ANALYSIS OF THREE-DIMENSIONAL VISCOUS INTERNAL FLOWS

K.N. GHIA
AND
U. GHIA

This research was supported by the Air Force Office of Scientific Research, under AFOSR-80-0160.

Distribution of this report is unlimited.

August 1982
ANALYSIS OF THREE-DIMENSIONAL VISCOSOUS INTERNAL FLOWS

K.N. GHIA*

AND

U. GHIA**

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

This research was supported by the Air Force Office
of Scientific Research, Bolling Air Force Base,
under AFOSR Grant No. 80-0160 with Dr. James D. Wilson
as Technical Monitor.

Distribution of this report is unlimited.

* Professor
** Research Associate Professor
ANALYSIS OF THREE-DIMENSIONAL VISCOUS INTERNAL FLOWS

Kirti N. Ghia and Urmila Ghia

Dept. of Aerospace Engineering & Applied Mechanics
University of Cincinnati
Cincinnati, Ohio 45221

Air Force Office of Scientific Research/NA
Building 410
Bolling Air Force Base, D.C. 20332

This report describes the technical progress achieved in research sponsored by the Air Force Office of Scientific Research during the period between March 1981 and February 1982. Three areas of research were pursued; two of these consist of (1) analysis of laminar and turbulent duct flows, and (2) study of laminar and turbulent separated flows. Both of these studies were aimed at acquiring a better understanding of isolated physical phenomena significant to turbomachinery applications via the use of appropriate model problems. The
third area of research pursued consisted of (7) the analysis of numerical methods with the goal of improving the efficiency and accuracy of the various methods developed and implemented. In the first area of research, fine-grid asymptotic solutions were obtained for laminar flow through curved ducts of simple cross sections; also marching solutions have been obtained for turbulent flow in the entrance region of curved ducts of simple cross sections. The subject of streamwise separation is examined using the laminar flow through a constricted asymmetric channel and the laminar and turbulent flows past a thick blunt plate as the model problems. In the third category, high-Re very fine-grid solutions have been provided for the shear-driven cavity problem using a multi-grid strongly implicit method. Finally, the block Gaussian elimination method is implemented to solve the unsteady Navier-Stokes equations to provide true transient internal viscous flow solutions.
ABSTRACT

This report describes the technical progress achieved in research sponsored by the Air Force Office of Scientific Research during the period between March 1981 and February 1982.

Three areas of research were pursued; two of these consist of (1) analysis of laminar and turbulent duct flows, and (2) study of laminar and turbulent separated flows. Both of these studies were aimed at acquiring a better understanding of isolated physical phenomena significant to turbomachinery applications via the use of appropriate model problems. The third area of research pursued consisted of (3) the analysis of numerical methods with the goal of improving the efficiency and accuracy of the various methods developed and implemented.

In the first area of research, fine-grid asymptotic solutions were obtained for laminar flow through curved ducts of simple cross sections; also marching solutions have been obtained for turbulent flow in the entrance region of curved ducts of simple cross sections. The subject of streamwise separation is examined using the laminar flow through a constricted asymmetric channel and the laminar and turbulent flows past a thick blunt plate as the model problems. In the third category, high-Re very fine-grid solutions have been provided for the shear-driven cavity problem using a multi-grid strongly implicit method. Finally, the block Gaussian elimination method is implemented to solve the unsteady Navier-Stokes equations to provide true transient internal viscous flow solutions.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>i</td>
</tr>
<tr>
<td>1 OBJECTIVES</td>
<td>1</td>
</tr>
<tr>
<td>2 DESCRIPTION OF SIGNIFICANT ACCOMPLISHMENTS</td>
<td>3</td>
</tr>
<tr>
<td>Laminar and Turbulent Duct Flows</td>
<td>3</td>
</tr>
<tr>
<td>Laminar and Turbulent Separated Flows</td>
<td>5</td>
</tr>
<tr>
<td>Numerical Methods</td>
<td>7</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>10</td>
</tr>
<tr>
<td>3 JOURNAL PAPERS PUBLISHED AND IN PREPARATION.</td>
<td>12</td>
</tr>
<tr>
<td>4 PROFESSIONAL PERSONNEL</td>
<td>13</td>
</tr>
<tr>
<td>5 SCIENTIFIC INTERACTIONS - SEMINAR AND PAPER PRESENTATIONS</td>
<td>14</td>
</tr>
<tr>
<td>6 TECHNICAL APPLICATIONS</td>
<td>16</td>
</tr>
<tr>
<td>TABLES</td>
<td>17</td>
</tr>
<tr>
<td>FIGURES</td>
<td>19</td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Values for Streamline and Vorticity Contours in Figures 6a and 6b</td>
<td>17</td>
</tr>
<tr>
<td>2</td>
<td>Properties of Primary and Secondary Vortices</td>
<td>18</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>1a</td>
<td>Comparison of Streamwise Velocity Profiles; AR = 1, R = 100</td>
<td>19</td>
</tr>
<tr>
<td>1b</td>
<td>Effect of Dean Number on Streamwise Velocity for Polar Curved Ducts; AR = 1, R = 100</td>
<td>20</td>
</tr>
<tr>
<td>2</td>
<td>Effect of Dean Number on Secondary Flow for Polar Ducts - (A) Cross-Flow Streamline Contours, (B) Cross-Flow Velocity Profiles</td>
<td>21</td>
</tr>
<tr>
<td>3</td>
<td>Effect of Aspect Ratio on Secondary Flow for Polar Ducts; K = 100, R = 100</td>
<td>22</td>
</tr>
<tr>
<td>4a</td>
<td>Steady-State Stream-Function Contours for Re = 1,000, ( \Delta \psi = 0.002 ) within Separation Bubble; ( \Delta \psi = 0.1 ) Elsewhere</td>
<td>23</td>
</tr>
<tr>
<td>4b</td>
<td>Steady-State Vorticity Contours for Re = 1,000, ( \Delta \omega = 2.0 )</td>
<td>24</td>
</tr>
<tr>
<td>5a</td>
<td>Axisymmetric Flow Configuration and Coordinate System</td>
<td>25</td>
</tr>
<tr>
<td>5b</td>
<td>Comparison Between Predicted and Experimental Data for Velocity Profiles for Blunt Plates</td>
<td>26</td>
</tr>
<tr>
<td>6a</td>
<td>Streamline Pattern for Primary, Secondary and Additional Corner Vortices</td>
<td>27</td>
</tr>
<tr>
<td>6b</td>
<td>Vorticity Contours for Flow in Driven Cavity</td>
<td>27</td>
</tr>
</tbody>
</table>
SECTION 1

OBJECTIVES

The objective of the present study was to develop analyses and improved understanding of viscous internal flows in a class of complex three-dimensional configurations related to turbo-machinery, using appropriate model problems.

1. Laminar and turbulent three-dimensional internal viscous flows were to be studied using the model problem of curved ducts with simple cross sections. The asymptotic form of this laminar flow was to be studied to investigate the occurrence of Dean's instability by varying the Dean number, K, and the aspect ratio of the duct cross section. An additional goal of studying this flow configuration was to provide a set of accurate asymptotic flow solutions, so that the marching solutions calculated earlier using the parabolized Navier-Stokes equations could be independently verified.

2. Separated flows were to be studied with two different objectives. For laminar flow, the model problem of a doubly infinite channel with an asymmetric constriction was to be studied to obtain solutions for high-Reynolds number flows. On the other hand, a turbulent separated-flow analysis was to be initiated to further understand the low-Reynolds number turbulence modelling techniques required at the boundaries in the second-order closure formulation of the time-averaged Navier-Stokes equations.
3. The efficiency and accuracy of semi-implicit and implicit methods were to be investigated using model problems. In the first category, the strongly implicit (SI) scheme was to be studied using the model problem of flow in a shear-driven cavity in order to improve its convergence rate. Also, the multi-grid technique was to be employed to further improve the convergence rate of semi-implicit solution techniques. In the latter category, a fully implicit scheme was to be developed for enabling direct solution of the Poisson equation in generalized orthogonal curvilinear coordinates.

The research performed in each of these areas is described briefly in the next section; the main results and conclusions obtained are also summarized.
SECTION 2

DESCRIPTION OF SIGNIFICANT ACCOMPLISHMENTS

All three areas of research proposed were initiated and the specific achievements made in these studies during this reporting period are briefly described in the following subsections.

2.1 Laminar and Turbulent Duct Flows

Asymptotic flows inside curved ducts of rectangular as well as polar cross section were analyzed using the Navier-Stokes equations in terms of the axial velocity, the axial vorticity and the cross-flow stream function. One of the objectives of this study was to investigate, for the polar duct, the possibility of Dean's instability which, for curved rectangular ducts, is characterized by the occurrence of an additional pair of secondary-flow vortices. The significance of this phenomenon is that this second pair of streamwise vortices creates additional pressure losses. To achieve this goal, the asymptotic form of the flow equations was used to calculate some benchmark solutions which serve as the only available quantitative check on the accuracy of the developing-flow numerical solutions for this class of flow problems. From the investigators' earlier work on this problem, it was felt that, for highly curved configurations, the strong coupling between the primary and the secondary flow should have to be honored by the numerical solution technique employed. In fact, Ghia et al. (1980) had observed, using an alternating-direction
implicit (ADI) method, that the simultaneous solution of the three differential equations governing the primary and the secondary flows in these highly curved configurations was essential for computational efficiency. Also, the initial conditions employed were observed to have a significant influence on the stability of the numerical scheme, particularly for cases with high Dean number \( K \).

The strongly implicit (SI) scheme was developed to facilitate efficient high-Dean-number solutions of the coupled flow equations. Numerical experiments were conducted with all three governing equations being solved simultaneously as well as with the axial velocity equation being solved sequentially with a coupled solution of the vorticity and stream function equations. From the limited experiments conducted, it was observed that the coupling of all three equations was not crucial for the SI scheme; this was contrary to the findings for the ADI method. Further, a combined multi-grid–strongly implicit (MG-SI) scheme was also developed; brief remarks about this method will be made later in this section. Using this new method, fine-grid results were obtained by K. Ghia, U. Ghia and Shin (1981). Figures 1-3 show some of their typical results. The details of these results are given in a technical paper which was presented at the ASME Winter Annual Meeting, November 1981, in Washington, D.C. Additional details of the MG-SI method were also prepared and added to this study and the revised manuscript prepared has been submitted for journal publication.
A technical paper based on the analysis developed and the results obtained for turbulent flow inside curved ducts of regular cross section was completed by Goyal, K. Ghia and U. Ghia (1982) and has been submitted for journal publication. The wall-function approach and the low-Reynolds number modelling approach are carefully evaluated therein.

2.2 Laminar and Turbulent Separated Flow

Laminar incompressible flow with streamwise separation was studied further with the help of the model problem of a doubly infinite channel with an asymmetric constriction. The use of a semi-implicit method, such as the alternating-direction implicit (ADI) method, leads to poor convergence for this flow, particularly as the grid size is refined. To circumvent this difficulty, the ADI method for the vorticity-transport equation was coupled with the block-Gaussian elimination (BGE) method for the stream-function equation, to obtain accurate and efficient solutions for the channel flow problem. Since the analysis used the derived variables, namely, the vorticity \( \omega \), and the stream function \( \psi \), it also provided an independent check for the results of U. Ghia et al. (1979b) which had been obtained using primitive variables. The present results for separated flow agreed well with those of U. Ghia et al. (1979b) for \( \text{Re} = 100 \). However, for a mildly separated flow configuration with \( \text{Re} = 1000 \), the present results show a very different internal structure. The results of the present true transient analysis, obtained using the ADI-BGE method, appeared to be
converging to steady-state values. These results for $Re = 1000$ are shown in Fig. 4. The streamline contours in Fig. 4a show four discrete eddy structures, all rotating in the clockwise sense, a transient characteristic of unstable flows. This stable series of vortices formed on the lee side of the channel constriction shows a qualitative resemblance with an unstable series of like-rotating vortex structures in the separating boundary layer on the tail end of a blunt body shown by Prandtl and Tietjens (1934). The corresponding vorticity contours for the $Re = 1000$ flow configuration are shown in Fig. 4b. A careful reexamination of the results in the transformed plane revealed that the grid used is not adequate between the reattachment point and downstream infinity. Preliminary results have been obtained using a modified grid where the discrete eddies combine to a single eddy, a characteristic of steady flow in the constricted asymmetric channel.

A turbulent separated-flow analysis was developed for flow past a class of two-dimensional and axisymmetric blunt bodies as shown in Fig. 5a. The time-averaged Navier-Stokes equations for these flows were derived in surface-oriented conformal coordinates $(\xi, \eta)$ in terms of similarity-type vorticity and stream-function variables. Turbulence closure was achieved by means of a two-equation ($k, \varepsilon$) turbulence model which enables determination of the isotropic eddy viscosity $v_t$. The coupled vorticity and stream-function equations were solved simultaneously using an incremental formulation of the factored ADI scheme. Numerical solutions were obtained for a thin flat
plate and compared with available experimental and analytical data. Also, results were obtained for flow over a parabola and compared with the flat-plate results. Finally, solutions were obtained for flow past a two-dimensional semi-infinite body with a shoulder, at $Re_d = 24,000$. Typical mean-flow velocity profiles for the blunt plate were compared with the experimental data of Ota and Itasaka (1976) and are shown in Fig. 5b. All of the computed results have the same general trend as the experimental data of Ota and Narita (1978); possible causes for the differences within the separated-flow region were carefully examined. Some of these results were presented briefly by Abdelhalim, U. Ghia and K. Ghia (1982) at an AIAA Mini-Symposium at the Air Force Institute of Technology, Ohio. A full-length paper based on these results has been prepared by Abdelhalim, U. Ghia and K. Ghia (1982) for presentation at an ASME meeting.

2.3 **Numerical Methods**

A number of isolated effects were studied using model problems so as to maintain the accuracy and efficiency of the algorithms developed. This approach not only facilitates the assessment of various algorithms, but also provides benchmark solutions for some of the model problems used. With an eventual goal of solving flow through complex turbomachinery passages, every effort is made, during each grant period, to
improve the numerical methods already used and to develop new methods which can further enhance the convergence rate. The solution convergence rate is strongly dependent on many problem parameters, such as the Reynolds number, the mesh size and the total number of computational points. This led to carefully examining the recently emerging multi-grid (MG) technique as a useful means for enhancing the convergence rate of iterative numerical methods for solving discretized equations at a number of computational grid points so large as to be considered impractical previously.

The vorticity-stream function formulation of the two-dimensional incompressible Navier-Stokes equations was used to study the effectiveness of the coupled strongly-implicit multi-grid (CSI-MG) method in the determination of high-Re fine-mesh flow solutions. The driven flow in a square cavity was used as the model problem. Solutions were obtained for configurations with Reynolds number as high as 10,000 and meshes consisting of as many as (257 x 257) points. Figure 6a shows the streamline contours for the cavity-flow configuration with Re = 10,000. A magnified view of the various secondary vortices is also included. The values of ψ along the contours shown are listed in Table 1. For this case of Re = 10,000, the present results are in excellent agreement with those reported by Keller (1981), and are computationally very efficient. Figure 6b shows the corresponding vorticity contours with the values of ω along these contours listed in Table 1. This figure shows
that, in addition to the boundary layers at the walls, free-
shear layers with high vorticity gradients appear in the
interior of the cavity in a very complex manner. It was because
of this complex flow structure that uniform mesh refinement
was used in the present study. Earlier, some of these results
were presented in a paper by U. Ghia, K. Ghia and Shin (1981)
at a Multi-Grid Symposium held at NASA-Ames Research Center.
The original manuscript was revised to include additional results
for the convergence history of the MG-SI solution procedure.
In the revised form, the paper has been accepted for publication
in Journal of Computational Physics.

Towards the same goal of efficient numerical methods, the
two-dimensional unsteady Navier-Stokes equations, in terms of
vorticity and stream function, and generalized orthogonal
coordinates, were used to analyze a fully implicit scheme
developed for the general Poisson equation in this study. The
vorticity-transport equation was solved using an ADI method,
whereas the Dirichlet Poisson problem for the stream function was
solved using a direct block Gaussian elimination (BGE) method.
The BGE method was compared with the semi-direct (SD) method of
Martin (1978) for the general Poisson problem for accuracy and
efficiency and was found to yield a direct one-step solution,
irregardless of the degree of grid clustering, with considerably
improved efficiency as compared to the SD method. Osswald and
K. Ghia (1981) presented detailed results of this study at a
Multi-Grid Symposium held at NASA-Ames Research Center. The
manuscript is being revised to include accurate fine-grid results for the Re = 1000 flow configuration. The revised paper will be submitted shortly for journal publication.

REFERENCES


SECTION 3

JOURNAL PAPERS PUBLISHED AND IN PREPARATION


SECTION 4

PROFESSIONAL PERSONNEL

The principal investigators for the research reported herein were Professors K.N. Ghia and U. Ghia, of the Department of Aerospace Engineering and Applied Mechanics, University of Cincinnati. They were assisted, periodically, by Mr. A.A. Abdelhalim, Mr. G.A. Osswald and Mr. C.T. Shin, graduate students pursuing their advanced degrees in the same Department. Drs. A.G. Mikhail and R.K. Goyal, formerly graduate students in the Aerospace Engineering and Applied Mechanics Department, contributed by aiding in the preparation and presentation of technical papers based on their Ph.D. dissertations completed earlier.
SECTION 5

SCIENTIFIC INTERACTIONS - SEMINAR AND PAPER PRESENTATIONS

Invited Lectures


Contributed Papers


14


SECTION 6

TECHNICAL APPLICATIONS

Of the various areas of research pursued, two appear to be most useful to the technical community. The multi-grid solution procedure formulated for the Navier-Stokes equations for determining fine-grid results for high-Re flows is a unique capability developed in the present research. This is particularly useful because, although the multi-grid procedure is generally recognized as beneficial for accelerating convergence, its adaptation to the solution of high-Re viscous flows has been extremely limited thus far. Secondly, the unsteady-flow solution procedure using time marching and block-Gaussian elimination yields useful information about transient separated internal flows. Both of these developments provide highly accurate benchmark solutions for the problems to which these have been applied so far. Both of these programs are developed in modular form, and several of the modules are prepared for general-purpose use and can be easily implemented in other applications. Some other researchers have already requested for some of these modules.
<table>
<thead>
<tr>
<th>Contour Letter</th>
<th>Value of $\psi$</th>
<th>Contour Number</th>
<th>Value of $\psi$</th>
<th>Contour Number</th>
<th>Value of $\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>$-1.0 \times 10^{-10}$</td>
<td>0</td>
<td>$1.0 \times 10^{-8}$</td>
<td>0</td>
<td>0.0</td>
</tr>
<tr>
<td>b</td>
<td>$-1.0 \times 10^{-7}$</td>
<td>1</td>
<td>$1.0 \times 10^{-7}$</td>
<td>1</td>
<td>$\pm 0.5$</td>
</tr>
<tr>
<td>c</td>
<td>$-1.0 \times 10^{-5}$</td>
<td>2</td>
<td>$1.0 \times 10^{-6}$</td>
<td>2</td>
<td>$\pm 1.0$</td>
</tr>
<tr>
<td>d</td>
<td>$-1.0 \times 10^{-4}$</td>
<td>3</td>
<td>$1.0 \times 10^{-5}$</td>
<td>3</td>
<td>$\pm 2.0$</td>
</tr>
<tr>
<td>e</td>
<td>$-0.0100$</td>
<td>4</td>
<td>$5.0 \times 10^{-5}$</td>
<td>4</td>
<td>$\pm 3.0$</td>
</tr>
<tr>
<td>f</td>
<td>$-0.0300$</td>
<td>5</td>
<td>$1.0 \times 10^{-4}$</td>
<td>5</td>
<td>4.0</td>
</tr>
<tr>
<td>g</td>
<td>$-0.0500$</td>
<td>6</td>
<td>$2.5 \times 10^{-4}$</td>
<td>6</td>
<td>5.0</td>
</tr>
<tr>
<td>h</td>
<td>$-0.0700$</td>
<td>7</td>
<td>$5.0 \times 10^{-4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>i</td>
<td>$-0.0900$</td>
<td>8</td>
<td>$1.0 \times 10^{-3}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>j</td>
<td>$-0.1000$</td>
<td>9</td>
<td>$1.5 \times 10^{-3}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>k</td>
<td>$-0.1100$</td>
<td>10</td>
<td>$3.0 \times 10^{-3}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>l</td>
<td>$-0.1150$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>m</td>
<td>$-0.1175$</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>No.</td>
<td>Property</td>
<td>Re</td>
<td>100</td>
<td>400</td>
<td>1000</td>
</tr>
<tr>
<td>-----</td>
<td>----------</td>
<td>-----</td>
<td>------</td>
<td>------</td>
<td>------</td>
</tr>
<tr>
<td>Primary</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{min}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{max}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( H_L )</td>
<td>( V_L )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1st</td>
<td>BL</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{min}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{max}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( H_L )</td>
<td>( V_L )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2nd</td>
<td>BL</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{min}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{max}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( H_L )</td>
<td>( V_L )</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3rd</td>
<td>SH</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{min}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \psi_{\text{max}} )</td>
<td>( \omega_{\text{v.c.}} )</td>
<td>Location, x, y</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>( H_L )</td>
<td>( V_L )</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Mark Units**: 18.35, 18.08, 31.56, 78.25, 70.8125, 68.50

**CPU Seconds**: 55.59, 215.05, 92.27, 207.26, 736.49, 705.62

**Map Order**: 129, 247, 129, 129, 257, 247
FIG. 1a. COMPARISON OF STREAMWISE VELOCITY PROFILES; AR = 1, R = 100.
FIG. 1b. EFFECT OF DEAN NUMBER ON STREAMWISE VELOCITY FOR SQUARE CURVED DUCTS; AR = 1, R = 100.
Fig. 2. Effect of Dean Number on Secondary Flow for Polar Ducts -
(A) Cross Flow Streamline Contours, (B) Cross Flow Velocity Profiles.
FIG. 3. EFFECT OF ASPECT RATIO ON SECONDARY FLOW FOR POLAR DUCTS; \( K = 100, R = 100 \).
FIG. 4a. STREAM FUNCTION CONTOURS FOR Re = 1,000.

$\Delta \psi = 0.002$ WITHIN SEPARATION BUBBLE; $\Delta \psi = 0.1$ ELSEWHERE.
CONTOUR PLOT
VORTICITY
RE = 1000
(85, 33) MESH
A² = 0.2100
H = 0.7620
SR = 1.3281
D = 0.4097
TIME = 44.0000

FIG. 4b. VORTICITY CONTOURS FOR Re = 1,000. Δω = 2.0.
FIG. 5a. AXISSYMMETRIC FLOW CONFIGURATION AND COORDINATE SYSTEM
FIG. 6a. STREAMLINE PATTERN FOR PRIMARY, SECONDARY AND ADDITIONAL CORNER VORTICES.

FIG. 6b. VORTICITY CONTOURS FOR FLOW IN DRIVEN CAVITY.