ANALYSIS OF THREE-DIMENSIONAL VISCOUS INTERNAL FLOWS

Approved for public release; distribution unlimited.

K.N. GHIA*
Department of Aerospace Engineering and Engineering Mechanics

and

U. GHIA*
Department of Mechanical and Industrial Engineering

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.
A five-year multi-tasked research project was pursued by the present investigators to study complex viscous internal flows under AFOSR sponsorship between 1980-1985. The major objective of this study was to acquire improved understanding of viscous internal flows related to turbomachinery components by analyzing appropriate model flow problems. In the process of achieving this objective, significant effort was directed towards developing basic computational methods which were made available to interested researchers involved in computational fluid dynamics (CFD) research and over -
19. (Continued)

Block Gaussian Elimination Method.

20. (Continued)

to users involved in the design of turbomachinery components. Several analyses were developed and include an asymptotic analysis for the fully developed three-dimensional flow in curved ducts, a parabolized Navier-Stokes analysis for developing flow in curved ducts, an unsteady Navier-Stokes analysis for internal and external flows, adaptive grid generation for one- and two-dimensional viscous flows, analysis of the Neumann problem in generalized orthogonal coordinates, efficient semi-implicit solution techniques consisting of the alternating-direction implicit multi-grid (ADI-MG) and strongly implicit multi-grid (SI-MG) methods, the direct block Gaussian elimination (BGE) method for solution of the Poisson equation in generalized orthogonal coordinates and the ADI-BGE and SI-BGE methods for the unsteady Navier-Stokes analysis of incompressible flows. For the flow inside a shear-driven cavity, the asymptotic flow in curved ducts and the flow in doubly infinite backstep channel, the predicted results provided clarity for interpretation of the available corresponding experimental results and have now become benchmark solutions for these problems. The adaptive-grid generation procedure developed is unique and effectively treats multiple critical regions of high gradients typical of nonlinearities such as boundary layers, shear layers, shock waves, etc. For the incompressible Navier-Stokes equations, the coupled ADI-MG and SI-MG semi-implicit analyses provide the most efficient steady-state methods available in the literature; this is also true for the ADI-BGE and SI-BGE fully-implicit methods for solution of the unsteady Navier-Stokes equations.
ABSTRACT

A five-year multi-tasked research project was pursued by the present investigators to study complex viscous internal flows under AFOSR sponsorship between 1980-1985. The major objective of this study was to acquire improved understanding of viscous internal flows related to turbomachinery components by analyzing appropriate model flow problems. In the process of achieving this objective, significant effort was directed towards developing basic computational methods which were made available to interested researchers involved in computational fluid dynamics (CFD) research and to users involved in the design of turbomachinery components. Several analyses were developed and include an asymptotic analysis for the fully developed three-dimensional flow in curved ducts, a parabolized Navier-Stokes analysis for developing flow in curved ducts, an unsteady Navier-Stokes analysis for internal and external flows, adaptive grid generation for one- and two-dimensional viscous flows, analysis of the Neumann problem in generalized orthogonal coordinates, efficient semi-implicit solution techniques consisting of the alternating-direction implicit multi-grid (ADI-MG) and strongly implicit multi-grid (SI-MG) methods, the direct block Gaussian elimination (BGE) method for solution of the Poisson equation in generalized orthogonal coordinates and the ADI-BGE and SI-BGE methods for the unsteady Navier-Stokes analysis of incompressible flows. For the flow inside a shear-driven cavity, the asymptotic flow in curved ducts and the flow in doubly-infinite backstep channel, the predicted results provided clarity for interpretation of the available corresponding experimental results and have now become benchmark solutions for these problems. The adaptive-grid generation procedure developed is unique and effectively treats multiple critical regions of high gradients typical of nonlinearities such as boundary layers, shear layers, shock waves, etc. For the incompressible Navier-Stokes equations, the coupled ADI-MG and SI-MG semi-implicit analyses provide the most efficient steady-state methods available in the literature; this is also true for the ADI-BGE and SI-BGE fully-implicit methods for solution of the unsteady Navier-Stokes equations.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>1</td>
</tr>
<tr>
<td>1 OBJECTIVES</td>
<td>1</td>
</tr>
<tr>
<td>2 DESCRIPTION OF SIGNIFICANT ACCOMPLISHMENTS</td>
<td>4</td>
</tr>
<tr>
<td>2.1 Viscous Flow in Curved Ducts: Examination of Asymptotic Flow and Turbulence Models</td>
<td>4</td>
</tr>
<tr>
<td>2.2 Two-Dimensional and Axisymmetric Separation for Steady Flow</td>
<td>7</td>
</tr>
<tr>
<td>2.3 Two-Dimensional and Axisymmetric Separation for Unsteady Flow</td>
<td>9</td>
</tr>
<tr>
<td>2.4 Numerical Grid Generation</td>
<td>11</td>
</tr>
<tr>
<td>2.5 Semi-Implicit Numerical Methods:</td>
<td>12</td>
</tr>
<tr>
<td>Accuracy and Efficiency</td>
<td></td>
</tr>
<tr>
<td>2.6 Fully-Implicit Numerical Methods:</td>
<td>15</td>
</tr>
<tr>
<td>Semi-Direct and Direct Methods</td>
<td></td>
</tr>
<tr>
<td>REFERENCES</td>
<td>18</td>
</tr>
<tr>
<td>3 JOURNAL PAPERS PUBLISHED AND IN PREPARATION</td>
<td>22</td>
</tr>
<tr>
<td>4 SCIENTIFIC INTERACTIONS - SEMINAR AND PAPER PRESENTATIONS</td>
<td>25</td>
</tr>
<tr>
<td>5 STUDENT DEGREE THESES AND DISSERTATIONS</td>
<td>29</td>
</tr>
<tr>
<td>6 TECHNICAL APPLICATIONS</td>
<td>30</td>
</tr>
<tr>
<td>TABLE</td>
<td>31</td>
</tr>
<tr>
<td>FIGURES</td>
<td>32</td>
</tr>
<tr>
<td>Table</td>
<td>Page</td>
</tr>
<tr>
<td>-------</td>
<td>------</td>
</tr>
<tr>
<td>1</td>
<td>31</td>
</tr>
<tr>
<td>Comparison of Subroutines for the 2-D Dirichlet Poisson Problem, Following Schumann (1977)</td>
<td></td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1a</td>
<td>Curved-Duct Geometry and Coordinate System</td>
<td>32</td>
</tr>
<tr>
<td>1b</td>
<td>Effect of Dean Number on Secondary Flow for Square Ducts - Cross-Flow Streamline Contours</td>
<td>33</td>
</tr>
<tr>
<td>1c</td>
<td>Effect of Dean Number on Secondary Flow for Polar Ducts - Cross-Flow Streamline Contours</td>
<td>34</td>
</tr>
<tr>
<td>2a</td>
<td>Comparative Study of Isotropic and Anisotropic Turbulent Models</td>
<td>35</td>
</tr>
<tr>
<td>2b</td>
<td>Effect of Mass Transfer on Laminar and Turbulent Velocity Profiles</td>
<td>36</td>
</tr>
<tr>
<td>3</td>
<td>Stream-Function Contours for Constricted Channel Flow. Re = 1000, ( \Delta \psi = 0.002 ) Within Separation Bubble; ( \Delta \psi = 0.1 ) Otherwise</td>
<td>37</td>
</tr>
<tr>
<td>a.</td>
<td>Steady-State Solution (at ( T=45 )) with Inappropriate Mesh.</td>
<td></td>
</tr>
<tr>
<td>b(i)</td>
<td>Transient Solution (at ( T=8 )) with Improved Mesh.</td>
<td></td>
</tr>
<tr>
<td>b(ii)</td>
<td>Steady-State Solution (at ( T=41 )) with Improved Mesh.</td>
<td></td>
</tr>
<tr>
<td>4a</td>
<td>Comparison of Measured and Computed Recirculation Lengths for Blunt Plates and Longitudinal Cylinders</td>
<td>38</td>
</tr>
<tr>
<td>4b</td>
<td>Comparison Between Predicted and Experimental Variation of Turbulent Kinetic Energy Within Reduced Separation Bubble for Blunt Plates</td>
<td>39</td>
</tr>
<tr>
<td>5a</td>
<td>Similarity Study of Primary Reattachment Length</td>
<td>40</td>
</tr>
<tr>
<td>5b</td>
<td>Secondary Separation and Reattachment Length</td>
<td>40</td>
</tr>
<tr>
<td>5c</td>
<td>Separation and Reattachment Lengths After Ref. [2]</td>
<td>40</td>
</tr>
<tr>
<td>5d</td>
<td>Comparison of Primary Reattachment Length in Transition Regime</td>
<td>40</td>
</tr>
<tr>
<td>6a</td>
<td>Stream-Function Contours for Unsteady Flow in Transition Regime; Re = 2000, ( \Delta \psi = 0.0536 ) in Bubbles</td>
<td>41</td>
</tr>
<tr>
<td>6b</td>
<td>Time-Averaged Stream-Function Contours for ( 95.02 \leq T \leq 117 ); Re = 2000, ( \Delta \psi = 0.1 )</td>
<td>41</td>
</tr>
<tr>
<td>6c</td>
<td>Time-Averaged Vorticity Contours for ( 95.02 \leq T \leq 117 ); Re=2000, ( \Delta \omega = 2.0 )</td>
<td>41</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>Transient Flow Through Orifice Pipe; $Re_D = 1000$; $B = 0.6$</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>(a) Streamfunction Contours; $\Delta \psi = 0.05$</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>(b) Vorticity Contours; $\Delta \omega = 5.0$</td>
<td>42</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>43</td>
<td></td>
</tr>
<tr>
<td>Limit-Cycle Analysis: $Re = 1,000$, $\alpha = 15^\circ$</td>
<td>43</td>
<td></td>
</tr>
<tr>
<td>9a</td>
<td>44</td>
<td></td>
</tr>
<tr>
<td>Grid-Point Paths for 1-D Nonlinear Viscous Burgers' Equation. (i) Adaptive Grid; (ii) Adaptive Grid (Modified)</td>
<td>44</td>
<td></td>
</tr>
<tr>
<td>9b</td>
<td>45</td>
<td></td>
</tr>
<tr>
<td>Solution of 1-D Viscous Burgers' Equation, With $Re$ Ranging from 1 to 10,000</td>
<td>45</td>
<td></td>
</tr>
<tr>
<td>9c</td>
<td>46</td>
<td></td>
</tr>
<tr>
<td>Solution of 1-D Viscous Internal Flow Model Problem, With $Re = 1$ to $Re = 10,000$</td>
<td>46</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>47</td>
<td></td>
</tr>
<tr>
<td>Convergence History of Multigrid Method for Neumann Pressure Problem in Developing Flow Through Curved Polar Duct With Grid Clustering</td>
<td>47</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>48</td>
<td></td>
</tr>
<tr>
<td>Flow in Shear Driven Cavity</td>
<td>48</td>
<td></td>
</tr>
<tr>
<td>(a) Streamline Pattern for Primary, Secondary and Additional Corner Vortices in Shear-Driven Cavity Flow, $Re = 10000$</td>
<td>48</td>
<td></td>
</tr>
<tr>
<td>(b) Vertical Extent of Secondary Downstream Eddy - Comparison Between Computations and Measurements</td>
<td>48</td>
<td></td>
</tr>
</tbody>
</table>
SECTION 1

OBJECTIVES

The development of computational fluid dynamics (CFD) analyses for complex viscous interacting flows was pursued by the present investigators under AFOSR sponsorship during March 1980 - February 1985. The broad objective was to gain a better understanding of the basic fluid flow phenomena present in complex multi-dimensional internal viscous flows. To realize this objective, the approach used was to study, initially, isolated problem areas and flow features through the simulation of model flow problems. Subsequently, as the necessary information was either generated by the present investigators or made available through the efforts of other researchers, more complex model flow problems were treated to better understand some of the dominant flow features in turbomachinery type of flow fields and, eventually, develop an analysis to solve the three-dimensional flow in rotating blade passages.

The major research thrust was directed in analyzing the dominant features of the flow fields in diffusers, ducts, blade passages, etc. Significant effort was directed in carefully analyzing

- Geometrical Complexities and Three-Dimensionality,
- Secondary Recirculating Flows and Streamwise Separation,
- Unsteadiness,

whereas some effort was also made in better understanding

- Turbulence and Compressibility.

Several major CFD analyses were initiated to study these flow features; these are grouped into six different categories and are briefly outlined here.

1. For a class of three-dimensional viscous flow problems with a predominant flow direction and no streamwise separation, asymptotic flow fields were to be determined using the derived
variables vorticity and velocity \((\omega, \vec{V})\). This totally independent formulation was to be used to verify, at least in part, the significant amount of developing-flow predictions determined earlier. Also, a systematic study of Dean's instability was to be carried out for curved square ducts.

Further, for the curved duct flow configuration with low-speed flow at relatively high-Reynolds number, turbulence was to be carefully modelled by the two-equation \((k-\epsilon)\) differential model. The Wall-Function (WF) approach as well as the Low Reynolds-number Modelling (LRM) approach were to be studied carefully to determine which of the two treatments of the wall-region lead to more accurate and efficient solutions. The \((k-\epsilon)\) model itself needed further study in order to modify its usage with flow configurations encountering secondary flows of the second kind, namely, those generated by the turbulence itself; a corner-flow configuration was to be used as the model flow problem for this purpose.

ii. Two-dimensional viscous separation encountered in steady flows with adverse pressure gradient was to be studied using both the Navier-Stokes (NS) equations as well as the approximate NS equations referred to as the semi-elliptic set of equations, using primitive variables. An asymmetric channel with a constriction, as well as a thick plate, were used as model problems. Both laminar as well as turbulent flow separation were to be investigated. An improved semi-elliptic analysis was to be developed, with appropriate treatment of the inflow and outflow boundary conditions.

iii. The effect of unsteadiness in the flow fields was to be studied using an unsteady Navier-Stokes analysis. This analysis was to use derived variables, namely, vorticity \(\omega\) and stream function \(\psi\), and was to be applied to both two-dimensional and axisymmetric geometries such as the doubly-infinite backstep channel, channel with sudden expansion, pipe with sudden expansion, center-body combustor geometry, pipe orifice, etc.

iv. Surface-oriented coordinates were to be used so as to permit approximations to the NS equations, if needed. Analytically as well as numerically generated grids were to be developed, depending on the specific body shape in question. An adaptive grid-generation analysis was to be developed for treating multiple critical regions. The adaptive grid procedure to be developed was to have been applied to model configurations.

v. Accuracy and efficiency of the numerical analysis developed was to be continually improved by careful examination of both the continuous and discretized problems. Boundary and initial conditions were to be developed so the mathematical problems are well posed and these conditions were to be implemented correctly. Higher-order spline methodology was to be developed for improving the solution accuracy. The efficiency of the numerical solutions was improved by the use of semi-implicit methods with increasing
degree of implicitness such as alternating direction implicit (ADI) method and strongly implicit (SI) procedure. For enhancing solution convergence, the multi-grid (MG) technique was to be coupled with the existing semi-implicit methods for two-dimensional flows. The various operators of the multi-grid methods were to be carefully studied so as to lead to an efficient solution procedure. The multi-grid method was to be implemented in a three-dimensional PNS analysis. The solution convergence was also to be enhanced by use of a local time-stepping method.

vi. Direct solvers available for solving the Poisson equation in uniform Cartesian coordinates were to be extended for use with clustered curvilinear orthogonal coordinates. Cauchy-Riemann solvers were to be developed for the solution of Poisson equations in rectangular and generalized curvilinear coordinates. The block Gaussian elimination (BGE) method was to be developed for the solution of the Poisson equation in generalized orthogonal coordinates.
SECTION 2
DESCRIPTION OF SIGNIFICANT ACCOMPLISHMENTS

All of the areas of research initiated and the progress as well as the specific achievements made in these studies during the five-year grant period are briefly summarized in the following subsections.

2.1 Viscous Flow in Curved Ducts: Examination of Asymptotic Flow and Turbulence Models

A generalized formulation was developed to study three-dimensional asymptotic viscous flow in curved ducts using the streamwise vorticity $\zeta$, the streamwise velocity $w$ and the cross-flow velocities in terms of a cross-flow stream function $\psi$. The analysis readily permits the consideration of straight as well as curved ducts of rectangular as well as polar cross sections, as shown in Fig. 1a. For these flows, the similarity parameter of significance is the Dean number $K$ rather than the Reynolds number $Re$. 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 be honored by the numerical solution technique employed. Consequently, simultaneous numerical solutions of the three second-order coupled partial differential equations (PDE's) governing the flow were obtained using semi-implicit methods. Initially, an ADI method was used; subsequently, the strongly implicit (SI) method was necessary for proper convergence of cases with larger values of $K$. For curved ducts of square cross section, the use of very fine grids revealed that Dean's instability occurs at $K=125$, as shown in Fig. 1b, when an additional pair of secondary vortices first makes its appearance. The PNS analysis of K. Chia and Sokhey (1977) and, later, of U. Chia, K. Chia and Goyal (1979) had
predicted this instability to occur at \( K = 143 \), whereas Cheng et al. (1976) had used the vorticity-velocity variables \((\zeta, w, \psi)\) and predicted this instability to occur at \( K = 202 \). In the latter investigation, it was also predicted that this additional pair of secondary vortices disappears for \( K > 520 \) whereas, in the present study, the additional pair of secondary vortices persists even for \( K = 900 \). The significance of this phenomenon is that this second pair of streamwise vortices creates additional pressure losses. The results for the asymptotic flow in curved ducts of polar cross section are shown in Fig. 1c. The cross-flow streamline contours for \( K = 100, 200 \) and 300 exhibit no symmetry and the primary vortex pair shows that the upper vortex is slightly weaker than the lower vortex. Most of the results for laminar flow obtained earlier using the PNS analysis withstood the test of comparison with these asymptotic results, except for the cross-flow velocities. In general, even the quantitative agreement between the two approaches is good, thereby providing reliance in the PNS marching analysis. The detailed asymptotic results for these flows were given by K. Ghia, U. Ghia and Shin (1981). The fine-grid solutions for this flow problem as well as this model problem itself have become benchmarks, as other investigators are using this flow configuration to validate their results and analyses by comparing with the present investigators' solutions.

The PNS analysis developed by U. Ghia, K. Ghia and Goyal (1979) for laminar flow in curved ducts was extended to turbulent flow configurations. Turbulence closure was achieved by a two-equation \((k, \epsilon)\) differential model. A comparative study of the wall-region treatment by the wall-function (WF) method and the Low-Reynolds number Modeling (LRM) method was carried out by Goyal, K. Ghia and U. Ghia (1980) using a curved circular pipe configuration with \( \text{Re}_D = 25,000 \). Here, \( \text{Re}_D \) denotes Reynolds number based on hydraulic
diameter. The mean flow as well as turbulence quantities were examined in detail but the study was inconclusive due to the following. The form of the law of the wall used did not account for the effects of curvature; on the other hand, the LRM method satisfies the wall-boundary conditions more accurately, but has the disadvantage of being limited to flows with moderate Reynolds number and requiring much finer computational grids. The detailed flow results, showing the effect of the duct aspect ratio and curvature ratio on the overall flow, were given by Goyal, K. Ghia and U. Ghia (1981). Flow results were obtained for curved circular pipes with $Re_D = 25,000$ and $236,000$. For curved rectangular ducts, results were obtained for $Re_D = 0.706 \times 10^6$. All of these results compare well with other existing experimental and numerical results.

The $90^\circ$ axial corner flow configuration, representative of the flow near the junction of a turbomachinery blade with the hub, was analyzed, using a velocity-vorticity formulation, to study the effect of turbulence, compressibility and mass transfer on the developing flow along an axial corner. The governing equations for this quasi-three-dimensional flow were obtained as the limiting equations derived from the general corner-layer equations formulated earlier. These asymptotic equations were solved using a semi-implicit second-order accurate marching scheme. The turbulence was modeled by using a Cebeci-Smith (1974) type two-layer algebraic model in which isotropy is assumed. The turbulence stresses were also modeled using a modified form of the Gessner-Emery (1976) anisotropic model. The skin-friction coefficient obtained using the two different turbulence models is presented in Fig. 2a. The conformity between these two sets of results led to the conclusion that the effect of anisotropy is not significant in the
corner-flow configuration. Figure 2b shows the variations in the streamwise as well as the cross-flow velocities due to suction and injection at the wall for both laminar as well as turbulent flow. Additional results for this flow configuration were presented by Mikhail and Ghia (1981) for a range of Mach numbers between 0 and 2.0, with adiabatic as well as heat-transfer boundary conditions at the corner walls; the effects of suction and injection were also included.

2.2 Two-Dimensional and Axisymmetric Separation For Steady Flow

Steady laminar incompressible separated flow was studied by U. Ghia, K. Ghia, Rubin and Khosla (1981). Their semi-elliptic analysis was formulated using the primitive variables \((\overline{V}, p)\). The model problem was that of a doubly infinite channel with an asymmetric constriction. This approach was demonstrated to be very promising by comparing the results with those of the Navier-Stokes analysis of the authors. Further, Osswald and Ghia (1981) used a totally different formulation using the derived variables, namely, the vorticity \(\omega\) and the stream function \(\psi\), and showed that their results agree with those of U. Ghia et al. (1981). Hence, further work was carried out using the semi-elliptic analysis. However, with a larger region and a finer grid, the numerical method experienced convergence difficulties. In hindsight, it seems that, if the semi-elliptic analysis had been used in conjunction with a staggered grid and a multi-grid procedure, the steady separated flow could have been studied more effectively. Also, it should be pointed out that the present investigators were later able to develop an Interacting Parabolized Navier-Stokes (IPNS) analysis which overcomes these difficulties. This latter analysis was developed originally under NASA sponsorship and, hence, the details are not
presented here. Figure 3 shows the typical stream function contours for separated flow computed using the Navier-Stokes analysis of Osswald and K. Ghia (1981); additional results are also given in this reference. A second model problem, namely, that of flow past a class of two-dimensional blunt bodies was also initiated to study flow separation. The Navier-Stokes equations in conformal coordinates and similarity-type variables were solved, using the approximate factorization scheme of Beam and Warming (1977). The effect of Reynolds number on the length of the recirculation region was studied and satisfactory agreement was obtained with the data of Lane and Loehrke (1980), as shown in Fig. 4a. The detailed flow results, including those for the axisymmetric case, were given by Chia and Abdelhalim (1982). The corresponding turbulent separated flow was studied using second-order closure via the (k-ε) two-equation model. The wall region was treated using the LRM method. Figure 4b shows the distribution of the turbulent kinetic energy obtained from this analysis for a slightly blunt-shouldered body and compared with the measurements of Ota and Narita (1978) for a completely sharp-shouldered thick plate. As expected, the computed separation was milder than the measured one. When the streamwise dimension of the predicted separation bubble was scaled up to match with the measured separation bubble, the computed results within the separated-flow region compared well with the corresponding data, as shown in Fig. 4b. The detailed flow results were given by Abdelhalim, U. Ghia and K. Ghia (1983).
2.3 Two-Dimensional and Axisymmetric Separation for Unsteady Flow

The effect of unsteadiness on the flow field was studied using the model problem of flow over a backstep in a doubly infinite channel. The unsteady Navier-Stokes equations were formulated in terms of vorticity \( \omega \) and stream function \( \psi \). Clustered conformal grids were used and nearly optimum grid distribution was arrived at to attempt to honor the multiple length scales of the separated-flow problem. K. Ghia, Osswald and U. Ghia (1983) gave the detailed flow results and also provided comparisons with the data of Denham and Patrick (1974) as well as Armaly and Durst (1980). Figure 5a shows this comparison for the reattachment length of the primary bubble on the lower wall. At \( \text{Re}_3 = 212 \), the calculations show, for the first time, the appearance of a secondary bubble near the upper wall and, simultaneously, a marked deviation from the data of Armaly and Durst (1980). This information was communicated to those authors and the response received was that spanwise variation had indeed been observed and, hence, the experimental data were really three-dimensional. These results were carefully examined by Osswald, K. Ghia and U. Ghia (1983) who also obtained predictions of their own, as given in Figs. 5b-5d; their deviation from the data of Armaly and Durst (1980) represents the effect of three-dimensionality in the flow field. The computed results for a configuration well within the transition regime exhibits a 'near-limit-cycle' behavior, as shown in Fig. 6. This is an example of self-sustaining oscillatory motion, where disturbance mechanisms are developed without any external excitation. This creates an instability which subsequently leads to transition. Figures 7a and 7b show the time-averaged stream-function and vorticity contours, respectively. The overall mechanism causing the unsteadiness was also explained by the present investigators. The Strouhal number of the vortex shedding is 0.38 and the large-scale coherent structure observed has been
recently reproduced by Patera (1985) in his direct numerical simulation using the spectral-element method; for a similar configuration, Roose and Kegelman (1985) also observed this coherent structure in their experiments for the first time.

The separated flow through two-dimensional and axisymmetric sudden expansions was studied by Osswald, K. Ghia and U. Ghia (1984). The results of their analysis agreed well with the data of Durst, Melling and Whitelaw (1974) for the plane sudden expansion and with the analysis of Macagno and Hung (1967) for the axisymmetric case. The transient results were presented for the cold flow in an axisymmetric centerbody combustion chamber. McGreehan, K. Ghia, U. Ghia and Osswald (1984) simulated the flow inside a pipe orifice to study this persistently unsteady flow and analyze the nature of vortex shedding. The collective vortex interaction phenomenon of Ho (1981) was observed in this configuration.

The cornerstone of any internal aerodynamic analysis is the ability to predict the lift and drag on lifting surfaces such as airfoils, or cascades of airfoils where concerned with turbomachinery applications. For compressor cascades, it is very vital to understand the effect of unsteadiness on the flow field involving phenomena such as rotating stall, individual blade stall, stall flutter, compressor surge, etc. Common to all of these physical phenomena is the occurrence of unsteady flow separation. When the flow separates over a body surface, a 'strong' interaction region appears locally where the pressure field in the flow is determined by the viscous layer rather than the inviscid flow. Accurate and efficient strong viscous-inviscid analyses are still under development. Therefore, it was decided to modify the objectives for the last year of the proposal and include in it a study of flow over an isolated airfoil. At low speed, the
local 'strong' interaction regions arise in this latter flow due to boundary-layer separation and rapid flow acceleration immediately aft of the trailing edge. The Joukowski airfoil, with its sharp trailing edge, is an ideal candidate for this study, since it also permits the use of analytical transformation metrics, thereby avoiding the error incurred due to the use of numerically computed metrics. Therefore, in order to carefully analyze persistently unsteady flow with massive separation, K. Ghia, Osswald and U. Ghia (1985a) extended their earlier analysis for internal flows and applied it to an isolated airfoil at high angle of attack. For moderate Re, these authors were successful in simulating the flow structure in the highly unsteady vortex-dominated wake region. The results of a limit cycle analysis for flow past a Joukowski airfoil are presented in Fig. 8. Such limit-cycle solutions represent realizations of a strange attractor, as all phase-space trajectories are ultimately attracted to the time-asymptotic limit-cycle solution. The detailed evolution of the pairwise shedding of vortices from the airfoil surface was discussed by K. Ghia, Osswald and U. Ghia (1985b). The Strouhal number of this shedding motion was $S = 0.18$ which agrees well with the universal wake-based number of Roshko (1954).

2.4 Numerical Grid Generation

The problem of the resolution of high-gradient and high-curvature regions in computational fluid dynamics is important not only from considerations of truncation errors but sometimes simply for correct prediction of the flow details in these regions. Most nonlinear phenomena have a tendency to occur in very thin regions which may or may not be associated with boundaries, as shown by K. Ghia, U. Ghia and Shin (1981) and U. Ghia, K. Ghia and Shin (1982). In the simulation of viscous flows at
high Reynolds number in these thin regions, the nonlinear convective terms became large in comparison with the viscous terms. In this circumstance, it became difficult to obtain a wiggle-free solution using a central-difference discretization for the convective terms. A new analysis was developed for generating an adaptive grid which took into account the influence of the problem geometry, the prevailing physical flow phenomena, the flow parameters, as well as the discretization parameters of the problem. The grid constraint used minimized the coefficient of the convective terms in the transformed flow equations in a rational manner and led to grid equations which were analogous to the inverted form of the Poisson equations used in elliptic grid-generation techniques. The method was demonstrated with the help of nonlinear one-dimensional as well as two-dimensional model problems by comparing the predicted solutions with the corresponding exact solutions. As shown in Fig. 9a, for the flow-dependent adaptive grid, the desired number of grid points have migrated to the region of the high gradient so as to limit the magnitude of the truncation errors. Figures 9b and 9c show the computed and analytical results for \( Re \) up to \( 10^4 \) for the nonlinear viscous Burgers' equation and a model internal flow problem, respectively, and the agreement with the exact analytical solution is excellent; to the authors' knowledge, these calculations were the first of their kind. The detailed results for a few one-dimensional and two-dimensional model problems were given by K. Chia, U. Chia and Shin (1983).

2.5 Semi-Implicit Numerical Methods: Accuracy and Efficiency

For improving the accuracy of a given numerical method, it was decided, early in this research program, to seek solutions with accuracy higher than second order by using spline methodology, thereby also minimizing the number
of grid points needed for obtaining solutions for viscous flow problems with high-gradient regions. A subsequent period of this research program saw the development of a multi-grid (MG) method which improves the solution accuracy by utilizing very fine grids while still retaining rapid convergence behavior. This MG method as well as the spline method are discussed next.

It was shown by K. Ghia, Shin and U. Ghia (1979), using primitive variables ($\bar{V}, p$), and by Shin, K. Ghia and U. Ghia (1981), using derived variables, that for low-speed viscous flow problems with localized high-gradient regions, although the resulting solutions themselves may be smooth, their first- and/or second-order derivatives were frequently not smooth. This led to the development of a new spline technique employing a quartic spline polynomial $S(4,2)$ of deficiency two, i.e., with two continuous derivatives.

The integrated form of the governing equation, which is generally used in finite-volume techniques, was employed in this analysis to complete the equation set. Some detailed flow results obtained using the spline $S(4,2)$ method were given by Turner, K. Ghia, U. Ghia and Keith (1982). The accuracy of spline $S(4,2)$ is comparable to, if not better than, that of the fourth-order box scheme used by Wornom (1977) and the compact differencing scheme of Hirsh (1975). In a review of higher-order methods for the solution of incompressible viscous flows, K. Ghia and U. Ghia (1982) had suggested spline $S(4,2)$ as a potential means for fourth-order accurate solutions of Navier-Stokes equations and their approximation forms.

To improve accuracy and simultaneously achieve superior convergence, U. Ghia, K. Ghia and Shin (1981) developed a multi-grid (MG) method for the solution of the two-dimensional Navier-Stokes equations. A coupled strongly implicit multi-grid (CSI-MG) method was used for simultaneous solution of the vorticity and stream function equations. The model problem used was
that of the flow inside a shear-driven cavity. The potential of the method was demonstrated via efficient computation of solutions for Reynolds number up to $10^4$ using a very fine grid. Because of the appearance of one or more secondary vortices in the flow field, uniform mesh refinement was preferred to the use of one-dimensional grid clustering coordinate transformations. This method was further extended by K. Ghia, U. Ghia and Shin (1981) for use in determining asymptotic three-dimensional flow in curved ducts and by U. Ghia, K. Ghia and Ramamurti (1983) for determining the developing three-dimensional flow in curved ducts. In this latter parabolized Navier-Stokes analysis, the MG method was advanced for use with Neumann boundary-value problems in clustered curvilinear coordinates. This comprised an important step in the analysis of viscous flows using the velocity-pressure formulation of the Navier-Stokes equations. With successive over-relaxation (SOR) as the smoothing operator and with suitably formulated restriction and coarse-grid-correction operators, a 4-grid MG procedure enhanced the efficiency of fine-grid solutions of the Neumann problem approximately by a factor of four. U. Ghia, K. Ghia and Ramamurti (1983) also carefully examined the influence of the smoothing operator by employing the ADI and SI techniques in place of SOR. For the curved polar duct with a clustered grid, Fig. 10 shows the convergence history of the MG-SI method for the Neumann Poisson problem. The computational advantage of the MG procedure with increasing refinement of the finest grid is very obvious from this figure.

As stated earlier in this section, the MG-SI method was applied to the shear-driven cavity problem. The MG-SI method developed by U. Ghia, K. Ghia and Shin (1982) had about fourteen times faster convergence rate as compared to the available solvers, including that of Benjamin and Denny (1979). The
detailed structure of the various secondary eddies at $Re = 10^4$ is shown in Fig. 11a; these have become the benchmark solutions for this fairly standard model problem as they were obtained using a fine grid of $(257 \times 257)$ points. Further, K. Ghia and U. Ghia (1984) suggested, via the results presented in Fig. 11b, that the original experiments of Pan and Acrivos (1967) were three-dimensional in nature. Koseff and Street (1983) repeated the experiment with a spanwise aspect ratio of three and confirmed that the predictions of U. Ghia, K. Ghia and Shin (1982) are correct; these new experimental results are also shown in Fig. 11b.

The convergence of the conventional ADI method was also accelerated by using a local time-step method suggested by K. Ghia (1975). Abdelhalim, U. Ghia and K. Ghia (1983) found that it was almost essential to use this procedure to obtain converged results for their problem of flow past a blunt plate including a separation bubble. The approach can be viewed as a means of preconditioning the iteration matrix of the relaxation scheme and corresponds to a more uniform Courant number throughout the flow field.

2.6 Fully-Implicit Numerical Methods: Semi-Direct and Direct Methods

The development of semi-direct and direct Poisson solvers was also given considerable attention. The fast direct solvers of Hockney (1970) and Buneman (1969) were widely used for two-dimensional flow problems, involving separable Poisson equations. It was felt that these methods were not easily extendable to the generalized Poisson equation or to other generalized elliptic equations. Therefore, initially, a Cauchy-Riemann solver which leads to a semi-direct (SD) method of solution was developed. This method followed that of Martin (1978) such that the form of the generalized operator in non-orthogonal coordinates was made to fit the form of the
Cauchy-Riemann operator in rectangular coordinates. The results of this feasibility study were given by Osswald, K. Ghia and U. Ghia (1980). These results showed that the degree of non-orthogonality and grid clustering strongly influenced the solution convergence rate. However, it remains a viable approach for efficient solutions of the Poisson equation in a variety of orthogonal and non-orthogonal coordinate systems.

For solution of the Poisson equation in generalized orthogonal coordinates, Osswald and K. Ghia (1981) developed a direct block Gaussian elimination (BGE) method. Block-Gaussian elimination is a direct extension of the Gaussian elimination procedure to block matrices. The BGE procedure provides the effective inversion of an \([NM] \times [NM]\) matrix through the actual inversion of a predetermined sequence of \([N(M \times M)]\) sub-matrices. The BGE method was compared with the SD method of Martin (1978) for the general Poisson problem for evaluation of accuracy and efficiency and was found to yield a direct one-step solution, irregardless of the degree of grid clustering, with considerably increased efficiency as compared to the SD method. A comparison of various existing subroutines for solving the two-dimensional Dirichlet Poisson problem was also provided and, as shown in Table 1, direct BGE provides a solver with efficiency comparable to other available direct solvers, but with the added advantage that it is applicable for unsteady flow analysis in generalized orthogonal coordinates. This BGE solver was also extended by Osswald, K. Ghia and U. Ghia (1983) for solution of the Neumann Poisson problem encountered for the pressure field in an unsteady primitive-variable Navier-Stokes analysis. The extension of the BGE method to the solution of the Poisson equation for axisymmetric flow was also provided by Osswald, K. Ghia and U. Ghia (1984), whereas the solution of the Poisson equation for a doubly-infinite region was provided by
K. Chia, Osswald and U. Ghia (1985a). The solution of the Poisson equation by the BGE method has proved to be very vital in the overall success of Navier-Stokes analysis of unsteady flow.
REFERENCES


SECTION 3

PAPERS AND REPORTS PUBLISHED

BOOKS AND MONOGRAPHS


PAPERS AND REPORTS


SECTION 4

SCIENTIFIC INTERACTIONS - SEMINARS AND PAPER PRESENTATIONS

SEMINARS AND INVITED LECTURES


25


PRESENTATIONS


SECTION 5

STUDENT DEGREE THeses AND DISSERTATIONS

M.S. DEGREE THeses


PH.D. DEGREE DISSERTATIONS


Of the various CFD analyses developed, some were of direct use to the technical community. Although to our knowledge, none of these analyses were used in the development of any specific hardware, they are being used in preliminary design studies by analysts in the industry. Some of these analyses are also being used by other researchers at governmental laboratories to improve their analyses. The following is a list of the CFD analyses and the organizations using them.

<table>
<thead>
<tr>
<th>ANALYSIS</th>
<th>ORGANIZATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>The Parabolized Navier-Stokes Analysis for Internal</td>
<td>General Electric Co., Cincinnati, OH;</td>
</tr>
<tr>
<td>Flows</td>
<td>AVCO Corp., Everett, MA.</td>
</tr>
<tr>
<td>The Unsteady Navier-Stokes Analysis for Internal</td>
<td>General Electric Co., Cincinnati, OH.</td>
</tr>
<tr>
<td>Flows</td>
<td></td>
</tr>
<tr>
<td>The Unsteady Navier-Stokes Analysis for External</td>
<td>McDonnell Aircraft, Co., St. Louis, MO;</td>
</tr>
<tr>
<td>Flows</td>
<td>NASA-Langley Research Cntr., Hampton, VA.</td>
</tr>
<tr>
<td>Adaptive Grid Generation Analysis</td>
<td>Sverdrup Technology, Inc., Arnold</td>
</tr>
<tr>
<td></td>
<td>Air Force Station, TN.</td>
</tr>
<tr>
<td>Navier-Stokes Analysis in Primitive Variables</td>
<td>NASA Langley Research Center, Hampton, VA.</td>
</tr>
<tr>
<td>SUBROUTINE</td>
<td>REFERENCE</td>
</tr>
<tr>
<td>------------</td>
<td>-------------------</td>
</tr>
<tr>
<td>PSOLVE</td>
<td>Temperton (1979)</td>
</tr>
<tr>
<td>POISSX</td>
<td>Schumann &amp; Sweet (1976)</td>
</tr>
<tr>
<td>BLKTRI</td>
<td>Swarztauber (1974)</td>
</tr>
<tr>
<td>OPD</td>
<td>Current Work</td>
</tr>
<tr>
<td>CG</td>
<td>Khosla (1983)</td>
</tr>
<tr>
<td>MGSI</td>
<td>Ramamurti (1983)</td>
</tr>
<tr>
<td>ADI</td>
<td>Schumann (1977)</td>
</tr>
<tr>
<td>SOR</td>
<td>Schumann (1977)</td>
</tr>
</tbody>
</table>

* (N+1, M+1) Uniform Cartesian Grid with N = 128 and M = 32. IBM 370/168 Fortran H-Extended Single Precision.

+ Effective IBM 370/168 Time, computed from AMDAHL 470/V6 Time, using IBM Time/AMDAHL time = 1.6.
Aspect Ratio
AR = $a/b$

Hydraulic Diameter
$D = 2ab/(a+b)$

J = 0: Rectangular Cross Section
Reynolds Number, $Re = \frac{w_{avg}D}{v}$
Dean Number, $K = Re/\sqrt{R}$

J = 1: Polar Cross Section

FIG. 1a. CURVED-DUCT GEOMETRY AND COORDINATE SYSTEM.
FIG. 1c. EFFECT OF DEAN NUMBER ON SECONDARY FLOW FOR POLAR DUCTS - CROSS-FLOW STREAMLINE CONTOURS.
FIG. 1c. EFFECT OF DEAN NUMBER ON SECONDARY FLOW FOR POLAR DUCTS - CROSS-FLOW STREAMLINE CONTOURS.
$M_\infty = 0$, ADIABATIC WALL.

- MODIFIED GESSNER-EMERY MODEL
- 3-D, TWO-LAYER CEBECI'S MODEL

$C_f \times 10^3$

$Re_x \times 10^{-6}$

FIG. 2a. COMPARATIVE STUDY OF ISOTROPIC AND ANISOTROPIC TURBULENT MODELS.
FIG. 2b. EFFECT OF MASS TRANSFER ON THE LAMINAR AND TURBULENT VELOCITY PROFILES.
(a) STEADY-STATE SOLUTION (AT T = 45) WITH INAPPROPRIATE MESH.

b(i) TRANSIENT SOLUTION (AT T = 8) WITH IMPROVED MESH.

b(ii) STEADY-STATE SOLUTION (AT T = 41) WITH IMPROVED MESH.

FIG. 3. STREAM-FUNCTION CONTOURS FOR CONSTRUCTED CHANNEL FLOW.
Re = 1000, $\Delta \psi = 0.002$ WITHIN SEPARATION BUBBLE;
$\Delta \psi = 0.1$ OTHERWISE.
FIG. 4a. COMPARISON OF MEASURED AND COMPUTED RECIRCULATION LENGTHS FOR BLUNT PLATES AND LONGITUDINAL CYLINDERS.
FIG. 4b. COMPARISON BETWEEN PREDICTED AND EXPERIMENTAL VARIATION OF TURBULENT KINETIC ENERGY WITHIN REDUCED SEPARATION BUBBLE FOR BLUNT PLATES.
FIG. 5a. SIMILARITY STUDY OF PRIMARY REATTACHMENT LENGTH.

FIG. 5b. SECONDARY SEPARATION AND REATTACHMENT LENGTHS.

FIG. 5c. SEPARATION AND REATTACHMENT LENGTHS AFTER REF. [11].

FIG. 5d. COMPARISON OF PRIMARY REATTACHMENT LENGTH IN TRANSITION REGIME.
Fig. 6a. Stream-function contours for unsteady flow in transition regime; Re = 2000, $\Delta \psi = 0.1$ for main flow; $\Delta \psi = 0.0536$ in bubbles.

Fig. 6b. Time-averaged stream-function contours for $95.02 \leq T \leq 117.00$; Re = 2000, $\Delta \psi = 0.1$.

Fig. 6c. Time-averaged vorticity contours for $95.02 \leq T \leq 117.00$; Re = 2000, $\Delta \omega = 2.0$. 

Page 41
(a) STREAMFUNCTION CONTOURS; $\Delta \psi = 0.05$.

(b) VORTICITY CONTOURS; $\Delta \omega = 5.0$.

FIG. 7. TRANSIENT FLOW THROUGH ORIFICE PIPE; $Re_D = 1000$; $\beta = 0.6$. 

42
FIG. 8. LIMIT-CYCLE ANALYSIS: RE = 1,000, $\alpha = 15^\circ$. 

43
FIG. 9a. GRID-POINT PATHS FOR 1-D NONLINEAR VISCOUS BURGERS' EQUATION

(i) ADAPTIVE GRID
(ii) ADAPTIVE GRID (MODIFIED)
DIFFERENTIAL EQUATION: \( u_{xx} - (u-U)Re \ u_x = Re \ u_t \) for \(-5 \leq x \leq 5\) and \( U = 0.5 \)

B.C.: \( u(-5,t) = 1, \ u(5,t) = 0; \) I.C.: \( u(x,0) = \begin{cases} 1.0 & -5 \leq x < 0 \\ 0.5 & x = 0 \\ 0.0 & 0 < x \leq 5 \end{cases} \)

EXACT SOLUTION: \( u(x) = U[1 - \tanh\left(\frac{U \ Re \ x}{2}\right)] \)

(i) Coordinate Transformation  (ii) Computed Velocity Profile

(a) Adaptive Grid and Flow Solution  (b) Analytical Velocity Profile

(c) Adaptive Grid (Modified) and Flow Solution  (d) Velocity Profile Computed With Fixed Uniform Grid of 51 Points.

FIG. 9b. SOLUTION OF 1-D VISCOUS BURGERS' EQUATION, WITH RE RANGING FROM 1 TO 10,000.
DIFFERENTIAL EQUATION: \( u_{xx} + (1-2x)Re \ u_x + (1-u^2) \ Re = Re \ u_t \) for \( 0 \leq x \leq 1 \)

B.C.: \( u(0,t) = 0, \ u(1,t) = 0 ; \) I.C.: \( u(x,0) = 0 \)

EXACT SOLUTION: \( u(x) = \tanh \left( \frac{Re \ x(1-x)}{2} \right) \)

Fig. 9c. SOLUTION OF 1-D VISCOUS INTERNAL FLOW MODEL PROBLEM, WITH \( RE = 1 \) TO \( RE = 10,000 \).
FIG. 10. CONVERGENCE HISTORY OF MULTIGRID METHOD FOR NEUMANN PRESSURE PROBLEM IN DEVELOPING FLOW THROUGH CURVED POLAR DUCT WITH GRID CLUSTERING.
(a) STREAMLINE PATTERN FOR PRIMARY, SECONDARY AND ADDITIONAL CORNER VORTICES IN SHEAR-DRIVEN CAVITY FLOW, Re = 10000.

(b) VERTICAL EXTENT OF SECONDARY DOWNSTREAM EDDY - COMPARISON BETWEEN COMPUTATIONS AND MEASUREMENTS

FIG. 11. FLOW IN SHEAR DRIVEN CAVITY.
END

12-86

DTTC