thermal conduction are significant in the lateral direction. The solution procedure remains unchanged for the main and lateral momentum equations, but the pressure correction equation at each marching step contains terms which link the pressure correction in a given lateral plane to upstream and downstream pressure corrections. In both methods, the velocity components are obtained from the momentum equations, and the continuity equation is satisfied indirectly by the pressure field. This indirect approach to satisfying the continuity equation is common to all parabolized and partially parabolized methods.

The full Navier-Stokes equations are required to model flows with significant separation or shear layers not aligned with one of the coordinates. In addition, the parabolized and partially parabolized methods are not suitable for obtaining solutions to flow fields in which viscous phenomena significantly affect pressure distribution. Several methods have been developed for the solution of the time-dependent form of the governing equations.

A large number of numerical methods have been developed over the years for the solution of internal viscous flow fields. The earliest solutions obtained for the parabolized Navier-Stokes equations. Several successful implicit iterative solution procedures have since been developed for the numerical solution of the parabolized Navier-Stokes equations in primitive variables for incompressible, compressible, subsonic, supersonic, laminar, and turbulent internal flows.
The results of the through flow computations were shown to be dependent on the difference schemes for the continuity equation, especially in the prediction of the secondary flow development. The numerical solution fluttered on the inside of the head, suffered a loss of mass and failed to fully converge, and the author suggested that more attention to the non-linearities of the flow may possibly alleviate the last two problems. Other methods developed for the elliptic solution of the steady state full Navier-Stokes equations are based on the extension of the well-known 2-D vorticity transport formulation to three dimensions. Nils and Hallums defined a vector potential to identify satisfy the equation of conservation of mass in three dimensional flow fields. The vector potential vorticity formulation has been used to solve the problem of laminar natural convection. In this formulation, the resulting governing equations consist of three 2-D Poisson equations for the vector potential and three vorticity transport equations. The main advantage of this formulation, in two dimensions, namely the smaller size of the governing equations is actually reversed in three dimensions. Since three-dimensional differential equations have to be solved for the three components of the vector potential.

One can conclude from the preceding discussions that existing space-elliptic solvers of the 2-D Navier-Stokes equations are very costly in terms of CPU time and storage requirements. In addition, some of these methods which were developed for simple convection problems have not been very successful in through flow calculations. The presented work represents a new formulation for the 3-D Navier-Stokes equations that leads to an elliptic solution. The formulation is based on the use of the 2-D streamlike functions to identify satisfy the equation of conservation for 3-D rotational flows. The governing equations consist of the vorticity transport equation and 2-D Poisson equations for the streamlike functions. The present method is very general, in that inviscid flow solutions can be obtained in the limit when \( \nu \to \infty \). In fact, numerical solutions have been obtained for inviscid rotational incompressible and compressible flows in curved ducts and it was demonstrated that the method can predict significant secondary flow and streamline vorticity development due to inlet vorticity. The following work represents the generalization of this formulation to internal three-dimensional viscous flow problems.

**Analysis**

The governing equations consist of the vorticity transport equation and the equation of conservation of mass which are written in the following dimensionless form, for incompressible viscous flow:

\[
(\nabla \times \mathbf{v}) \cdot \mathbf{h} = (\nabla \times \mathbf{v}) \cdot \mathbf{h} - \frac{1}{Re} \nabla^2 \mathbf{h} 
\]

and

\[
\mathbf{v} \cdot \nabla \mathbf{v} = 0
\]

In the above equations \( \mathbf{h} = \nabla \mathbf{v} / \nu \) when the velocities are normalized by \( \nu^2 \), the space dimension by \( D \) and the vorticity by \( \nu^2 / D \).

The solution to the three dimensional viscous flow is obtained in terms of the three vorticity components and three streamlike functions which are defined to identically satisfy the equation of conservation of mass for general three-dimensional rotational flow fields. Unlike the traditional stream function solutions which must be obtained on stream surfaces, the following streamlike function velocity relations permit the definition of these two dimensional functions \( x_1(x,y), x_2(y,z), x_3(z,x) \) on fixed non-stream surfaces in the flow field.

**Streamlike Functions Velocity Relations**

The streamlike function formulation was developed by the authors to model internal three-dimensional flow fields. More details and general definitions in curvilinear coordinates of the streamlike function can be found in reference 17. For the sake of simplicity, the equations will be presented here for incompressible flow using Cartesian coordinates.

**Definition of \( x_1 \)**

\[
\frac{\partial x_1}{\partial y} = u_1 + \int_0^x \frac{\partial w_2}{\partial z} \, dx, \quad \frac{\partial x_1}{\partial x} = -v_1
\]

**Definition of \( x_2 \)**

\[
\frac{\partial x_2}{\partial y} = -w_2 - \int_0^x \frac{\partial u_1}{\partial z} \, dx, \quad \frac{\partial x_2}{\partial z} = v_2
\]

**Definition of \( x_3 \)**

\[
\frac{\partial x_3}{\partial z} = -u_3 + \int_0^x \frac{\partial w_2}{\partial x} \, dx, \quad \frac{\partial x_3}{\partial x} = w_3 - \int_0^y \frac{\partial u_1}{\partial x} \, dx
\]
The governing equations (7)-(9) and (13)-(15) are solved for the vorticity components \( \eta, \xi, \zeta \) and the streamlike functions \( x_1, x_2, x_3 \), respectively. The boundary conditions used for the solution of these equations are given for the viscous flow in a square duct. Because of symmetry, only one quarter of the square duct is considered in the following derivations. The coordinate \( y \) along the straight duct is measured from the duct entrance, while \( x \) and \( s \) represent the coordinates in the cross sectional planes measured from the duct centerline as shown in Fig. 1.

**Boundary Conditions**

At the inlet station which extends far upstream of the duct entrance, the flow velocity is taken to be uniform \( (v = v_1, u, w = 0) \), leading to the following boundary conditions:

\[
\eta = \xi = \zeta = 0
\]

and

\[
\frac{\partial x_1}{\partial y} = \frac{\partial x_2}{\partial y} = 0
\]

At the duct boundaries, the no slip condition is used to obtain the boundary conditions for the vorticity components, while the zero flux condition is used to obtain the boundary conditions for the streamlike functions.

The streamlike function boundary conditions are simplified through the appropriate choice of the reference coordinates \( x_0, s_0 \) in the lower limit of the integrals in equations (3)-(5). The following boundary conditions result when \( x_0 = s_0 = 0 \).
The elliptic system of equations are solved using an iterative procedure. At each global iteration, the linear equations were solved by successive relaxation methods. Numerical computations in a straight duct with $L/D_D = 0.1$ were performed using a uniform grid with $Ax/D_D = Ay/D_D = 0.001$ and $Ay/D_D = 0.0033$. Due to symmetry, the computations were only carried out in one quarter of the duct for $x_1$, $x_2$, $\xi$ and $\xi$ since $\xi(x,y,z) = -\eta(x,y,z)$ and $x_1(x,y) = -x_3(s,y)$. Relaxation parameters of 1.6 for $x_1$, 1.9 for $x_3$ and 0.4 for $\xi$ and $\xi$ were used in the inner iterations with a convergence criteria of $\varepsilon_1 = 1 \times 10^{-4}$, $\varepsilon_3 = 1 \times 10^{-5}$, $\varepsilon_4 = 1 \times 10^{-5}$, $\varepsilon_1 = 5 \times 10^{-6}$, $\varepsilon_3 = 5 \times 10^{-6}$ according to the following equation:

$$\frac{1}{h} \sum \left( |u_{i,j}^{n+1} - u_{i,j}^n | \right) \leq \varepsilon_n$$

The numerical solutions required 30 global iterations and a CPU time of 2 minutes and 13 seconds on AMDahl 370 using an $11 \times 11 \times 14$ uniform grid. The overall number of iterations was 35 for the vorticity equations and 179 for the streamfunction equations. The numerical solution domain extended 1.67 diameters upstream of the duct inlet, where the flow velocity was taken to be uniform and equal to one. The results of the numerical computations are presented at $y/D_D = 0.0$, 0.01 and 0.10. The through flow contours at the duct inlet are presented in Fig. 2. The flow development from a uniform through velocity to the profile of Fig. 2 at the duct inlet is accompanied by lateral flow displacements due to the secondary velocities. The secondary velocity contours at the duct inlet are shown in Fig. 3 for the vertical velocity component $w$. The ellipticity of the numerical solution is demonstrated in the velocity contours at the duct inlet, and in the velocity fields up to 0.83 diameters upstream of the duct inlet. Figure 4 shows the contours for the secondary velocity component, $w$, at $y/D_D = 0.01$. A comparison of Figs. 3 and 4 reveals the change in both the magnitude and the location of the maximum secondary velocities along the duct. The development of the secondary velocity component, $w$, along the plane of symmetry, $x = 0$, is presented in Fig. 5. One can see that the maximum secondary velocities are found near the solid boundaries at the duct inlet. As the flow proceeds towards fully developed conditions, the secondary velocities decrease and the location of the maximum values moves toward the center of the duct. The results of the numerical computations at the duct exit are shown in Fig. 6 for the
The vorticity and streamlike function contours are shown in Figs. 7 through 11. The contours for the vorticity component (y/200 = 0.0 and 0.1 in Figs. 7 and 6. Horizontal sections through the center of the streamlike function x_3 at y/200 = 0.0 and 0.1. The figures show a change in the sign of the streamlike function between these two sections. The contours for the streamlike function x_1 at y/200 = 0.0 are shown in Fig. 11. One can see that x_1 reaches values less than -0.5 inside the duct near the walls.

The computed through flow velocity profiles along the plane of symmetry, z = 0, are compared with the experimental measurements of reference 19 in Fig. 11. One can see that the computed and measured are in good agreement with the experimental measurements. The computed through flow velocity development along the duct centerline are compared with the experimental measurements of reference 18 in Fig. 11. The agreement between the computed results and the experimental data shown in Figs. 12 and 13 is very satisfactory, in view of the uniform coarse grid used in the numerical calculations.

**Conclusions**

This paper presents a new method for the three-dimensional elliptic solution of the Navier-Stokes equations which is based on the streamlike-function vorticity formulation. The computed results for the three-dimensional viscous flow in a square duct are presented and compared with experimental measurements. The results demonstrate that the streamlike function can successfully model viscous effects in the three dimensional flow field computations. Since the same formulation has been successfully used in inviscid rotational flows, to model secondary flow development due to inlet shear velocity under the effect of curvature, one can conclude that the present method can be effectively developed to obtain efficient numerical elliptic solutions of internal viscous flow in curved passages.

**Acknowledgement**

This work was sponsored by U.S. Air Force Office of Scientific Research under Contract No. 86-0242.

The authors wish to express their appreciation to Mr. M. Mansour for his help in performing the numerical computations.

**References**


**Fig. 1.** Coordinate System in the Duct.

**Fig. 2.** Through Flow Velocity Contours at the Duct Inlet (y/Dr=0.00).
Fig. 3. Secondary Velocity Contours at the Dust Inlet (y/Drn = 0.00).

Fig. 4. Secondary Velocity Contours at y/Drn = 0.01.

Fig. 5. Through Flow Velocity Contours at y/Drn = 0.1.

Fig. 6. Through Flow Velocity Contours at y/Drn = 0.1.
Fig. 7. Vorticity, \( \zeta \) Contours at the Duct Entrance \( (y/DRe = 0.0) \).

Fig. 9. Streamlike Function, \( x_3 \) Contours at the Duct Inlet \( (y/DRe = 0.0) \).

Fig. 8. Vorticity, \( \zeta \) Contours at \( y/DRe = 0.1 \).

Fig. 10. Streamlike Function, \( x_3 \) Contours at \( y/DRe = 0.01 \).
**Fig. 11.** Streamline Function, $\chi_1$ Contours at $z = 0$.  

**Fig. 12.** Development of the Through Flow Velocity Profile at the Central Plane $x_1 = 0$.  

**Fig. 13.** Through Flow Velocity Development Along the Duct Centerline.
Appendix 5

The Elliptic Solution of 3-D Internal Viscous Flow Using the Streamlike Function
1. INTRODUCTION

The prediction of the complex 3-D flow field in turbo-machinery blade passages continues to be the subject of many viscous and inviscid flow studies. Recently developed 3-D inviscid methods [1-3] are capable of predicting 3-D flow characteristics such as secondary flows [3]. While the secondary flow is caused by vorticity which is produced by viscous forces, these inviscid methods cannot predict the viscous produced losses in the blade passage. Internal viscous flow solution methods have been developed using parabolised Navier-Stokes equations. While these methods are very fast, their application is limited due to their inability to simulate downstream blockage and strong curvature effects. Partially parabolised methods maintain the advantages of the parabolised methods in that the streamwise diffusion of mass, momentum and energy are still neglected but the elliptic influence is transmitted upstream through the pressure field. These well developed methods are not discussed here since the reader can refer to the extensive review of Davis and Rubin [4] and Rubin [5]. Our following discussion will be limited to the fully elliptic methods for the solution of the 3-D Navier-Stokes equations for internal flows.

Axia and Mallums [6] developed the vector potential-vorticity formulation, and used it to solve the problem of laminar natural convection. This approach is an extension of the well known 2-D stream function vorticity formulation to 3-D problems. The equation of conservation of mass is identically satisfied through the definition of the vector potential. In this formulation, the resulting governing equations consist of three 3-D Poisson equations for the vector potential and three vorticity transport equations. Williams [7] used a time marching method for the solution of the laminar incompressible flow field due to thermal convection.
A rotating annulus using primitive variables. In this pressure field is computed from the solution of a equation with Neumann boundary conditions. Both variables require the solution of three par

transport equations. The first method requires in the solution of the 3-D Poisson equations with boundary conditions along two boundaries and zero conditions along the third boundary, while a

additional 3-D Poisson equation for the third boundary conditions over all the [9] discussed the relative merits of these in terms of CPU time, computer storage require-

development time when iterative and direct

used in the solution of Poisson equation.

an elliptic solution for viscous flow in a [10] from the governing

primitive variables using a new finite difference

problems were encountered in the application of the tendency to break flow calculations, the tendency of the

stream to decrease and a loss of mass between the duct inlet

point and the computed results. More recently, Dodge [11]

split procedure in which the velocity

split into viscous and potential components, and the

field is determined from the potential velocity

The governing equation for the viscous velocity

vector component is obtained from the momentum equation with the pressure gradient expressed in terms of the derivatives of the velocity potential vector component, while the govern-

ation for the velocity potential vector component

obtained from continuity. Beyond this formulation,

Dodge’s numerical solution procedure is partially parabolic

since he neglected the streamwise diffusion of momentum in

a marching solution for the viscous velocity com-

ents. The analysis itself, in terms of the type of the
governing equations and their boundary conditions, is com-

parable to the velocity pressure formulation since the govern-
ing equation for the velocity potential is a 3-D Poisson

equation. Dodge did not discuss the boundary conditions for

the velocity potential. He only mentioned that it can be

complex and that he used zero potential gradient normal to

the wall in his numerical solution. Other formulations that

can lead to elliptic solution, were described in references

[5] and [8], however they will not be discussed here since

they have not yet been applied to 3-D flow computations.

In summary, existing elliptic solvers for the Navier-

Stokes equations require the solution of one or three 3-D

Poisson equations in addition to the three momentum or the

vorticity transport equations. In the case of the

velocity pressure formulation, the velocity vector is evalu-

ated from the momentum equation and the continuity equation

satisfied indirectly in the pressure equation. The con-

vergence of the iterative numerical procedure would be very
The present approach are not used for the solution of the
problem with Neumann boundary conditions for the
velocity. On the other hand, the continuity equation
is satisfied in the vector potential-vorticity
form and it leads to three 3-D Poisson equations
for the vector potential equations. Convergence of the iterative
process presents a problem in this case, since Dirichlet
boundary conditions are not imposed on parts of the boundary, but computer
resources are greatly increased by two additional

The work represents a new formulation for the
problem equations that leads to a very economical
formulation. The formulation is based on the use of
potential functions [13] to identify the
three 3-D rotational flows. In addition to
the continuity equation, the present formulation
for the problem equations with Dirichlet boundary con-
tains additional similarity functions. The presented method
is such that inviscid flow solutions can be obtained
from initial values for \( u = 0 \). In fact, numerical solu-
tions obtained for inviscid rotational incompressible
and compressible [14] flows in curved ducts and it
was demonstrated that the method predicts the secondary flow
with an exception to the inert vorticity. The validity of the
method is established by considering secondary flow which are indirectly
obtained by the analysis of vorticity have been demon-
strated in inviscid flow [13, 14]. The following work repre-
sents an extension of this formulation to viscous flow
problems. The analytical formulation is developed for general
viscous cases only the results of the computation are presented for
nearly viscous flow problem of a uniform entrance flow in
a curved duct. The final goal is to develop a method for
predicting the three dimensional flow field, and the losses
in internal flow fields with large surface curvature and
significant downstream effects as in the case of turbine blade
passages.

2. ANALYSIS

The governing equations are the vorticity transport
equation and the equation of conservation of mass, which are
given below in nondimensional vector form for incompressible
flow [15]

\[
\frac{\partial \Omega}{\partial t} + (\mathbf{v} \cdot \nabla) \Omega - (\mathbf{v} \cdot \nabla) v - \frac{1}{Re} \nabla^2 \Omega = 0
\]  

(1)

and

\[
\nabla \cdot \mathbf{v} = 0
\]  

(2)

where

\[
\Omega = \nabla \times \mathbf{v}
\]
\[ \mathbf{\Omega} = \mathbf{\omega} \times \mathbf{\Omega} \]  

(3)

The Reynolds number \((Re = U \rho D / \nu)\), appears in the equations as a result of normalising the velocities by \(U\), the space dimensions by \(D\) and the vorticity by \(U \rho / \nu\), where \(\nu\) is the kinematic viscosity.

Equations (3-8) are written in Cartesian coordinates as follows.

**Vorticity Transport Equations**

\[ \frac{\partial \Omega_x}{\partial x} = - \nabla \cdot \mathbf{\omega} + \Omega \cdot \nabla u + \frac{1}{Re} \Omega^2 \]  

(4)

\[ \frac{\partial \Omega_y}{\partial y} = - \nabla \cdot \mathbf{\omega} + \Omega \cdot \nabla v + \frac{1}{Re} \Omega^2 \]  

(5)

\[ \frac{\partial \Omega_z}{\partial z} = - \nabla \cdot \mathbf{\omega} + \Omega \cdot \nabla w + \frac{1}{Re} \Omega^2 \]  

(6)

where \(\Omega_x, \Omega_y, \Omega_z\) are the components of the vorticity vector, \(\mathbf{\Omega}\), in the \(x, y\) and \(z\) directions, respectively; \(u, v, w\) are the components of the velocity vector \(\mathbf{v}\).

**Equation of Conservation of Mass**

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \]  

(7)

**Vorticity Velocity Relations**

\[ \eta = \frac{\partial \omega}{\partial y} - \frac{\partial v}{\partial z} \]  

(8)

\[ \zeta = \frac{\partial \omega}{\partial z} - \frac{\partial w}{\partial x} \]  

(9)

\[ \zeta = \frac{\partial \omega}{\partial x} - \frac{\partial u}{\partial y} \]  

(10)

**The Streamlike Function Formulation**

Three streamlike functions \(\chi_n(x,y), \chi_v(y,z), \chi_c(x,z)\) are defined to identically satisfy the equation of conservation of mass in the case of internal rotational flows [13, 14]. The velocity field is determined from the streamlike functions according to the following relations:
Semi-infinite Functions Velocity Relations

\[
\begin{align*}
\mathbf{u} & = \frac{2}{\pi} - \int_{0}^{2} \frac{\mathbf{v}}{3z} \, dz \quad \text{(11)} \\
\mathbf{v} & = -\frac{3}{2\pi} \quad \text{(12)} \\
\mathbf{w} & = -\frac{3}{2\pi} - \int_{0}^{2} \frac{\mathbf{w}}{3z} \, dz \quad \text{(13)} \\
\mathbf{v} & = \frac{3}{2\pi} \quad \text{(14)} \\
\mathbf{w} & = -\frac{3}{2\pi} - \int_{0}^{2} \frac{\mathbf{w}}{3z} \, dz \quad \text{(15)} \\
\mathbf{v} & = \frac{3}{2\pi} + \int_{0}^{2} \frac{\mathbf{w}}{3z} \, dz \quad \text{(16)}
\end{align*}
\]

where the subscripts \( h \), \( v \) and \( w \) refer to solutions on the horizontal, vertical and cross-sectional planes respectively with

\[ u = u_{h} + u_{v}, \quad v = v_{h} + v_{v}, \quad \text{and} \quad w = w_{v} + w_{w}, \quad \text{(17)} \]

Equations (11-17) satisfy the equation of conservation of mass (7) identically,

\textbf{Streamline Functions Equations} \hspace{1cm} \text{Substituting equations (11)-(17) into equations (8)-(10) one obtains}

\[
\begin{align*}
\frac{3^{2}}{2\pi} + \frac{3^{2}}{2\pi} = \epsilon - \xi + \frac{3}{3z} \int_{z=0}^{2} \frac{\mathbf{v}}{3z} \, dz \\
\frac{3^{2}}{2\pi} + \frac{3^{2}}{2\pi} = n - \eta - \frac{3}{3z} \int_{z=0}^{2} \frac{3u_{h}}{3z} \, dz \\
\frac{3^{2}}{2\pi} + \frac{3^{2}}{2\pi} = \epsilon - \xi + \frac{3}{3z} \int_{z=0}^{2} \frac{\mathbf{v}}{3z} \, dz - \frac{3}{3z} \int_{z=0}^{2} \frac{3u_{h}}{3z} \, dz
\end{align*}
\]
The governing equations (4)-(6) and (18)-(20) are solved for the vanishing components $\eta, \xi, \zeta$ and the streamlike components $\phi_y, \phi_v$ and $\phi_q$, respectively. The boundary conditions from the solution of these equations are given for the incoming flow in a square duct. The inlet station is extended far upstream where uniform incoming flow is assumed (Fig. 1). This case simulates flow in cascades with zero exit, where periodic conditions apply over the extended boundaries up to the duct entrance. Because of symmetry, only one quarter of the square duct is considered in the determination of the boundary conditions. The coordinates $x$ and $s$ are measured from the duct centerline and $y$ from the duct entrance as shown in Fig. 1.

DUCT GEOMETRY AND DIMENSIONS

Fig. 1. A Schematic of the Duct with Cascade Entrance.
Boundary Conditions for the Streamlike Functions

i. At the duct boundaries:

As indicated in Fig. 3, the boundary conditions for 
the equations (11)-(17) are obtained from the substitution of the no 
slip boundary conditions. In addition, the symmetry condition at the duct boundaries

$$\frac{\partial x_c}{\partial s} = 0$$

are used in the determination of the 
streamlike functions in the planes coinciding with the duct boundaries.

![Diagram](image)

Figure 2

ii. Along the cascade entry:

Figure 3 shows the boundary conditions for the stream-
like functions at the extension of the duct boundaries.
The conditions at the planes of symmetry are unchanged.
iii. At the inlet station:

The following relations satisfy the inlet condition 

\[ u = v = 0 \]

\( \frac{3x_a}{l_T} = 0 \)

\( \frac{3x_r}{l_T} = 0 \)

iv. At exit:

Fully developed flow conditions are assumed, leading to

\( \frac{3x_a}{l_T} = 0 \)

\( \frac{3x_r}{l_T} = 0 \)
Boundary Conditions for the Vorticity Components

Along the boundaries:

The no-slip condition is satisfied through the vorticity boundary conditions. The symmetry conditions are used along the vertical and horizontal central planes to determine the boundary conditions shown in Fig. 4.

\[ \eta = -\frac{2\psi}{\partial \xi}, \zeta = 0, \zeta = \frac{3\eta}{\partial x} \]

\[ \eta = 0 \]

\[ \zeta = -\frac{3\eta}{\partial x} \]

\[ \zeta' = \frac{3\eta}{\partial x} \]

\[ \eta = 0, \zeta = 0, \frac{\partial \zeta'}{\partial x} = 0 \]

Figure 4

ii. Along the cascade entrance

Figure 5 shows the boundary conditions at the extension of the duct boundaries.

iii. At the inlet station:

\[ \zeta = 0 \]

\[ \eta = 0 \]

\[ \zeta = 0 \]

iv. At exit:

Fully developed flow conditions are assumed, leading to:

57
3. RESULTS AND DISCUSSION

The results of the numerical computations are presented in one quarter of a square duct. In this case, only two of the vorticity equations [eqs. (4) and (5)] and two of the streamlike function equations [eqs. (19) and (20)] are solved, since \( \chi_\theta = \chi_r \) and \( \zeta(x,y,z) = \eta(x,y,z) \). Referring to Fig. 1, the solution was obtained for \( \text{Re} = 50 \) in a duct with \( L/D_re = 0.1 \) and \( L_e/D_re = 0.01 \) using SOR and a (11 x 11 x 34) grid.

Figures 6a and 6b show the through flow velocity contours at the duct entrance and exit. The influence of the cascade entering on the elliptic solution is demonstrated in contours of Fig. 6a.

The contours for the secondary velocity component, \( w \), are shown at \( y/D_re = 0.0 \) and \( 0.0075 \) in Figs. 7a and 7b. From these figures, one can see a large change in both the magnitude and the location of the maximum secondary velocities along the duct. The development of the through flow velocity profiles along the plane of symmetry, \( x = 0 \), is shown in...
Fig. 6a. Through Flow Velocity Contours at the Duct Entrance.

Fig. 6b. Through Flow Velocity Contours at the Duct Exit ($y/DRe = 0.1$).
Fig. 7a. Secondary Velocity Contours at $y/Dr = 0.0$

Fig. 7b. Secondary Velocity Contours at $y/Dr = 0.0075$
Fig. 8 for the computed results and the experimental measurements of reference [16]. One can see that the computed results are in good agreement with the experimental measurements at $y/D_{in} = 0.0075$ and 0.02, but that the computed through flow velocities at $y/D_{in} = 0.1$ did not reach the experimentally measured fully developed profile.

![Graph showing experimental data and computed results.](image)

Fig. 8. Development of the Through Flow Velocity Profile at the Central Plane $x = 0$.

Figure 9 shows the development of the secondary velocity component, $w$, along the duct plane of symmetry, $x = 0$. No experimental measurements are available for comparison with the computed secondary velocities. Figure 9 shows that the maximum secondary flow is initially located near the solid boundaries, then moves towards the center of the duct and decreases as the flow proceeds towards fully developed conditions. The computed through flow velocity development along the duct centerline are compared with the experimental measurements of reference [16] in Fig. 10. One can see in this figure that the elliptic solution predicts an increase in the centerline through flow velocity in the cascade entry region preceding the actual duct entry. Figure 10 shows that the computations slightly underestimate the centerline velocity. Considering the coarse grid used in the numerical computations ([11x11x34] grid points in the duct), the agreement of the computed results with the experimental data as shown in Figures 8 and 10 is very satisfactory.
Fig. 9. The Development of Secondary Flow Velocity Profile at the Central Plane $x = 0$.

Fig. 10. Through Flow Velocity Development Along the Duct Centerline.
This paper presents a fast efficient method for the 3-D solution of the Navier-Stokes equations. It is based on a streamfunction-vorticity formulation which leads to six equations with Dirichlet boundary conditions for the three streamlike functions, in addition to the vorticity transport equations. The method is more economical than viscosity Navier-Stokes elliptic solvers, yet does not suffer limitations of the parabolized and partially parabolized procedures. It offers a useful tool for the numerical simulation of internal viscous flow fields where surface curvature and downstream effects are significant, as in turbine main passages. The results of the computations are presented for an average flow in a constant area duct, as a corner stone for more general future applications.

ACKNOWLEDGEMENT

This work was supported by the Air Force Office of Scientific Research under Grant No. 80-0242.

REFERENCES


**Definitions**

- $K$: dust height
- $L$: dust length
- Re: Reynolds number
- $t$: time
- $u$, $v$, $w$: velocity components in $x$, $y$ and $z$ directions
- $\mathbf{u}$: velocity vector
- $v_{in}$: maximum velocity at inlet
- $x$, $y$, $z$: Cartesian coordinates
- $x^*$, $y^*$, $z^*$: streamlike functions
- $\nu$: kinematic viscosity
- $\mathbf{a}$: vorticity vector
- $\alpha_x$, $\alpha_y$, $\alpha_z$: vorticity components in $x$, $y$ and $z$ directions

**Superscripts**

- $a$: refers to cross sectional plane
- $h$: refers to horizontal plane
- $i$: refers to inlet plane
- $v$: refers to vertical plane
Appendix 6

LVD Measurements of Three-Dimensional Flow
Development in a Curved Rectangular Duct
with Inlet Shear Profile
AIAA 17th Fluid Dynamics, Plasma Dynamics, and Lasers Conference

June 25-27, 1984/Snowmass, Colorado
LOW MEASUREMENTS OF THREE-DIMENSIONAL FLOW DEVELOPMENT IN A CURVED RECTANGULAR DUCT WITH INLET SHEAR PROFILE

A. Nasr* and N. Malak**

Department of Aerospace Engineering and Applied Mechanics
University of Cincinnati
Cincinnati, Ohio 45221

Abstract

The results of an experimental investigation of the three-dimensional flow development in a highly curved duct with inlet shear profile are presented. The three components of the air velocity in a curved duct with a rectangular cross-section are measured using Laser Dopper Anemometry. Significant through velocity contour rotations are reported with secondary velocity development of magnitudes up to 0.25 of mean inlet velocity in the 90° turning angle curved duct.

Introduction

The secondary flow affects the overall turbomachinery performance through its influence on the angle and the energy distribution of the flow leaving the blade rows. A deviation in the exiting flow angles from those predicted by the blade element analysis results from the secondary velocities. The secondary losses are also caused by the redistribution of the low energy flow by the same secondary velocities. These cross velocities are associated with the secondary velocity development in the streamwise direction through the turning of the flow with nonuniform inlet conditions in the blade passages.

Secondary flow in compressor and turbine cascades has been the subject of several theoretical and experimental investigations. In most of the experimental secondary flow investigations, the flow measurements have been limited to cascade inlet and exit conditions to provide empirical correlations for secondary flow losses and exit flow angles.

Lansonett obtained detailed measurements showing the general characteristics of the end wall flow downstream of the channel and upstream of the turbine cascade passages. Moore measured the total pressure and flow directions downstream of the turbine cascade trailing edge. Several investigators measured the development of secondary flow in curved ducts from a fully developed inlet velocity profile. These studies demonstrate the secondary flow development in the absence of added complexities of the cascade blade leading edges.

In the present work detailed measurement of the three flow velocity components are obtained in a curved duct with a nearly linear shear flow inlet profile produced using a grid of 0.120 x 0.003" wire mesh with varying spacing. Under these conditions, the development of the secondary velocities associated with the passage vortex is not limited to the boundary layer region near the wall, but extends instead through the whole passage sections. The experimental measurements of this complex flow field are based on the use of a two-color back scatter Laser Dopper Velocimeter.

EXPERIMENTAL SET-UP

The experimental set-up is shown schematically in Fig. 1. It consists of the tunnel, the seeding particle atomiser, the LDV, optical and data acquisition systems.

Tunnel

The high pressure air supply from the storage tanks is regulated to a lower pressure before entering the 12.75" diameter settling chamber. A 1.5" thick honeycomb of 0.187" cell and 0.003" wall thickness is placed in a 4" diameter PVC tube to condition the flow. The latter extends 18" inside the chamber and blends smoothly into a 22.75" long rectangular channel preceding the curved duct. The duct is shown schematically in Fig. 2 and consists of a 90° bend of 6" mean radius and a 2 x 4" rectangular cross-section. The duct walls are made of plexiglass. The thickness of the curved wall is equal to 1/6" while the plane walls are 3/4" thick. The curved duct is connected to straight ducts downstream and upstream where the shear flow is produced using a grid of parallel rods with varying spacing. The grid imposes a resistance to the flow that varies across the section so as to produce variation in the flow velocity, without introducing an appreciable gradient in static pressure. The basic relations between rod size, spacing and the produced velocity gradient were derived first by Owen and Bollenkamps and modified later by Livesey and Turner and by Elder for more generalised profiles. The basic relations for a grid to generate a uniform

* Professor; Associate Fellow AIAA.
** Graduate Research Assistant; Student Member AIAA.

Copyright © Aerospace Institute of Aeronautics and Astronautics, Inc., 1969. All rights reserved.
The high pressure air supply from air storage tanks was regulated to a pressure of 2.5684 x 10^5 N/m^2 (30 psig) at the orifice meter, using a regulating valve, to give an air mass flow of 0.1143 kg/sec (0.252 lb/sec). This corresponds to Reynolds number of 1.3 x 10^5, based on the height of the duct and the mean inlet velocity of 30 m/sec (66 ft/sec), upstream of the shear velocity generator.

The experimental velocity measurements were obtained at sections B, C, D and E as shown in Fig. 2. Sections C and D are located in the curved duct at the 45° and 75° turning angles, while the first and last measuring stations B and E are located in the straight portions of the tunnel. A quarter inch spacing between the measuring points in the radial direction and also in the direction normal to the duct plane walls was kept in the measurements at all four cross sections. The velocity measurements were, therefore, obtained at 105 points of a 7 x 15 grid in every section. In order to determine the three velocity components at each measuring point the measurements were obtained once with the Laser-optics axis perpendicular to the duct plane wall, then repeated with the Laser-optics axis parallel to the curved wall. The first set of measurements provided the through flow velocity U, in the direction normal to the tunnel cross-

<table>
<thead>
<tr>
<th>TABLE 1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>LOD CHARACTERISTICS</strong></td>
</tr>
<tr>
<td>----------------------------------</td>
</tr>
<tr>
<td></td>
</tr>
<tr>
<td>Wavelength</td>
</tr>
<tr>
<td>Fringe spacing</td>
</tr>
<tr>
<td>Diameter of measuring volume at the 1/ε^-2</td>
</tr>
<tr>
<td>Length of measuring volume at the 1/ε^-2</td>
</tr>
<tr>
<td>Number of stationary fringes</td>
</tr>
</tbody>
</table>

Two signal processors and a DIGITAL MINC 11/73 computer were used on line to acquire synchronized data for the simultaneous measurements of the two velocity components.

**RESULTS AND DISCUSSION**

The laser and optical systems are located on a table designed such that the optical axis can be traversed in the horizontal plane, as well as along and across the horizontal plane. The whole apparatus is located on a platform that rotates about a vertical axis. This 3 degrees of freedom is used to allow measurements through the outer curved wall. Finally the mountings of the sending and receiving optics are designed to allow for rotation around the optical axis up to 90°, in order to obtain the measurements of the two velocity components in any specified direction.
sections, and the radial velocity components, \( V_r \), while the second set of measurements give the third velocity component \( V_z \), and also the through flow velocity \( u_0 \). The through-flow velocity at the three measurement sections were compared to determine the repeatability of the data after the duct is rotated 90° relative to the settling chamber. The difference between the two values was not found to exceed 1.5% in the reported measurements. The results of the experimental measurements of the three velocity components normalized with respect to the mean flow velocity \( u_0 \), upstream of the shear generating grid, are presented in Figures 4 through 11.

The profiles of the normalized through-flow velocity \( (u_0/u_0) \) are presented at seven concentric cylindrical surfaces between the duct inner and outer curved walls at sections E, C, D and B in Figures 4, 5, 6 and 7 respectively. One can see that the shear generating grid produced the desired linear velocity profile along the duct axis except in the region near the upper wall where the velocity gradient is higher. This deviation was due to the influence of the distance of the last grid wire from the upper wall. This factor was not found to have significant effect near the lower wall where the wire grid spacings are smaller. Other shear velocity generating grids with different wire diameters (0.039" and 0.139") have been investigated. The discussed effect was even more pronounced in the case of larger wire diameter grid. On the other hand, the velocity profiles produced by the grids of the smaller wire diameter were found to produce velocity variations along the wire length. Careful examination of the grid revealed non-uniformities in the wire spacings in this direction, which was found difficult to control.

Figures 5 through 7 demonstrate the change in the through flow velocity profile with the duct turning angle. Initially, the flow accelerates along the inner wall and decelerates along the outer wall to approach potential free stream velocity distribution. This can be seen by comparing the velocity profiles in Figures 4 and 5, at \( x = 1.75" \) and \( x = 0.25" \) respectively. Later on the flow accelerates along the inner wall and decelerates along the outer wall. This pattern is also reinforced by the secondary flow velocity development, which tends to transfer the slower moving flow towards the inner wall as can be seen by comparing the profiles at \( x = 1.75" \) in Figures 6 and 7.

The profiles of the normalized secondary velocity in the radial direction \( (V_r/u_0) \) are presented at seven concentric cylindrical surfaces which are equally spaced at 0.25" between the inner and outer curved walls of the duct. Figure 8 combines all the profiles at sections C, D, and B, as these did not indicate any significant secondary velocities at section E. On the other hand, the profiles for the normalized secondary velocity for the normalized secondary velocity in the vertical direction \( (V_z/u_0) \) are presented at 15° parallel planes which are equally spaced at 0.25" between the duct plane walls as shown in Fig. 9. The same symbols which were used for the through flow velocity profiles at sections C, D, E and B are also maintained in presenting the measured secondary velocities in Figures 8 and 9. The vertical component of the secondary velocity, \( V_z \), was not measured at section D-D, due to the deterioration of the quality of the duct curved outer wall after obtaining the measurements at the other sections. The maximum secondary velocity components were measured at section E-E, due to the lower wall, and \( 0.128 \) in the vertically upward direction near the duct upper wall.

The two measured secondary velocity components were combined to produce the secondary flow patterns at sections C, D and E, which are shown in Figures 10 and 11, respectively. From these figures the development of the passage vortex due to the nearly linear shear inlet flow profile can be observed throughout the duct cross-sections. The secondary velocities associated with this vortex tend to move the slower flow in the lower duct sections towards the inner curved wall and the faster flow in the upper duct sections, towards the outer curved wall. One can see from Fig. 11 that, the center of the passage vortex at the 45° turning angle is nearly in the middle between the inner and outer curved walls but closer to the duct upper wall. A close examination of Figures 10 and 11 reveals that the center of the passage vortex moves towards the duct lower wall and also in the outward radial direction as the flow turning angle increases from 45° to 90° between sections C and B.

**CONCLUSIONS**

LDV measurements were presented for the three velocity components of the flow in a rectangular curved duct with shear inlet velocity profiles. Secondary velocities of magnitudes greater than 25% of the main velocity, were measured after the 90° flow turning angle. The results demonstrate the passage vortex development throughout the duct cross-sections with the flow turning angles. These experimental results can therefore be used to validate both viscous and inviscid codes for internal three dimensional rotational flow fields.

**ACKNOWLEDGEMENT**

This research was sponsored by the U.S. Air Force Office of Scientific Research under Grant No. 80-0242.
REFERENCES


FIG. 1. SCHEMATIC SHOWING THE EXPERIMENTAL SET-UP.
Fig. 4. Schematic of the flow tunnel showing location of shear velocity generator and measuring stations.

Fig. 6. The shear velocity generator and coordinate system.
FIG. 4. NORMALIZED THROUGH FLOW VELOCITY AT SECTION B-B

FIG. 5. NORMALIZED THROUGH FLOW VELOCITY AT SECTION C-C
FIG. 6. NORMALIZED THROUGH FLOW VELOCITY AT SECTION D-D

FIG. 7. NORMALIZED THROUGH FLOW VELOCITY AT SECTION E-E
FIG. 8. NORMALIZED SECONDARY FLOW VELOCITY COMPONENT IN RADIAL DIRECTION

FIG. 9. NORMALIZED SECONDARY FLOW VELOCITY COMPONENT IN THE VERTICAL DIRECTION
Section C-C

Section E-E

FIG. 10. SECONDARY FLOW VELOCITY AT SECTION C-C

FIG. 11. SECONDARY FLOW VELOCITY AT SECTION E-E
END

FILMED

1-85

DTIC