Lewis presentations at the NASA CFD Conference hosted by NASA Ames Research Center
Moffett Field, California
March 12 to 14, 1991
Computational Fluid Dynamics

Lewis presentations at the NASA CFD Conference hosted by NASA Ames Research Center Moffett Field, California March 12 to 14, 1991

NASA
National Aeronautics and Space Administration
Office of Management
Scientific and Technical Information Program
1992
PREFACE

The first NASA Computational Fluid Dynamics (CFD) Conference was held at NASA Ames Research Center on March 7 through 9, 1989. Conference objectives were to disseminate CFD results to industry and university researchers, to promote synergy among NASA CFD researchers, and to permit feedback from researchers outside of NASA concerning issues facing the discipline of CFD. The focus of the 1989 conference was on the application of CFD technology. As a result of three panel sessions held at the 1989 conference, the following conclusions were reached:

- NASA's program was too heavily focused on applications.
- CFD developers need to better understand the needs of the users.
- Industry needs more reliable and cost effective CFD tools.
- Three critical areas of research were identified:
  - More algorithm research, particularly for Navier-Stokes solvers with unstructured grids.
  - More research on geometric modeling with rapid, accurate, and effective surface definition methods.
  - More research on grid generation stressing faster, more efficient, and improved quality with emphasis on reduced set-up time and complexity.

Additional concerns resulting from the panels may be found in NASA Conference Publication 10038, Vol. 1, p. xiv.

The objectives of the second NASA CFD Conference, held at the Ames Research Center on March 12 through 14, 1991, were the same as the 1989 conference: to disseminate CFD research results to industry and university CFD researchers, to promote synergy among NASA CFD researchers, and to permit feedback from researchers outside NASA on issues pacing the discipline of CFD. However, the focus of the 1991 conference was on the elements that comprise the discipline of CFD, rather than on CFD applications. These elements are

- Algorithm development and analysis
- Algorithms for real gas/combustion modeling
- Algorithms for advanced computer architectures
- Turbulence modeling/flow physics
- Scientific visualization
- Grid generation
- Other CFD methodologies

The intent of the 1991 conference was to emphasize CFD development underway at NASA Centers, rather than the applications activity.
This publication is a collection of 20 presentations made by members of the NASA Lewis Research Center staff; the members of the Institute for Computational Mechanics in Propulsion (ICOMP); Sverdrup Corporation, Lewis Research Group; and visiting academics at Lewis. Presentation abstracts, (published in bound form at the conference) as well as viewgraph material used during the conference are included in this publication. The intent is to display research underway on the fundamentals of CFD at Lewis.

The conference was sponsored by the Aerodynamics Division, Office of Aeronautics, Exploration and Technology (OAET), NASA Headquarters, Washington, D.C.

This preface is based on an adaptation of that appearing in NASA Conference Publication 10038, proceedings of the CFD Conference held at Ames Research Center, 1989.

Louis A. Povinelli
NASA Lewis Research Center
April 8, 1991
CONTENTS

Computational Fluid Dynamics at the Lewis Research Center—An Overview
  L.A. Povinelli ............................................. 1

ALGORITHM DEVELOPMENT -- UPWIND/TVD/ENO METHODS

Development of New Flux Splitting Schemes
  M.-S. Liou and C.J. Steffan, Jr. .......................... 19

ALGORITHM DEVELOPMENT -- NEW METHODS AND ANALYSIS

An Iterative Implicit DDADI Algorithm for Solving the Navier-Stokes Equation
  S.C. Chen, N.S. Liu, and H.D. Kim .......................... 29

ALGORITHM DEVELOPMENT -- CODES AND METHODS

The Proteus Navier-Stokes Code
  C.E. Towne and J.R. Schwab ................................. 47

Accurate Upwind-Monotone (Nonoscillatory) Methods for Conservation Laws
  H.T. Huynh ................................................. 57

Numerical Simulation of Conservation Laws
  S. Chung-Chang and W.M. To ................................. 65

ALGORITHMS FOR REAL GAS

A New Lagrangian Method for Real Gases at Supersonic Speed
  C.Y. Loh and M.-S. Liou .................................. 75

A Time-Accurate Implicit Method for Chemical Non-Equilibrium Flows at All Speeds
  J.-S. Shuen ................................................ 83

Study of Shock-Induced Combustion Using an Implicit TVD Scheme
  S. Yungster ................................................ 93

Upwind Schemes and Bifurcating Solutions in Real Gas Computations
  A. Suresh and M.-S. Liou .................................. 103

ALGORITHMS FOR ADVANCED ARCHITECTURES

High Order Parallel Numerical Schemes for Solving Incompressible Flows
  A. Lin, E.J. Milner, M.-F. Liou, and R.A. Belch ............... 115
An Efficient and Robust Algorithm for Time Dependent Viscous Incompressible Navier-Stokes Equations
J.W. Goodrich ........................................ 133

TURBULENCE MODELING/FLOW PHYSICS

Advances in Engineering Turbulence Modeling
T.-H. Shih ........................................... 143

On Recontamination and Directional-Bias Problems in Monte Carlo Simulation of PDF Turbulence Models
A.T. Hsu ............................................. 163

Implementation of a $k-\varepsilon$ Turbulence Model to RPLUS3D Code
T. Chitsomboon ..................................... 175

Turbulence and Deterministic Chaos
R.G. Deissler ....................................... 181

Stability of Compressible Taylor-Couette Flow
K. Kao and C. Chow ................................. 193

SCIENTIFIC VISUALIZATION

Techniques for Animation of CFD Results
J.G. Horowitz and J.C. Hanson ..................... 205

Distributed Visualization for Computational Fluid Dynamics
D.J. Sosoka and A.A. Facca .......................... 219

GRID GENERATION

Techniques for Grid Manipulation and Adaptation
Y.K. Choo, P.R. Eisemann, and K.D. Lee ............. 225
COMPUTATIONAL FLUID DYNAMICS
AT THE
LEWIS RESEARCH CENTER

- AN OVERVIEW

LOUIS A. POVINELLI
DEPUTY CHIEF
INTERNAL FLUID MECHANICS DIVISION
OVERVIEW OUTLINE

- ORGANIZATIONAL STRUCTURE
- MAIN THRUSTS
- LEVEL OF EFFORT
- COMPUTATIONAL FACILITIES
- TECHNICAL HIGHLIGHTS
- FUTURE DIRECTIONS

ORGANIZATIONAL STRUCTURE

- LeRC Director's Office
- Research Acad.
- Aero Directorate
- CFD Branch
- Comp. Tech. Branch
- Heat Transfer Branch
- Aerothermochem. Branch
- Turbo-mach. Branch
- Inf., Ducts & Nozzles Branch
- Space Comp. Branch
INSTITUTE FOR COMPUTATIONAL MECHANICS IN PROPULSION (ICOMP)

OBJECTIVE: TO DEVELOP ADVANCED COMPUTATIONAL METHODS REQUIRED FOR THE SOLUTION OF INTERNAL FLUID MECHANICS AND STRUCTURAL MECHANICS PROBLEMS

ORGANIZATION

PROGRAM DIRECTOR: Dr. Louis Povineill
EXECUTIVE OFFICER: Dr. Charles Feller
STEERING COMMITTEE: Prof. E. Reshotko, Chairman

Dr. H. Brandhorst  Dr. L. Diehl
Dr. M. Goldstein  Prof. I. Greber
Prof. R. Mullen  Dr. L. Nichols
Dr. L. Pinson  Dr. L. Reid

ICOMP STATISTICS

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>PEOPLE</td>
<td>23</td>
<td>43</td>
<td>50</td>
<td>46</td>
<td>47</td>
</tr>
<tr>
<td>SEMINARS</td>
<td>10</td>
<td>27</td>
<td>39</td>
<td>30</td>
<td>37</td>
</tr>
<tr>
<td>REPORTS</td>
<td>2</td>
<td>9</td>
<td>22</td>
<td>32</td>
<td>25</td>
</tr>
<tr>
<td>WORKSHOPS/LECTURE SERIES PRESENTATIONS</td>
<td>1</td>
<td>0</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>UNIVERSITIES ORGANIZATIONS</td>
<td>7</td>
<td>0</td>
<td>21</td>
<td>14</td>
<td>15</td>
</tr>
</tbody>
</table>

Figure 4. - ICOMP statistics (1986 to 1990)
ICOMP/LeRC Joint Activities

<table>
<thead>
<tr>
<th>ICOMP Member</th>
<th>LeRC Civil Servant</th>
<th>Activity</th>
</tr>
</thead>
<tbody>
<tr>
<td>AFOLABI (Purdue-Ind)</td>
<td>STEFKO</td>
<td>BLADE AEROELASTICITY-MODAL INTERACTIONS</td>
</tr>
<tr>
<td>AGGARWAL (ILL-Chicago)</td>
<td>BULZAN</td>
<td>SPRAY COMBUSTION MODELS</td>
</tr>
<tr>
<td>AKL (Louisiana Tech)</td>
<td>KIRALY</td>
<td>FINITE ELEMENT MODEL MAPPING ONTO</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PARALLEL MICROCOMPUTING ARCHITECTURE</td>
</tr>
<tr>
<td>ARNONE (Florence)</td>
<td>LIOU</td>
<td>TRANSONIC FLOW CALCULATIONS/NON-</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PERIODIC C-GRIDS</td>
</tr>
<tr>
<td>BALSA (Arizona)</td>
<td>GOLDSTEIN</td>
<td>INSTABILITIES IN SUPERSONIC MIXING LAYERS</td>
</tr>
<tr>
<td>COWLEY (Cambridge)</td>
<td>HULTGREN</td>
<td>NON-LINEAR BOUNDARY LAYER PROBLEMS-</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SEPARATION</td>
</tr>
<tr>
<td>DEMUREN (Old Dom)</td>
<td>CLAUS</td>
<td>FULL REYNOLDS STRESS TURBULENCE MODELS</td>
</tr>
<tr>
<td>EISEMAN (Columbia)</td>
<td>CHOD</td>
<td>ADAPTIVE GRID GENERATION</td>
</tr>
<tr>
<td>GAJJAR (Exeter)</td>
<td>GOLDSTEIN</td>
<td>UNSTEADY NON-LINEAR CRITICAL LAYERS</td>
</tr>
<tr>
<td>HAGSTROM (New Mexico)</td>
<td>GOODRICH</td>
<td>OUTFLOW BOUNDARY CONDITIONS IN LOW MACH NO.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>COMBUSTION</td>
</tr>
</tbody>
</table>

ICOMP Statistics

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>People</td>
<td>23</td>
<td>43</td>
<td>50</td>
<td>46</td>
<td>47</td>
</tr>
<tr>
<td>Seminars</td>
<td>10</td>
<td>27</td>
<td>39</td>
<td>30</td>
<td>37</td>
</tr>
<tr>
<td>Reports</td>
<td>2</td>
<td>9</td>
<td>22</td>
<td>32</td>
<td>25</td>
</tr>
<tr>
<td>Workshops/Lecture Series</td>
<td>1</td>
<td>0</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Presentations</td>
<td>7</td>
<td>0</td>
<td>21</td>
<td>14</td>
<td>15</td>
</tr>
<tr>
<td>Universities or Organizations</td>
<td>20</td>
<td>28</td>
<td>43</td>
<td>35</td>
<td>36</td>
</tr>
</tbody>
</table>

Figure 4. - ICOMP statistics (1986 to 1990)
CENTER FOR MODELING OF TURBULENCE AND TRANSITION (CMOTT)

TECHNICAL LEADER: DR. T. SHIH

OBJECTIVE: DEVELOP VALIDATED ENGINEERING CLOSURE MODELS FOR IMPLEMENTATION IN PROPULSION CFD

SCOPE:
- MODELING OF COMPRESSIBLE TURBULENT FLOWS
  - TWO-EQUATION MODELS
  - SECOND ORDER CLOSURE MODEL
  - RENORMALIZATION GROUP THEORY (RNG)
  - DIRECT INTERACTION APPROXIMATION (DIA)
- TURBULENCE/CHEMISTRY INTERACTION
  - PDF PLUS MOMENT CLOSURE MODELS
- BOUNDARY LAYER TRANSITION
  - MODELING OF BYPASS TRANSITION
  - IMPROVEMENT OF NEAR WALL TURBULENCE MODELS
  - DNS OF BYPASS TRANSITION

INTERNAL FLUID MECHANICS DIVISION

INTERNAL FLUID MECHANICS DIVISION
Dr. Lonnie Reid, Chief
Dr. Louis A. Povinelli, Deputy Chief

COMPUTATIONAL FLUID DYNAMICS BRANCH 2610
Dr. D. Raam Reddy, Chief

HEAT TRANSFER BRANCH 2630
Dr. Robert J. Simoneau, Chief
Frederick F. Simon, Deputy Chief

AEROTHERMOCHEMISTRY BRANCH 2650
Dr. Edward J. Mularz, Chief
Erwin A. Lezberg, Deputy Chief

COMPUTATIONAL METHODS FOR SPACE BRANCH 2670
Dr. Robert M. Stubbs, Chief

COMPUTATIONAL TECHNOLOGY BRANCH 2620
John R. Szuch, Chief
Richard A. Blech, Deputy Chief

TURBOMACHINERY FLOW PHYSICS BRANCH 2640
Dr. Raymond E. Gaugler, Chief
Jerry R. Wood, Deputy Chief

INLET, DUCT & NOZZLE FLOW PHYSICS BRANCH 2660
John M. Abbott, Chief
Dr. James R. Scott, Deputy Chief
FUNDING DISTRIBUTION

BASE RTOP (505-62-52)

- 66% IN-HOUSE
- 22% ICOMP
- 12% UNIVERSITY
**LEVEL OF EFFORT**

**IFMD PERSONNEL**

- 9 GRANTEES
- 16 IOOMP
- 26 SUPPORT SERVICE CONTRACTORS
- 103 CIVIL SERVANTS

**LeRC COMPUTATIONAL RESOURCES ON NETWORK**

- CRAY Y-MP (6 proc., 128 MW)
- CRAY X-MP (2 proc., 8 MW)
- CONVEX C220 (2 proc., 256 MB)
- SCIENTIFIC VAXcluster (5 proc., 448 MB)
- VM/CMS/UTS (2 proc., 128 MB)
- GRAPHICS & VISUALIZATION (G-VIZ) LAB
- ADVANCED COMPUTATIONAL CONCEPTS LAB (ACCL)
  - INTEL IPSC/860 (16 nodes, 256 MB)
  - ALLIANT FX/80 (8 proc., 128 MB)
  - STELLAR GS1000 & IRIS 4D/320 (2 proc.)
IFMD COMPUTATIONAL RESOURCES

- HIGH-END GRAPHICS WORKSTATIONS (11)
- PERSONAL GRAPHICS AND COMPUTER WORKSTATIONS (30)
- PERSONAL COMPUTERS (26)
- COLOR HARD-COPY UNITS (16)
- ADMINISTRATION OF NAS ACCOUNTS (26,000 HRS.)
  - IN-HOUSE, ~50%
  - UNIVERSITY, INDUSTRY, ~50%

IFMD GOAL

DEVELOP VALIDATED, NUMERICAL SIMULATIONS FOR PROPULSION SYSTEMS

- GAS TURBINES
- ROCKETS
- HYPERSONIC AIR BREATHING SYSTEMS

BY CONDUCTING CLOSELY COUPLED EXPERIMENTAL/COMPUTATIONAL RESEARCH ON:

- INLETS, DUCTS, NOZZLES
- COMBUSTORS
- TURBOMACHINERY COMPONENTS
IFMD MAIN THRUSTS

- ALGORITHM DEVELOPMENT
  - NAVIER-STOKES
  - LOW SPEED COMPRESSIBLE
  - BOLTZMANN & DSMC

- CLOSURE MODELING
  - TURBULENCE & TRANSITION
  - PDF MODELING FOR REACTING FLOWS

- REAL-TIME INTEGRATION OF CFD/EXPERIMENT (ICE)
Algorithm Development

M. Liou, C. Steffen, "Development of New Flux Splitting Schemes"
S. Chen, N. Liu, H. Kim, "An Iterative Implicit DDADI Algorithm for Solving the Navier-Stokes Equations"
C. Towne, J. Schwab, "The PROTEUS Navier-Stokes Code"
H. Huynh, "Accurate Upwind-Monotone Methods for Conservation Laws"
S. Chang, W. To, "Numerical Simulation of Conservation Laws"

Algorithms For Real Gas

C. Loh, M. Liou, "A New Lagrangian Method for Real Gases at Supersonic Speed"
J. Shuen, "A Strongly Implicit Method for Chemical Non-Equilibrium Viscous Flows at all Speeds"
S. Yungster, "Study of Shock-Induced Combustion Using An Implicit TVD Scheme"
A. Suresh, M. Liou, "Upwind Schemes and Bifurcating Solutions in Real Gas Computations"

Algorithms for Advanced Architectures

A. Lin, E. Milner, M. Liou, R. Biech, "High Order Parallel Numerical Scheme for Solving Incompressible Flows"
J. Goodrich, "An Efficient and Robust Algorithm for Time Dependent Incompressible Navier-Stokes Equations"

Turbulence Modeling/Flow Physics

T. Shih, "Advances in Engineering Turbulence Modeling"
A. Hsu, "On the Recontamination Problem In Monte Carlo Simulation of PDF Turbulence Models"
T. Chitsomboon, "Implementation of a K-ε Turbulence Model to RPLUS3D Code"
R. Deissler, "Turbulence and Deterministic Chaos"
K. Kao, C. Chow, "Stability of Compressible Taylor-Couette Flow"

Scientific Visualization

J. Horowitz, J. Hanson, "Techniques for Animation of CFD Results"
D. Sosoka, T. Facca, "Distributed Visualization for CFD"
Algorithms for Advanced Architectures

A. Lin, E. Milner, M. Liou, R. Blech, "High Order Parallel Numerical Scheme for Solving Incompressible Flows"
J. Goodrich, "An Efficient and Robust Algorithm for Time Dependent Viscous Incompressible Navier-Stokes Equations"

Turbulence Modelling/Flow Physics

T. Shih, "Advances in Engineering Turbulence Modeling"
A. Hsu, "On the Recontamination Problem in Monte Carlo Simulation of PDF Turbulence Models"
T. Chitsomboon, "Implementation of a K-ԑ Turbulence Model to RPLUS3D Code"
R. Deissler, "Turbulence and Deterministic Chaos"
K. Kao, C. Chow, "Stability of Compressible Taylor-Couette Flow"

Scientific Visualization

J. Horowitz, J. Hanson, "Techniques for Animation of CFD Results"
D. Sosoka, T. Facca, "Distributed Visualization for CFD"

Grid Generation

Y. Choo, P. Eiseman, K. Lee, "Techniques for Grid Manipulation and Adaptation"
THE LONG RANGE VISION IN PROPULSION ANALYSIS

Numerical Algorithms and Models

High-Speed Interactive Computing

Detailed Experiments

Validated Multidisciplinary Codes

High-Speed Interactive Computing

Numerical Test Cell for Propulsion

INTEGRATED CFD AND EXPERIMENTS (ICE) TECHNOLOGY

NUMERICAL PROPULSION SYSTEM SIMULATION (NPSS) TECHNOLOGY
Maximizing both accuracy and efficiency has been the primary objective in designing a numerical algorithm for computational fluid dynamics (CFD). This is especially important for solution of complex 3D systems of Navier-Stokes equations which often include turbulence modeling and chemistry effects. Recently, upwind schemes have been well received for both their capability of resolving discontinuities and their sound theoretical basis in characteristic theory for hyperbolic systems. With this in mind, we present two new flux splitting techniques for upwind differencing.

The first method is based upon High-Order Polynomials Expansions (HOPE) of the mass flux vector. The present splitting results in positive and negative mass-flux components that vanish at $M = 0$. Thus the error in the Van Leer scheme which results in the diffusion of the boundary layer is eliminated. We also introduce several choices for splitting the pressure and examine their effects on the solution.

The second new flux splitting is based on the Advection Upwind Splitting Method (AUSM for short). In Navier-Stokes calculations, the diffusion error present in Van Leer's flux-splitting scheme corrupts the velocity vector near the wall. In the AUSM, a proper splitting of the advective velocity component leads to an accurate resolution of the interface fluxes. The interface velocity is defined using the Mach number polynomial expansion in the mass flux, then the convective fluxes follow directly. Again, several choices of pressure splitting are possible among which a simple Mach number splitting according to characteristics appears to be the best in terms of accuracy. The scheme has yielded results whose accuracy rivals, and in some cases surpasses that of Roe's method, at reduced complexity and computational effort.

The calculation of the hypersonic conical flow demonstrates the accuracy of the splittings in resolving the flow in the presence of strong gradients. The second series of tests involving the 2D inviscid flow over a NACA 0012 airfoil demonstrate the ability of the AUSM to resolve the shock discontinuity at transonic speed and the level of entropy generation at the stagnation point.

In the third case we calculate a series of supersonic flows over a circular cylinder. The Roe splitting in all conditions and grids tested yielded anomalous solutions (sometimes referred to as the carbuncle phenomenon), which could appear as nonsymmetric, protuberant, or indented contours, see Figs. 1. The AUSM gave expected solutions in all calculations.

The fourth test deals with a 2D shock wave/boundary layer interaction. This provides an opportunity to accurately resolve a laminar separation region and to compare the ability to resolve a non grid-aligned shock with other methods, as shown in Figs. 2.
Fig. 1 Mach contours for a Mach 6 flow over a circular cylinder: (a) the AUSM solution, and (b) the Roe solution displaying a protuberant, two-shock solution. Note the calculations were performed for the front half plane, thus no symmetry was imposed.

Fig. 2 Mach contours for a shock wave/boundary layer interaction problem: (a) the AUSM solution showing sharp resolutions of various waves, and (b) the Roe solution showing more diffusive waves.
Motivation

To couple Accuracy and Efficiency together within an Upwind Flux-Splitting Scheme

Two Possible Improvements

1. Less Diffusive FVS Technique
   - High Order Polynomial Expansion (HOPE)
   - Unresolved Stability Concerns

2. New Advective Velocity Splitting Technique
   - Advection Upwind Splitting Method (AUSM)
   - Simple, Accurate, Efficient and Robust

Upwinding in 1D

Conservation Law

\[
\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} = 0
\]

\[
u_{i}^{n+1} - u_{i}^{n} = -\frac{\Delta t}{\Delta x} \left( F_{i+\frac{1}{2}}^{*} - F_{i-\frac{1}{2}}^{*} \right)
\]

Numerical Flux Function

Flux Vector Splitting (FVS)

\(\Rightarrow\) (more diffusive, more efficient)

\(F_{i+\frac{1}{2}}^{*} \Rightarrow \) Flux Difference Splitting (FDS)

(less diffusive, less efficient)
Quasi-2D Viscous Conical Flow

Geometry

\[ M_\infty = 7.95 \]
\[ Re_\infty = 0.42 \times 10^6 \]
\[ Pr = 1.0 \]

Adiabatic Wall Boundary Condition
AUSM in 1D

Euler Equations

\[
\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{F}}{\partial x} = 0, \quad \mathbf{U} = \left\{ \begin{array}{c} \rho \\ \rho u \\ \rho E \end{array} \right\}, \quad \mathbf{F} = \left\{ \begin{array}{c} \rho u \\ \rho uu \\ \rho Hu \end{array} \right\} + \left\{ \begin{array}{c} 0 \\ p \end{array} \right\}
\]

Convective Flux

\[ \mathbf{F}_c = \text{(scalar)} \text{(convective velocity)} \]

\[ \mathbf{F}_{c,i+\frac{1}{2}} = \text{(scalar)} \text{upwind} (u)_{i+\frac{1}{2}}, \quad (u)_{i+\frac{1}{2}} = u(U_i, U_{i+1}) \]

Pressure Flux

\[ \mathbf{F}_p = \text{(scalar pressure)} \]

\[ \mathbf{F}_{p,i+\frac{1}{2}} = (p^+)_i + (p^-)_{i+1}, \quad (p^\pm)_i = p(U_i) \]

Conical Flow: First Order Accurate Results

Pressure Profile

\begin{center}
\begin{tabular}{c c}
\textbf{Pressure Profile} & \textbf{Temperature Profile} \\
\includegraphics[width=0.4\textwidth]{pressure_profile} & \includegraphics[width=0.4\textwidth]{temperature_profile}
\end{tabular}
\end{center}

- Van Leer
- Roe
- AUSM
- \( M = 7.95 \)
- \( \text{Pr} = 1.0 \)
Conical Flow: First Order Accurate Results

Convective Velocity Profile

![Convective Velocity Profile](image)

2D Shock Wave - Boundary Layer Interaction

Geometry

Impinging Shock

Reflecting Shock

Weak Wave

M = 2.0

Re = 2.96 \times 10^5

Shock Angle = 32.58°
Conical Flow: Convergence Histories

First Order Accurate Scheme  
Second Order Accurate Scheme

Shock Wave/Boundary Layer: Mach Number Contours

AUSM Scheme $O(\Delta^2)$  
Roe FDS Scheme $O(\Delta^2)$
Mach 6.0 Blunt Body Flow: Carbuncle Phenomenon

Velocity Vectors

Mach 3.0 Blunt Body Flow: Mach Number Contours

AUSM Scheme \( O(\Delta) \)

Grid

Roe FDS Scheme \( O(\Delta) \)
Supersonic Blunt Body Flow

Geometry

Inviscid Flow Assumed

Mach 6.0 Blunt Body Flow: Mach Number Contours

Roe FDS Scheme $O(\Delta)$

Grid

AUSM Scheme $O(\Delta)$
Shock Wave/Boundary Layer:

Friction Coefficient

Pressure at the Wall

*Experiment by Hakkinen, Greber, Trilling and Abarbanel

Conclusions

- Accuracy of AUSM rivals FDS schemes (i.e. Roe Splitting)

- Efficiency and Simplicity of AUSM rivals FVS schemes (i.e. VanLeer Splitting)

- AUSM is a new family of Upwind Flux Splitting Techniques
An algorithm utilizing a first-order upwind split flux technique and the diagonally dominant treatment is proposed to be the temporal operator for solving the Navier-Stokes equations. During the factorization process, if care is taken to ensure the symmetry of the operator, the resulting algorithm will be stable when spatial derivatives of the right hand side residual are evaluated by the central differencing scheme as well as the upwind differencing Roe scheme. Temporal accuracy is assured by the inner Newton-iteration process first introduced by Rai (1986).

Given the limit of a five-point stencil, the right hand side flux derivatives are formulated by several commonly used central and upwind schemes. Their performances are studied through a test case of free vortex convection in a uniform stream. From these results, a superior treatment for evaluating the flux term is proposed and compared with the rest.

The application of the proposed algorithm to the full Navier-Stokes equations is demonstrated through a calculation of flow over a backward facing step. Results are compared against the calculation done by using the fourth-order central differencing scheme with artificial damping.
OBJECTIVE

GIVEN A STENCIL OF 5 POINTS, FIND A SCHEME THAT COMBINES THE MERITS OF BOTH CENTRAL DIFFERENCING AND UPWIND DIFFERENCING SCHEMES.
Fig. 1. Free vortex convection in a uniform stream of sonic speed.
(a) Vorticity contours at the initial and final stations.
(b) Mach contours at the initial and final stations.
(c) Center line pressure profile at different time frames.
Temporal Operator

\[
\begin{bmatrix}
\alpha I + D_i & \Delta t + S_j & \Delta t + \Delta t & I_i & + \Delta t & U_j & U_i \\
\alpha I & D_j & \Delta t + \Delta t & I_j & + \Delta t & U_j & U_j \\
\alpha I & D_k & \Delta t + \Delta t & I_0 & + \Delta t & U_k & U_k \\
\end{bmatrix}^{(P)} \Delta Q
\]

\[
= \Delta t \cdot \text{RHS}
\]

\[
\text{RHS} = -\alpha \sum_{q=1}^{p-1} \frac{\Delta Q}{\Delta t} + (\alpha - 1) \frac{\Delta Q}{\Delta t}
\]

\[
- \frac{\partial E}{\partial \xi} - \frac{\partial F}{\partial \eta} - \frac{\partial G}{\partial \xi} + \frac{\partial (Ev)}{\partial \eta} + \frac{\partial (Fv)}{\partial \xi} + \frac{\partial (Gv)}{\partial \eta}
\]

Where

\[ D = A^+ - A^- + B^+ - B^- + C^+ - C^- \]

\[ S = \partial (\text{source terms})/\partial Q \]

L - Lower half of the flux Jacobian operators

U - Upper half of the flux Jacobian operators

**Algorithm: In 2 Consecutive Steps**

\[
[\alpha I + D\Delta t + S\Delta t + (L\Delta t)] \Delta Q = \Delta t \cdot \text{RHS}^{(n)}
\]

\[
[\alpha I + D\Delta t + S\Delta t] \Delta Q + [\Delta t L(\Delta Q) + \Delta t U(\Delta Q)]\Delta t = \Delta t \cdot \text{RHS}^{(n+1)}
\]

\[
[\alpha I + D\Delta t + S\Delta t + (U\Delta t)] \Delta Q = \Delta t \cdot \text{RHS}^{(n+1)}
\]

\[
[\alpha I + D\Delta t + S\Delta t] \Delta Q + [\Delta t L(\Delta Q) + \Delta t U(\Delta Q)]\Delta t = \Delta t \cdot \text{RHS}^{(n+1)}
\]
Formal Error in 2 Time Steps is

\[ 2 \cdot E_{RR} \sim -\Delta t^2 \cdot U \left( \frac{\delta \Delta Q^*}{\delta t} \right) \]

\[ + \Delta t^2 \cdot \left\{ L \left[ \alpha I + \Delta t(D+S) \right]^{-1} U \right\} (\Delta Q^*) \]

\[ - \Delta t^2 \cdot L \left( \frac{\delta \Delta Q^{**}}{\delta t} \right) \]

\[ + \Delta t^2 \cdot \left\{ U \left[ \alpha I + \Delta t(D+S) \right]^{-1} L \right\} (\Delta Q^{**}) \]

Define

\[ \frac{\delta Q}{\delta t} \equiv \alpha \frac{\Delta Q}{\Delta t}^{(n)} - (\alpha - 1) \frac{\Delta Q}{\Delta t}^{(n-1)} \]

\[ = \alpha \sum_{q=1}^{P} \frac{\Delta Q}{\Delta t}^{(q)} - (\alpha - 1) \frac{\Delta Q}{\Delta t}^{(n-1)} \]

Then

\[ \text{RHS} = -\frac{\delta Q}{\delta t} - \alpha \frac{\Delta Q}{\Delta t}^{(P)} \]

\[ = -\frac{\delta E}{\delta \xi} - \frac{\delta F}{\delta \eta} - \frac{\delta G}{\delta \xi} + \frac{\delta (Ev)}{\delta \xi} + \frac{\delta (Fv)}{\delta \eta} + \frac{\delta (Gv)}{\delta \zeta} \]
Geometry

\[ \Delta x = \Delta y = 1/4 \quad (\text{Vortex core radius} = 1) \]
\[ \Delta t = 0.05 \]
\[ \text{CFL} \sim 0.7 \]
\[ M \infty = 0.5 \]
\[ \gamma' = -0.5 \]
\[ \text{Innerm} = 4; \text{NORDR} = 2 \]
\[ \text{Inviscid} \]

\[ U, V, P, T = f_n(t) \]

Travel a total of 45 length in 900 steps.

Test 1 4th Order Central

Flux derivative in 1-D difference for

\[ E_i = \left[ \frac{2}{3} \frac{E_{i+1} - E_{i-1}}{h} - \frac{1}{12} \frac{E_{i+2} - E_{i-2}}{h} \right] \]

\[ - \sum_{n=5}^{N,2} \frac{1}{3} \frac{(4 - 2^{n-1})}{n!} E_i^{(n)} h^{n-1} (N \to \infty) \]

* Contains only the odd derivatives leading error is of order "4"
Pressure

STATIC PRESSURE
Vorticity Magnitude

Static Pressure

\begin{figure}
\centering
\begin{tikzpicture}
\draw[->, thick] (-2.5,0) -- (5,0);
\draw[->, thick] (0,-2.5) -- (0,5);
\end{tikzpicture}
\end{figure}
Consider:

\[ E'_{i} = [4\text{th order central differencing}] \]

\[ - \sum_{n=5}^{N} \frac{1}{3} \frac{(4 - 2^{n-1})}{n!} E_{i}^{(n)} h^{n-1} \]  \hspace{1cm} (A)

and

\[ E'_{i} = [3\text{rd order upwind biased differencing}] \]

\[ - \sum_{n=5}^{N} \frac{1}{3} \frac{(4 - 2^{n-1})}{n!} E_{i}^{(n)} h^{n-1} \]

\[ - \sum_{n=4}^{N} \frac{1}{3} \frac{(2 - 2^{n-1})}{n!} E_{i}^{(n)} h^{n-1} \]  \hspace{1cm} (B)

The Dissipative Equivalence is

\[ \text{Diss} = (A) - (B) \]

i.e.,

\[ \text{Diss} = - \sum_{n=4}^{N} \frac{1}{3} \frac{(2 - 2^{n-1})}{n!} E_{i}^{(n)} h^{n-1} \]

\[ = [3\text{rd order upwind difference}] \]

\[ - [4\text{th order central difference}] \]
Let \[ \frac{\delta E}{\delta x} = 4\text{th central} + \gamma \cdot \text{DISS} \]

Then =

\[ E_i = \{4\text{th order central differencing} \]
\[ + \gamma \cdot [ - \sum_{n=4}^{N,2} \frac{1}{3} \frac{(2-2^{n-1})}{n!} E_i^{(n)} h^{n-1}] \}
\[ - \sum_{n=5}^{N,2} \frac{1}{3} \frac{(4-2^{n-1})}{n!} E_i^{(n)} h^{n-1} \]

= \{(1-\gamma) \cdot [4\text{th order central difference}] \]
+ \gamma \cdot [3\text{rd order upwind biased difference }] \}
+ \text{Trunc. error} \\

Test 5. 3rd Order Upwind Biased Scheme

\[ E_i = \left[ \frac{1}{3h} (E_{i+1} - E_{i-1}) + \frac{1}{6h} (-3 E_i + 4 E_{i+1} - E_{i+2}) \right] \]

\[ - \sum_{n=5}^{N,2} \frac{1}{3} \frac{(4-2^{n-1}) (n)}{n!} E_i^{(n)} h^{n-1} \]
\[ (\text{odd derivatives}) \]

\[ - \sum_{n=4}^{N,2} \frac{1}{3} \frac{(2-2^{n-1})}{n!} E_i^{(n)} h^{n-1} \]
\[ (\text{even derivatives}) \]

* Leading error is of order "3"

Test 6

Set \( \gamma = 0.18 \) so that leading error term has a coefficient of 0.015
STATIC PRESSURE

![Graph showing static pressure trajectory](image)

Vortex core pressure trajectory

- Blended 3rd upwind ( Roe)
- Proposed scheme ($\alpha = 0.18$)
- 4th Central ($\beta = 0.015$)
- Analytical level
Case 2
Flow over a backward-facing step

\[
\begin{array}{c|c|c}
\text{Re}_d & 100 & 100 \\
& 389 & 389 \\
& 500 & 1000 \\
& 600 & \\
& 1000 & \\
\end{array}
\]

\(\uparrow\) 4th order central + artificial viscosity \((\beta = 0.5)\)
\(\uparrow\) Proposed scheme \((\gamma = 0.5)\)

Max. \(L_2\) - Residual \(< 5 \times 10^{-5}\) for all cases

\[
\begin{align*}
\text{M}_\infty &= 0.18 \\
\text{CFL} &= 50.0 \\
\text{Pr} &= 0.72 \\
\text{NORDR} &= 1 \\
\gamma &= 1.4 \\
\text{Innerm} &= 1 \\
\text{Air}, \text{viscous flow} &
\end{align*}
\]
Reattachment and Detachment Length I

Reattachment and Detachment Length II


42
Measurements

- Proposed scheme
- 4th Order central

**V - Velocity profiles**

Re = 100

- Measurements
- Proposed scheme
- 4th Order central

**V - Velocity profiles**

Re = 389
Measurements

Proposed scheme

4th Order control

V - Velocity profiles

Re = 1000

Static Pressure - 4th Order Central

Re = 1000

Re = 600

Re = 500

Re = 389

Re = 100

44
**CONCLUDING REMARKS**

* CENTRAL DIFFERENCING:
  SUSCEPTIBLE TO SPURIOUS OSCILLATIONS, ARTIFICIAL DAMPING NEEDS TO BE ADDED.

* UPWIND DIFFERENCING:
  INTRINSIC YET EXCESSIVE NUMERICAL DISSIPATION.

* PROPOSED SCHEME:
  USES 4th ORDER CENTRAL DIFFERENCING SCHEME WITH INTRINSIC NUMERICAL DISSIPATION PROVIDED BY THE UPWIND DIFFERENCING SCHEME.
An effort is currently underway at NASA Lewis to develop two- and three-dimensional Navier-Stokes codes, called Proteus, for aerospace propulsion applications. The emphasis in the development of Proteus is not algorithm development or research on numerical methods, but rather on the development of the code itself. The objective is to develop codes that are user-oriented, easily-modified, and well-documented. Well-proven, state-of-the-art solution algorithms are being used. Code readability, documentation (both internal and external), and validation are being emphasized.

Proteus solves the Reynolds-averaged, unsteady, compressible Navier-Stokes equations in strong conservation law form. Turbulence is modeled using a Baldwin-Lomax based algebraic eddy viscosity model (AIAA Paper 78-257.) The governing equations are written in Cartesian coordinates and transformed into generalized nonorthogonal body-fitted coordinates. They are solved by marching in time using a fully-coupled ADI solution procedure (Briley and McDonald, J. Comp. Phys., Aug. 1977) with generalized first- or second-order time differencing (Beam and Warming, AIAA J., Apr. 1978.) The boundary conditions are also treated implicitly, and may be steady or unsteady. All terms, including the diffusion terms, are linearized using second order Taylor series expansions.

In addition to solving the full set of Reynolds-averaged equations, options are available to solve the thin-layer or Euler equations, and to eliminate the energy equation by assuming constant stagnation enthalpy. Artificial viscosity is used to minimize the odd-even decoupling resulting from the use of central spatial differencing for the convective terms, and to control pre- and post-shock oscillations in supersonic flow. Two artificial viscosity models are available - a combination implicit/explicit constant coefficient model (Stegel, AIAA J., July 1978), and an explicit non-linear coefficient model designed specifically for flows with shock waves (Jameson, Schmidt, and Turkel, AIAA Paper 81-1259.) At NASA Lewis, the code is run on the Cray X-MP and Y-MP computers, and is highly vectorized.

An extensive series of validation cases have been run, primarily using the two-dimensional planar/axisymmetric version of the code. Several flows were computed for which exact solutions to the Navier-Stokes equations exist, including fully-developed channel and pipe flow, Couette flow with and without a pressure gradient, unsteady Couette flow formation, flow near a suddenly accelerated flat plate, flow between concentric rotating cylinders, and flow near a rotating disk. Additional validation cases that have been successfully run include flat plate laminar and turbulent boundary layers, boundary layers with confined separation, 2-D and 3-D driven cavities, normal and oblique shock-boundary layer interactions, steady and unsteady flows past a circular cylinder, developing laminar and turbulent pipe flows, and steady and unsteady flows in a transonic diffuser. The figure shows computed Mach number contours and the static pressure distribution for flow through a transonic diffuser.

The documentation consists of a three-volume user’s manual (Towne, Schwab, Benson, and Suresh, NASA TM’s 102551-102553). Volume 1 is the Analysis Description, and presents the equations and solution procedure. It describes in detail the governing equations, the turbulence model, the linearization of the equations and boundary conditions, the time and space differencing formulas, the ADI solution procedure, and the artificial viscosity models. Volume 2 is the User’s Guide, and contains information needed to run the program. It describes the program’s general features, the input and output, the procedure for setting up initial conditions, the computer resource requirements, the diagnostic messages that may be generated, the job control language used to run the program, and several test cases. Volume 3 is the Programmer’s Reference, and contains detailed information useful when modifying the program. It describes the program structure, the Fortran variables stored in common blocks, and the details of each subprogram.
THE PROTEUS NAVIER-STOKES CODE

By

C. TOWNE
Inlet, Duct & Nozzle Flow Physics Branch
And

J. SCHWAB
Turbomachinery Flow Physics Branch

NAVIER-STOKES ANALYSIS
FOR TRANSONIC DIFFUSER

STATIC PRESSURE

-5.0 -2.5 0.0 2.5 5.0 7.5 10.0
X-COORDINATE
PROTEUS NAVIER-STOKES CODE

• Objective
  — Develop user-oriented, easily-modified, well-documented, 2- and 3-dimensional Navier-Stokes codes for aerospace propulsion applications.

• Approach
  — Use well-proven, state-of-the-art algorithms.
  — Use a consistent, modular code structure.
  — Emphasize documentation, code readability, and validation.

PROTEUS NAVIER-STOKES CODE
ANALYSIS SUMMARY

• Reynolds-averaged, unsteady, compressible Navier-Stokes equations.

• Strong conservation-law form.

• Generalized nonorthogonal body-fitted coordinates.

• Convection and diffusion terms linearized using second-order Taylor series expansion.

• Baldwin-Lomax algebraic turbulence model.

• Fully-coupled ADI solution procedure, Beam-Warming generalized time differencing.

• Implicit steady/unsteady boundary conditions.
PROTEUS NAVIER-STOKES CODE
CODE FEATURES

- 2-D, axisymmetric with swirl, or 3-D flow.
- Thin-layer and Euler options.
- Wide variety of boundary conditions.
- Constant stagnation enthalpy option.
- Constant-coefficient or adaptive artificial viscosity.
- First- or second-order time differencing.
- Variety of time step selection methods.
- Output files for CONTOUR and PLOT3D.
- Highly vectorized for Cray computers.
- Extensively commented.
- Three-volume documentation set.
  — Analysis Description.
  — User's Guide.
  — Programmer's Reference.

PROTEUS NAVIER-STOKES CODE
VALIDATION CASES

- Exact Navier-Stokes solutions.
  — Fully-developed channel and pipe flow.
  — Couette flow w/wo pressure gradient.
  — Couette flow formation.
  — Flow near a suddenly-accelerated flat plate.
  — Flow between concentric rotating cylinders.
  — Flow near a rotating disk.
PROTEUS NAVIER-STOKES CODE
VALIDATION CASES

- Blasius flat plate boundary layer.
- Flat plate boundary layer with confined separation.
- Turbulent flat plate boundary layer w/wo heat transfer.
- Flow over a rearward-facing step.
- 2-D and 3-D driven cavity.
- Normal and oblique shock formation.
- Steady and unsteady flow past a circular cylinder.
- Flow past an airfoil.
- Developing laminar and turbulent flow in a channel and pipe w/wo swirl.
- Developing 3-D flow in a square duct.
- Steady and unsteady turbulent flow in a transonic diffuser.
Developing Turbulent Pipe Flow

![Graph showing x-velocity vs. x-coordinate for various radial distances (r/R).]

Flow Past A Circular Cylinder

![Graph showing static pressure coefficient (c_p) vs. circumferential location (θ) for different methods like Proteus, viscous, Proteus, Euler, and Exact Potential Flow.]
NAVIER-STOKES ANALYSIS FOR TRANSONIC DIFFUSER

PROTEUS NAVIER-STOKES CODE

HISTORY

• 4-86 Start.
• 9-87 First master file.
• 9-88 Emphasis placed on 2-D code development.
• 9-89 Version 1.0 of 2-D code released.
• 3-90 Documentation published for version 1.0 of 2-D code.
• 9-90 Version 1.2 of 2-D code released.
• 3-91 Preliminary Version 1.0 of 3-D code ready for testing.
PROTEUS NAVIER-STOKES CODE
CURRENT STATUS

- Version 1.2 of 2-D code available.
- 3-D code and documentation being upgraded.
- Two-equation $k$-$\epsilon$ turbulence model available in 2-D code.
- Grid patching (embedded boundary) capability available for 2-D code.
- Validation studies underway.
  - Laminar/turbulent compressible flat plate boundary layers
  - Laminar/turbulent flow past an airfoil.
  - Turbulent flow in transonic diffusers.

PROTEUS NAVIER-STOKES CODE
NEAR-TERM PLANS

- Complete upgrade of 3-D code and documentation.
- Investigate additional algebraic and pde turbulence models.
- Create diagonalized version.
- Continue validation of 2-D code, emphasizing turbulent flow and flow with heat transfer.
- Begin more intensive validation of 3-D code.
- Release 3-D code in fall of 1991.
PROTEUS NAVIER-STOKES CODE
LONG-TERM PLANS

- Continue validation of 2-D and 3-D codes.
- Add zonal gridding capability.
- Investigate convergence acceleration techniques.
- Add algebraic Reynolds stress turbulence model.
- Investigate adaptive mesh schemes.
- Hold user's workshop after 3-D code release.
The well-known MUSCL scheme of Van Leer (J. Comp. Phys., 1979)—a second-order extension of Godunov's first-order upwind-difference method—is constructed using a piecewise linear approximation. In order that the solutions are free from oscillations, a constraint is imposed on the slopes of the piecewise linear reconstruction. The MUSCL scheme is second-order accurate at the smooth part of the solution except at extrema where the accuracy degenerates to first-order due to the monotonicity constraint. Harten and Osher (SIAM J. Numer. Anal., 1987) resolved this loss of accuracy by introducing the concept of nonoscillatory reconstruction. Their UNO2 scheme involves no constraint, is uniformly second-order accurate, and does not create oscillations. The UNO2 scheme is, however, very diffusive, e.g., it smears contact discontinuities.

To construct accurate schemes which are free from oscillations, we first introduce the concept of upwind-monotonicity. Next, in a geometric framework in which the median function plays a crucial role, we present Van Leer's constraint and the UNO2 scheme. In this framework, the constraint is extended in such a manner that extrema are not "clipped", while upwind-monotonicity is preserved. At monotone part of the data, the new constraints reduce to that of Van Leer for the second-order case, and in the third order case, that of Colella and Woodward (J. Comp. Phys., 1984). Several classes of schemes, which are upwind-monotone and of uniform second or third-order accuracy, are then presented. These schemes also satisfy the nonoscillatory property of the UNO2 scheme. It is shown that the MUSCL, UNO2 and all the popular TVD (total variation diminishing) schemes (Harten, J. Comp. Phys., 1983; or Sweby, SIAM J. Numer. Anal., 1984) are members of the new class. Moreover, each TVD scheme corresponds to a uniform second-order accurate upwind-monotone scheme. The gain of accuracy is obtained by a few extra lines of Fortran programming. These new schemes are also extensions of the authors SONIC schemes (NASA TM 102010, 1989; also Yokota & Huynh, NASA TM 102354, 1990).

Results for advection with constant speed are shown below. The initial profile consists of a sin² wave, a square wave, a triangle wave and a semi-ellipse wave. Each wave contains 20 grid points on a uniform grid of 200 points. Using periodic boundary conditions, the profile is advected one period with CFL number 0.5. The results after 400 time steps are shown in Figs. (A) for a member of the new class, (B) for the UNO2, (C) for the PPM (Colella and Woodward, J. Comp. Phys., 1984), and (D) for Roe's "Superbee" schemes. It can be seen that the new scheme compares favorably with the state-of-the-art methods.
Upwind Shock-Capturing Methods

Why new methods?

TVD (Total Variation Diminishing) schemes
+ Capture shocks with high resolution
- First-order accurate near extrema

UNO2 scheme
+ Uniformly second-order accurate
+ Nonoscillatory, i.e., no new strict extrema
- Somewhat diffusive

New Class of Methods

New geometric framework

Existing schemes (TVD, UNO)
are presented in the new framework

New schemes
* Uniformly second-order accurate
* Upwind-monotone and nonoscillatory
* Extensions of TVD and UNO2 schemes

\[ \frac{\partial u}{\partial \tau} + a \frac{\partial u}{\partial x} = 0, \quad a > 0 \]
\[ u(x, 0) = u_0(x). \]

Approach

Projection or Reconstruction,
Evolution or Upwinding + Time stepping
The task for piecewise linear reconstruction: given the data \( \{u_j\} \), define \((\text{slope})_j\)

\[ m(s, t) = \text{median}(s, t, 0) \]

\[ = \frac{1}{2} [\text{sgn}(s) + \text{sgn}(t)] \min(|s|, |t|) \]
The pictures

Constraint: \((\text{slope})_j\) lies between 0 and \(2r\),

\[ r = m(s, t). \]

Algorithm: \((\text{slope})_j \leftarrow m\left[(\text{slope})_j, 2r\right]\)

TVD schemes

Minmod scheme

\[(\text{slope})_j = r = m(s, t)\]

Average limiter

\[(\text{slope})_j = m\left(\frac{s + t}{2}, 2r\right)\]

Superbee scheme

\[(\text{slope})_j = m\left[\maxmod(s, t), 2r\right]\]

\[
\maxmod(s, t) = \frac{1}{2}[\sgn(s) + \sgn(t)] \max(|s|, |t|)
\]
The Peter-Paul Principle

Resolve shock well (Paul), 1st-order at extrema (Peter).

For TVD schemes, $u_j^{n+1}$ lies between $u_j^n$ and $u_{j-1}^n$.

The data are monotone in $[x_j, x_{j+1}]$ if

- $u_{j-1} \leq u_j \leq u_{j+1} \leq u_{j+2}$, or
- $u_{j-1} \geq u_j \geq u_{j+1} \geq u_{j+2}$.

A scheme is upwind-monotone if monotonicity of the data in $[x_{j-1}, x_j]$ implies $u_j^{n+1}$ lies between $u_j$ and $u_{j-1}$.

**UNO2 scheme**

(slope)$_j = \tilde{r} = m(\tilde{s}, \tilde{t})$

+ Second-order accurate
+ Nonoscillatory
- Diffusive
New methods

New constraint:

\[(\text{slope})_j \text{ lies in } I[0, 2r, \tilde{r}] = I[0, r_{\text{max}}]\]

\[r_{\text{max}} = \text{sgn}(\tilde{r}) \max (2|r|, |\tilde{r}|).\]

New algorithm: \((\text{slope})_j \leftarrow m \left[(\text{slope})_j, r_{\text{max}}\right].\)

**UNO** \((\text{slope})_j = \tilde{r} = m(\tilde{s}, \tilde{t})\)

**SONIC-A** \((\text{slope})_j = m(\frac{\tilde{s} + \tilde{t}}{2}, r_{\text{max}})\)

**SONIC-B** \((\text{slope})_j = m(\max \text{mod}(\tilde{s}, \tilde{t}), r_{\text{max}})\)

Second-order accurate and upwind-monotone
Conclusions

A new framework for shock-capturing methods was presented
  . Simple
  . Geometric
  . Uses the median function

The approach leads to new classes of methods:
  . Compare favorably with existing methods
  . Resolve discontinuities with high resolution
  . Uniformly second-order accurate
  . Upwind-monotone (therefore, nonoscillatory)
  . Generalization of TVD and UNO2 schemes
  . Vectorize on Cray
A new numerical framework for solving conservation laws is being developed. This new approach differs substantially from the well established methods, i.e., finite difference, finite volume, finite element and spectral methods, in both concept and methodology. The key features of the current explicit scheme include: (a) direct discretization of the integral forms of conservation laws, (b) treating space and time on the same footing, (c) flux conservation in space and time, and (d) unified treatment of the convection and diffusion fluxes.

The model equation considered in the preliminary study is the standard 1-D unsteady constant-coefficient convection-diffusion equation. In a stability study, it is shown that the principal and spurious amplification factors of the current scheme, respectively, are structurally similar to those of the leapfrog/DuFort-Frankel scheme. As a result, the current scheme has no numerical diffusion in the special case of pure convection and is unconditionally stable in the special case of pure diffusion.

Assuming smooth initial data, it will be shown theoretically and numerically that, by using an easily determined optimal time-step, the accuracy of the current scheme may reach a level which is several orders of magnitude higher than that of the MacCormack scheme, with virtually identical operation count.
Introduction

The focus of the current method development is entirely on the integral form of conservation laws. Key features of the current explicit scheme include:

- Direct discretization of the integral forms of conservation laws.
- Treating space and time on the same footing.
- Flux conservation in space and time.
- Unified treatment of the convection and diffusion fluxes.
- Low-operation count and high accuracy.

Unsteady 1-D Convection-Diffusion Equation

\[
\frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} - \mu \frac{\partial^2 u}{\partial x^2} = 0
\]

or

\[
\vec{\nabla} \cdot \vec{h} = 0
\]

with

\[
\vec{h} \overset{\text{def}}{=} \left( au - \mu \frac{\partial u}{\partial x} , u \right)
\]

or

\[
\oint_{S(V)} \vec{h} \cdot d\vec{s} = 0
\]

\( V = \text{Any volume in space-time.} \)
\( S(V) = \text{Boundary of } V. \)
\( \vec{h} \cdot d\vec{s} = \text{Flux of } \vec{h} \text{ leaving } V \text{ through surface element } d\vec{s}. \)
Basic Conservation Domains (BCD's)

For each BCD, it is required that

\[ \text{Sum of incoming fluxes} = \text{sum of outgoing fluxes} \quad <1> \]

As a result, \(<1>\) is also true for the union of any collection of BCD's.

Space - Time Mesh and BCD's

\[ b \overset{\text{def}}{=} \left( \begin{array}{c} dx \\ dt \end{array} \right) \text{ along a } j \text{ mesh line} \]

\[ x^n_j = j \Delta x + n \cdot b \Delta t, \quad t^n = n \Delta t \]
The Solution in a BCD

\[ D(j,n) = \text{BCD centered at } (x_j^n, t^n). \]
\[ D'(j,n) = \text{Interior of } D(j,n). \]

\[ u(x,t; j,n) = \text{Approximation of } u(x,t) \text{ in } D'(j,n) \]
\[ = \alpha_j^n (x-x_j^n) + \beta_j^n (t-t^n) + \gamma_j^n \]

Since total flux leaving \( D'(j,n) = 0 \)
\[ \beta_j^n = -\alpha_j^n \]

Thus
\[ u(x,t; j,n) = \alpha_j^n [(x-x_j^n) - a(t-t^n)] + \gamma_j^n \]

Fundamental Relations (1)

\[ u(x,t; j,n) = \alpha_j^n [(x-x_j^n) - a(t-t^n)] + \gamma_j^n \]
\[ u(x_j^n, t^n; j,n) = \gamma_j^n \]

\[ \frac{\partial u(x,t; j,n)}{\partial x} = \alpha_j^n \]
\[ x = x_j^n, t = t^n \]

\[ (\tau,8) \quad \frac{\Delta x}{2} \gamma_j^n \left( \frac{\Delta x}{2} \right)^2 \]
\[ \alpha_j^n \]

\[ (\tau,8) \quad f^{(1)}_j(j,n), f^{(1)}_2(j,n) \]

\[ (\tau,8) \quad f^{(0)}_1(j,n), f^{(0)}_2(j,n) \]

\( \tau = \frac{(a-b)\Delta t}{\Delta x} \)
\( \delta = \frac{4\mu\Delta t}{(\Delta x)^2} \)
Invariant Property (1)

Let
\[ u = u_0(x,t; a, \mu) \quad <1> \]
be a solution of
\[ \frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} - \mu \frac{\partial^2 u}{\partial x^2} = 0 \quad <2> \]

Let \( x_* \) and \( t_* \) be any real constants. Then
\[ u = u_0(2x_* - x, 2t_* - t; a, -\mu) \quad <3> \]
is also a solution of <2>. As an example, consider
\[ u_0(x, t; a, \mu) \overset{\text{def}}{=} e^{-4\pi^2 \mu t} \sin[2\pi(x-at)] \quad <4> \]
The above property may be referred to as the p-t-\( \mu \) (parity-time reversal-\( \mu \) reversal) invariance.

Invariant Property (2)

The main scheme of the Leapfrog/DuFort-Frankel method i.e.,
\[ \frac{u_{j+1}^n - u_j^{n-1}}{2\Delta t} + a \frac{u_{j+1}^n - u_{j-1}^n}{2\Delta x} = \mu \frac{u_{j+1}^n - u_{j-1}^n + u_j^{n-1} - u_j^n}{(\Delta x)^2} \quad <5> \]
is also p-t-\( \mu \) invariant. In other words, if
\[ u_j^n = u_0(j, n; a, \mu) \]
satisfy <5>, then so does \((j, n)\) and \(n\) are constant integers)
\[ u_j^n = u_0(2j_*, j, 2n_* - n; a, -\mu) \]
The present scheme is p-t-\( \mu \) invariant also. Other classical schemes generally are not p-t-\( \mu \) invariant. p-t-\( \mu \) invariance has an impact on the stability property.
Stability Condition

1. The stability condition for the present scheme is

\[ 1 \geq \tau^2 \]

i.e., stability is not dependent on the viscosity \( \mu \).

2. The present scheme has no numerical diffusion in the special case of pure convection, and is unconditionally stable in the special case of pure diffusion.

Inconsistency

- A smooth solution which satisfies the present or the Leapfrog/ Dufort-Frankel scheme at the mesh points will satisfy

\[ \frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} - \mu \frac{\partial^2 u}{\partial x^2} = 0 \quad (\mu \neq 0) \quad <1> \]

as \( \Delta x \to 0 \) and \( \Delta t/\Delta x \to 0 \).

- A characteristic line of <1> is \( t = \text{constant} \).

Thus a discrete solution may converge to an analytical solution only if \( (\Delta t/\Delta x) \to 0 \).

- For a typical classical scheme, stability requires that

\[ \frac{\mu \Delta t}{(\Delta x)^2} \equiv \frac{\mu (\Delta t/\Delta x)}{\Delta x} \] be bounded. Thus, \( (\Delta t/\Delta x) \to 0 \) as \( \Delta x \to 0 \).
Space - Time Mesh and BCD's

\[ b \text{def } \frac{dx}{dt} \text{ along a } j \text{ mesh line} \]

\[ x_j^n = j \Delta x + n \cdot b \Delta t, \quad t^n = n \Delta t \]

Problem Definition (1)

Analytical Solution: \[ u = u_a(x, t) = e^{-4\pi^2 \mu t} \sin[2\pi(x - at)] \]

\[ \gamma_j^0 = u_a(j \Delta x, 0) = \sin(2\pi j \Delta x), \quad \alpha_j^0 = \frac{\partial u_a(j \Delta x, 0)}{\partial x} = 2\pi \cos(2\pi j \Delta x) \]

wavelength = \(K\Delta x = 1\) \[ u(x, t; j, n) = \alpha_j^n \left[ (x - x_j^n) - a(t - t^n) \right] + \gamma_j^n \]
Problem Definition (2)

A problem is defined by specifying the values of (i) physical parameters $a$ and $\mu$, (ii) the mesh parameters $b$, $\Delta t$ and $\Delta x (= 1/K)$, and (iii) total running time $t$. The total time-step number

$$n = \text{IN}(t/\Delta t) \overset{\text{def}}{=} \text{the integer nearest to the ratio } (t/\Delta t)$$

Error of a problem is measured by:

$$E(b,n,\Delta t,K) \overset{\text{def}}{=} \log_{10} \left[ \frac{1}{e^{-4\pi^2\mu t}} \cdot \frac{1}{K} \left| \sum_{j=1}^{k} u_j^n - u_a(x_j^n,t^n) \right| \right]$$

Accuracy of present method is optimized when either

(i) $\tau = 0$ or (ii) $1 - \tau^2 = \sqrt{3} \delta$.

$$\tau \overset{\text{def}}{=} (a - b)\Delta t/\Delta x, \quad \delta \overset{\text{def}}{=} 4\mu \Delta t/(\Delta x)^2$$

![Graph](image)

**Parameters**

- $a = 1$, $\mu = 0.1$
- $b = 0$, $K = 60$
- $t = 0.5$
- $\Delta t$ varies

**Re**

$$\text{Re} \overset{\text{def}}{=} \tau / \gamma$$

$$= \frac{(a-b)\Delta x}{4\mu} = \frac{1}{24}$$

$$\tau = \frac{(a-b)\Delta t}{\Delta x} = 60\Delta t$$

**Optimization:**

$$1 - \tau^2 = \sqrt{3} \gamma$$
Special Property

Let $K$ be the number of spatial mesh points. Then the present marching scheme may be expressed as

\[ \tilde{u}^{n+1} = A \tilde{u}^n + \tilde{b}^n, \quad n = 0, 1, 2, \ldots \]

Here (i) $\tilde{u}^n$ and $\tilde{u}^{n+1}$, respectively, are the $2K \times 1$ column matrices representing $2K$ numerical variables at the time levels $n$ and $n+1$, (ii) $\tilde{b}^n$ is a given $2K \times 1$ column matrix at the time level $n$, and (iii) $A$ is a give $2K \times 2K$ sparse matrix.

Unusual property: $A^{-1}$ is also sparse and is known explicitly
Conclusions (1)

- Flux conservation is the focus of method development.
- Unified treatment of space and time, also of convection and diffusion fluxes.
- Stability is not dependent on the viscosity \( \mu \). The present scheme has no numerical diffusion if \( \mu = 0 \) and is unconditionally stable if \( a = 0 \).
- The sparse matrix \( A \) in the present marching scheme \( \tilde{u}^{n+1} = A \tilde{u}^n + \tilde{e}^n \) has the unusual property that \( A^{-1} \) is also sparse and is known explicitly.
- The present scheme has the same invariant property of the physical equation it models.

Conclusions (2)

- The present scheme is most accurate and stable when \( \tau = 0 \), i.e., \( b = a \)
- For the present scheme, mesh refinement may be made locally without impacting global structure.
- Since initial data required include both \( u_j^0 \) and \( (\partial u/\partial x)^o \), initial-value specification of the present method is more accurate than that of traditional methods.
- Assuming smooth, initial data, it is shown that, by using an easily determined optimal time-step, the accuracy of the current scheme may reach a level which is several orders of magnitude higher than the MacCormack scheme, with virtually identical operation count.
A New Lagrangian Method for Real Gases at Supersonic Speed

C.Y. Loh and Meng-Sing Liou
Internal Fluid Mechanics Division
NASA Lewis Research Center

Abstract

With the renewed interest in high-speed flights, the real gas effect is of theoretical as well as practical importance. In general, this real gas phenomenon is seen in the regions of high gradients and discontinuities. The ability of numerically capturing them accurately, viz with minimal numerical dissipation and oscillations, has been the main challenge to the computational fluid dynamist as well as numerical analyst over the past four decades. In the past decade, upwind splittings or Godunov-type Riemann solutions have received tremendous attention and as a result significant progress has been made both in the ideal and non-ideal gas. However, almost all of these efforts have been exclusively formulated in the framework of Eulerian description of fluid motion. Although the Lagrangian description has been used in several reports, they are always followed by a remap step from a Lagrangian grid back to the original Eulerian one. This not only involves tedious interpolation procedure but also can corrupt the accuracy which was gained by the choice of the Lagrangian approach over the Eulerian one.

However, since all upwind schemes used by today's major CFD codes are strictly formulated in 1-D framework, and only formally extended to multi-dimensions in space. Consequently, the attractive property of crisp representation of discontinuities is lost and search for genuine multi-dimensional approach has just been undertaken by several leading researchers, but still based on Eulerian description. In this paper, we propose a new alternative that is formulated using the Lagrangian description, for the calculation of supersonic/hypersonic real-gas inviscid flows. This new formulation avoids grid generation step which is automatically obtained as solution procedure marches in the "time-like" direction. As a result, no remapping is required and the accuracy is faithfully maintained in the Lagrangian level. Moreover, the ability of preserving the sharpness of discontinuities in muti-dimensions is vividly demonstrated. In this paper, we will give numerical results for a variety of real-gas problems consisting of essential elements in the high-speed flows, such as shock waves, expansion waves, slip surfaces and their interactions. Finally, calculations for flows in a generic inlet and a nozzle are presented.
Outline

. New Lagrangian formulation and the conservation form for 2-D steady supersonic flow (for perfect gas)

. Advantages of the new Lagrangian method

. Real gas version of the new Lagrangian method

. Godunov scheme and TVD scheme

. Numerical examples for supersonic real gas flow

. Concluding Remark

New Lagrangian formulation and the conservation form for 2-D steady supersonic flow (perfect gas)

Independent variables of Eulerian formulation: x, y

Independent variables of the new Lagrangian formulation: \( \tau, \xi \) (Hui and Van Roessel, 1985)

\[ \tau -- Lagrangian \ time \]

\[ \xi -- stream \ function \]


\[ \frac{\partial E}{\partial \tau} + \frac{\partial F}{\partial \xi} = 0 \quad (1) \]
where
\[ u = \frac{\partial x}{\partial \tau}, \quad v = \frac{\partial y}{\partial \tau} \]
\[ U = \frac{\partial x}{\partial \xi}, \quad V = \frac{\partial y}{\partial \xi} \]
\[ K = \rho(uV - vU) \]
\[ H = \frac{u^2 + v^2}{2} + h(p, \rho) \]
\[ h(p, \rho) = \frac{\gamma p}{(\gamma - 1)\rho} \]

Advantages of the new Lagrangian method

(1) Truly multi-dimensional computation (no time splitting), a computational cell is literally a fluid particle, physics is closely followed.

(2) Crisp slip line (contact) resolution. (1 point)

(3) No grid generation is needed, grids are automatically generated following streamlines.

(4) Particularly appropriate for hypersonic computation

(5) Ready for parallel computation.

Real gas version of
the new Lagrangian method

- Accurate real gas Riemann solver
- Decoding of \( E \) to find \( u,v,p,\rho \)

Tannehill’s EOS is used (NASA CR 2470, 1974)
Godunov/TVD scheme

1. Initialize $u, v, p, \rho, U$ and $V$
2. Riemann solver
   - Compute flux vector $F$
   - Compute new vector $E$
   - Upgrade $E$ by TVD limiter (Sweby)
3. TVD scheme
4. Decode $E$ and output $u, v, p, \rho, U$ and $V$
5. Output $x$ and $y$ coordinates

Time lines and streamlines

Physical plane

Computational plane

Boundary cell $j = 1$

Computational Domain and Mesh

Pressure, density and temperature contours for the real gas channel problem: (a) isobars, (b) isopycnics
Numerical examples for real gas supersonic flows

. Riemann problem

. Shock collision in converging channel

. Generic nozzle flow
Pressure, density and temperature contours for a sophisticated real gas outlet problem, showing interactions among the waves and shock reflection on the body surface; (a) schematic sketch, (b) isobars, (c) isotherms and (d) isopycnics.

Concluding Remarks

Accurate yet efficient real gas effect computation:

- As accurate as Tannehill’s EOS warrants
- Genuine steady approach saves CPU
- Bears all the advantages of the Lagrangian method mentioned above
A TIME-ACCURATE IMPLICIT METHOD FOR CHEMICAL NON-EQUILIBRIUM FLOWS AT ALL SPEEDS

Jian-Shun Shuen
Sverdrup Technology, Inc.
Lewis Research Center Group
NASA Lewis Research Center

Summary

A new time-accurate coupled solution procedure for solving the chemical non-equilibrium Navier-Stokes equations over a wide range of Mach numbers is described. The scheme is shown to be very efficient and robust for flows with velocities range from \( M \leq 10^{-10} \) to supersonic speeds.

Description

Chemically reacting flows in aeropropulsion systems are often not amenable to numerical algorithms developed for high speed compressible flows. Examples include the rocket motor flow which involves a wide range of Mach numbers (from near zero velocity at the closed end to supersonic at the nozzle exit) and flow in gas turbine combustor where velocity is in the low subsonic range, yet the variation in density is very large so as to preclude an incompressible approach. Unfortunately, most available solution methods are usually applicable to either compressible flow with moderate to high Mach numbers or incompressible flows, but not both. This difficulty points out the need for a single solution technique that can be employed in flows with wide Mach number range and large property variations.

There are two underlying reasons for the difficulty of low Mach number computations in compressible flows. When the Mach number becomes low, the eigenvalues of the system differ widely so that the equations becomes stiff, resulting in significant slowdown in the convergence rate. Another reason is the singular behavior of the pressure term in the momentum equations as Mach number approaches zero, causing large roundoff error and smearing the pressure variation field, and, consequently, preventing the convergence.

In the present numerical algorithm, methods to overcome these difficulties are proposed and successfully tested. The resulting code is then extended to consider the effects of non-equilibrium chemistry and real-gas properties. Included in the abstract are results from a linear stability analyses for the present algorithm (referred to as the pressure based scheme) and a conventional compressible flow algorithm (density-based algorithms), and convergence histories for the convergent-divergent nozzle flows. The transonic nozzle flow has an inlet Mach number of 0.03 and exit Mach number 3.0. The subsonic cases are obtained by adjusting the nozzle back pressure so that flow is not choked at the throat. These results demonstrate the efficiency of the present method for flows over a wide Mach number range.
MOTIVATION

- Reacting flows in aeropropulsion devices are often not amenable to numerical algorithms developed for high speed compressible flows, e.g.,
  - rocket motor – wide range of Mach numbers, from near zero velocity at closed end to supersonic flow at nozzle exit.
  - gas turbine combustor – low subsonic velocities, but large variation in density precludes incompressible approach.

- Most popular codes for gas turbine combustor and other low-speed reacting flows are TEACH-type codes, which were based on technologies developed more than 20 years ago.

- Compressible flow CFD technologies can be carried over to low-speed flow regime.

OBJECTIVES

1. Development of an efficient and robust Navier-Stokes code for chemically reacting flows at all speeds.

2. Implementation of liquid-fuel spray combustion and turbulent combustion closure models.

3. Demonstration of the capability of the code for aeropropulsion applications, including flows in gas turbine combustor, after burner, liquid propellant rocket combustion chamber, etc.
STABILITY ANALYSIS, CFL=100

Magnitude of the Maximum Eigenvalue of the Amplification Matrix G, Pressure-Based Scheme, CFL=100.

Magnitude of the Maximum Eigenvalue of the Amplification Matrix G, Density-Based Scheme, CFL=100.

Convergence History of the Pressure-Based Scheme for the C-D Nozzle Flow, CFL=100.

Convergence History of the Density-Based Scheme for C-D Nozzle Flow, CFL=100.
Difficulties with Compressible Flow Algorithms at Low Mach Numbers

- Disparities among system’s eigenvalues (stiffness), \( u, u + c, u - c \), resulting in significant slowdown in convergence rate.

- Singular behavior of pressure gradient term in momentum equations as Mach number approaches zero,

\[
\rho^* u \cdot \frac{u^*}{\gamma M_f^2}
\]

As Mach number is decreased, pressure variation \((\Delta p^* \propto M^2)\) becomes of similar magnitude as roundoff error of the large pressure gradient term \((p^* / \gamma M_f^2)\).

**METHOD OF APPROACH**

**Pressure Singularity Problem**

- Pressure decomposed into two parts:

\[
p = p_o + p_g
\]

\( p_g \) replaces \( p \) in momentum equations and retains \( p_g \) as one of the unknowns.

- Employs conservative form of the governing equations, but uses primitive variables

\[
(p_g, u, v, h, Y_i)
\]

as unknowns. Conservation property preserved for flows at moderate to high Mach numbers. Pressure field accurately resolved for low Mach number flows.

**Eigenvalue Stiffness Problem**

- Pressure rescaled so that all eigenvalues have the same order of magnitude. Physical acoustic waves removed and replaced by a set of pseudo-acoustic waves which travel at speed comparable to fluid convective velocity.
**Discretized Equations**

\[
(T - \Delta r D) + \frac{3 \Delta r}{2 \Delta t} T + \Delta r (\frac{\partial A}{\partial x} - \frac{\partial^2 A_u}{\partial x^2}) \Delta \tilde{Q} = -\Delta r (R)^p
\]

where

\[
R = \frac{3Q^p - 4Q^n + Q^{n-1}}{2\Delta t} + \frac{\partial (E - E_u)}{\partial x} - H
\]

\[
A = \frac{\partial E}{\partial Q}, \quad A_u = \frac{\partial E_u}{\partial Q},
\]

\[
D = \frac{\partial H}{\partial Q}, \quad T = \frac{\partial Q}{\partial Q}.
\]

- Use dual-time stepping technique for time-accurate solutions.
- Fully implicit, fully coupled solution method. Use LU factorization for multi-dimensional flows.

**ONE DIMENSIONAL GOVERNING EQUATIONS**

\[
\Gamma \frac{\partial \tilde{Q}}{\partial \tau} + \frac{\partial Q}{\partial t} + \frac{\partial (E - E_u)}{\partial x} = H
\]

\[
\dot{Q} = A_f \begin{pmatrix} p \cr u \cr h \cr \rho \end{pmatrix}, \quad Q = A_f \begin{pmatrix} \rho \cr \rho u \cr \rho e \cr \rho Y_1 \end{pmatrix}, \quad \Gamma = \begin{pmatrix} 1 & 0 & 0 & 0 & \cdots & 0 \\ u & 0 & 0 & 0 & \cdots & 0 \\ h & 0 & 0 & 0 & \cdots & 0 \\ Y_1 & 0 & 0 & 0 & \cdots & 0 \\ \vdots & \vdots & \vdots & \vdots & \ddots & \vdots \\ Y_{N-1} & 0 & 0 & \cdots & \cdots & 0 \end{pmatrix}
\]

\[
E = A_f \begin{pmatrix} \rho u \cr \rho u^2 + p \cr (\rho + p)u \cr \rho u Y_1 \cr \vdots \cr \rho u Y_{N-1} \end{pmatrix}, \quad E_u = A_f \begin{pmatrix} 0 \\ \frac{\partial \rho}{\partial x} \\ q_e \\ q_i \cr \vdots \cr q_{N-1} \end{pmatrix}, \quad H = \begin{pmatrix} 0 \\ \rho \frac{\partial x}{\partial x} \\ 0 \\ A_f \omega_1 \\ \vdots \cr A_f \omega_{N-1} \end{pmatrix}
\]

\[
\lambda_1 = u, \quad \lambda_{2,3} = \frac{1}{2} \left[ \alpha \pm \sqrt{\alpha^2 - 4\beta(M^2 - 1)} \right], \quad \alpha = u \left( \frac{3\beta}{c^2} + 1 \right)
\]

\[
\beta = \frac{u^2}{d}, \quad \text{As } M \rightarrow 0, \quad \lambda_{\text{max}} / \lambda_{\text{min}} = 4.8
\]
Convergence History for the C-D Nozzle Flow, CFL=100.

Non-Equilibrium Air Dissociation/Recombination Chemistry
5 Species, 11 Reaction Steps

Stability Analysis
Linear stability analysis of 1-D inviscid equations using Euler implicit time marching technique.

Magnitude of the Maximum Eigenvalue of the Amplification Matrix G, Pressure-Based Scheme, CFL=100.

Magnitude of the Maximum Eigenvalue of the Amplification Matrix G, Density-Based Scheme, CFL=100.
CONVERGENT-DIVERGENT NOZZLE FLOW, $\text{Mi}=0.01$

Non-Equilibrium Air Dissociation/Recombination Chemistry
5 Species, 11 Reaction Steps

COMBUSTION OF n-PENTANE FUEL DROPLET

Non-Equilibrium n-Pentane - Air Combustion Chemistry
5 Species, 1 Global Reaction Step

FLAME ZONE

FUEL

OXYGEN

$T_f = 300k$
$T_w = 1000k$
$D_o = 100\mu m$
DROPLET COMBUSTION, $M \approx 10^{-4} - 10^{-6}$

Non-Equilibrium n-Pentane - Air Combustion Chemistry
5 Species, 1 Global Reaction Step

DROPLET COMBUSTION, $M \approx 10^{-4} - 10^{-6}$

Molar Fraction

Reduced Radial Distance, $r/R(t)$

$\text{CO}_2$

$\text{H}_2\text{O}$
CONCLUSIONS

- An efficient and robust numerical algorithm has been developed for chemically reacting flows at all speeds.

FUTURE PLAN

1. Development and validation of the 2D/3D all-speed code for reacting flows.
2. Implementation of zonal capability.
3. Implementation of turbulence, turbulence/reaction closure, spray combustion, and thermal radiation models.
4. Reacting flow simulation for gas turbine combustor, rocket motor combustion chamber, etc.
Study of Shock-Induced Combustion Using an Implicit TVD Scheme

Shayne Yungster
Institute for Computational Mechanics in Propulsion
NASA Lewis Research Center

The supersonic combustion flowfields associated with various hypersonic propulsion systems, such as the ram accelerator (a ramjet-in-tube concept), the oblique detonation wave engine, and the scramjet, are being investigated using a new CFD code. The code solves the fully coupled Reynolds-averaged Navier-Stokes equations and species continuity equations in an efficient manner. It employs an iterative method that is based on the lower-upper symmetric successive overrelaxation (LU-SSOR) implicit factorization scheme, and a second order symmetric total variation diminishing (TVD) differencing scheme. To accelerate the convergence of the basic iterative procedure, this code can be combined with vector extrapolation methods, such as the Minimal Polynomial (MPE) and the Reduced Rank (RRE) Extrapolation. The extrapolation procedure solves a linear least squares problem and produces a sequence of approximations that, in general, has better convergence properties than the sequence obtained from the iterative scheme alone. Two different formulations of the LU-SSOR factorization scheme are currently implemented. In one formulation, the implicit operator includes the full Jacobian matrix of the chemical source term, leading to a preconditioner matrix of size \( n_s \times n_s \), where \( n_s \) is the number of species, which has to be inverted at every grid point. If the number of species considered is large, inverting this preconditioner can be very expensive. Therefore, a second formulation has been introduced in which the Jacobian matrix is replaced by a diagonal matrix that is designed to approximate the time scaling effects obtained by using the full Jacobian. In this formulation, no matrix inversions are required. Figure 1 shows the density residual history obtained with the two formulations (with and without extrapolation) for the case of a supersonic flow past a compression corner. The chemical nonequilibrium processes are simulated by means of a 9-species, 18-step finite-rate \( H_2 \)-air combustion model. When extrapolation is used, it is started after \( N_0 \) iterations, and is implemented in the so called “cycling” mode, using a sequence of \( K_{\text{max}} \) vectors obtained from the iterative scheme. The overhead in CPU time due to the use of extrapolation is very small (less than 1%) in the present case. The results indicate that savings of up to 40% in the overall computational work required to reach convergence can be realized by using RRE in combination with the basic iterative scheme (using both formulations). Similar results are obtained with MPE. The results also indicate that for the present chemistry model, the diagonal formulation is less efficient, requiring approximately 45% more CPU time than the Full Jacobian formulation to reach convergence. The code is currently being applied to study shock wave/boundary layer interactions in premixed combustible gases, and to investigate the ram accelerator concept. Results obtained for a ram accelerator configuration (Fig. 2) indicate a new combustion mechanism in which a shock wave induces combustion in the boundary layer, which then propagates outwards and downstream. The combustion process creates a high pressure region over the back of the projectile resulting in a net positive thrust force.
Objective

- To investigate shock-wave/boundary layer interactions involving premixed combustible gases.

- To compute (based on the Navier-Stokes equations) the supersonic combustion flowfield in a ram accelerator, a ramjet-in-tube concept.

Figure 1: Supersonic reacting flow past a compression corner; (a) pressure contours; (b) Convergence history of $L_2$ density residual (grid, 80 x 50).

Figure 2: Nondimensional pressure (top) and temperature (bottom) contours for a ram accelerator, a ramjet-in-tube device for accelerating projectiles to ultrahigh velocities. Mixture: stoichiometric $H_2$-air, $M = 6.7$, $P_m = 1$ atm, $T_m = 300^\circ$K, $T_w = 600^\circ$K. The vertical direction is magnified by a factor of 2.
Equations and Transport properties

- 2D/axisymmetric Reynolds-Averaged Navier-Stokes equations (general coordinates).
- Species specific heat, conductivity and viscosity obtained from fourth order polynomials of temperature.
- Conductivity and viscosity of mixture - Wilke's mixing rule.
- Binary mass diffusivities - Chapman-Enskog theory.

Combustion and Turbulence Models

- 9-species, 18-step chemistry model for hydrogen-oxygen combustion.
- Baldwin-Lomax Turbulence Model

Numerical Method

- Finite difference method.
- LU-SSOR implicit factorization scheme.
- Second-order symmetric TVD differencing scheme.
- Use of extrapolation techniques for convergence acceleration (i.e. Reduced Rank and Minimal Polynomial Extrapolation).

Reduced Rank Extrapolation Method (RRE)

- Given the vector sequence \( Q^0, Q^1, ..., Q^{k+1} \), with \( k = K_{\text{MAX}} \), compute the differences
  \[
  \Delta Q^j = Q^{j+1} - Q^j, \quad j = 0, 1, ..., k. \tag{1}
  \]
- Next, determine the scalars \( \gamma_0, \gamma_1, ..., \gamma_k \) by solving the constrained least-squares problem
  \[
  \text{minimize} || \sum_{j=0}^{k} \gamma_j \Delta Q^j || \tag{2}
  \]
  subject to \( \sum_{j=0}^{k} \gamma_j = 1 \).
- Finally, set
  \[
  S_{0,k} = \sum_{j=0}^{k} \gamma_j Q^j \tag{3}
  \]

Although the original definition of RRE is different, the definition given above is equivalent to it and results in an implementation that is more stable numerically.
Numerical Method

- LU-SSOR implicit factorization scheme:

\[ LT^{-1}U\delta Q = -\Delta t[RHS] \]

\[ L = \frac{1}{1+\beta\Delta t}(D_1 A^+ + D_2 B^+ - A^+ - B^+) \]

\[ U = \frac{1}{1+\beta\Delta t}(D_1 A^- + D_2 B^- + A^- + B^-) \]

\[ T = \frac{1}{1+\beta\Delta t}(A^+ + B^+ - A^- - B^-) \]

- Second-order symmetric TVD differencing scheme:

\[ [RHS] = (F_{j+1,k} - F_{j-1,k} + G_{j+1,k} - G_{j-1,k} + W_{j,k} + (R_{j+1,k-1} - R_{j-1,k-1} + R_{j+1,k+1} - R_{j-1,k+1}) \]

Where

\[ \phi_{j+1}^l = \frac{1}{2} a_{j+1}^l (a_{j+1}^l - \tilde{Q}_{j+1}^l) \]

\[ \psi(z) = \begin{cases} |z| & \text{if } |z| \geq \delta_l \\ (z^2 + \delta_l^2) / 2\delta_l & \text{if } |z| < \delta_l \end{cases} \]

\[ (\delta_l)_{j+1} = \delta [\min(U_{j+1}) + \max(V_{j+1}) + \frac{1}{2}(\sqrt{\epsilon^2 + \delta_l^2} + \sqrt{\eta^2 + \delta_l^2}) \times a_{j+1}] \]

\[ 0 < \delta < \Delta \]

Reacting flow past a compression corner

\[ H_2 - \text{Air} \]
\[ \Phi = 1 \]
\[ P_{\infty} = 1 \text{ atm} \]
\[ T_{\infty} = 900^\circ \text{ K} \]
\[ M = 4.5 \]
Convergence History of $L_2$ Density Residual

Schematic of "Oblique Detonation" Ram Accelerator

$(L = 15 \text{ cm}; d_p = 2.5 \text{ cm}; d_t = 3.8 \text{ cm}; \theta = 14^\circ)$
NONDIMENSIONAL PRESSURE (TOP) AND TEMPERATURE (BOTTOM) CONTOURS
Stoichiometric Hydrogen-Air; M=6.7; P=1 atm; T=300 K.

TEMPERATURE

NONDIMENSIONAL TEMPERATURE (TOP) AND MACH NUMBER (BOTTOM) CONTOURS
Stoichiometric Hydrogen-Air; M=7.2; P=1 atm; T=300 K.

MACH NUMBER

ORIGINAL PAGE
COLOR PHOTOGRAPH
Conclusions

- A new CFD code that efficiently solves the Navier-Stokes equations with finite-rate chemistry was presented.

- The application of vector extrapolation methods to viscous, chemically reacting flows was demonstrated.

- Results indicate that significant savings in computational work can be realized by using vector extrapolation methods in combination with the basic iterative scheme.

- Ram Accelerator Concept: results indicate that viscous effects are of primary importance. The combustion processes in the boundary layer strongly affect the entire flowfield.
UPWIND SCHEMES AND BIFURCATING SOLUTIONS IN REAL GAS COMPUTATIONS

Ambady Suresh
Sverdrup Technology, Inc.
Lewis Research Center Group
NASA Lewis Research Center

and

Meng-Sing Liou
Internal Fluid Mechanics Division
NASA Lewis Research Center

The area of high speed flow is seeing a renewal of interest due to advanced propulsion concepts such as the NASP, Space Shuttle, and future civil transport concepts. Upwind schemes to solve such flows have become increasingly popular in the last decade due mainly to their excellent shock capturing properties. Within this class of upwind schemes the Osher scheme has a few distinct advantages, such as a continuously differentiable numerical flux (making it especially attractive for implicit schemes), entropy satisfying solutions and the exact resolution of stationary shocks and contacts.

High speed flow is generally accompanied by changes in the chemical and thermodynamic composition of the gas which need to be taken into account. In the first part of this paper we present the extension of the Osher scheme to equilibrium and non-equilibrium gases. For the equilibrium case, the extension proceeds by choosing the pressure and entropy as the independent thermodynamic variables so that the Riemann invariants can be reduced to quadratures. Efficiency is achieved by noting that the quadrature need be only as accurate as the discretization error. In the non-equilibrium case, the determination of the intermediate points using Riemann invariants involves iterative solution of a transcendental equation. As this is computationally inefficient, an approximate method which involves no iteration is presented. For simplicity, the source terms are treated explicitly. In both cases, the formal extension is unique unlike the Roe scheme where there is a one parameter family of solutions with no clear choice among them.

Computations based on the above scheme are presented to demonstrate the feasibility, accuracy and efficiency of the proposed scheme. One of the test problems is a Chapman-Jouguet detonation problem for which numerical solutions have been known to bifurcate into spurious weak detonation solutions on coarse grids. Our results indicate that the numerical solution obtained depends both on the upwinding scheme used and the limiter employed to obtain second order accuracy. For example, the Osher scheme gives the correct CJ solution when the super-bee limiter is used but gives the spurious solution when the Van Leer limiter is used. With the Roe scheme the spurious solution is obtained for all limiters.
OVERVIEW

- Extension of the Osher Scheme for reacting flows.
  - 2 species system
  - Intermediate and Sonic points
  - Numerical results

- Bifurcating $Z - N - D$ Detonations.
  - Theory
  - Numerical Results

THEORY [Colella, Majda, Royturd (86)]

Simplified Model:

$$u_t + \left(\frac{1}{2}u^2 + \Delta h z\right)_x = 0$$

$$u \sim p/\rho$$

$$z_n = k\phi(u)z$$

For large $k\Delta x$ and/or $\Delta h$ non-physical weak detonation waves exist which travel at mesh speed $\frac{\Delta x}{\Delta t}$.

NUMERICAL RESULTS

- Riemann Problems (ideal gas)
- Reacting nozzle flow
- $Z - N - D$ detonations
Sonic Points

Using invariants is expensive due to exponents.

\[
\tau_s = \frac{\lambda_L^1}{(\lambda_L^1 - \lambda_N^1)}
\]

\[
U_{LN}^S = U_L(1 - \tau_s) + U_R \tau_s
\]

Freeze eigenvectors at \((U_L + U_R)/2\) and evaluate wave strengths as:

\[
\Delta \tau_{1,4} = \frac{1}{2a^2} [\Delta p \mp pa \Delta u]
\]

\[
\Delta \tau_2 = \frac{1}{a^2} [-\Delta p + a^2 \Delta p]
\]

\[
\Delta \tau_3 = \Delta z
\]

Intermediate states obtained by translation from left and right.
For multispecies $U_N$ and $U_P$ can be obtained only iteratively:

$$a \rho_N^\alpha + b \rho_N^\beta = c$$

However, $U_N, U_P$ need only be as accurate as the overall discretization error.

1) Source terms do not change wave pattern.

2) Along $x = 0$, solution approaches homogeneous Riemann problem near origin. Li and Yu [1985]

Use homogeneous Riemann problem [Gottlieb, etc.] for upwinding and add in source terms later.
Riemann Problem: no longer self-similar.

\[ R(x, t, U_L, U_R) \neq R(\frac{x}{t}, U_L, U_R) \]

Interface flux \( F_{i+\frac{1}{2}} \) not well defined.

However . . .

2 Species System:

\[
\begin{bmatrix}
\rho \\
\rho u \\
\rho E \\
\rho z
\end{bmatrix}_t + \begin{bmatrix}
\rho u \\
\rho u^2 + p \\
\rho E u + pu \\
\rho u z
\end{bmatrix}_x = \begin{bmatrix}
0 \\
0 \\
0 \\
-\rho k
\end{bmatrix}
\]

\( z = \text{mass fraction.} \quad \frac{z}{1} = 1 \text{ (unburnt)} \quad \frac{z}{0} = 0 \text{ (burnt)} \)

\( k = k(z, T) = \text{reaction rate} \)

\( E = \frac{p}{(\gamma(z)-1)p} + \frac{1}{2}u^2 - z\Delta h \)

\( \gamma(z) = \frac{\eta_1 C_{v1} z + \eta_2 C_{v2}(1-z)}{C_{v1} z + C_{v2}(1-z)} \)
**BIFURCATING Z - N - D DETONATIONS**

\[ \begin{align*}
\rho &= \rho_2 \\
p &= p_2 \\
u &= u_2 \\
z &= 0
\end{align*} \quad \rightarrow D_{cJ} \quad \begin{align*}
\rho &= \rho_1 \\
p &= p_1 \\
u_1 &= 0 \\
z_1 &= 1
\end{align*} \]

\( r(z) = \text{constant}. \)

Solved on moving grid.

\( k \) - Step function of \( T \).

**NUMERICAL RESULTS**

- Bifurcation depends on limiter used and Riemann Solver.

<table>
<thead>
<tr>
<th>R. S \ Limiter</th>
<th>1st Order</th>
<th>L.L. Limiter</th>
<th>Superbee</th>
</tr>
</thead>
<tbody>
<tr>
<td>Osher-N</td>
<td>B</td>
<td>B</td>
<td>NB</td>
</tr>
<tr>
<td>Osher-R</td>
<td>B</td>
<td>B</td>
<td>NB</td>
</tr>
<tr>
<td>Roe</td>
<td>B</td>
<td>B</td>
<td>B</td>
</tr>
</tbody>
</table>

108
IDEAL GAS SHOCK TUBE PROBLEM
OSHER SCHEME NATURAL ORDER

Fig. 2: Ideal gas shock tube problem. Frozen solution.
Fig. 3: Chapman-Jouquet detonation problem on a fine grid.
Reaction zone = 10 cells
G - J DETONATION PROBLEM
OSHER SCHEME NATURAL ORDER

Fig. 4a: Chapman-Jouguet detonation problem on a coarse grid. Reaction zone = 1/10 cell, Superbee Limiter.
Fig. 4b: Chapman-Jouguet detonation problem on a coarse grid.
Reaction zone = 1/10 cell, Vanleer Limiter.
Fig. 5: Divergent Nozzle Problem. Reaction zone = 20 cells.
CONCLUSIONS

- The Osher scheme can be extended efficiently to reacting flows.

- Bifurcations in reacting flows require further study.
HIGH ORDER PARALLEL NUMERICAL SCHEMES 
FOR SOLVING INCOMPRESSIBLE FLOWS

Avi Lin
University of Pennsylvania

Edward J. Milner, May-Fun Liou, and Richard A. Belch
Internal Fluid Mechanics Division
NASA Lewis Research Center

The use of parallel computers for numerically solving flow fields has gained a lot of importance in recent years. In most cases, the parallel machine executes modified standard serial CFD codes, taking advantage of parallelism by using special constructs, directives, system commands and calls. In other instances, numerical schemes and computational algorithms have been changed or redesigned to increase performance of the overall algorithm. This paper presents a new high order numerical scheme for CFD specifically designed for parallel computational environments.

A distributed MIMD system gives the flexibility of treating different elements of the governing equations with totally different numerical schemes in different regions of the flow field. This heterogeneous parallel numerical approach is sometimes technically complicated in terms of programming and administering the various processes, but it is quite efficient and contains enough free parameters for optimizing execution speedup.

The parallel decomposition of the governing operator to be solved is the primary parallel split. The decomposition of the physical domain of the flow into subdomains is the secondary, or optional, split. At the coarsest parallel level, each of the processors is assigned to solve a suboperator of the original PDE operator over a given subdomain. An iterative numerical procedure is employed. All of these splittings are designed to ensure a high rate of convergence.

The primary parallel split was studied using a hypercube-like architecture having clusters of shared-memory processors at each node. The approach is demonstrated using as examples some simple steady-state incompressible flows. Future studies should investigate the secondary split because, depending on the numerical scheme that each of the processors applies and the nature of the flow in the specific subdomain, it may be possible for a processor to seek a better, or higher order, scheme for its particular subcase.
OBJECTIVE:

TO REDUCE THE TIME REQUIRED TO SOLVE CFD PROBLEMS BY USING PARALLEL PROCESSING TECHNIQUES

DISCUSSION FRAMEWORK

- PARALLEL PROCESSING TECHNIQUE FOR SOLVING A BLOCK TRIDIAGONAL SYSTEM
- SOME APPLICATIONS & RESULTS
- PARALLEL PROCESSING TECHNIQUE FOR INVERTING A MATRIX
A Parallel, Scalable, 2D Navier-Stokes Solver

Stream function, driven cavity, 50 by 50 grid

- Parallel algorithm design provides efficient implementation on distributed or shared memory machines
  - 74% efficiency achieved on Hypercluster test bed
- Scalable and portable algorithm
  - basic building blocks (e.g. matrix solvers)
  - total solver
- Second-order accurate

ASSUME THAT

1) $B_{m,1} = K_{m,1} \ast B_{1,1}$

2) $B_{m,2} = K_{m,2} \ast B_{2,2}$

3) $B_{m,3} = K_{m,3} \ast B_{3,3}$
THEN

1) \((A_{1,1} + A_{1,2}K_{x,1} + A_{1,3}K_{x,3})\) \(\times B_{1,1} = I\)
2) \((A_{2,1}K_{x,2} + A_{2,2} + A_{2,3}K_{x,3})\) \(\times B_{2,2} = I\)
3) \((A_{3,1}K_{x,3} + A_{3,2}K_{x,3} + A_{3,3})\) \(\times B_{3,3} = I\)
4) \(A_{1,1}K_{y,1} + A_{1,2} + A_{1,3}K_{y,3} = 0\)
5) \(A_{2,1}K_{y,2} + A_{2,2}K_{y,2} + A_{2,3} = 0\)
6) \(A_{3,1} + A_{3,2}K_{y,1} + A_{3,3}K_{y,3} = 0\)
7) \(A_{1,1}K_{y,1} + A_{1,2}K_{y,2} + A_{1,3} = 0\)
8) \(A_{2,1} + A_{2,2}K_{y,2} + A_{2,3}K_{y,1} = 0\)
9) \(A_{3,1}K_{y,1} + A_{3,2} + A_{3,3}K_{y,2} = 0\)

THUS,

1) \(B_{1,1} = (A_{1,1} + A_{1,2}K_{x,2} + A_{1,3}K_{x,3})^{-1}\)
2) \(B_{2,2} = (A_{2,1}K_{x,2} + A_{2,2} + A_{2,3}K_{x,3})^{-1}\)
3) \(B_{3,3} = (A_{3,1}K_{x,3} + A_{3,2}K_{x,3} + A_{3,3})^{-1}\)
4) \(K_{y,1} = -A_{1,1}K_{y,1} - A_{1,2} + A_{1,3}K_{y,3}\)
5) \(K_{y,2} = -A_{2,1}K_{y,2} - A_{2,2} + A_{2,3}K_{y,3}\)
6) \(K_{y,3} = -A_{3,1}K_{y,3} - A_{3,2} + K_{y,2}\)
7) \(K_{x,1} = (A_{1,1} - A_{1,2}A_{2,2}A_{2,3})^{-1}(A_{1,2}A_{2,2}A_{2,3} - A_{1,3})\)
8) \(K_{x,2} = (A_{2,1} - A_{2,2}A_{3,3}A_{3,3})^{-1}(A_{2,2}A_{3,3} - A_{3,1} - A_{2,1})\)
9) \(K_{x,3} = (A_{3,1} - A_{3,1}A_{1,1}A_{1,3})^{-1}(A_{3,1}A_{1,1}A_{1,3} - A_{3,3})\)
SUMMARY

THE PARALLEL SOLVER:

+++):
1) IS GENERAL
2) CONVERGES VERY QUICKLY
3) IS READILY EXPANDABLE
   a) CAN BE USED WITH ONLY ONE PROCESSOR
   b) WILL USE AS FEW OR AS MANY PROCESSORS
      AS ARE AVAILABLE

---):
1) PERFORMS SOME REDUNDANT CALCULATION
   BECAUSE OF OVERLAP OF THE DATA

BLOCK TRIDIAGONAL SYSTEM

\[
\begin{bmatrix}
BC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
ABC \\
\end{bmatrix} \times \begin{bmatrix}
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\* \\
\end{bmatrix} = \begin{bmatrix}
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
D \\
\end{bmatrix}
\]

COMPUTATIONAL TECHNOLOGIES BRANCH
\[ \begin{bmatrix} BC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \\ ABC \end{bmatrix} \times \begin{bmatrix} u \\ u \\ u \\ u \\ u \\ u \\ u \\ u \\ u \\ u \\ u \\ u \end{bmatrix} = \begin{bmatrix} D \\ D \\ D \\ D \\ D \\ D \\ D \\ D \\ D \\ D \\ D \\ D \end{bmatrix} \]
BLOCK TRIDIAGONAL SYSTEM

Given the matrices and the equation:

\[
\begin{bmatrix}
BC & ABC & ABC \\
ABC & ABC & ABC \\
ABC & ABC & ABC \\
ABC & ABC & ABC \\
\end{bmatrix}
\begin{bmatrix}
u \\
v \\
v \\
v \\
\end{bmatrix}
= 
\begin{bmatrix}
D \\
D \\
D \\
D \\
\end{bmatrix}
\]

The solution process involves solving for the unknown vector \( \mathbf{v} \) using the given matrices and the right-hand side vector \( \mathbf{D} \).
FLOW NEAR A ROTATING DISK

EQUATIONS:

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

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

\[ \frac{\partial w}{\partial r} + u \frac{\partial w}{\partial z} = - \frac{1}{r} \frac{\partial p}{\partial r} + \frac{\partial}{\partial z} \left( \frac{\partial u}{\partial z} + \frac{\partial v}{\partial z} \right) + \frac{\partial}{\partial z} \left( \frac{\partial w}{\partial z} + \frac{\partial v}{\partial z} \right) \]

\[ \frac{\partial u}{\partial r} + \frac{u^2}{r} + \frac{\partial u}{\partial z} = 0. \]

<table>
<thead>
<tr>
<th>ITERATION</th>
<th>MAXIMUM ERROR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.15019E+02</td>
</tr>
<tr>
<td>2</td>
<td>0.52285E+01</td>
</tr>
<tr>
<td>3</td>
<td>0.23837E+03</td>
</tr>
<tr>
<td>4</td>
<td>0.10327E+01</td>
</tr>
<tr>
<td>5</td>
<td>0.39308E+00</td>
</tr>
<tr>
<td>6</td>
<td>0.94527E-01</td>
</tr>
<tr>
<td>7</td>
<td>0.47686E-02</td>
</tr>
<tr>
<td>8</td>
<td>0.14872E-04</td>
</tr>
<tr>
<td>9</td>
<td>0.78692E-09</td>
</tr>
<tr>
<td>10</td>
<td>0.60396E-13</td>
</tr>
</tbody>
</table>

Flow in the neighborhood of a disk rotating in a fluid at rest

Velocity components: u-radial, v-axial, w-axial
A layer of fluid is carried by the disk riding in the motion of rotation. The centripetal force in this layer gives rise to secondary flow which is directed radially outward.
FLOW NEAR A ROTATING DISK

- DISTRIBUTED MEMORY SIMULATION
- SERIAL MATRIX INVERTER

COMPUTATIONAL TECHNOLOGIES BRANCH

BOUNDARY LAYER ALONG A PLATE

EQUATIONS:
\[
\begin{align*}
\frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= \frac{\partial^2 u}{\partial y^2}, \\
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0,
\end{align*}
\]

\[y = 0: \ u = e = 0; \quad y = \infty: \ u = U_{\infty}\]

ITERATION | RESIDUAL ERROR
--- | ---
1 | MAXIMUM ERROR = 0.14258E+01
2 | MAXIMUM ERROR = 0.26990E+00
3 | MAXIMUM ERROR = 0.26832E-01
4 | MAXIMUM ERROR = 0.24873E-03
5 | MAXIMUM ERROR = 0.16154E-07
6 | MAXIMUM ERROR = 0.11084E-11

COMPUTATIONAL TECHNOLOGIES BRANCH
Matrix Inverse

\[
\begin{bmatrix}
A_{1,1} & A_{1,2} & A_{1,3} \\
A_{2,1} & A_{2,2} & A_{2,3} \\
A_{3,1} & A_{3,2} & A_{3,3}
\end{bmatrix}
\begin{bmatrix}
B_{1,1} & B_{1,2} & B_{1,3} \\
B_{2,1} & B_{2,2} & B_{2,3} \\
B_{3,1} & B_{3,2} & B_{3,3}
\end{bmatrix}
= \begin{bmatrix}
1 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1
\end{bmatrix}
\]
A Parallel, Scalable, 2D Navier-Stokes Solver

Stream function, driven cavity, 50 by 50 grid

- Parallel algorithm design provides efficient implementation on distributed or shared memory machines
  - 74% efficiency achieved on Hypercluster test bed
- Scalable and portable algorithm
  - basic building blocks (e.g. matrix solvers)
  - total solver
- Second-order accurate
RELATIONSHIPS

1) \[ A_{1,1} \cdot B_{1,1} + A_{2,1} \cdot B_{2,1} + A_{3,1} \cdot B_{3,1} = 1 \]

2) \[ A_{1,2} \cdot B_{1,2} + A_{2,2} \cdot B_{2,2} + A_{3,2} \cdot B_{3,2} = 1 \]

3) \[ A_{1,3} \cdot B_{1,3} + A_{2,3} \cdot B_{2,3} + A_{3,3} \cdot B_{3,3} = 1 \]

4) \[ A_{1,1} \cdot B_{1,1} + A_{2,1} \cdot B_{2,1} + A_{3,1} \cdot B_{3,1} = 0 \]

5) \[ A_{1,2} \cdot B_{1,2} + A_{2,2} \cdot B_{2,2} + A_{3,2} \cdot B_{3,2} = 0 \]

6) \[ A_{1,3} \cdot B_{1,3} + A_{2,3} \cdot B_{2,3} + A_{3,3} \cdot B_{3,3} = 0 \]

7) \[ A_{1,1} \cdot B_{1,1} + A_{2,1} \cdot B_{2,1} + A_{3,1} \cdot B_{3,1} = 0 \]

8) \[ A_{1,2} \cdot B_{1,2} + A_{2,2} \cdot B_{2,2} + A_{3,2} \cdot B_{3,2} = 0 \]

9) \[ A_{1,3} \cdot B_{1,3} + A_{2,3} \cdot B_{2,3} + A_{3,3} \cdot B_{3,3} = 0 \]
An Efficient and Robust Algorithm For
Time Dependent Viscous Incompressible Navier-Stokes Equations

John W. Goodrich
Computational Fluid Dynamics Branch
NASA Lewis Research Center

A recently developed finite difference algorithm is presented for unsteady incompressible Navier-Stokes calculations. The algorithm is extremely robust with respect to Reynolds number, and has been used to directly compute incompressible flows with smoothly resolved streamfunction, kinetic energy and vorticity contours for Reynolds numbers as high as \( Re = 100,000 \) without requiring any subscale modelling. The accompanying figure shows contour plots for the streamfunction and vorticity at nondimensional times \( t = 83 \) and \( t = 84 \) from a transient calculation at \( Re = 25,000 \), with a \( 256 \times 256 \) grid and \( \Delta t = \frac{1}{400} \). The algorithm is second order accurate in both time and space, with a Crank-Nicolson differencing for the diffusion terms, with a lagged second order Adams-Basforth differencing for the convection terms, and with central differencing for all space derivatives. There is no constraint on the time step size from diffusion time scales, but the convection time scales impose a stability constraint of \( \frac{\Delta t}{\Delta x} < 1 \), and typically a CFL number of 0.75 or 0.80 is used. This algorithm is based on the fourth order Partial Differential Equation for incompressible fluid flow which uses the streamfunction as the only dependent variable,

\[
\frac{\partial \Delta \psi}{\partial t} + \frac{\partial \psi}{\partial y} \frac{\partial \psi}{\partial x} - \frac{\partial \psi}{\partial x} \frac{\partial \psi}{\partial y} - \frac{1}{Re} \Delta^2 \psi = 0, \quad \text{for} \ x \in \Omega, \ \text{and} \ t > 0.
\]

Notice that the vorticity does not enter into this formulation. Although the algorithm currently is only for incompressible flows in two space dimensions, the algorithm can be and is being extended to handle temperature dependent density, subsonic compressible flows, and flows in three space dimensions. The algorithm produces discretely divergence free velocity fields on a nonstaggered grid. The streamfunction discretization is related to a primitive variable discretization on a staggered grid, so that primitive variable velocity boundary conditions can be implemented for the discrete streamfunction. The algorithm is extremely efficient with respect to use of both CPU time and physical memory. A typical time dependant calculation with a \( 128 \times 128 \) spatial grid and \( \Delta t = \frac{1}{160} \) requires approximately 2 MegaBytes of memory, and approximately 2.3 CPU seconds per time step on an IBM RS/6000 model 530 workstation. Codes implementing the algorithm have been run for problems of various scales on systems ranging from PC's and workstations to CRAY supercomputers. The algorithm has been implemented on an INTEL TOUCHSTONE parallel processor. Solutions will be shown for cavity and channel flows at various Reynolds numbers.
THE TIME DEPENDENT INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

\[
\frac{\partial u}{\partial t} + \nabla \cdot (uu) - \frac{1}{Re} \Delta u = -\nabla p + F, \quad \text{for } x \in \Omega, \text{ and } t > 0,
\]
\[
\nabla \cdot u = 0, \quad \text{for } x \in \Omega, \text{ and } t > 0.
\]

STREAMFUNCTION EQUATION FOR UNSTEADY INCOMPRESSIBLE FLOW

\[
\frac{\partial \psi}{\partial t} = \frac{1}{Re} \Delta \psi + \frac{\partial \psi}{\partial x} \frac{\partial \Psi}{\partial y} - \frac{\partial \Psi}{\partial y} \frac{\partial \psi}{\partial x}, \quad \text{for } x \in \Omega, \text{ and } t > 0;
\]

with
\[
u(x,t) = \frac{\partial \psi}{\partial y}, \quad \text{and} \quad v(x,t) = -\frac{\partial \psi}{\partial x}, \quad \text{for } x \in \Omega, \text{ and } t \geq 0.
\]

* Initial and boundary conditions must be supplied.

\[\Rightarrow\] For \( \Omega \subseteq \mathbb{R}^2 \).
\[\Rightarrow\] Vorticity and pressure DO NOT enter into the streamfunction formulation.
\[\Rightarrow\] A single equation for a single scalar unknown.
\[\Rightarrow\] The velocity solution is always divergence free, and so incompressible.

THE STREAMFUNCTION ALGORITHM FOR UNSTEADY INCOMPRESSIBLE FLOW

\[
La(\ddot{z}^{n+1}) - \frac{\Delta t}{2Re} Bi(\ddot{z}^{n+1}) = La(\ddot{z}^{n}) + \frac{\Delta t}{2Re} Bi(\ddot{z}^{n}) - \frac{3\Delta t}{2} \left[ \delta_x \left( \delta_x (\ddot{z}^{n}) La(\ddot{z}^{n}) \right) - \delta_y \left( \delta_y (\ddot{z}^{n}) La(\ddot{z}^{n}) \right) \right]
\]
\[
+ \frac{\Delta t}{2} \left[ \delta_x \left( \delta_x (\ddot{z}^{n-1}) La(\ddot{z}^{n-1}) \right) - \delta_y \left( \delta_y (\ddot{z}^{n-1}) La(\ddot{z}^{n-1}) \right) \right],
\]

with
\[
u_{i,j} = \frac{1}{2\Delta y} (z_{i,j+1}^{n} - z_{i,j-1}^{n}), \quad \text{and} \quad v_{i,j} = -\frac{1}{2\Delta x} (z_{i+1,j}^{n} - z_{i-1,j}^{n}).
\]

* \( \ddot{z}^{n} \approx \psi(\cdot,t_n) \) on the discrete grid.
* The discretization uses central spatial differencing throughout.
* All variables are defined at each grid point.
* Banded LU decomposition and multigrid solvers have been used.

\[\Rightarrow\] The variable \( \psi \) is as smooth "as possible".
\[\Rightarrow\] The implicit equation is elliptic, for all Reynolds numbers.
\[\Rightarrow\] The local domain of dependance is the large symmetric 13 point discretization stencil.
\[\Rightarrow\] The discrete solution is exactly incompressible, \( \delta_x (u_{i,j}^{n}) + \delta_y (v_{i,j}^{n}) = 0 \).
\[\Rightarrow\] The stability limit is a CFL constraint, \( \frac{\|u^{n+1}\| \Delta t}{\Delta x} < 1 \).
A MULTIGRID SOLVER FOR THE LINEAR IMPlicit EQUATIONS

\[ La(\tilde{z}^{n+1}) - \frac{\Delta t}{2Re} Bi(\tilde{z}^{n+1}) = \text{Source Term}(\tilde{z}^{n}, \tilde{z}^{n-1}) \]

* The Biharmonic operator is factored as two Laplacians.
* Point Gauss-Seidel smoothing (or Red Black Gauss-Seidel).
* Linear restriction and prolongation.
* A V-cycle with 3 iterations per grid level while coarsening, none while refining.

\[ \Rightarrow \omega = \Delta \psi \text{ is introduced ONLY for iterating with the biharmonic operator.} \]
\[ \Rightarrow \text{Implemented on scalar, vector and parallel systems.} \]

TYPICAL PERFORMANCE DATA

CAVITY PROBLEM (IBM RS/6000 model 530 workstation)

* Square cavity transient from \( t = 0 \) to \( t = 1 \), at \( Re = 9600 \).
* 10 to 15 iteration cycles reduce residuals to less than \( 5.0 \times 10^{-12} \).

(1) \( 256 \times 256 \) grid, \( \Delta t = \frac{1}{400} \), 7 grid levels, 6.6 MBytes storage, 10.2 sec/ts;
(2) \( 192 \times 192 \) grid, \( \Delta t = \frac{1}{256} \), 7 grid levels, 3.8 MBytes storage, 5.5 sec/ts;
(3) \( 128 \times 128 \) grid, \( \Delta t = \frac{1}{128} \), 6 grid levels, 1.7 MBytes storage, 2.5 sec/ts.

\[ \Rightarrow \text{CPU time increases "linearly" with the number of grid points:} \]
\( (1) \ 1.54 \times 10^{-4} \) sec/ts per grid point on the \( 256 \times 256 \) grid;
\( (2) \ 1.48 \times 10^{-4} \) sec/ts per grid point on the \( 192 \times 192 \) grid;
\( (3) \ 1.50 \times 10^{-4} \) sec/ts per grid point on the \( 128 \times 128 \) grid.

\[ \Rightarrow \text{Required memory increases "linearly" with the number of grid points:} \]
\( (1) \ 104.78 \) Bytes per grid point on the \( 256 \times 256 \) grid;
\( (2) \ 106.97 \) Bytes per grid point on the \( 192 \times 192 \) grid;
\( (3) \ 107.12 \) Bytes per grid point on the \( 128 \times 128 \) grid.

\[ \Rightarrow \text{The number of iteration cycles/ts is independent of resolution and Reynolds number.} \]
STREAMFUNCTION MAX AND MIN COMPARATIVE DATA
The square driven cavity at Re=5000

The Streamfunction Minimum

<table>
<thead>
<tr>
<th>Source</th>
<th>Grid</th>
<th>$\psi_{\text{min}}$</th>
<th>$x_{\text{min}}$</th>
<th>$y_{\text{min}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Kim and Moin</td>
<td>96 x 96</td>
<td>$-1.12 \times 10^{-1}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Goodrich</td>
<td>128 x 128</td>
<td>$-1.15 \times 10^{-1}$</td>
<td>$65/128$</td>
<td>$59/128$</td>
</tr>
<tr>
<td>Goodrich</td>
<td>256 x 256</td>
<td>$-1.18 \times 10^{-1}$</td>
<td>$132/256$</td>
<td>$137/256$</td>
</tr>
<tr>
<td>Ghia, Ghia, and Shin</td>
<td>256 x 256</td>
<td>$-1.19 \times 10^{-1}$</td>
<td>$121/256$</td>
<td>$137/256$</td>
</tr>
</tbody>
</table>

The Streamfunction Maximum

<table>
<thead>
<tr>
<th>Source</th>
<th>Grid</th>
<th>$\psi_{\text{max}}$</th>
<th>$x_{\text{max}}$</th>
<th>$y_{\text{max}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Goodrich</td>
<td>128 x 128</td>
<td>$3.44 \times 10^{-3}$</td>
<td>$102/128$</td>
<td>$9/128$</td>
</tr>
<tr>
<td>Goodrich</td>
<td>256 x 256</td>
<td>$3.13 \times 10^{-3}$</td>
<td>$206/256$</td>
<td>$19/256$</td>
</tr>
<tr>
<td>Ghia, Ghia, and Shin</td>
<td>256 x 256</td>
<td>$3.08 \times 10^{-3}$</td>
<td>$207/256$</td>
<td>$19/256$</td>
</tr>
</tbody>
</table>

TRANSIENT DRIVEN CAVITY
$Re = 25,000$, 256 x 256 grid, $t = 10.25$
$Re = 5000, 256 \times 256 \text{ grid, } t = 748.45$

$v$ at $x=0.5$ as a function of $y$

$v$ at $y=0.5$ as a function of $x$

$vorticity at y=1.0$ as a function of $x$

$Re = 5000, 128 \times 128 \text{ grid, } \Delta t = \frac{1}{140}, \ t = 748.45$, calculated data,


137
TYPICAL PERFORMANCE DATA

CHANNEL PROBLEM (IBM RS/6000 model 530 workstation)

* Backward facing half height step with a parabolic inflow profile on the step corner.
* Transient from \( t = 0 \) to \( t = 100 \), \( Re = 800 \), \( u_{max} = 1.5 \) for inflow profile, \( \Delta t = \frac{1}{100} \).
* Outflow Boundary Conditions are:

\[
\frac{\partial^2 \psi}{\partial x^2} = 0 \quad \text{and} \quad \frac{\partial^2 \omega}{\partial x^2} = 0 \quad \Leftrightarrow \quad \frac{\partial^2 \psi}{\partial x^2} = 0 \quad \text{and} \quad \frac{\partial^4 \psi}{\partial x^4} = 0
\]

* 10 to 35 iteration cycles reduce residuals to less than \( 5.0 \times 10^{-12} \)
* 30 iterations per cycle on the coarse grid.
* Coded for \( 1024 \times 32 \) grid with 4 grid levels using 3.5 MBytes storage.

(1) \( 800 \times 32 \) grid, 25 diameter domain, 4.7 s/ts;
(2) \( 480 \times 32 \) grid, 15 diameter domain, 3.1 s/ts;
(3) \( 224 \times 32 \) grid, 7 diameter domain, 1.7 s/ts.

\[\Rightarrow\] Relative CPU time decreases with the length of the domain:
(1) \( 1.79 \times 10^{-4} \) sec/ts per grid point on the \( 800 \times 32 \) grid;
(2) \( 1.95 \times 10^{-4} \) sec/ts per grid point on the \( 480 \times 32 \) grid;
(3) \( 2.23 \times 10^{-4} \) sec/ts per grid point on the \( 224 \times 32 \) grid.
STREAM FUNCTION CONTOURS
Re=800, 64*448 grid, 0<=x<=7, t=30, ul=1.5
Backward facing step with parabolic inflow, code g4a32vc3.f

BACKWARD FACING STEP
Re = 2000, 32 x 480 grid, 0 <= x <= 15

Stream function contours for different times:
- t = 25
- t = 35
- t = 45
- t = 150
SUMMARY

* Vorticity and pressure DO NOT enter into the streamfunction formulation.
* A single equation for a single scalar unknown in $\mathbb{R}^2$.
* The variable $\psi$ is as smooth "as possible".

Second order accuracy in both time and space.

The discrete solution is exactly incompressible, $\delta_t (u_i^n) + \delta_x (v_i^n) = 0$.

The stability limit is a CFL constraint, $\frac{\|w\|}{\Delta t} < 1$.

Implemented on scalar, vector and parallel systems.

CPU time increases "linearly" with the number of grid points.

Required memory increases "linearly" with the number of grid points.

The number of iteration cycles/ths is independent of resolution and Reynolds number.

Extremely robust with respect to Reynolds number.

GOALS

* Develop and adapt numerical methods.
* Investigate dynamic phenomena (instable, transitional, unsteady time asymptotic).

STEPS

* Incompressible flow (robust, fast, validated).
* Artificial outflow boundary conditions (Tom Hagstrom, Univ. of N.M.).
* Parallel processor implementation on INTEL iPSC (Rodger Dyson, NASA Lewis).
* Temperature dependent density (Boussinesq approximation).
* Chemistry models (Marty Rabinowitz, NASA Lewis).
* Compressible subsonic flow.
* Higher order differencing scheme.
* Three dimensional version.

APPLICATIONS

* Development of models for simplified chemical kinetics.
* Development of models for chemistry/turbulence interaction.
* Prediction of chemical emissions (High Speed Research Program).
* Understanding instabilities (flashback, flameout and acoustic/combustion interaction).
Some new developments in two-equation models and second order closure models will be presented. Two-equation models (e.g., $k$-$\varepsilon$ model) have been widely used in CFD for engineering problems. In most low-Reynolds number two-equation models, some wall-distance damping functions are used, especially in the eddy viscosity, to account for the effect of wall on turbulence. However, this often causes the confusions and difficulties in computing flows with complex geometry and also needs an ad hoc treatment near the separation and reattachment points. In this paper, modified two-equation models are proposed to remove abovementioned shortcomings. The calculations using various two-equation models are compared with direct numerical simulations of channel flows and flat boundary layers.

Development of second order closure model will be also discussed with emphasis on the modeling of pressure related correlation terms and dissipation rates in the second moment equations. All the existing models poorly predict the normal stresses near the wall and fail to predict the 3 dimensional effect of mean flow on the turbulence (e.g., decrease in the shear stress caused by the cross flow in the boundary layer). The newly developed second order near-wall turbulence model to be described in this paper is capable of capturing the near-wall behavior of turbulence as well as the effect of three dimension mean flow on the turbulence.
Two-Equation Near-Wall Turbulence Models

Second-Order Closure Models

Modeling of Three-Dimensional Turbulent Flows

$k$-$\epsilon$ model

The eddy viscosity $\nu_T$ is assumed in two-equation models as follows:

$$\nu_T = C_\mu f_\mu \frac{k^2}{\epsilon}$$

$$\nu_T = C_\mu f_\mu k\tau$$

where $\tau = k/\epsilon$.

The general $k$-$\epsilon$ (or $k$-$\tau$) model equations are of the following forms:

$$k_t + U_j k_{,j} = \left[ \left( \frac{\nu_T}{\sigma_k} + \nu \right) \frac{k_{,j}}{\epsilon} \right] + \Pi + \nu_T (U_{i,j} + U_{j,i}) U_{i,j} - \epsilon + D$$

$$\epsilon_t + U_j \epsilon_{,j} = \left[ \left( \frac{\nu_T}{\sigma_\epsilon} + \nu \right) \epsilon_{,j} \right] + C_1 f_1 \frac{\epsilon}{k} \nu_T (U_{i,j} + U_{j,i}) U_{i,j} - C_2 f_2 \frac{\epsilon \epsilon}{k} + E$$

$$\tau_t + U_j \tau_{,j} = \left[ \left( \frac{\nu_T}{\sigma_{\tau}} + \nu \right) \tau_{,j} \right] + \frac{2}{k} \left( \frac{\nu_T}{\sigma_{\tau,1}} + \nu \right) k_{,i} \tau_{,i} - \frac{2}{\tau} \left( \frac{\nu_T}{\sigma_{\tau,1}} + \nu \right) \tau_{,i} \tau_{,i}$$

$$+ (1 - C_\epsilon) \frac{k}{\tau} \nu_T (U_{i,j} + U_{j,i}) U_{i,j} + (C_2 f_2 - 1)$$
The damping function $f_\mu$:

$$f_\mu = f\left(\frac{k^2}{\nu} \right) = f(R_t) \quad \text{— Jones and Launder model}$$

$$f_\mu = f\left(\frac{u'v'}{\nu} \right) = f(y^+) \quad \text{— for all other } k-\epsilon \text{ models}$$

The parameter $R_t$ and $y^+$ are used for counting the wall effect.

The problems with the form of $f_\mu(y^+)$ are following:

Near the separation region, $f_\mu \to 0$, $\nu_T = 0$;

In the flows with complex geometry, the $y$ is not well defined.

$y^+$ is an unacceptable parameter.
In the present study, we suggest another parameter:

\[ R = \frac{k^{3/2} |U|}{\epsilon \nu} \]

and the following modifications in Shih's model:

\[ f_\mu = 1 - \exp \left\{ C_3 \left[ 1 - \exp(C_6 R^{1/4}) \right] \right\} \]
\[ \Pi = \left[ \frac{C_0 \nu T}{f_\mu^2 \sigma_k} \right]_{ij} \]
\[ \bar{\epsilon} = \epsilon \left[ 1 - \exp(-R_t^{1/2}) \right] \]

where \( R_t = k^2/\nu \epsilon \), \( C_0 = .004 \), \( C_3 = .0004 \), \( C_6 = 1.2 \)
Turbulent flow past a hill
(Re≈1.4 \cdot 10^6)
Another alternative:

$$R_u = \frac{U^4}{\nu \varepsilon}$$

and

$$f_\mu = 1 - \exp(-a_1 R_u^{1/4} - a_2 R_u^{1/2} - a_3 R_u)$$

$$\Pi = \left[ \frac{C_0 \nu T}{f^2 \sigma_k \kappa} \right]_{k,j}$$

$$\bar{\varepsilon} = \varepsilon \left[ 1 - \exp(-R_t^{1/2}) \right]$$

where $R_t = k^2/\nu \varepsilon$, $a_1 = 5 \cdot 10^{-3}$, $a_2 = 10^{-5}$, $a_3 = 8 \cdot 10^{-7}$, $C_0 = .01$.

---

**Second order modeling of near-wall turbulence**

The exact equation for the Reynolds stress tensor is:

$$\frac{D}{Dt} \langle u_i u_j \rangle = P_{ij} + T_{ij} + D_{ij}^{(\nu)} + \Pi_{ij} - \varepsilon_{ij}$$

where $\langle \rangle$ stands for an ensemble average, $D/Dt = \partial / \partial t + U_k \partial / \partial x_k$.

- $P_{ij} = -\langle u_i u_k \rangle U_{j,k} - \langle u_j u_k \rangle U_{i,k}$
- $T_{ij} = -\langle u_i u_j u_k \rangle_{,k}$
- $D_{ij}^{(\nu)} = \nu \langle u_i u_j \rangle_{,kk}$
- $\Pi_{ij} = -\frac{1}{\rho} \langle u_i p_{,j} + u_j p_{,i} \rangle$
- $\varepsilon_{ij} = 2\nu \langle u_{i,k} u_{j,k} \rangle$
Mean velocity profile for channel flow, $Re_\tau = 180$.

Turbulent kinetic energy for channel flow, $Re_\tau = 180$. 

153
Turbulent kinetic energy for flat plate boundary layer, $Re_\theta = 1410$.

Mean velocity profile for flat plate boundary layer, $Re_\theta = 1410$. 
Using the near-wall asymptotic behavior of turbulence\textsuperscript{[10]} as model constraints, we formed a set of modeled transport equations for the Reynolds-stress tensor and the dissipation rate of turbulent kinetic energy. The proposed near-wall model for $\Pi_{ij} - \epsilon_{ij}$ is:

$$\Pi_{ij} - \epsilon_{ij} = -f_w \frac{\epsilon}{\langle q^2 \rangle} [2\langle u_i u_j \rangle + 4(\langle u_i u_k \rangle n_j n_k + \langle u_j u_k \rangle n_i n_k) + 2\langle u_k u_l \rangle n_k n_l n_i n_j]$$

where $n_i$ is a unit vector normal to the surface, and $f_w = \exp(- (R_t/C_1)^2)$, $R_t = \langle q^2 \rangle / \nu$, $C_1 = 1.358 R_{\tau}^{0.44}$, $R_{\tau} = u_{\tau} \delta / \nu$. $u_{\tau}$ is the friction velocity, $\delta$ is the thickness of the boundary layer or the half width of the channel.

Away from the wall, the velocity pressure-gradient correlation $\Pi_{ij}$ is split into the rapid part $\Pi_{ij}^{(1)}$ and the slow part $\Pi_{ij}^{(2)}$:

$$\Pi_{ij} = \Pi_{ij}^{(1)} + \Pi_{ij}^{(2)}$$

The proposed model for $\Pi_{ij}^{(2)} - \epsilon_{ij}$ is:

$$\Pi_{ij}^{(2)} - \epsilon_{ij} = -\epsilon (\beta b_{ij} + \frac{2}{3} \delta_{ij}) (1 - f_w)$$

where

$$\beta = 2 + \frac{F}{9} \left[ \frac{72}{R_t^{1/2}} + 80.1 \ln[1 + 62.4 (-II + 2.3III)] \right] \exp \left( - \frac{7.77}{R_t^{1/2}} \right)$$

$$F = 1 + 27III + 9II$$

$$II = -\frac{1}{2} b_{ij} b_{ji}$$

$$III = \frac{1}{3} b_{ij} b_{jkl} b_{ki}$$

$$b_{ij} = \langle u_i u_j \rangle / \langle q^2 \rangle - \delta_{ij} / 3$$

156
The rapid part of velocity pressure-gradient, $\Pi_{ij}^{(1)}$ is modeled as follows (Shih and Lumley\cite{11,12}): 

$$
\Pi_{ij}^{(1)} = \left(\frac{1}{5} + 2a_5\right)(q^2)(U_{i,j} + U_{j,i}) - \frac{2}{3}(1 - a_5)(P_{ij} - \frac{2}{3}P\delta_{ij})
+ \left(\frac{2}{3} + \frac{16}{3}a_6\right)(D_{ij} - \frac{2}{3}P\delta_{ij}) + \frac{2}{15}(P_{ij} - D_{ij}) + \frac{6}{5}b_{ij}P
+ \frac{2}{5(q^2)}[(u_i u_k) U_{j,i} + (u_j u_k) U_{i,j} - (u_i u_p) (u_j u_q) (U_{p,q} + U_{q,p})]
$$

where,

$$
P_{ij} = -(u_i u_k) U_{j,k} - (u_j u_k) U_{i,k}
$$

$$
D_{ij} = -(u_i u_k) U_{k,i} - (u_j u_k) U_{k,i}
$$

$$
P = \frac{1}{2}P_{ii}
$$

$$
a_5 = -\frac{1}{10}(1 + C_2k^{1/2})
$$

$$
C_2 = 0.8[1 - \exp(-(R_t/40)^2)]
$$

**Dissipation rate equation**

The modeled dissipation rate equation derived in this work is:

$$
\epsilon, t + U_i \epsilon, i = (\nu \epsilon, i - <\epsilon u_i>); i - \psi_0 \frac{\epsilon}{(q^2)}
- \psi_1 \frac{\epsilon}{<q^2>} (u_i u_j) U_{i,j} - \psi_2 \frac{\nu(q^2)}{\epsilon} (u_k u_l) (U_{i,j} - U_{i,j}) U_{i,j}
$$

where

$$
\psi_0 = \frac{14}{5} + 0.98[1 - 0.33 \ln(1 - 55II)] \exp(-2.83R_t^{-1/2})
$$

$$
\psi_1 \approx 2.1
$$

$$
\psi_2 = -1.15(1 - F')
$$

$$
\tilde{\epsilon} = \epsilon - \frac{\nu(q^2,i q^2,i)}{4(q^2)}
$$

157
Three-dimensional boundary layer

The effect of three-dimensional flow on:

- Reduction in Reynolds stress.
- Lag development between the stress and strain rate.
Direct numerical simulation of 3D channel flows:
Turbulence modeling: Launder and Shima model

Turbulent shear stress

\[ \tau_{xy} \]

\[ y^+ \]

Turbulent intensity

\[ u', v', w' \]

\[ y^+ \]

Turbulence modeling: Lai and So model

\[ \langle u \rangle \text{ PROFILE} \]

\[ y^+ \]

\[ u, v, w \text{ -- RMS OF FLUCTUATING VELOCITY} \]

\[ y^+ \]
Turbulence modeling: Present model

Performance of existing models in modeling of 3-D flows:

- Two-equation (e.g. $k$-$\varepsilon$) models — incapable
- Reynolds-stress models — possible
  - Launder and Shima model — no
  - Lai and So model — no
  - Present model — yes
On Recontamination and Directional-Bias Problems in Monte Carlo Simulation of PDF Turbulence Models
Andrew T. Hsu
Sverdrup Technology, Inc.
Lewis Research Center Group
NASA Lewis Research Center

Turbulent combustion cannot be simulated adequately by conventional moment closure turbulence models. The difficulty lies in the fact that the reaction rate is in general an exponential function of the temperature, and the higher order correlations in the conventional moment closure models of the chemical source term cannot be neglected, making the applications of such models impractical. The probability density function (pdf) method offers an attractive alternative: in a pdf model, the chemical source terms are closed and do not require additional models.

The partial differential equation for the probability density function, \( \bar{P} \), can be written as

\[
\frac{\partial \bar{P}}{\partial t} + \rho \bar{\nabla} \cdot \bar{\nabla} \bar{P} + \bar{P} \sum_{i=1}^{N} \partial \psi_{i}(\psi_{i}, ..., \psi_{N}) \bar{P} = -\partial \psi(\bar{P} < \psi_{i} > \bar{P}) - \rho \sum_{i=1}^{N} \partial \psi_{i}(\psi_{i}, ..., \psi_{N}) \bar{P} < \epsilon_{i} > \bar{P}
\]

where the terms represent the time derivative, mean convection, chemical reaction, turbulent convection, and molecular mixing, respectively. The fact that the pdf equation has a very large dimensionality renders finite difference schemes extremely demanding on computer memory and CPU time and thus impractical, if not entirely impossible. A logical alternative is the Monte Carlo scheme, wherein the number of computer operations increases only linearly with the increase of number of independent variables, as compared to the exponential increase in a conventional finite difference scheme.

A grid dependent Monte Carlo scheme following that of J.Y. Chen and W. Kollmann has been studied in the present work. In dealing with the convection and diffusion of the pdf, the pdf equation is discretized on a given grid, e.g.,

\[
\bar{P}_{x+dx,j} = \alpha_{j} \bar{P}_{x,j+1} + \beta_{j} \bar{P}_{x,j} + \gamma_{j} \bar{P}_{x,j-1}
\]

However, if this is the only restriction satisfied by the numerical algorithm, the mass fractions may not be conserved due to re-contamination, and directional-bias also appears. These phenomena are illustrated in Figure 1: Consider a mixing layer; use white balls to represent contaminants in the upper stream and black balls to represent contaminants in the lower stream. As the two streams move toward right, the location of the white balls and black balls are interchanged randomly to simulate convection and diffusion. From Figure 1, it is clear that directional-bias caused by recontamination caused the center of the mixing layer to drift downward. (Directional-bias can be partially corrected by changing sweeping directions.) One also notices that after the first marching step, the conservation law is violated, reflected in the Figure as missing white or black balls.

It is found that in order to conserve the mass fractions absolutely, one needs to add further restriction to the scheme, namely

\[
\alpha_{j} + \beta_{j} + \gamma_{j} = 1
\]

A new algorithm was devised that satisfies this restriction in the case of pure diffusion or uniform flow problems. Using the same example, it is shown that absolute conservation can be achieved. This result is shown in Figure 2. One can see that the diffusion process is symmetric, and the problem of directional-bias is eliminated.

Although for non-uniform flows absolute conservation seems impossible, the present scheme has reduced the error considerably.
Figure 1. Illustration of directional-bias and recontamination problems in Monte Carlo simulation.

Figure 2. Solution obtained by using additional constraint in the solution procedure.
CLOSURE PROBLEM:

\[ u_i^\tau u_j^\tau, \quad - \text{Turbulence Modeling} \]

\[ Y_i^\tau u_j^\tau, \quad - \text{Analogy of shear stress: diffusion model.} \]

\[ \rho \bar{w}_i \quad - \text{???} \]

\[ \rho w_i = \rho w_1 Y_1, \ldots, Y_n, T, \rho \]

But in general:

\[ \rho \bar{w}_i \neq \rho w_1 \bar{Y}_1, \ldots, \bar{Y}_n, \bar{T}, \bar{\rho} \]
The Simulation of Convection and Diffusion

\[ \rho \tilde{v}_i \partial_n \tilde{P} = -\partial_n (\rho < v''_i \psi_i > \tilde{P}) \]

Discretize:

\[ \tilde{P}^{*}_{x+dx,j} = \alpha_j \tilde{P}^{*}_{x,j+1} + \beta_j \tilde{P}^{*}_{x,j} + \gamma_j \tilde{P}^{*}_{x,j-1} \]

Where the coefficients must satisfy:

\[ \alpha_j + \beta_j + \gamma_j = 1 \]

**MONTE CARLO METHOD**

(1) Use an ensemble of stochastic particles to represent the PDF

(2) The particles move in such a way that the PDF evolution equation is satisfied.

(3) Three steps: convection/diffusion; molecular mixing; chemical reaction.
PDF EVOLUTION EQUATION

\[
\tilde{p} \partial_t \tilde{P} + \tilde{p} \tilde{v}_n \partial_n \tilde{P} + \tilde{p} \sum_{i=1}^{N} \partial_{\psi_i} \left\{ w_i (\psi_1, ..., \psi_N) \tilde{P} \right\} \\
= -\partial_n (\tilde{p} < \psi^n_i | \tilde{P} >) - \tilde{p} \sum_{i=1}^{N} \sum_{j=1}^{N} \partial_{\psi_i} \psi_j \left( < \epsilon_{ij} | \psi_k > \tilde{P} \right)
\]

(1) Mean Convection
(2) Chemical Reaction
(3) Turbulent Convection
(4) Molecular Mixing

Dimension: N+5 finite difference not feasible.

PROBABILITY DENSITY FUNCTION MODEL

\[
\rho \tilde{w}_i = \int ... \int \rho w_i (Y_1, ..., Y_n, T, \rho) P(Y_1, ..., Y_n, T, \rho) dY_1 ... dY_n dT d\rho
\]

EXAMPLES:

ASSUMED PDF
MOMENT CLOSURE MODEL

Example:

\[ w_F = -BT^n \exp\left(-\frac{T_n}{\bar{T}}\right) \rho^2 Y_F Y_O \]

\[ T = \bar{T} + T' \]
\[ Y_F = \bar{Y}_F + Y'_F \]
\[ Y_F = \bar{Y}_F + Y'_F \]

- \( w_F \) can be expended into series and terms such as \( Y_F^r \bar{Y}_O^l \), \( Y_F^r T' \), \( Y_O^l T' \), and \( \bar{T}^{n2} \) have to be modeled.

Restrictions: \( T' \ll \bar{T}, \ T_n \sim \bar{T} \)
\[ \text{Ran}(\alpha_{j+1}) = \text{Ran}(\gamma_j) \]

\[ \text{iRan}(\alpha_{j+1}) = \text{iRan}(\gamma_j) \]
Fig. 1  Velocity distribution

Fig. 2  Turbulent kinetic energy
Fig. 1 Temperature distribution: $H_2 + F_2 = 2HF$

Fig. 2 Mass fraction: $H_2 + F_2 = 2HF$
Fig. 1 Temperature distribution: $H_2 + F_2 = 2HF$

- Experiment, Hermanson & Dimotakis
- Present PDF solution.
Fig. 2 probability mass function.

\[ P_1(\Delta T / \Delta T_{ad}) \]

- \( y/\delta_t = -0.42 \)
- \( y/\delta_t = -0.15 \)
CONCLUDING REMARKS

(1) For absolute conservation, the number of particles received in a cell must equal to the number that are sent out from the same cell:

$$a_j + \gamma_j = a_{j-1} + \gamma_{j+1}$$

(2) The particles that are replaced must be the particles that were sent out.

(3) Store at least 3 arrays of random indices and move them successively.

(4) Eliminated the problem of recontamination and directional-bias in the cases of uniform flow or pure turbulent diffusion. Reduced error.

(5) In case of non-uniform flow, convection is needed, and thus (1) can not be satisfied.

(6) The capability and accuracy of the present scheme are demonstrated through the example of a turbulent non-premixed flame.
Implementation of a k-ε Turbulence Model to RPLUS3D Code

Tawit Chitsomboon
Institute for Computational Mechanics in Propulsion
NASA Lewis Research Center

The RPLUS3D code has been developed at the NASA Lewis Research Center to support the national aerospace plane project. The code has the capability to solve three dimensional flowfields with finite rate combustion of hydrogen and air. The combustion processes of the hydrogen-air system are simulated by an 18-reaction path, 8-species chemical kinetic mechanism. The code uses a Lower-Upper (LU) decomposition numerical algorithm as its basis, making it a very efficient and robust code. Except for the Jacobian matrix for the implicit chemistry source terms, there is no inversion of a matrix even though a fully implicit numerical algorithm is used.

A k-epsilon turbulence model has recently been incorporated into the RPLUS3D code. Initial validations have been conducted for a flow over a flat plate. Results of the validation studies will be shown. Some difficulties in implementing the k-epsilon equations to the RPLUS3D code will also be discussed.
MOTIVATION

- MOST APPLICATIONS ESPECIALLY FLOWS IN SCRAMJET COMBUSTORS ARE TURBULENT
- COMPLEX FLOWS IN SCRAMJET COMBUSTORS (e.g., NORMAL H₂ INJECTION INTO SUPERSONIC AIR) WILL REQUIRE AT THE MINIMUM A k-ε MODEL

OBJECTIVE

- IMPLEMENT A HIGH REYNOLDS NUMBER k-ε TURBULENCE MODEL
- A LOW REYNOLDS NUMBER MODEL IS NOT PRACTICAL FOR A 3D CHEMICALLY REACTING CFD CODE

SOME FEATURES OF RPLUS3D

- 3D CFD CODE DEVELOPED AT NASA LEWIS RESEARCH CENTER
- FULLY IMPLICIT FINITE VOLUME L-U SCHEME
- CENTRAL-DIFFERENCE SCHEME WITH JAMESON-TYPE A.V.
- FINITE-RATE HYDROGEN-AIR COMBUSTION (8-SPECIES, 18-STEP)
- EFFICIENT: NO INVERSION OF MATRICES EXCEPT FOR COMBUSTING FLOWS
- DEVELOPED ESPECIALLY FOR NASP COMBUSTORS
- COULD BE USED FOR GENERAL PROBLEMS BY SETTING VARIOUS SWITCHES
K-EPsILON TURBULENCE MODEL

- HIGH REYNOLDS NUMBER FORM
- ECONOMICAL FOR 3D COMBUSTING FLOWS
- LAW OF THE WALL
- SOLVE 2 PDE'S FOR $k$ AND $\epsilon$
- USE L-U ALGORITHM (THE SAME AS THE FLOW SOLVER)
- DECOUPLE FROM THE FLOW SOLVER
- MODULAR PROGRAM: ONLY ONE SUBROUTINE CALL FROM THE FLOW SOLVER

HIGH REYNOLDS NUMBER $k$-$\epsilon$ MODEL

- STANDARD $k$-$\epsilon$ MODEL

\[
\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \rho P - \rho \epsilon + \nabla \cdot (\mu + \frac{\mu_t}{\sigma_k}) \nabla k
\]

\[
\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho \epsilon u_i}{\partial x_i} = \frac{C_1 \rho P}{k} - \frac{C_2 \rho \epsilon}{k} + \nabla \cdot (\mu + \frac{\mu_t}{\sigma_\epsilon}) \nabla \epsilon
\]

WHERE

\[
P = -u_i \frac{\partial u_i}{\partial x_j}
\]

AND,

\[
-u_i u_j = \nu_t (u_{i,j} + u_{j,i}) - (2/3) \delta_{ij} (k + \nu_t \nabla \cdot \vec{V})
\]
BOUNDARY CONDITIONS FOR k-ε MODEL

- **ASSUMPTIONS**
  - LOG LAW PROFILE
    \[ u^+ = \frac{1}{\kappa} \ln(Ey^+) \]
  - LOCAL EQUILIBRIUM
    \[ \text{dissipation} = \text{production} \]

- **PSEUDO CODE**
  - INTEGRATE FLOW EQUATIONS
  - INTEGRATE k-ε EQUATIONS
  - FIND \( u_r \) FROM LOG-LAW PROFILE BY NEWTON'S ITERATION
  - FIND \( k \) and \( \epsilon \) AT THE FIRST CELL CENTER
    \[ k = \frac{(u_r)^2}{\kappa^+} \quad \epsilon = \frac{(u_r)^3}{\kappa y} \]
  - FIND \( \nu_\varepsilon = C_\omega k^2 / \epsilon \)

DIFFICULTIES ENCOUNTERED

- **RUN AT LOWER CFL NO. THAN LAMINAR CASES**
  - DUE TO EXPLICIT VISCOUS TERMS IN THE FLOW SOLVER
- **IMPLICIT FORMULATION FOR BOTH CONVECTION AND DIFFUSION TERMS**
- **PARTIALLY IMPLICIT SOURCE TERMS**
- **POSITIVITY OF k-ε IMPOSED AUTOMATICALLY**
  \[ \epsilon = \epsilon^* k^{n+1} / k^n \]
- **REQUIRES ARTIFICIAL DAMPING**
- **NO DAMPING AT TWO POINTS ADJACENT TO THE WALL**
LAMINAR FLOW OVER A FLAT PLATE

• MACH NO. = 0.5; 31 X 31 GRID

Figure 1. Turbulent boundary layer flow over a flat plate.
FUTURE PLANS

• MORE VALIDATIONS
  – DOUBLE NORMAL JETS IN CROSS FLOW (UVA)
  – SLANT JET IN CROSS FLOW (NASA LEWIS)

• COMPRESSIBILITY EFFECTS

• PROBABILITY DENSITY FUNCTION (PDF)
TURBULENCE AND DETERMINISTIC CHAOS

Robert G. Deissler
LeRC Research Academy
NASA Lewis Research Center

ABSTRACT

Several turbulent and nonturbulent solutions of the Navier-Stokes equations are obtained. The unaveraged equations are used numerically in conjunction with tools and concepts from nonlinear dynamics, including time series, phase portraits, Poincare sections, largest Liapunov exponents, power spectra, and strange attractors.

Initially neighboring solutions for a low-Reynolds-number fully developed turbulence are compared. The turbulence, which is fully resolved, is sustained by a nonrandom time-independent external force. The solutions, on the average, separate exponentially with time, having a positive Liapunov exponent. Thus, the turbulence is characterized as chaotic.

In a search for solutions which contrast with the turbulent ones, the Reynolds number (or strength of the forcing) is reduced. Several qualitatively different flows are noted. These are, respectively, fully chaotic, complex periodic (see the accompanying figure), weakly chaotic, simple periodic, and fixed-point. Of these, we classify only the fully chaotic flows as turbulent. Those flows have both a positive Liapunov exponent and Poincare sections without pattern. By contrast, the weakly chaotic flows, although having positive Liapunov exponents, have some pattern in their Poincaré sections. The fixed-point and periodic flows are nonturbulent, since turbulence, as generally understood, is both time-dependent and aperiodic.

Besides the sustained (forced) flows, a flow which decays as it becomes turbulent is examined. As in the sustained case, the flow is extremely sensitive to small changes in initial conditions. The sensitivity increases with improved spatial resolution. For the finest grid (128³ points) the spatial resolution appears to be quite good.

As a final note, the variation of the velocity-derivative skewness of a Navier-Stokes flow as the Reynolds number goes toward zero is calculated numerically. The value of the skewness, which has been somewhat controversial, is shown to become small at low Reynolds numbers, in agreement with intuitive arguments that nonlinear terms should be negligible.
TURBULENCE PROBABLY STILL MAIN UNSOLVED PROBLEM IN FLUID DYNAMICS.

BUT CONSIDERABLE ADVANCES BEING MADE:

- NUMERICAL SOLUTIONS OF INSTANTANEOUS NAVIER-STOKES EQUATIONS

- UTILIZATION OF CONCEPTS AND TOOLS FROM NONLINEAR DYNAMICS, INCLUDING DETERMINISTIC CHAOS. "DETERMINISTIC" REFERS TO THE EVOLUTION EQUATIONS (NO RANDOM COEFFICIENTS.)

THESE TWO GO HAND IN HAND. IN FACT, USE OF NONLINEAR DYNAMICS IN STUDY OF TURBULENCE (OR IN THE STUDY OF ANYTHING) DID NOT GET FAR UNTIL ADVENT OF HIGH-SPEED COMPUTERS, BECAUSE THE NONLINEAR EVOLUTION EQUATIONS (NAVIER-STOKES EQUATIONS IN CASE OF TURBULENCE) COULD NOT PREVIOUSLY BE EFFECTIVELY USED.

HERE WE WANT TO TAKE A LOOK AT TURBULENCE BY USING CFD, TOGETHER WITH CONCEPTS AND TOOLS FROM NONLINEAR DYNAMICS.

OBTAINED SEVERAL SOLUTIONS OF INSTANTANEOUS NAVIER-STOKES EQUATIONS AT LOW REYNOLDS NUMBERS-BOTH TURBULENT AND NONTURBULENT FLOW KEPT FROM DYING OUT BY TIME-INDEPENDENT FORCING TERM. ALTHOUGH FORCING WAS STEADY, RESULTING FLOW WAS OFTEN HIGHLY UNSTEADY AND RANDOM IN APPEARANCE.

THIS IS A REMARKABLE FEATURE OF MANY TURBULENT FLOWS—E.G., A TURBULENT (TIME DEPENDENT) FULLY DEVELOPED PIPE FLOW MAY BE PRODUCED BY A STEADY PRESSURE GRADIENT.
SOLVED NUMERICALLY THE UNAVERAGED THREE-DIMENSIONAL
NAVIER-STOKES EQUATIONS WITH A TIME-INDEPENDENT
FORCING TERM

\[
\frac{\partial u_i}{\partial t} = -\frac{\partial (u_i u_k)}{\partial x_k} - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_k \partial x_k} + F_i
\]

WHERE

\[
F_i = \sum_{n=1}^{3} n a_i \, n q^2 \cos(n \hat{q} \cdot \hat{x})
\]

WHERE

\[
\begin{align*}
1 a_i &= k(2, 1, 1) \quad 2 a_i = k(1, 2, 1) \quad 3 a_i = k(1, 1, 2) \\
1 q_i &= (-1, 1, 1) \quad 2 q_i = (1, -1, 1) \quad 3 q_i = (1, 1, -1)
\end{align*}
\]

AND

\[
\frac{1}{\rho} \frac{\partial^2 p}{\partial x_i \partial x_j} = -\frac{\partial^2 (u_i u_k)}{\partial x_k \partial x_k}
\]

PLOT OF FORCING TERM \(F_i\) ON PLANE THROUGH GRID CENTER

MAGNITUDE OF FORCING VECTOR ON PLANE THROUGH GRID CENTER
USED A CUBICAL COMPUTATIONAL GRID (0 TO $2\pi$ ON A SIDE) WITH FOURTH-ORDER SPATIAL DIFFERENCING. 32$^3$ GRID POINTS

USED A PREDICTOR-CORRECTOR TIME DIFFERENCING (SECOND-ORDER LEAPFROG PREDICTOR AND THIRD-ORDER ADAMS-MOULTON CORRECTOR) BOUNDARY CONDITIONS: PERIODIC

OBTAINED ASYMPTOTIC (LONG-TERM) SOLUTIONS TO INSURE EXCITATION OF AS MANY MODES AS POSSIBLE AT A GIVEN REYNOLDS NUMBER, INITIAL CONDITIONS ON MOST OF FLOWS WERE SPATIALLY CHAOTIC

MAGNITUDE OF CHAOTIC INITIAL VELOCITY VECTOR ON PLANE THROUGH GRID CENTER
CALCULATED SPATIAL VARIATION OF VELOCITY ON PLANES THROUGH GRID CENTER

\[ Re_a = 13.3 \]

PLOT OF PROJECTION OF VELOCITY-VECTOR FIELD ON \( x_1 \times x_2 \) PLANE THROUGH GRID CENTER
Time series

ASYMPTOTIC REYNOLDS NUMBER

\[ \text{Re}_a = 4.78 \]

SIMPLE PERIODIC FLOW

\[ \text{Re}_a = 6.24 \]

\[ \text{Re}_a = 6.72 \]

Phase portrait

\[ \text{Re}_a = 4.78 \]
UNCONVERGED

\[
\begin{align*}
\text{SIMPLE} & \quad \text{PERIODIC} \\
\text{FLOW} & \quad \text{LIMIT} \\
\text{CYCLE} & 
\end{align*}
\]

\[
\begin{align*}
\text{CONVERGED} & \quad \text{Re}_a = 6.24 \\
\end{align*}
\]

\[
\begin{align*}
\text{Re}_a = 6.24 \\
\end{align*}
\]

\[
\begin{align*}
u_1 & \quad u_2 \\
(\pi, \pi, \pi) & \quad (\pi, \pi, \pi) \\
\end{align*}
\]

\[
\begin{align*}
\text{Re}_a = 6.24 \\
\end{align*}
\]
Apparent flow

Apparent edge of sheets

aperiodic flow
COMPLEX PERIODIC FLOW

WEAKLY CHAOTIC FLOW

FULLY CHAOTIC FLOW

Liapunov exponent plots
(aperiodic flows)
WEAKLY CHAOTIC FLOW

\[ \text{Re}_a = 6.72 \]

FULLY CHAOTIC FLOW

\[ \text{Re}_a = 6.93 \]

Power spectra
(Periodic flows)
Poincaré sections

Re_a = 6.24

PERIODIC FLOWS

Re_a = 6.72

aperiodic flows
SUMMARY

FIXED POINT

LIMIT CYCLE, SIMPLE PERIODIC (PERIOD 1)

LIMIT CYCLE, COMPLEX PERIODIC (PERIOD 2)

WEAKLY CHAOTIC FLOW (LIAPOUNOV EXPONENT POSITIVE, POINCARE'S SECTION HAS SOME PATTERN)

LIMIT CYCLE, COMPLEX PERIODIC (PERIOD 4) (PERIODIC WINDOW)

FULLY CHAOTIC FLOW (LIAPOUNOV EXPONENT POSITIVE, POINCARE'S SECTION HAS NO APPARENT PATTERN, TURBULENT)
OBJECTIVES

- To develop both analytical and numerical tools that can be used to predict the onset of instability and subsequently to simulate the transition process by which the originally laminar flow evolves into a turbulent flow.

- To conduct the preliminary investigations with the purpose of understanding the mechanisms of the vortical structure of the compressible flow between two concentric cylinders.
INCOMPRESSIBLE TAYLOR VORTICES

- If the angular velocity of the inner cylinder is larger than a critical value, Couette flow becomes unstable so that a secondary flow starts to appear with nonvanishing radial and axial velocity components; this new flow, called Taylor-Couette flow, is in the form of opposite toroidal vortices arranged next to each other in the axial direction.

COMPRESSIBLE TAYLOR VORTICES

- In the presence of heat transfer and angular motions, the influence of compressibility is nontrivially coupled and its effect on stability has yet to be determined.

- The addition of compressibility to the stability analysis greatly increases its complexity, since even in the linearized analysis there will now exist fluctuations not only in velocity and pressure but also in density, temperature, viscosity and thermal-conductivity.
GOVERNING EQUATIONS

• Mean Flow

momentum:

\[
\frac{\partial p}{\partial r} = \frac{w^2}{r}
\]

\[
\frac{\partial}{\partial r} \left[ \mu \left( \frac{\partial w}{\partial r} - \frac{w}{r} \right) \right] + \frac{2\mu}{r} \left( \frac{\partial w}{\partial r} - \frac{w}{r} \right) = 0
\]

density:

\[
\frac{1}{r} \frac{\partial}{\partial r} \left[ \alpha r \frac{\partial T}{\partial r} \right] + \mu M^2 (\gamma - 1) \left( \frac{\partial w}{\partial r} - \frac{w}{r} \right)^2 = 0
\]

\[
M = \frac{R_1 T_1^*}{\sqrt{\gamma R_e T_e}}
\]

• Boundary Conditions

at \( r = 1 \):

\[ w = 1, T = 1, P = \frac{1}{\gamma M^2}, \rho = 1 \]

and at \( r = \frac{R_2}{R_1} \):

\[ w = \frac{R_2 T_2^*}{R_1 T_1^*}, T = \frac{T_2^*}{T_1^*} = \phi \]

GOVERNING EQUATIONS

• Small Perturbation

\[
\phi(\xi, \eta, \zeta, t) = \Phi + \tilde{\phi}(\eta) \exp[i(\alpha \xi + \beta \zeta - \omega t)]
\]

where

\( \xi, \eta, \zeta \): computational coordinates in axial, radial, and circumferential directions, respectively

\( \Phi \): steady unperturbed flow,

\( \tilde{\phi} \): complex disturbances

\( \alpha \): disturbance wave number in \( \xi \) direction

\( \beta \): disturbance wave number in \( \zeta \) direction

\( \omega \): complex frequency
**Boundary Conditions for Disturbances**

- at \( r = 1 \):
  \[ \tilde{u}, \tilde{v}, \tilde{w}, \tilde{\tau} = 0 \]

and at \( r = \frac{R_2}{R_1} \):
  \[ \tilde{u}, \tilde{v}, \tilde{w}, \tilde{\tau} = 0 \]

Note that no boundary conditions are required for pressure fluctuations when the equations are solved on a pressure staggered mesh.

**METHOD OF SOLUTION**

- Numerical solutions for the mean flow
- Chebyshev collocation spectral method for stability analysis

**Momentum and Energy Equations:**

Gauss-Lobatto points for \( \tilde{u}, \tilde{v}, \tilde{w}, \tilde{\tau} \)

**Continuity Equation:**

Gauss points for \( \bar{p} \)

**Momentum and Energy Equations:**

\[
A_{GL} \phi + B_{GL} (\phi + I_{GL}^G p) + C_{GL} (\phi + I_{GL}^G p) = \omega \left[ D_{GL} (\phi + I_{GL}^G p) + E_{GL} (\phi + I_{GL}^G p) \right]
\]

**Continuity Equation:**

\[
B_{GL} I_{GL}^G \phi + C_{GL} (I_{GL}^G \phi + p) = \omega E_{GL} (I_{GL}^G \phi + p)
\]

where

\[
\phi = \begin{pmatrix} \tilde{u} \\ \tilde{v} \\ \tilde{w} \\ \tilde{\tau} \end{pmatrix}, \quad p = (\bar{p})
\]

and

- \( A, B, C, D, E \): coefficient matrices
- \( L \): spectral differentiation operator
- \( I \): spectral interpolation matrix

198
VERIFICATION

Comparison of the Critical Reynolds Number for Temporal Stability of Taylor-Couette Flow ($R_1^2/R_2^2=0.5$)

<table>
<thead>
<tr>
<th>Sparrow et al. \textsuperscript{20}</th>
<th>Present</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Omega_2^2/\Omega_1^2$</td>
<td>$Re_c$</td>
</tr>
<tr>
<td>0.0000</td>
<td>68.19</td>
</tr>
<tr>
<td>0.0600</td>
<td>73.02</td>
</tr>
<tr>
<td>0.1250</td>
<td>84.30</td>
</tr>
<tr>
<td>0.1800</td>
<td>107.21</td>
</tr>
<tr>
<td>0.2350</td>
<td>221.33</td>
</tr>
<tr>
<td>-0.10</td>
<td>66.18</td>
</tr>
<tr>
<td>-0.20</td>
<td>70.03</td>
</tr>
<tr>
<td>-0.30</td>
<td>80.93</td>
</tr>
<tr>
<td>-0.35</td>
<td>88.81</td>
</tr>
<tr>
<td>-0.50</td>
<td>114.24</td>
</tr>
</tbody>
</table>

RESULT AND DISCUSSION

MEAN VELOCITY PROFILES

$R_1^2/R_2^2=0.5$ and $\Omega_2^2=0$
RESULT AND DISCUSSION

MEAN TEMPERATURE PROFILES

$R_1^2/R_2^2 = 0.5$ and $\Omega_1^2 = 0$

EFFECT OF MACH NUMBER

$Re=100$ and $\beta=1.0$

EFFECT OF REYNOLDS NUMBER

$M=1.0$ and $\beta=1.0$
RESULT AND DISCUSSION

EFFECT OF TEMPERATURE RATIO

\( \alpha = 3.182, R_t^* = 0 \)

\[ E_{\tau} = \frac{E_{\tau}}{E_{\tau}} \]

\[ \text{Re}=600 \]
\[ \text{Re}=500 \]
\[ \text{Re}=200 \]
\[ \text{Re}=100 \]

\[ M=3.0 \]

RESULT AND DISCUSSION

EFFECT OF SPIN MOTION

\( M=1.0 \) and \( Re=500 \)

\[ \frac{\alpha_{\eta}}{\eta_{\tau}} = 0 \]
\[ \frac{\alpha_{\eta}}{\eta_{\tau}} = 0.18 \]
\[ \frac{\alpha_{\eta}}{\eta_{\tau}} = 0.5 \]

MAXIMUM GROWTH RATE

VS

TEMPERATURE RATIO

\( \alpha = 6.2, M=1.0, \) and \( Re=500 \)
RESULT AND DISCUSSION

\[ M=1.0, \, Re=800, \, \Omega^*/\Omega_1^*=0.0 \]
\[ \alpha=6.2, \, \text{and} \, \beta=1.0 \]

Radial-Velocity fluctuation

cell pattern of \( \tilde{v} \)

cell pattern of \( \tilde{u} \)

RESULT AND DISCUSSION

\[ M=1.0, \, Re=800, \, \Omega^*/\Omega_1^*=0.18 \]
\[ \alpha=6.2, \, \text{and} \, \beta=1.0 \]

Radial-Velocity fluctuation

cell pattern of \( \tilde{v} \)

cell pattern of \( \tilde{u} \)

202
RESULT AND DISCUSSION

\[ M=1.0, \: Re=800, \: \Omega_1/\Omega_2=0.5 \]
\[ \phi=6.2, \: and \: \delta=1.0 \]

Radial Velocity fluctuation

CONCLUDING REMARKS

- For the case of rotating inner cylinder, the higher Mach number flows are unstable to a larger range of wave numbers.

- By increasing the temperature ratio, the flow between counter rotating cylinders is stabilized with the growth rate decaying in a linear fashion. However, the effect of increasing temperature ratio is to amplify the maximum growth rate in the flow with a positive rotating speed ratio.

- Further examinations can be made on those effects of axial flow, gap width, finite axial length, and other possible influential factors.
Intro

Video animation is becoming increasingly vital to the CFD researcher, not just for the presentation results, but for recording and comparing dynamic visualizations that are beyond the current capabilities of even the most powerful graphic workstations. To meet these needs, NASA Lewis Research Center has recently established a facility to provide users with easy access to advanced video animation capabilities. However, producing animation that is both visually effective and scientifically accurate involves various technological and aesthetic considerations that must be understood both by the researcher and by those supporting the visualization process. Many of these considerations will be addressed in the presentation.

Scan Conversion

Conversion of high-resolution workstation images to low-resolution television images is performed by hardware that either converts only a portion of the screen, or invokes various pixel-averaging techniques. The former can result in excessive aliasing, whereas the latter can result in smearing and disappearance of points and lines. Both can have trouble displaying semi-transparent objects from workstation using bit-mask transparency algorithms. Scan conversion is a major problem in accurately portraying Computational Fluid Dynamics grid geometry.

Color Conversion

Translation of the workstation's red-green-blue component color space to the composite NTSC color system can generate various artifacts. This can be a major problem when color is used to convey scientific information. Several techniques are used at LeRC to overcome these problems, including: color-table adjustment to avoid over-saturation, and the use of edge-outlined contour plots to emphasize subtle color differences.

Spatial Ambiguities

Since passive viewers are denied the interactivity of workstations, that allows researchers to accurately determine spatial relationships and resolve visual ambiguities, it is critical that recorded data be easily understood. Careful attention must be paid to relative object and 'camera' movements and scene content. In addition, advanced display techniques such as solid, smooth-shaded stream-tubes instead of line-segment tracers can significantly improve flow visualization.
LeRC advanced Graphics & VISualization laboratory
(Video Animation Subsystem)

Local User

- RGB Monitor
- Local Viewing (RGB)
- Local Control
- Ethernet
- Remote Image X-fer & Control
- Abekas A60
- NTSC
- LINK (Labwide CATV)
- Remote Viewing (NTSC)
- TV Monitor
- VTR's

Remote Users

---

Video as a CFD Research Tool

* Visualize Time-Dependent Results
* Record Interactions with Data
* Workstations too slow for complex data
* Video cost effective & ubiquitous
Successful CFD Video Visualizations:

Technology -
- Scan Conversion
- Color Conversion
- Recording media

Aesthetics -
- Resolving Spatial Ambiguities
- Motion Control
- Script

Problems:

* Most researchers are unaware of steps necessary to go from workstation to video

* Most visualization software don’t provide all necessary tools

* ’Video-ifying’ the visualization usually occurs at the last minute
Scan Conversion

Workstation RGB Color Domain

Scanline

Red Signal

Green Signal

Blue Signal

Workstation
1024 Scanlines
60 Full Frames/sec
Non-Interlaced

Video
486 Scanlines
50 Full Frames/sec
2 Interlaced Fields
NTSC Color Domain
Susceptible to adjacent pixel color interference

Possible Effects of Scan Conversion

Hi-res Workstation  Lo-res Video
Effect of Scan Conversion on Texture Patterns

Workstation

Video

NTSC Color Artifacts

* Zippers
* Dot Crawl
* Saturation

Image at risk
Modify Colormaps to avoid over-saturation

Luminance:
\[ Y = 0.29 \cdot r + 0.59 \cdot g + 0.11 \cdot b \]

Chrominance:
\[ I = 0.59 \cdot r - 0.27 \cdot g - 0.32 \cdot b \]
\[ Q = 0.21 \cdot r - 0.52 \cdot g + 0.31 \cdot b \]

Saturation:
\[ S = (I^2 + Q^2)^{1/2} \]

O.K. If \(-0.25 < (Y-S), (Y+S) < 1.0\)
Resolve Spatial Ambiguities:
Video is passive -- viewer cannot manipulate objects.

Line tracers  Tube tracers

advanced Graphics & VISualization laboratory’s
digital video facility

Local User

Local Viewing (RGB)

RGB Monitor

Local Control

Ethernet

Remote Image X-fer & Control

Remote Viewing (NTSC)

Remote Users

Abekas A60

VTR’s

NTSC

LINK (Labwide CATV)
Techniques for CFD Animation

* Use thick lines when possible
* Outline contour edges if possible
* Use nonsaturating colors
* Use shaded tracers
* Preview images on TV monitor
* Avoid simultaneous viewpoint and data motion
Distributed concurrent visualization and computation in Computational Fluid Dynamics is not a new concept. Specialized applications such as RIP (Realtime Interactive Particle-tracer) and vendor specific tools like DGL (Distributed Graphics Language) have been in use for some time. Little work however, has been done to put together a concise, easy to use set of routines which would allow the CFD code developer to take advantage of this technology. Recent advances in the speeds of the supercomputer and graphics workstations have made it possible to realize many of the applications for distributed visualization which have long been envisioned. Among these are the ability to "watch" a CFD code as it runs on a supercomputer, to perform interactive grid generation on a local workstation interacting with a solver running on the supercomputer, and ultimately, to "steer" the CFD run interactively from the workstation. For any of these applications to be developed, a set of tools are required to provide this distributed capability to the scientist. Three components can be identified as being necessary to make this capability useful to the CFD researcher. First, a FORTRAN-callable set of network routines must be provided to "link-up" the distributed processes and provide continuous process-to-process communication. Data is moved from the computational process to the visualization process transparently. Secondly, a FORTRAN-callable set of visualization functions must also be provided to the CFD researcher. These functions must allow the researcher to easily view the data in a meaningful, understandable way. Finally, some empirical data should be available to provide guidance to the researcher on the type of applications which would benefit from this kind of distributed processing. Several problems must be addressed in providing each of these three components. In dealing with the network, capacity is always a concern. Binary data compatibility between machines with vastly different architectures is a major factor which must be addressed. Most importantly, ease of use from the CFD code developers point of view must be maintained in delivering a useful solution.

This paper describes a current project underway at NASA Lewis Research Center to provide the CFD researcher with an easy method for incorporating distributed processing concepts into program development. Details on the FORTRAN callable interface to a set of network and the visualization functions are presented along with some results from initial CFD case studies that employ these techniques.
OBJECTIVE:

PROVIDE CFD RESEARCHERS AN EASY METHOD TO INCORPORATE DISTRIBUTED VISUALIZATION TECHNIQUES INTO THEIR PROGRAM DEVELOPMENT.

DISTRIBUTED VISUALIZATION:

CONCURRENT GRAPHICAL RENDERING ON ONE SYSTEM OF RESULTS, AS THEY ARE COMPUTED ON ANOTHER.
APPLICATIONS FOR DISTRIBUTED VISUALIZATION IN CFD

- Convergence / Status Checking (PEEKER)
- Interactive / Batch Grid Generation
- Immediate Visualization of Time-Dependent Calculations
- Interactive "Steering" of Remote Computations

![Diagram](SupercomputerPersonalWorkstation)

DEGREES OF DISTRIBUTED VISUALIZATION

- All computation, scientific visualization, and graphics performed remotely.
  
  ex: Ultra Frame Buffer

- Computations and scientific visualization done remotely, graphics performed by the workstation.
  
  ex: DGL for SGI workstations

- Numeric Computations done remotely, all scientific visualization and graphics performed by the workstation.
  
  most difficult to implement
  - requires network interface capable of transferring transparent binary data
  
  highest payoff
  - provides greatest degree of parallelism
  - no static datasets required to visualize results
TOOLS REQUIRED FOR DISTRIBUTED VISUALIZATION

- FORTRAN-Callable Set of Network Interface Routines
  - establish network connection between processes
  - provide continuous process-to-process communication

- FORTRAN-Callable Set of Visualization Functions
  - incorporate state-of-the-art rendering technology
  - utilize non-proprietary graphics interface
  - encompass all current CFD post-processing capability

Design Goal: Make Routines Easy to Use From a CFD Code Developers Point of View

Open Close Routines:
OPNCON -- Open a "connect" on a port
OPNLIS -- Open a "listen" on a port

Send Routines:
PUTRA1 -- Send array of real numbers
PUTIA1 -- Send array of integer numbers
PUTSTR -- Send character string
PUTRA2 -- Send 2-dimensional array real numbers
PUTIA2 -- Send 2-dimensional array of integers

Receive Routines:
GETRA1 -- Receive array of real numbers
GETIA1 -- Receive array of integer numbers
GETSTR -- Receive character string
GETRA2 -- Receive 2-dimensional array real numbers
GETIA2 -- Receive 2-dimensional array of integers
c Sample SEND program..
  
  c establish network connection
  CALL OPNCON( *** )
  
  c transfer 2D computational array
  CALL PUTRA2( *** )

---

c Sample RECEIVE program..
  
  c accept network connection
  CALL OPNLIS( *** )
  
  c receive 2D array from server
  CALL GETRA2( *** )

---

! *** SVP/NETWORK RECEIVE CODE
PARAMETER (ID=80, JD=70)
REAL xgrd(ID,JD), ygrd(ID,JD)
REAL ws(ID,JD)

! *** access the network
  call OPNLIS

! *** get data from the network
100 CONTINUE
  call GETIA1(1,nx,istat)
  call GETIA1(1,ny,istat)
  call GETRA2(nx,ny,2,ws,istat)
  call GETRA2(nx,ny,2,xgrid,istat)
  call GETRA2(nx,ny,2,ygrid,istat)

! *** introduce data to SVP
  call GDATA(nx,ny,nx,ny,xgrid,ygrid)
  call FDATA(ws,' ')
  call GENCON(' ')
  call CLRsvP
GOTO 100
STOP
END
Two approaches have been taken to provide systematic grid manipulation for improved grid quality.

One is the control point form (CPF) of algebraic grid generation. It provides explicit control of the physical grid shape and grid spacing through the movement of the control points. The control point array, called a control net, is a sparse grid-type framework in physical space. The CPF-based grid manipulation has many merits. The method is efficient and easy to implement, since its formulation is algebraic and concise. Grid quality can be easily enhanced by numerous local and global grid distribution strategies. It is also compatible with various complementary operations needed in a quality grid-generation procedure. It works well in the interactive computer graphics environment and hence can be a good candidate for integration with other emerging technologies.

The other approach is grid adaptation using a numerical mapping between the physical space and a parametric space. Grid adaptation is achieved by modifying the mapping functions through the effects of grid control sources. The source strengths are extracted from the distribution of flow or geometric properties on the initial grid. Grids can be made adaptive to geometry, flow solution, or grid quality, depending on what property is used to define the source strengths. One advantage of the method is that the basic characteristics of the initial grid can be retained while adapting it to grid quality. The use of grid control sources allows for linear combinations of different controls based on the superposition principle. And the grid can be adapted to more than one property through a series of mappings. The source formulation promotes smooth variations in the grid, even with irregularly distributed sources. The adaptation process can be repeated in a cyclic manner if satisfactory results are not achieved after a single application.
OUTLINE

1. EXPLICIT GRID MANIPULATION BY USER

   TECHNICAL APPROACH - CONTROL POINT FORM (CPF)
   OBJECTIVES / ADVANTAGES
   STATUS / RESULTS
   FUTURE DIRECTION

2. GRID ADAPTATION TO SOLUTIONS, QUALITY

   TECHNICAL APPROACH - NUMERICAL MAPPING with GRID CONTROL SOURCE
   OBJECTIVES / ADVANTAGES
   STATUS / RESULTS
   FUTURE DIRECTION

OBJECTIVES

TO DEVELOP EASY, EXPLICIT GRID MANIPULATION CAPABILITY

ADVANTAGES

THE CPF-BASED GRID GENERATION:

- IS EFFICIENT AND EASY TO IMPLEMENT
- CAN EASILY ENHANCE GRID QUALITY BY VARIOUS LOCAL/GLOBAL GRID DISTRIBUTION STRATEGIES
- CAN BE USED AS A COMPLEMENTARY TOOL AS WELL AS A STAND-ALONE TOOL
- WORKS WELL IN THE INTERACTIVE COMPUTER GRAPHICS ENVIRONMENT
- ALLOWS TO MANIPULATE GEOMETRY (FREE-FORM BOUNDARY)
- MAKES INTERACTIVE CFD ATTRACTIVE
TECHNICAL APPROACH - CONTROL POINT FORM (CPF)

CONSTRUCTION OF A CURVE & A CONTROL NET

\[ Q(r, t) = T(r, t) + \alpha_1[1 - C_1(r)][P(1, t) - F_1(t)] + \alpha_2 G_{N-1}(r)[P(N-1, t) - F_{N}(t)] + \alpha_3[1 - H_1(t)][P(r, 1) - E_1(r)] + \alpha_4 H_{M-1}(t)[P(r, M-1) - E_{M}(r)] \]

TURBO/I INTERACTIVE PROCESS
TURBO/I INTERACTIVE PROCESS - CONTINUED

CHANGING THE NUMBER OF CONTROL POINTS

TURBO/I INTERACTIVE PROCESS - CONTINUED

STRETCHING
GEOMETRY MODIFICATION USING FREE-FORM BOUNDARY

TURBO/I AS A COMPLEMENTARY TOOL
FUTURE DIRECTION

SURFACE GRID CONTROL USING CONTROL POINT FORM

INTERACTIVE ADAPTIVE GRID GENERATION

TECHNICAL APPROACH
NUMERICAL MAPPING WITH GRID CONTROL SOURCE

initial grid

(s,t) parametric domain

σ† distribution

solution-adapted grid

isobars on initial grid

σ† distribution

(s',t') parametric domain

isobars on adapted grid
Grid control sources definition:

\[ \sigma_{s_{kl}} = w_0^s \phi + w_1^s \frac{\partial \phi}{\partial s} + w_2^s \frac{\partial^2 \phi}{\partial s^2} \]

\[ \sigma_{t_{kl}} = w_0^t \phi + w_1^t \frac{\partial \phi}{\partial t} + w_2^t \frac{\partial^2 \phi}{\partial t^2} \]

where \( w_i \) = weighting factors

\( \phi \) = a grid quality parameter

Mapping modification:

\[ s'_{ij} = s_{ij} + \sum_{k,l} K_{ijkl}^s \sigma_{s_{kl}} \]

\[ t'_{ij} = t_{ij} + \sum_{k,l} K_{ijkl}^t \sigma_{t_{kl}} \]

where \( (s', t') \) = modified parametric coordinates

\( K_{ijkl}^s, K_{ijkl}^t \) = influence coefficients
OBJECTIVES

TO DEVELOP GRID ADAPTATION CAPABILITY TO SOLUTIONS, GRID QUALITY, AND GEOMETRY TO OBTAIN ACCURATE PREDICTION OF COMPLEX FLOWS

ADVANTAGES

THE APPROACH IS EASY TO IMPLEMENT AS ONE OF THE REDISTRIBUTION SCHEME.

THE USE OF GRID CONTROL SOURCES ALLOWS TO ADAPT THE GRID BY USING LINEAR COMBINATIONS OF FLOW PROPERTIES.

THE APPROACH CONTROLS BOTH GRID DENSITY AND QUALITY WITHOUT ANY ADVERSE INFLUENCE OF ONE TO THE OTHER.

THE GRID ADAPTATION SCHEME CAN BE INTEGRATED WITH WELL-DEVELOPED FLOW SOLVERS AS SUBROUTINES.

NEEDS

FULL 3D ADAPTATION WITH REDUCED COST

STATUS / RESULTS

2D ADAPTATION TO SOLUTIONS, GRID QUALITY

APPLICATION OF 2D ADAPTATION TECHNIQUE TO 3D PROBLEMS
CORNER FLOW
GRID COMPARISON

Initial grid
@ x = 0.407

Initial grid
@ x = 0.688

Initial grid
@ x = 1.000

Adapted grid
@ x = 0.407

Adapted grid
@ x = 0.688

Adapted grid
@ x = 1.000

CORNER FLOW
SOLUTION COMPARISON

Density contours
on initial grid
@ x = 0.407

Density contours
on initial grid
@ x = 0.688

Density contours
on initial grid
@ x = 1.000

Density contours
on adapted grid
@ x = 0.407

Density contours
on adapted grid
@ x = 0.688

Density contours
on adapted grid
@ x = 1.000

233
FUTURE DIRECTION

EXTEND IT TO FULL 3D ADAPTATION

EXPLORE AND IMPLEMENT COST REDUCTION SCHEMES

EXPLORE POSSIBLE BENEFITS FROM PARALLEL COMPUTING

APPLY IT TO CHALLENGING PROPULSION PROBLEMS

INTERACTIVE GRID GENERATION

Program TURBO

CONSTRUCT A SIMPLE CONTROL NET
GENERATE SURFACE GRID

COMPUTE & EXAMINE INITIAL GRID

SELECT A CONTROL POINT TO MODIFY GRID

EXAMINE NEW GRID
-ACCEPT OR REPEAT THE PROCESS

COMPUTE NEW GRID

TRANSLATE THE CONTROL POINT
This collection of papers was presented at the Computational Fluid Dynamics (CFD) Conference held at Ames Research Center in California on March 12 through 14, 1991. It is an overview of CFD activities at NASA Lewis Research Center. The main thrust of computational work at Lewis is aimed at propulsion systems such as gas turbines, rocket engines, and hypersonic propulsion systems. Specific issues related to propulsion CFD and associated modeling will also be presented. Examples of results obtained with the most recent algorithm development will also be presented.