Tenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion

Compiled by
R. W. Williams
George C. Marshall Space Flight Center
Marshall Space Flight Center, Alabama

Proceedings of a workshop held at
NASA George C. Marshall Space Flight Center
Huntsville, Alabama
April 28–30, 1992

NASA
National Aeronautics and Space Administration
Office of Management
Scientific and Technical Information Program
1992
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>TECHNOLOGY TEST BED REVIEW (H.V. McConnaughey)</td>
<td>1</td>
</tr>
<tr>
<td>ADVANCED SOLID ROCKET MOTOR PROJECT STATUS (K.D. Coates)</td>
<td>27</td>
</tr>
<tr>
<td>SPACE TRANSPORTATION MAIN ENGINE (J.C. Monk)</td>
<td>45</td>
</tr>
<tr>
<td>THE IMPACT OF TIME STEP DEFINITION ON CODE CONVERGENCE AND ROBUSTNESS (S. Venkateswaran, J.M. Weiss, and C.L. Merkle)</td>
<td>83</td>
</tr>
<tr>
<td>COMPARISON BETWEEN THE PISO ALGORITHM AND PRECONDITIONING METHODS FOR COMPRESSIBLE FLOW (C.L. Merkle, P.E.O. Buelow, and S. Venkateswaran)</td>
<td>123</td>
</tr>
<tr>
<td>A COMPARISON OF ARTIFICIAL COMPRESSIBILITY AND FRACTIONAL STEP METHODS FOR INCOMPRESSIBLE FLOW COMPUTATIONS (D.C. Chan, A.D. Darian, and M.M. Sindir)</td>
<td>147</td>
</tr>
<tr>
<td>CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER (G.C. Cheng, Y.S. Chen, and R.W. Williams)</td>
<td>201</td>
</tr>
<tr>
<td>CFD APPLICATIONS IN PUMP FLOWS (C. Kiris, L. Chang, and D. Kwak)</td>
<td>219</td>
</tr>
<tr>
<td>IMPELLER TANDEM BLADE STUDY WITH GRID EMBEDDING FOR LOCAL GRID REFINEMENT (G. Bache')</td>
<td>259</td>
</tr>
<tr>
<td>THREE-DIMENSIONAL FLOW FIELDS INSIDE A SHROUDED INDUCER AT DESIGN AND OFF-DESIGN CONDITIONS (CFD STUDY) (C. Hah, O. Kwon, D.A. Greenwald, and R. Garcia)</td>
<td>289</td>
</tr>
<tr>
<td>EFFECTS OF CURVATURE AND ROTATION ON TURBULENCE IN THE NASA LOW-SPEED CENTRIFUGAL COMPRESSOR IMPELLER (J.G. Moore and J. Moore)</td>
<td>315</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMIC DESIGN OF ROCKET ENGINE PUMP COMPONENTS (W.C. Chen, G.H. Prueger, D.C. Chan, and A.H. Eastland)</td>
<td>339</td>
</tr>
</tbody>
</table>
TABLE OF CONTENTS (Continued)

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>SSME HPOTP IMPELLER BACKCAVITY CFD ANALYSIS (W.W. Hsu and S.J. Lin)</td>
<td>361</td>
</tr>
<tr>
<td>NLS CLUTCHING BEARING CAVITY FLOW ANALYSIS (K. Tran, D.C. Chan, and A. Darian)</td>
<td>389</td>
</tr>
<tr>
<td>CFD ANALYSIS TO OPTIMIZE A DESIGN MODIFICATION OF BSMT (M. Ratcliff, R. Avva, and R. Williams)</td>
<td>419</td>
</tr>
<tr>
<td>COMBUSTION INSTABILITY ANALYSIS FOR LIQUID PROPELLANT ROCKET ENGINES (Y.M. Kim, C.P. Chen, and J.P. Ziebarth)</td>
<td>441</td>
</tr>
<tr>
<td>INVERSE DESIGN OF A PROPER NUMBER, SHAPES, SIZES, AND LOCATIONS OF COOLANT FLOW PASSAGES (G.S. Dulikravich)</td>
<td>467</td>
</tr>
<tr>
<td>NUMERICAL ANALYSIS OF THE HOT-GAS-SIDE AND COOLANT-SIDE HEAT TRANSFER IN LIQUID ROCKET ENGINE COMBUSTORS (T.S. Wang and V. Luong)</td>
<td>487</td>
</tr>
<tr>
<td>AN EFFICIENT AND ROBUST GRID OPTIMIZATION ALGORITHM (B.K. Soni and S. Yang)</td>
<td>503</td>
</tr>
<tr>
<td>ENHANCEMENTS TO THE GRIDGEN STRUCTURED GRID GENERATION SYSTEM FOR INTERNAL AND EXTERNAL FLOW APPLICATIONS (J.P. Steinbrenner and J.R. Chawner)</td>
<td>543</td>
</tr>
<tr>
<td>CAGI: COMPUTER AIDED GRID INTERFACE—A WORK IN PROGRESS (B.K. Soni, T.-Y. Yu, and D. Vaughn)</td>
<td>577</td>
</tr>
<tr>
<td>USING ADAPTIVE GRID IN MODELING ROCKET NOZZLE FLOW (A.S. Chow and K.-R. Jin)</td>
<td>615</td>
</tr>
<tr>
<td>COMPLEX THREE-DIMENSIONAL INTERNAL FLOWS IN THE ASRM AND RSRM AFT END SEGMENTS (E.J. Reske, D.F. Billings, and J.W. Cornelison)</td>
<td>647</td>
</tr>
<tr>
<td>AN ANALYSIS OF THE FLOW FIELD IN THE REGION OF THE ASRM FIELD JOINTS (R.A. Dill and H.R. Whitesides)</td>
<td>663</td>
</tr>
<tr>
<td>EFFECT OF INCLUDING VARIABLE GAS PROPERTIES AND ENTRAINED PARTICLES IN THE FLOW ANALYSIS OF THE ASRM NOZZLE (C.D. Clayton)</td>
<td>689</td>
</tr>
<tr>
<td>A TWO-PHASE RESTRICTED EQUILIBRIUM MODEL FOR COMBUSTION OF METALIZED SOLID PROPELLANTS (J.S. Sabnis, F.J. de Jong, and H.J. Gibeling)</td>
<td>713</td>
</tr>
<tr>
<td>Title</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>CURRENT STATUS OF THE DEVELOPMENT OF AN IGNITION TRANSIENT MODEL</td>
<td>725</td>
</tr>
<tr>
<td>FOR SOLID ROCKET MOTORS (G.D. Luke and H.A. Dwyer)</td>
<td></td>
</tr>
<tr>
<td>SRMAFTE FACILITY CHECKOUT MODEL FLOW FIELD ANALYSIS (R.A. Dill</td>
<td>763</td>
</tr>
<tr>
<td>and H.R. Whitesides)</td>
<td></td>
</tr>
<tr>
<td>A COMPARATIVE STUDY OF THE EFFECTS OF INHIBITOR STUB LENGTH ON</td>
<td>787</td>
</tr>
<tr>
<td>SOLID ROCKET MOTOR COMBUSTION CHAMBER PRESSURE OSCILLATIONS: RSRM</td>
<td></td>
</tr>
<tr>
<td>AT T=80 SECONDS, PRELIMINARY RESULTS (D. Chasman, D. Burnette, J.</td>
<td></td>
</tr>
<tr>
<td>Holt, and R. Farr)</td>
<td></td>
</tr>
<tr>
<td>OVERVIEW OF THE RELEVANT CFD WORK AT THIOKOL CORPORATION</td>
<td>791</td>
</tr>
<tr>
<td>(P. Chwalowski and H.-T. Loh)</td>
<td></td>
</tr>
<tr>
<td>A STATUS OF THE ACTIVITIES OF THE NASA/MSFC COMBUSTION DEVICES</td>
<td>807</td>
</tr>
<tr>
<td>TECHNOLOGY TEAM (P.K. Tucker)</td>
<td></td>
</tr>
<tr>
<td>CFD ANALYSIS OF THE STME NOZZLE FLOWFIELD (A. Krishnan, P.K. Tucker)</td>
<td>831</td>
</tr>
<tr>
<td>NLS NOZZLE BASE FLOW CHARACTERISTICS (J.J. Erhart)</td>
<td>849</td>
</tr>
<tr>
<td>HEAT TRANSFER IN ROCKET ENGINE COMBUSTION CHAMBERS AND NOZZLES</td>
<td>863</td>
</tr>
<tr>
<td>(P.G. Anderson, Y.S. Chen, and R.C. Farmer)</td>
<td></td>
</tr>
<tr>
<td>APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO THE DESIGN OF THE</td>
<td>897</td>
</tr>
<tr>
<td>FILM COOLED STME SUBSCALE NOZZLE FOR THE NATIONAL LAUNCH SYSTEM</td>
<td></td>
</tr>
<tr>
<td>(J.L. Garrett)</td>
<td></td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF SPACE SHUTTLE MAIN ENGINE</td>
<td>923</td>
</tr>
<tr>
<td>MULTIPLE PLUME FLOWS AT HIGH-ALTITUDE FLIGHT CONDITIONS (N.S.</td>
<td></td>
</tr>
<tr>
<td>Dougherty, J.B. Holt, B.L. Liu, and S.L. Johnson)</td>
<td></td>
</tr>
<tr>
<td>DIRECT NUMERICAL SIMULATION OF A COMBUSTING DROPLET WITH CONVECTION</td>
<td>945</td>
</tr>
<tr>
<td>(P.Y. Liang)</td>
<td></td>
</tr>
<tr>
<td>A NUMERICAL MODEL FOR ATOMIZATION-SPRAY COUPLING IN LIQUID ROCKET</td>
<td>965</td>
</tr>
<tr>
<td>THRUST CHAMBERS (M.G. Giridharan, A. Krishnan, J.J. Lee, A.J.</td>
<td></td>
</tr>
<tr>
<td>Przekwas, and K. Gross)</td>
<td></td>
</tr>
<tr>
<td>NUMERICAL MODELING FOR DILUTE AND DENSE SPRAYS (C.P. Chen, Y.M. Kim,</td>
<td>987</td>
</tr>
<tr>
<td>H.M. Shang, J.P. Ziebarth, and T.S. Wang)</td>
<td></td>
</tr>
<tr>
<td>MODELING OF SSME FUEL PREBURNER ASI (P.Y. Liang)</td>
<td>1013</td>
</tr>
</tbody>
</table>
### TABLE OF CONTENTS (Continued)

<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD MODELING OF TURBULENT FLOWS AROUND THE SSME MAIN INJECTOR ASSEMBLY USING POROSITY FORMULATION (G.C. Cheng, Y.S. Chen, and J.H. Ruf)</td>
<td>1033</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF SSME PHASE II AND PHASE II+ PREBURNER INJECTOR ELEMENT HYDROGEN FLOW PATHS (J. H. Ruf)</td>
<td>1071</td>
</tr>
<tr>
<td>STME HYDROGEN MIXER STUDY (R. Blumenthal, D. Kim, and G. Bache’)</td>
<td>1093</td>
</tr>
<tr>
<td>AN EXPERIMENTAL STUDY OF THE FLUID MECHANICS ASSOCIATED WITH POROUS WALLS. (N. Ramachandran, J. Heaman, and A. Smith)</td>
<td>1117</td>
</tr>
<tr>
<td>TURBINE DISK CAVITY AERODYNAMICS AND HEAT TRANSFER (B.V. Johnson and W.A. Daniels)</td>
<td>1163</td>
</tr>
<tr>
<td>A NUMERICAL STUDY OF TWO-DIMENSIONAL VORTEX SHEDDING FROM RECTANGULAR CYLINDERS (A.H. Hadid, M.M. Sindir, and R.I. Issa)</td>
<td>1181</td>
</tr>
<tr>
<td>A STATUS OF THE TURBINE TECHNOLOGY TEAM ACTIVITIES (L.W. Griffin)</td>
<td>1205</td>
</tr>
<tr>
<td>A CRITICAL EVALUATION OF A THREE-DIMENSIONAL NAVIER-STOKES CFD AS A TOOL TO DESIGN SUPersonic TURBine STAGEs (C. Hah, O. Kwon, and M. Shoemaker)</td>
<td>1227</td>
</tr>
<tr>
<td>NAVIER-STOKES ANALYSIS OF AN OXIDIZER TURBINE BLADE WITH TIP CLEARANCE (H.J. Gibeling and J.S. Sabnis)</td>
<td>1243</td>
</tr>
<tr>
<td>NUMERICAL SIMULATION OF TURBOMACHINERY FLOWS WITH ADVANCED TURBULENCE MODELS (B. Lakshminarayana, R. Kunz, J. Luo, and S. Fan)</td>
<td>1275</td>
</tr>
<tr>
<td>DEVELOPMENT OF A CFD CODE FOR INTERNAL FLOWS IN LIQUID FUELED ENGINES (Y. Dakhoul)</td>
<td>1307</td>
</tr>
<tr>
<td>DEVELOPMENT OF THE KIVA-II CFD CODE FOR ROCKET PROPULSION APPLICATIONS (R.V. Shannon, Jr. and A.L. Murray)</td>
<td>1349</td>
</tr>
<tr>
<td>A COMPUTATIONAL DESIGN SYSTEM FOR RAPID CFD ANALYSIS (E.P. Ascoli, S.L. Barson, M.E. DeCroix, and M.M. Sindir)</td>
<td>1379</td>
</tr>
<tr>
<td>OPTIMUM DESIGN OF NINETY DEGREE BENDS (V. Modi)</td>
<td>1397</td>
</tr>
</tbody>
</table>
TABLE OF CONTENTS (Continued)

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A MULTIDOMAIN METHOD FOR SUBSONIC VISCOUS FLOWS (D.C. Chan and M.M. Sindir)</td>
<td>1427</td>
</tr>
<tr>
<td>LARGE EDDY SIMULATION OF COMPRESSIBLE TURBULENT CHANNEL FLOWS (R.A. Beddini and J.P. Ridder)</td>
<td>1453</td>
</tr>
<tr>
<td>TREATING CONVECTION IN SEQUENTIAL SOLVERS (W. Shyy and S. Thakur)</td>
<td>1469</td>
</tr>
</tbody>
</table>
An ignition transient model for solid rocket motors is currently being developed jointly by the Aerojet ASRM Division and the University of California at Davis. Though the CFD code will be general enough to predict the start transient for most solid rocket motors, the particular motive for the development of this code stems from the desire to analyze the flow field within the Advanced Solid Rocket Motor (ASRM) for the Space Shuttle with all its geometric complexity. The star grain configuration in the head end of the ASRM coupled with the multiport igniter creates a formidable problem which can only be modeled accurately using a three dimensional Navier-Stokes code if one wishes to preserve both volume and burning surface area as a function of axial position down the bore. The actual physical geometry is crucial in modeling the multiple wave interactions occurring within the combustion chamber as well as in predicting the correct amount of mass, momentum, and energy injected as a function of time and space at the propellant surface.

The primary objectives of the CFD code are to calculate the pressure rise rate, the thrust rise rate, and the maximum chamber pressure which occurs during the first second of the ASRM action time. And, more specifically, to determine the relative difference between the ignition transients produced using a single port igniter and that produced by a multiport igniter.

An implicit, three dimensional, time accurate, finite volume method is being developed to solve this problem. Current plans are to use an ADI technique, with replacement, to solve the set of analytically linearized full Navier-Stokes equations. However, to gain an understanding of which features of the code need more attention than others, and to provide an inexpensive, quick tool for studying the time accuracy of the selected solution algorithm, a one dimensional version of the code was written first. The various features of the one dimensional code were validated by comparing the numerical results to analytical results for problems where exact solutions were available. Then the one dimensional code was used to perform some sensitivity studies to help develop an understanding of the complex wave interactions occurring within the solid rocket motor.

This presentation will discuss the results obtained from the one dimensional code and the plans for the development of the 3-D code.
CURRENT STATUS OF THE DEVELOPMENT OF AN IGNITION TRANSIENT MODEL FOR SOLID ROCKET MOTORS

Gary D. Luke
Aerojet ASRM Division
Sacramento, California
and
Harry A. Dwyer
University of California
Davis, California

1992 Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion
OBJECTIVES

- Develop Ignition Transient Model(s) to Characterize the Start Transient for Solid Rocket Motors

- Determine Relative Differences Between the Ignition Transients Produced Using Single (RSRM) and Multiport (ASRM) Igniters
DEFINITION OF PROBLEM
NEED TO ACCURATELY PREDICT:

- Instantaneous Pressure Rise Rate
- Corresponding Thrust Rise Rate
- Maximum Chamber Pressure
- Details of Flow Field in Head End Where Fins and Igniter are Located for RSRM and ASRM
THREE PART MODELING APPROACH

1. Preliminary Analysis with Existing General 3-D Navier-Stokes Solver (CONTINUSYS Code)

2. Develop 1-D "Engineering Workhorse" Ignition Transient Code

3. Develop "Full" 3-D Ignition Transient Code Using Same Methodology as 1-D Code

Only 1-D Code Discussed Today
PURPOSE OF 1-D CODE

- Provide Quick, Efficient Tool for Performing Sensitivity Studies
- Establish Bounds for Problem Through Parametric Studies
- Evaluate Various Numerical Schemes Prior to Incorporation into 3-D Code
- Determine Which Factors are Most Influential, Requiring More Attention During Development of 3-D Code
DESCRIPTION OF NUMERICAL MODEL

- Implicit, Time Dependent, Finite Volume Method
- Modular Format for Flexibility
- Currently Contains Centered Scheme with Explicit Artificial Dissipation
- System of Equations Solved Using Alternating Direction Implicit (ADI) with Replacement (Essentially a Direct Solver for 1-D Case)
Governing Equations in Integral Form

Continuity

\[ \frac{\partial}{\partial t} \iiint_{v} \rho dV + \iint_{s} \rho \vec{u} \cdot \vec{n} dA = \iint_{A_{p}} \dot{\rho}_{p} dA_{p} \]

X-Momentum

\[ \frac{\partial}{\partial t} \iiint_{v} \rho \vec{u} dV + \iint_{s} \rho \vec{u} (\vec{u} \cdot \vec{n}) dA = -\vec{n} \iint_{s} \rho dA - \iint_{w} \tau_{w} dA_{w} - \iint_{s} \tau_{xx} dA \]

Energy

\[ \frac{\partial}{\partial t} \iiint_{v} \rho e dV + \iint_{s} [\rho e + P] (\vec{u} \cdot \vec{n}) dA = \iint_{w} q_{w} dA_{w} + \iint_{s} q_{x} dA + \iint_{A_{p}} \dot{\rho}_{p} H_{p} dA_{p} \]

where \( e = C_v T + u^2/2 \)

Equation of State

\[ P = \rho RT \]
Linearized Form of Discrete Equations

\[
\frac{A_i - 1/2}{2} \left[ \begin{array}{c} \frac{\varepsilon}{J_{i-1}} \\ \frac{\varepsilon}{J_i} \end{array} \right] (\Delta \hat{U}_{i+1}^n) + \left\{ \frac{A_i \Delta X_i}{\Delta t} \left[ \begin{array}{c} I_g \\ I_s \end{array} \right] \left( \frac{A_{i+1/2} - A_{i-1/2}}{2} \right) \left[ \begin{array}{c} \varepsilon_n \\ J_{i-1} \end{array} \right] \right\} (\Delta \hat{U}_i^{n+1}) + \frac{A_i + 1/2}{2} \left[ \begin{array}{c} \varepsilon_n \\ J_{i+1} \end{array} \right] (\Delta \hat{U}_{i+1}^{n+1})
\]

\[= \left[ \hat{S}_i^n \right] + \frac{A_i - 1/2}{2} \left[ \hat{F}_i^n + \hat{F}_{i-1}^n \right] - \frac{A_i + 1/2}{2} \left[ \hat{F}_{i+1}^n + \hat{F}_i^n \right] \]

where: \( \hat{U} \equiv \text{primitive variable vector} = \left[ \begin{array}{c} \rho \\ \rho u \\ \rho e \end{array} \right] \)

\( \hat{F} \equiv \text{flux vector} = \left[ \begin{array}{c} \rho u \\ \rho u^2 + P \\ (\rho e + P)u \end{array} \right] \)

\[\hat{S}_i = \text{Jacobian Matrix} = \left[ \frac{\partial \hat{F}}{\partial \hat{U}} \right] \]

\[
\hat{S} = \left[ \begin{array}{c} [\hat{F} \rho_p A_p]^n_i \\
- \left[ \frac{f}{2} \rho \left. u^2 \frac{u}{u} \right| A_w \right]^n_i - \left\{ \left[ \frac{\mu}{\partial x} \right]^n_{i+1/2} - \left[ \frac{\mu}{\partial x} A \right]^n_{i-1/2} \right\} + P_i^n \left[ A_{i+1/2} - A_{i-1/2} \right] \\
\left[ h(T-T_w) A_w \right]^n_i + \left\{ \left[ -k \left. \frac{\partial T}{\partial x} \right| A \right]^n_{i+1/2} - \left[ -k \left. \frac{\partial T}{\partial x} A \right]^n_{i-1/2} \right\} + [\hat{F} \rho_p H_p A_p]^n_i \end{array} \right]
\]

Analytically Linearized Using Newton's Method
Artificial Dissipation
2nd Order Terms Required to Dampen Out Oscillations Occurring at Steep Gradients

- Continuity Equation:
  \[ \eta \frac{\partial \rho}{\partial x} \]

- Momentum Equation:
  \[ \tau_{xx} = (\mu_R + \mu_A) \frac{\partial u}{\partial x}, \mu_A = \frac{\rho u \Delta x}{R_e} \]

- Energy Equation:
  \[ q_x = - (k_R + k_A) \frac{\partial T}{\partial x}, k_A = \frac{\rho C_p u \Delta x}{P_e} \]

R = Real, A = Artificial

Pe = Peclet Number (Re*Pr)
Re = Reynold’s Number
Pr = Prandtl Number
FEATURES OF 1-D CODE

- Heat Transfer
- Friction
- Variable Area
- Mass Injection Through Side Walls
BOUNDARY CONDITIONS

- ENTRANCE BOUNDARY CONDITIONS
  - Solid Wall
  - All Static Conditions Specified
  - Reservoir Conditions Specified
  - Solid Wall with Mass and Energy Source

- EXIT PLANE BOUNDARY CONDITIONS
  - Solid Wall
  - Static Pressure Specified
  - Frictionless, Constant Area Extension for Supersonic Outflow
  - Linear Extrapolation for Supersonic Outflow

- SIDE WALL BOUNDARY CONDITIONS
  - Impermeable
  - Specified Mass/Energy Injection Rate
  - Burning Propellant Surface
VALIDATION BY COMPARISON TO EXACT 1-D SOLUTIONS

- Fanno Flow
- Rayleigh-Line Flow
- Isentropic Nozzle Flow
- Mass Injection Through Walls of Constant Area Duct
- Shock Tube Problem
Flow Through Constant Area Duct With Friction

Inlet Boundary Conditions:
Reservoir Conditions
Pr = 3,000 psia
Tr = 700°R
Vr = 0 ft/sec

Outlet Boundary Condition:
Static Pressure Specified
Pex = Pa = 1,673 psia

Initial Conditions:
Pi = 3,000 psia
Ti = 700°R
Vi = 0 ft/sec
For: 1 ≤ i ≤ I MAX

Additional Input Required
γ = 1.4
R = 53.35 ft lb f/lbm °R
f = 0.00325 (Darcy-Weisbach)
L/D = 20
FLOW THROUGH CONSTANT AREA DUCT WITH FRICTION

Numerical results from AJITC1D Code using 200 cells
Analytical results interpolated from Fanno Flow Tables
FLOW THROUGH CONSTANT AREA DUCT WITH FRICTION

AXIAL LOCATION (in.)

NORMALIZED TEMPERATURE

NORMALIZED VELOCITY

— NUMERICAL  • ANALYTICAL

Numerical results from AJITC1D Code using 200 cells
Analytical results interpolated from Fanno Flow Tables
ISENTRPIC FLOW THROUGH A NOZZLE
NOZZLE CONFIGURATION

Numerical results from AJITC1D Code using 145 cells
Analytical results from 1-D Isentropic Tables
ISENTROPIC FLOW THROUGH A NOZZLE
PRESSURE vs AXIAL POSITION

Numerical results from AJITC1D Code using 145 cells
Analytical results from 1-D Isentropic Tables
ISENTROPIC FLOW THROUGH A NOZZLE
TEMPERATURE vs AXIAL POSITION

Numerical results from AJITC1D Code using 145 cells
Analytical results from 1-D Isentropic Tables
Radial Mass Injection Into Constant Area Duct

Gas Properties:
Air:
\( \gamma = 1.4 \)
\( R = 53.35 \text{ ft lbf/lbm}^2 \text{°R} \)

Initial Conditions:
\( P_i = 600 \text{ psia} \)
\( T_i = 5000 \text{°R} \)
\( V_i = 0.0 \text{ ft/sec} \)
For: \( 1 \leq i \leq I_{\text{MAX}} \)

Left Boundary Condition:
Solid Wall

Right Boundary Condition:
Static Pressure Specified
\( P_{ex} = P_a = 600 \text{ psia} \)

Reflective Boundary Condition Used

Lateral Surface Boundary Conditions:
\( w' = 0.3855 \text{ lbm/sec in.}^2 \)
\( A_w = 37,700 \text{ in.}^2 \)
\( \dot{w} = 14,533 \text{ lbm/sec} \)
\( T_i = 6000 \text{°R} \)
RADIAL MASS INJECTION INTO CONSTANT AREA DUCT

Numerical results from AJITC1D Code using 200 cells
Analytical results from 1-D formulas in Shapiro's
Numerical results from AJITC1D Code using 200 cells
Analytical results from 1-D formulas in Shapiro's
ONE DIMENSIONAL SHOCK TUBE PROBLEM
PRESSURE vs AXIAL POSITION PRIOR TO
SHOCK REFLECTION, t = 0.5 milliseconds

DIAPHRAGM ORIGINALLY LOCATED AT X/L = 0.75
LEFT: P = 400 psia, T = 530 degR, V = 0 ft/sec
RIGHT: P = 100 psia, T = 530 degR, V = 0 ft/sec
ONE DIMENSIONAL SHOCK TUBE PROBLEM
PRESSURE vs AXIAL POSITION AS SHOCK CONTACTS WALL, t = 1.0 milliseconds

DIAPHRAGM ORIGINALLY LOCATED AT X/L = 0.75
LEFT: P = 400 psia, T = 530 degR, V = 0 ft/sec
RIGHT: P = 100 psia, T = 530 degR, V = 0 ft/sec
ONE DIMENSIONAL SHOCK TUBE PROBLEM
PRESSURE vs AXIAL POSITION NEAR
STEADY STATE, t = 835 milliseconds

DIAPHRAGM ORIGINALLY LOCATED AT X/L = 0.75
LEFT: P = 400 psia, T = 530 degR, V = 0 ft/sec
RIGHT: P = 100 psia, T = 530 degR, V = 0 ft/sec
ONE DIMENSIONAL SHOCK TUBE PROBLEM
TEMPERATURE vs AXIAL POSITION PRIOR TO SHOCK REFLECTION, \( t = 0.5 \) milliseconds

**Graph Details:**
- **X-axis:** Axial Position, (in.)
- **Y-axis:** Temperature, (degR)
- **Graph Lines:**
  - Analytical
  - Numerical
- **Diaphragm Location:** Diaphragm originally located at \( x/l = 0.75 \)
- **Left Side:**
  - Pressure: 400 psia
  - Temperature: 530 degR
  - Velocity: 0 ft/sec
- **Right Side:**
  - Pressure: 100 psia
  - Temperature: 530 degR
  - Velocity: 0 ft/sec
ONE DIMENSIONAL SHOCK TUBE PROBLEM
TEMPERATURE vs AXIAL POSITION AS SHOCK CONTACTS WALL, t = 1.0 milliseconds

DIAPHRAGM ORIGINALLY LOCATED AT X/L = 0.75
LEFT: P = 400 psia, T = 530 degR, V = 0 ft/sec
RIGHT: P = 100 psia, T = 530 degR, V = 0 ft/sec
ONE DIMENSIONAL SHOCK TUBE PROBLEM
TEMPERATURE vs AXIAL POSITION NEAR
STEADY STATE, t = 835 milliseconds

DIAPHRAGM ORIGINALLY LOCATED AT X/L = 0.75
LEFT: P = 400 psia, T = 530 degR, V = 0 ft/sec
RIGHT: P = 100 psia, T = 530 degR, V = 0 ft/sec
PRELIMINARY SENSITIVITY STUDIES
MAXIMUM HEAD END $dP/dt$ AS A FUNCTION OF DUCT LENGTH AND NOZZLE CONTRACTION RATIO

- General Results Obtained Using a Cylindrical Duct with a Closed Aft End and an Increasing Mass Flow Rate Entering at the Head End:
  - as the Duct Length Increased, the Maximum $dP/dt$ at the Head End Increased
  - Closer Examination Revealed Sinusoidal Behavior Due to Complex Wave Interactions

- General Results Obtained Using a Cylindrical Duct with a Nozzle Attached at the Aft End:
  - as the Nozzle Contraction Ratio Increased, the Maximum $dP/dt$ at the Head End Increased
  - Contraction Ratio of 2.5:1 (ASRM) Produced Results within 10% of Worst Case Solid Wall
PRELIMINARY RESULTS FROM THE AJITC1D CODE
VERIFICATION OF SHOCK STEEPENING AS A
CONSEQUENCE OF AN INCREASING MASS FLOW RATE

<table>
<thead>
<tr>
<th>TIME (msec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(1) 14.33</td>
</tr>
<tr>
<td>(2) 24.83</td>
</tr>
<tr>
<td>(3) 34.78</td>
</tr>
<tr>
<td>(4) 44.36</td>
</tr>
<tr>
<td>(5) 53.65</td>
</tr>
<tr>
<td>(6) 62.71</td>
</tr>
<tr>
<td>(7) 71.58</td>
</tr>
<tr>
<td>(8) 80.29</td>
</tr>
</tbody>
</table>

AXIAL DISTANCE, (in.)

IGNITER PRESSURE RISE RATE = 75,000 psi/sec
PRELIMINARY RESULTS FROM THE AJITC1D CODE
VERIFICATION OF SHOCK STEEPENING AS A
CONSEQUENCE OF AN INCREASING MASS FLOW RATE

<table>
<thead>
<tr>
<th>TIME (msec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 14.33</td>
</tr>
<tr>
<td>2 24.83</td>
</tr>
<tr>
<td>3 34.78</td>
</tr>
<tr>
<td>4 44.36</td>
</tr>
<tr>
<td>5 53.65</td>
</tr>
<tr>
<td>6 62.71</td>
</tr>
<tr>
<td>7 71.58</td>
</tr>
<tr>
<td>8 80.29</td>
</tr>
</tbody>
</table>

IGNITER PRESSURE RISE RATE = 75,000 psi/sec
PRELIMINARY RESULTS FROM THE AJITC1D CODE
MAXIMUM $dP/dt$ AS A FUNCTION OF THE TOTAL DUCT LENGTH AND THE IGNITER MASS FLOW RATE
PRELIMINARY RESULTS FROM THE AJITC1D CODE
MAXIMUM $dP/dt$ AS A FUNCTION OF THE TOTAL DUCT LENGTH AND THE IGNITER MASS FLOW RATE
PRELIMINARY RESULTS FROM THE AJITC1D CODE

MAXIMUM $dP/dt$ AS A FUNCTION OF THE
DOWNSTREAM BOUNDARY CONDITION SPECIFIED

RESULTS OBTAINED USING AN INCREASING
IGNITER FLOW RATE (75,000 PSI/SEC)
CURRENT STATUS OF 1-D CODE

- All features of code validated individually by comparisons to exact analytical solutions
- Burning surface boundary condition added
- For immediate parametric study to establish bounds:
  - Simple percent of prop. lit vs T and X logic added
  - Can be adjusted to match single port RSRM data (lower limit for ASRM)
  - Worst case upper limit obtained by allowing fins to ignite as initial pulse passes
FUTURE PLANS FOR 1-D CODE

- NEED FLAME SPREADING MODEL FOR "FINNED" ZONE
  - DERIVE FROM RSRM DATA (LOTS OF VARIABLES)
  - INCORPORATE RESULTS FROM AUBURN UNIVERSITY HEAD END COLD FLOW TEST SERIES (EXTRAPOLATION PROBLEM)
  - CALCULATE MASS ADDITION RATE USING CONTINUSYS CODE (LIMITED TO INVISCID CALCULATIONS WITH USER SUPPLIED IGNITION TEMPERATURE)

- ADD EROSIVE BURNING MODEL
  - PENN STATE FUNDED FOR TEST PROGRAM
PLANS FOR 3-D IGNITION TRANSIENT MODEL

- Extend 1-D ADI technique to 3-D
- Use existing grid generators and coordinate transformation routines
- Probably not feasible to run full 3-D model (even with 22 planes of symmetry)
  - Complicated geometry in head end
  - Extremely long domain (L/D > 20)
  - Must calculate through entire 1st second
- Need to develop transition from 3-D to axisymmetric geometry
  - Must preserve wave mechanics
SRMAFTE Facility Checkout Model Flow Field Analysis

Richard A. Dill, ERC Incorporated
Harold R. Whitesides, ERC Incorporated

Abstract

The Solid Rocket Motor Air Flow Equipment (SRMAFTE) facility was constructed for the purpose of evaluating the internal propellant, insulation, and nozzle configurations of solid propellant rocket motor designs. This makes the characterization of the facility internal flow field very important in assuring that no facility induced flow field features exist which would corrupt the model related measurements. In order to verify the design and operation of the facility, a three-dimensional computational flow field analysis was performed on the facility checkout model setup.

The facility checkout model entails a straight constant diameter pipe in place of a specific solid propellant rocket motor internal component. This configuration was to provide a simple internal flow field for evaluation of any facility induced effects.

One-dimensional estimates of the checkout model flow field were available for comparison to the measurement data collected for the checkout model but the CFD results provided a comparative estimate in regions where one-dimensional estimates were not valid.

Since the facility was too large and complex to perform a complete three-dimensional analysis from end to end, the facility was divided into three major zones for analysis. 1) The header pipes, metering nozzle, nozzle exit, and diffuser. 2) The adapter chamber, transition, checkout model section, and checkout model nozzle. 3) The model nozzle exit, diffuser and muffler. The three-dimensional numerical calculation of the flow field was performed by Fluent/BFC. This code solves the full Navier-Stokes equations of fluid flow cast in a staggered grid formulation. The SIMPLER numerical algorithm is used in the solution process. The code utilizes wall functions instead of physically resolving the boundary layer and the standard k-ε turbulence model is used to close the system fluid dynamic equations.

The checkout model measurement data, one-dimensional and three-dimensional estimates were compared and the design and proper operation of the facility was verified. The proper operation of the metering nozzles, adapter chamber transition, model nozzle and diffuser were verified. The one-dimensional and three-dimensional flow field estimates along with the available measurement data are compared in this presentation.
SRMAFTE FACILITY CHECKOUT MODEL FLOW FIELD ANALYSIS

Richard A. Dill and R. Harold Whitesides
ERC, Inc.

Tenth Annual CFD Working Group Meeting

Session 8
NASA/MSFC

April 29, 1992
OBJECTIVES

- VERIFY CHECKOUT MODEL SYSTEM DESIGN AND OPERATIONAL PERFORMANCE PARAMETERS INCLUDING TWO AND 3-D EFFECTS

1) ASYMMETRIC FLOW EFFECTS CREATED BY THE MANIFOLD SYSTEM
2) METERING NOZZLE PERFORMANCE AND EXPANSION SECTION FLOW FIELD
3) ASYMMETRIC FLOW FIELD IN THE ADAPTER CHAMBER
4) UNIFORMITY OF FLOW IN THE CHECKOUT MODEL CHAMBER
5) MODEL NOZZLE REGION FLOW FIELD
6) DIFFUSER FLOW FIELD INCLUDING SHOCKS

- COMPARE FACILITY CHECKOUT MODEL TEST DATA WITH CFD RESULTS AS VERIFICATION OF THE CFD CODE
CFD METHODOLOGY

- GOVERNING EQUATIONS ARE THE 3-D ENSEMBLE-AVERAGED NAVIER STOKES EQUATIONS IN CONSERVATION FORM

- CLOSURE OF THE EQUATIONS BY THE STANDARD TWO-EQUATION $\kappa$-$\varepsilon$ MODEL OF TURBULENCE

- WALL FUNCTIONS USED TO DETERMINE NEAR WALL GRADIENTS

- DISCRETIZATION METHOD
  - GOVERNING EQUATIONS ARE WRITTEN IN COMPONENT FORM USING CONTRAVARIANT VELOCITY COMPONENTS
  - THIS ALLOWS THE USE OF A BOUNDARY FITTED CURVILINEAR COORDINATE SYSTEM
  - NUMERICAL METHOD IS FINITE VOLUME BASED
  - STAGGERED GRID STORAGE SYSTEM IS USED
  - CONVECTION AND DIFFUSION FLUXES ARE APPROXIMATED USING A POWER-LAW SCHEME
  - TIME DERIVATIVES ARE CALCULATED USING A FULLY IMPLICIT FIRST ORDER SCHEME

- PRESSURE-VELOCITY COUPLING IS ACCOMPLISHED BY USING THE SIMPLER ALGORITHM

- SOLVER USES LINEARIZED BLOCK IMPLICIT SCHEME
Checkout Model Cutaway

- Metering Nozzle, 4 typ.
- Model Chamber
- Model Nozzle
- Adapter Chamber
- Diffuser Duct
- Manifold, 2 typ.
BOUNDARY CONDITIONS USED FOR THE CHECKOUT MODEL HEADER PIPES AND METERING NOZZLE

**INLET CONDITIONS**

- Stagnation Pressure (psia) : 1200
- Stagnation Temperature (°R) : 530
- Turbulence Intensity (%) : 10
- Turbulence Length Scale (m) : 0.146

**GENERAL CONDITIONS MODELED**

- Ideal Gas Law Used
- Ratio of Specific Heats : 1.4
- Molecular Weight : 28.966
- Dynamic Viscosity (lbm/ft-sec) : $1.245 \times 10^{-5}$

**EXIT CONDITION**

- Static Pressure (psia) : 600
COMPUTATIONAL GRID FOR MANIFOLD SYSTEM AND METERING NOZZLE
<table>
<thead>
<tr>
<th>KEY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum = 0.00E+00</td>
</tr>
<tr>
<td>0.00E+00</td>
</tr>
<tr>
<td>0.33E-02</td>
</tr>
<tr>
<td>1.67E-01</td>
</tr>
<tr>
<td>2.50E-01</td>
</tr>
<tr>
<td>3.33E-01</td>
</tr>
<tr>
<td>4.17E-01</td>
</tr>
<tr>
<td>5.00E-01</td>
</tr>
<tr>
<td>5.83E-01</td>
</tr>
<tr>
<td>6.67E-01</td>
</tr>
<tr>
<td>7.50E-01</td>
</tr>
<tr>
<td>8.33E-01</td>
</tr>
<tr>
<td>9.17E-01</td>
</tr>
<tr>
<td>1.00E+00</td>
</tr>
<tr>
<td>1.00E+00</td>
</tr>
<tr>
<td>1.17E+00</td>
</tr>
<tr>
<td>1.25E+00</td>
</tr>
<tr>
<td>Maximum = 1.66E+00</td>
</tr>
</tbody>
</table>

**HEADER METERING NOZZLE (25X25X151) GRID**

**FLUENT/BFC V3.02**

**Raster Plot of MACH-NUMBER**

**Slices: J=13**

**3D Domain**

**Steady State**
Metering Nozzle Radial Pressure Ratio Profile

- Pressure Ratio (P/Po)
- Normalized Radial Distance, r/wall

Key parameters:
- Entrance Angle = 30 Deg
- Radius of Curvature = 4.17
- Contraction Ratio = 9.003
BOUNDARY CONDITIONS USED FOR THE CHECKOUT MODEL ADAPTER, SPOOL PIECES AND MODEL NOZZLE

• **INLET CONDITIONS**

  STAGNATION PRESSURE (psia) : 610  
  STAGNATION TEMPERATURE (°R) : 530  
  TURBULENCE INTENSITY (%) : 10  
  TURBULENCE LENGTH SCALE (m) : 0.146

• **GENERAL CONDITIONS MODELED**

  IDEAL GAS LAW USED  
  RATIO OF SPECIFIC HEATS : 1.4  
  MOLECULAR WEIGHT : 28.966  
  DYNAMIC VISCOSITY (lbm/ft-sec) : 1.245x10^{-5}

• **EXIT CONDITION**

  SUPersonic OUTLET BOUNDARY CONDITION
Axial Direction Grid
3D CHECKOUT MODEL FLOW FIELD (21X3X120) GRID FLUENT/BFC Y3.92

Roster Plot of PRESSURE

Steady State

KEY

Minimum Value = 3.75E-6
Maximum Value = 1.32E-5
<table>
<thead>
<tr>
<th>3-D CHECKOUT MODEL FLOW FIELD (21x30x170) GRID</th>
<th>FLUENT/BFC V3.02</th>
</tr>
</thead>
<tbody>
<tr>
<td>Profiles of W-VELOCITY</td>
<td>3D Domain</td>
</tr>
<tr>
<td>Slices: J=15</td>
<td>Steady State</td>
</tr>
</tbody>
</table>
Pressure Ratio and Mach Number Profiles at Model Nozzle Throat Plane

![Graph showing Pressure Ratio and Mach Number Profiles at Model Nozzle Throat Plane.](image)
BOUNDARY CONDITIONS USED FOR THE SRMAFTE DIFFUSER

• **INLET CONDITIONS**

  STAGNATION PRESSURE (psia) : 150
  STAGNATION TEMPERATURE (°R) : 530
  TURBULENCE INTENSITY (%) : 5
  TURBULENCE LENGTH SCALE (m) : 0.108

• **GENERAL CONDITIONS MODELED**

  IDEAL GAS LAW USED : 1.4
  RATIO OF SPECIFIC HEATS : 28.966
  MOLECULAR WEIGHT : 1.245x10^{-5}

• **EXIT CONDITION**

  STATIC PRESSURE (psia) : 15
COMPUTATIONAL GRID FOR THE DIFFUSER
KEY

4.4E+00
4.2E+00
4.1E+00
3.9E+00
3.8E+00
3.7E+00
3.6E+00
3.5E+00
3.4E+00
3.3E+00
3.2E+00
3.1E+00
3.0E+00
2.9E+00
2.8E+00
2.7E+00
2.6E+00
2.5E+00
2.4E+00
2.3E+00
2.2E+00
2.1E+00
2.0E+00
1.9E+00
1.8E+00
1.7E+00
1.6E+00
1.5E+00
1.4E+00
1.3E+00
1.2E+00
1.1E+00
9.7E-01
8.3E-01
7.0E-01
5.8E-01
4.7E-01
3.7E-01
2.7E-01
1.8E-02

RASTER PLOT OF MACH NO. (DIMENSIONLESS)

MAX. = 4.48647E+00
MIN. = 5.25557E-04

ORIENT = Z
PLANE = 1
2-D DOMAIN
Diffuser Performance Map

Model Aft Total Pressure/Ambient Pressure

- Mode A
- Mode B
CONCLUSIONS

THE CFD ANALYSES OF THE CHECKOUT MODEL, MANIFOLD SYSTEM AND DIFFUSER CONFIRMED THE DESIGN AND VERIFIED THAT:

1) THE FLOW CHOKES IN BOTH THE METERING AND MODEL NOZZLE AND THE MANIFOLD SYSTEM SHOULD PERFORM AS EXPECTED.

2) THE ADAPTER AND TRANSITION SECTION PERFORM WELL IN DELIVERING A UNIFORM FLOW PROFILE TO THE MODEL NOZZLE.

3) THE PREDICTED SHOCK STRUCTURE IN THE DIFFUSER INDICATES THE DESIGN SHOULD PERFORM AS EXPECTED.
A Comparative Study of the Effects of Inhibitor Stub Length on Solid Rocket Motor Combustion Chamber Pressure Oscillations: RSRM at T=80 Seconds. Preliminary Results

D. Chasman, D. Burnette, J. Holt
Rockwell International, Space Systems Division
Huntsville, Alabama 35806
R. Farr
NASA Science and Engineering Laboratory
George C. Marshall Space Flight Center

ABSTRACT

Results from a continuing, time-accurate computational study of the combustion gas flow inside the Space Shuttle Redesigned Solid Rocket Motor (RSRM) are presented. These CFD analyses duplicate unsteady flow effects which interact in the RSRM to produce pressure oscillations, and resulting thrust oscillations, at nominally 15, 30 and 45 Hz. Results of Navier-Stokes computations made at mean pressure and flow conditions corresponding to 80 seconds after motor ignition both with and without a protruding, rigid inhibitor at the forward joint cavity are presented here.

Previous studies by the authors have demonstrated that combustion chamber pressure oscillations in the RSRM are generated by flow/acoustic interactions which occur at the three field joint cavities [1,2,3]. Edge-tone, Hole-tone and Organ pipe-tone are all acoustic sources that play part in these interactions[4,5,6]. By constructive interference processes, the different acoustic sources interact, amplifying the pressure oscillation level at some instant during the RSRM burn (i.e. T+80 sec). This behavior is representative of an aerodynamic whistle of Class III [7,8].

However, the question remained as to whether the cavities alone are the main acoustic generators or whether protruding inhibitors dominate the system. With this in mind, two simulations have been conducted. The first simulation represent a full scale model of the RSRM with a rigid inhibitor perturbing the flow at the forward joint, while the second simulation was conducted without an inhibitor. All other flow conditions and grain geometries were kept constant for both simulations.

Fig. 1 shows a comparison of results between the two simulations. A full-length density contour plot of the simulation with forward inhibitor is shown in Fig. 1a., while that without the inhibitor is shown in Fig. 1d.. In both plots the shear layer which developed from the burning surfaces is apparent. High density values, corresponding to areas of high pressure amplitude, appear in red.

Details of the forward field joint area are shown for both simulations in Figs. 1b. and 1e.. Both show intense, but different, shear flow activity indicative of vortex dynamics. Streamlines shown in Fig. 1c. and Fig. 1f. single out individual vortices and further illustrate the difference between the two simulations.

Fig. 2 illustrates a comparison of pressure data time histories for three points common to both simulations. It can be seen that 30 Hz oscillations are evident in the midsection of the chamber, while 15 Hz is found at both the head and aft ends, with the head end being 180 degrees out of phase with the aft end. These results indicate classic organ pipe acoustics are found in both simulations. However, there is a marked difference in the peak-to-peak amplitude of these pressure oscillations. These preliminary results show pressure amplitude in the case without the inhibitor are about 14% of total chamber pressure, while those in the case with the inhibitor are only 10% of the total pressure.

While calculated peak-to-peak pressure amplitude values are significantly higher than levels measured during actual firing and flight, we feel these results can nevertheless be used for
comparative analyses of the effects of inhibitor stub height on RSRM combustion chamber acoustic pressure amplitudes. Specifically, these findings demonstrate that the inhibitors alone are not the dominant factor in amplifying combustion chamber pressure oscillations, but rather are included in secondary acoustic/flow interactions occurring at the three field joint cavities. Our results indicate that inhibitors, when present, actually act to damp such oscillations.

References


Figure 1. Simulations With and Without Forward Inhibitor

RSRM PC OSCILLATION

a) Simulation With Forward Inhibitor (Density)

b) Density Detail
With Forward Inhibitor

c) Streamline Detail
With Forward Inhibitor

d) Simulation Without Forward Inhibitor (Density)

e) Density Detail
Without Forward Inhibitor

f) Streamline Detail
Without Forward Inhibitor

Rockwell International
Space Systems Division
Huntsville Operations
RENG001746.0
Figure 2. Pressure data history of selected points in the RSRM chamber at T + 80 sec:
I. Point 1 is located in the head end
II. Point 6 is located in the middle
III. Point 9 is located in the aft end
Overview of the relevant CFD work at Thiokol Corporation

Pawel Chwalowski & Hai-Tien Loh
Thiokol Corporation, M/S L63, P.O. Box 707,
Brigham City, Utah 84302-0707

The use of computational fluid dynamics (CFD) in supporting the rocket propulsion designs at Thiokol Corporation has continuously increased in the past few years. An in-house developed proprietary advanced CFD code called SHARP* is a primary tool for many flow simulations and design analyses. The SHARP code is a time dependent, two-dimensional (2-D) axisymmetric numerical solution technique for the compressible Navier-Stokes equations. The solution technique in SHARP uses a vectorizable implicit, second order accurate in time and space, finite-volume scheme based on the following: 1) an upwind flux-difference splitting of a Roe-type approximated Riemann solver, 2) Van Leer's flux vector splitting, and 3) a fourth order artificial dissipation scheme with a preconditioning to accelerate the flow solution. Turbulence is simulated by an algebraic model, and ultimately the k-ε model. Some other capabilities of the code are 2-D two-phase Lagrangian particle tracking and cell blockages. Extensive development and testing has been conducted on the three-dimensional (3-D) version of the code with flow, combustion, and turbulence interactions.

The SHARP code has been applied in many areas of the solid rocket motor (SRM) design involving internal and external flow analysis. However, the internal flow analysis inside the motor and in the nozzle region are the most frequent. Usually, the results from these CFD analyses become the boundary conditions for thermal and structural computations. In the case of the internal nozzle flow calculations, SHARP computes the convective heat transfer coefficients and temperature distribution along the nozzle wall for the thermal erosion predictions, and the pressure distribution for the structural predictions. The pressure loads on the propellant grain surfaces inside the SRM obtained from SHARP are used as a boundary conditions to predict propellant grain deformation and displacement. The 2-D two-phase Lagrangian particle tracking gives the ability to predict solid particle impingement on the exit cone. Also, SHARP prediction of the slag accumulation in the aft dome region of the SRM agrees with the actual static test data.

The emphasis in the presentation will be put on the specific applications of SHARP in SRM design.

* SHock wave And Recirculation Program is a copyrighted acronym owned by Thiokol Corporation.
OVERVIEW OF THE RELEVANT CFD WORK AT THIOKOL CORPORATION

SPACE OPERATIONS

Pawel Chwalowski

CFD Workshop

April 28, 1992

INFORMATION ON THIS PAGE WAS PREPARED TO SUPPORT AN ORAL PRESENTATION AND CANNOT BE CONSIDERED COMPLETE WITHOUT THE ORAL DISCUSSION
AGENDA

CFD Tools Used at Thiokol Corp.
Description of the Primary CFD Code
Applications in the Solid Rocket Motor (SRM) Design
CFD TOOLS

PHOENICS (versions 1.4 and 1.5)

-well developed and tested for many incompressible and/or subsonic flow cases (includes heat conduction, multiphase flow, reactive flow, etc.)

-works well for incompressible and/or subsonic flow

-finite volume SIMPLE scheme (Semi-Implicit Method for Pressure Linked Equation)

-exhibits problems for high speed compressible and turbulent modeling

-lacks flexibility in BFC gridding

SHARP

-developed in-house by Thiokol; considered proprietary

-2D version extensively tested and operational

-3D version in development and testing
SHARP 2D DESCRIPTION

Flow Modeling

2D Planar/Axisymmetric Compressible Code

2D Lagrangian Particle Tracking

Solution Algorithm

Upwind Roe Flux Difference Splitting

Upwind Van Leer's Flux Vector Splitting

Central Difference with Artificial Dissipation
SHARP 2D DESCRIPTION (cont.)

Features

Finite Volume / Second Order Accuracy in Time

Use of Preconditioning to Accelerate Flow Solution

Cell Blockage

Baldwin-Lomax Algebraic Turbulence Model (Ultimately k-ε Model)

Unstructured Grid in Testing

SHARP 3D CODE

SHARP 3D in Development and Testing with Flow, Combustion, and Turbulence Interaction
APPLICATIONS IN SRM DESIGN

Internal Flow Modeling

Nozzle Flow

-heat transfer coefficients and temperature distribution along nozzle wall are calculated for the thermal erosion analysis

-pressure along nozzle wall is calculated for the structural analysis

-aluminum oxide particle motion is predicted and better understood
SHARP Results
Internal Flow Modeling
Nozzle Flow
Pressure Contour (N/m²)

Thiokol CORPORATION
SPACE OPERATIONS

Information on this page was prepared to support an oral presentation and cannot be considered complete without the oral discussion.
SHARP Results
Internal Flow Modeling
Nozzle Flow
Mach Contour
Temperature Contour (K)

Mach Number

<table>
<thead>
<tr>
<th>Mach Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.384E+00</td>
</tr>
<tr>
<td>2.708E+00</td>
</tr>
<tr>
<td>2.031E+00</td>
</tr>
<tr>
<td>1.354E+00</td>
</tr>
<tr>
<td>6.770E-01</td>
</tr>
<tr>
<td>7.711E-05</td>
</tr>
</tbody>
</table>

Temperature

<table>
<thead>
<tr>
<th>Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.369E+03</td>
</tr>
<tr>
<td>3.028E+03</td>
</tr>
<tr>
<td>2.687E+03</td>
</tr>
<tr>
<td>2.346E+03</td>
</tr>
<tr>
<td>2.005E+03</td>
</tr>
<tr>
<td>1.664E+03</td>
</tr>
</tbody>
</table>

Thiokol CORPORATION
SPACE OPERATIONS

INFORMATION ON THIS PAGE WAS PREPARED TO SUPPORT AN ORAL PRESENTATION AND CANNOT BE CONSIDERED COMPLETE WITHOUT THE ORAL DISCUSSION
SHARP Results
Internal Flow Modeling
Nozzle Flow

Shuttle SRM Aluminum Oxide Particle Motion At Different Burn Times

a) 35 seconds

b) 52 seconds

c) 79 seconds

d) 111 seconds

e) 120 seconds

Thiokol CORPORATION
SPACE OPERATIONS

INFORMATION ON THIS PAGE WAS PREPARED TO SUPPORT AN ORAL PRESENTATION AND CANNOT BE CONSIDERED COMPLETE WITHOUT THE ORAL DISCUSSION.
APPLICATIONS IN SRM DESIGN (cont.)

Internal Flow Modeling (cont.)

Chamber Internal Flow

-pressure loads on the propellant grain surfaces
   are calculated to predict propellant grain deformation
   and displacement
Internal Flow Modeling
Shuttle SRM Internal Flow
Pressure Contour (psi)

Propellant grain | Propellant grain | Propellant grain

Center segment | Center segment | Aft segment

Thiokol Corporation
Space Operations

Information on this page was prepared to support an oral presentation and cannot be considered complete without the oral discussion.
Internal Flow Modeling
Shuttle SRM Internal Flow
Velocity Vector Contour (ft/sec)

Thiokol CORPORATION
SPACE OPERATIONS

INFORMATION ON THIS PAGE WAS PREPARED TO SUPPORT AN ORAL PRESENTATION
AND CANNOT BE CONSIDERED COMPLETE WITHOUT THE ORAL DISCUSSION
Internal Flow Modeling
Shuttle SRM Internal Flow
Pressure Differential In The
Slot Area Between Center And Aft
Segments (psi)
Internal Flow Modeling
Shuttle SRM Internal Flow

Velocity Vectors In The Slot Area Between Center And Aft Segments (ft/sec)
SUMMARY

- efficient algorithm was developed / runs well on computer workstations
- SHARP 2D was extensively tested
- SHARP demonstrates capabilities in SRM design and flow analysis
- turbulent combustion development and testing is in progress
Submitted for the CFD Workshop - 1992

A Status of the Activities of the NASA/Marshall Space Flight Center Combustion Devices Technology Team

Kevin Tucker

The Consortium for Computational Fluid Dynamics (CFD) Applications in Propulsion Technology was established to focus CFD applications in propulsion. Specific areas of effort include developing the CFD technology required to address rocket propulsion issues, validating the technology, and applying the validated technology to design problems; all under peer review by experts in the field.

The Combustion Devices Technology Team was formed to implement the above objectives in the broad area of combustion-driven flows. In an effort to bring CFD to bear in the design environment, the team has focused its efforts on the Space Transportation Main Engine nozzle. The main emphasis has been on the film cooling scheme used to cool the nozzle wall. Benchmark problems have been chosen to validate CFD film cooling capabilities. CFD simulations of the subscale nozzle (to be tested 8/92) have been made. Also, CFD predictions of the base flow resulting from this type of nozzle have been made. A status of these calculations will be presented along with future plans.
Combustion-Driven Flow Analysis Technology

Overview

- Combustion Driven Flow Team Participants

- Background

- Programs
  - STME Nozzle Film Cooling
  - NLS Base Heating

- Summary
Combustion-Driven Flow Analysis Technology

Participants

- NASA/ Marshall Space Flight Center (MSFC)
- NASA/ Ames Research Center (ARC)
- NASA/ Lewis Research Center (LeRC)
- Air Force (Phillips Lab)
- Aerojet
- Pratt & Whitney (P&W)
- Rocketdyne (Rkdn)
- SECA
- Computational Fluid Dynamics (CFD) Research Corporation
- United Technology Research Center (UTRC)
- Calspan - University of Buffalo Research Center (CUBRC)
- Calspan - AEDC Operations
- W. J. Shafer Associates
- Remtech
- University of Tennessee Space Institute (UTSI)
- Pennsylvania State University
- The University of Alabama (UA)
- The University of Alabama in Huntsville (UAH)
Combustion-Driven Flow Analysis Technology

Background

**STME Nozzle**

- TQM activity identified TCA issues which could benefit from enhanced analytical capabilities
  - Evaluated issues based on:
    -- Need for improved design methodology (engine contractors)
    -- Potential for CFD impact in near to mid-term (CFD specialists)
- Identified nine generic technology issues in injector, chamber, and nozzle
- Identified major technology/development in STME TCA
  -- Nozzle film cooling
  -- Integral fuel mixer
  -- High aspect ratio coolant channels
- Compared general technology issues with STME TCA concerns
- Chose STME film/dump cooled nozzle for team focus
- Identified validation/verification requirements for supersonic film cooling
Combustion-Driven Flow Analysis Technology

Background

**NLS Base Heating**

- Base heating is a concern on every launch vehicle

- The hydrogen-rich film/dump coolant causes additional complexities/concerns
  - Dumping low energy hydrogen into the base region put the program out of historical database
  - Resulting high heating rate environments are based on conservative assumptions
  - Subscale tests with base and/or afterburning don't scale well

- CFD is being used to augment classical analysis and testing

- Extensive code validation plan has been developed
Combustion-Driven Flow Analysis Technology

Program - STME Nozzle Film Cooling

• Issues
  - Capability of the film to adequately cool the nozzle skirt
    -- Effect of accelerating core flow on film integrity
    -- Conditions at which film should be injected
  - Effect of film coolant/Injector delivery system on performance
    -- Shock losses
  - Environment definition in nozzle/MCC joint area

• Task Description
  - Validate CFD codes for supersonic film cooling
  - Use validated codes as design tool for subscale (40K) nozzle
  - Verify film cooling performance in subscale nozzle
  - Use validated codes in design of full scale nozzle
Combustion-Driven Flow Analysis Technology

Program - STME Nozzle Film Cooling

• Results to Date
  - Film cooling benchmark calculations complete
  - Analysis of film coolant network for subscale nozzle complete
  - Preliminary primary injector analysis complete
  - Analysis of cases from subscale test matrix underway; to be completed 8/92

• Impact
  - Demonstration of CFD as a nozzle design tool successful
  - Subscale secondary injector redesigned using CFD
  - Confidence gained in film cooling early in subscale design
  - Large film cooling analytical/test database being developed
Combustion-Driven Flow Analysis Technology

Film Cooling Benchmark

Case 45:
Matched Pressure Condition
Shock Generator Angle = 0°
STME Subscale Nozzle

H₂ film coolant manifold

Film coolant properties:
P = 290 PSIA
T = 70° F
W = 1.77 lbm/s

Water coolant exit tubes

Film Injector ring

9.00 dia.

21.900 dia.

21.437

Water coolant properties:
P = 300 PSIA
T = 70° F
W = 25 lbm/s

3 water coolant inlet manifolds
SST
CORE/SECONDARY COOLANT FLOW INTERACTION
Interaction Pressure Contours Indicate a Step—Induced Shock
CORE/SECONDARY COOLANT FLOW INTERACTION

Velocity Vectors / H₂ Mass Fraction Contours Show Vortex Mixing
CORE/SECONDARY COOLANT FLOW INTERACTION

Interaction Temperature Contours Indicate Entrainment of Core Gases
CORE/SECONDARY COOLANT FLOW INTERACTION

$H_2$ Mass Fraction Contours Indicate Vortex and Diffusion Mixing
TEMPERATURE DISTRIBUTION
DEGREES RANKINE

CONTOUR LEVELS
350.000
510.000
670.000
830.000
990.000
1150.000
1310.000
1470.000
1630.000
1790.000
1950.000
2110.000
2270.000
2430.000
2590.000
2750.000

0.000
0.02
0.04
0.06
0.08
0.1
0.12
0.14
0.16
0.18
0.2
0.22
0.24
0.26
0.28
0.3
0.32
0.34
0.36
0.38
0.4
0.42
0.44
0.46
0.48
0.5
0.52
0.54
0.56
0.58
0.6
0.62
0.64
0.66
0.68
0.7
0.72
0.74
0.76
0.78
0.8
0.82
0.84
0.86
0.88
0.9
0.92
0.94
0.96
0.98
1
1.02
1.04
1.06
1.08
1.1
1.12
1.14
1.16
1.18
1.2
1.22
1.24
1.26
1.28
1.3
1.32
1.34
1.36
1.38
1.4
1.42
1.44
1.46
1.48
1.5
1.52
1.54
1.56
1.58
1.6
1.62
1.64
1.66
1.68
1.7
1.72
1.74
1.76
1.78
1.8
1.82
1.84
1.86
1.88
1.9
1.92
1.94
1.96
1.98
2
2.02
2.04
2.06
2.08
2.1
2.12
2.14
2.16
2.18
2.2
2.22
2.24
2.26
2.28
2.3
2.32
2.34
2.36
2.38
2.4
2.42
2.44
2.46
2.48
2.5
2.52
2.54
2.56
2.58
2.6
2.62
2.64
2.66
2.68
2.7
2.72
2.74
2.76
2.78
2.8
2.82
2.84
2.86
2.88
2.9
2.92
2.94
2.96
2.98
3
3.02
3.04
3.06
3.08
3.1
3.12
3.14
3.16
3.18
3.2
3.22
3.24
3.26
3.28
3.3
3.32
3.34
3.36
3.38
3.4
3.42
3.44
3.46
3.48
3.5
3.52
3.54
3.56
3.58
3.6
3.62
3.64
3.66
3.68
3.7
3.72
3.74
3.76
3.78
3.8
3.82
3.84
3.86
3.88
3.9
3.92
3.94
3.96
3.98
4
• Issues
  - CFD has not been used extensively for base environments
  - Codes must be assessed for vehicle base flows
  - Decisions must be made on where, how, and when CFD codes will be used to supplement current predictive techniques

• Task Description
  - Establish validation plan based on physical processes involved in base heating
  - Perform single nozzle analyses for the following:
    -- Engineering models
    -- Support for subscale model design and subsequent scaling
  - Perform 3D/axisymmetric clustered nozzle calculations
  - Perform all nozzle analyses at altitudes spanning vehicle trajectory
Results to Date

- STME single nozzle "demonstration" calculations at low and high altitudes complete (4 codes)
- Benchmark calculations well underway in-house and at Rocketdyne
  -- Backward facing step
  -- MOC/Inviscid plume
  -- S-1C single nozzle
  -- 4-nozzle cluster
- Preliminary NLS/STME single nozzle calculations to be completed by 5/92
- Preliminary 3D NLS/STME base calculation to be completed by 5/92

Impact

- Results of "demonstration" calculations delivered to ED64 for heat flux calculations
- The 3D analyses are pushing computer limits; job turnaround is slow
NLS 1.5 Stage Base

NOTE:
BASE HEAT SHIELD NOT SHOWN

FAIRING

VIEW B-B

SECTION A-A

STME

B

VIEW LOOKING FWD
Combustion-Driven Flow Analysis Technology

Backward-Facing Step Benchmark

Flow Benchmark (Backstep flow $M = 2.996, H = 1.02$)
Clustered Nozzle Benchmark

Radial base pressure distribution

Static pressure on heat shield, psia

Distance from center of heat shield, inches
Summary

- Technology program geared toward STME TCA issues
  - Supported by government, industry, and universities

- CFD codes being validated for and applied to NLS/STME

- CFD contributing to STME design improvements

- Usefulness of analytical tools being enhanced via technology, experience, and peer review of results
CFD Analysis of the STME Nozzle Flowfield

Anantha Krishnan
CFD Research Corporation, Huntsville, AL
and
Kevin Tucker
NASA MSFC, Huntsville, AL

The Space Transporter Main Engine (STME) uses a gas generator cycle to cool the nozzle wall by a film-dump of the turbine exhaust. The ability to cool the skirt is a key concern in the design of the STME. CFD calculations were undertaken to predict the film cooling effectiveness and performance sensitivities for various design configurations, operating points and inlet conditions. The results in this study were obtained for the subscale nozzle. The computations were performed using REFLEQS.

The computational analysis showed that a chemical equilibrium model was necessary to obtain correct predictions of the specific impulse. The frozen composition model underpredicts the ISP by about 6%. It was also observed that the coolant film was successful in maintaining the nozzle wall well below the stagnation temperature of the core flow. The effect of the coolant flow on the performance of the engine was found to be negligible. The computed heat fluxes at the wall were in good agreement with the empirical data obtained by Pratt & Whitney.

Further test data from Pratt & Whitney are forthcoming for the subscale nozzle. Calculations will be performed to determine cooling efficiencies and nozzle performance over a range of conditions and the model predictions will be compared with experimental data.
CFD ANALYSIS OF THE STME FLOWFIELD

by
Anantha Krishnan
CFD Research Corporation

and
Kevin Tucker
NASA - MSFC

Tenth Annual CFD Workshop
NASA - MSFC
April 28-30, 1992
INTRODUCTION

- The STME Uses a Gas Generator Cycle to Cool the Nozzle Wall by a Film Dump of the Turbine Exhaust.

- The Film Dump is Split into a Subsonic Stream and Supersonic Stream Injected at Different Locations on the Nozzle Wall.

- CFD Calculations were Undertaken to Predict Film Cooling Effectiveness and Performance Sensitivities for Various Design Configurations and Operating Points.

- The Calculations were Done for the Subscale Nozzle.
Supply Coolant Splits between Injector Ring and Primary Coolant

- H₂ Supply Holes
- 36 Subsonic Orifices
- Annular Groove
- 135 Primary Injector Slots
- Subsonic Cavity
- 40K Chamber
CFD METHODOLOGY

- REFLEQS Code has been Adapted for Rocket Thrust Chambers

- Solving for Reynolds Average Navier-Stokes Equations with
  - k-ε Turbulence Model
  - Chemical Equilibrium
  - BFC Grid (210x58)
  - Colocated Variables and Solving for Cartesian Velocity Components (strong conservation form)
  - Implicit Solver for Skew Grids
  - Finite Volume Discretization

- Validated for Large Number of Benchmark Problems (REFLEQS Validation Manual)
STME NOZZLE COMPUTATIONAL GRID

- Structured 210 x 58 Grid
Table 1. Inlet Conditions

<table>
<thead>
<tr>
<th>Mass Flow Rate (lb/sec)</th>
<th>Pressure (Psia)</th>
<th>Temperature (°R)</th>
<th>Velocity (ft/sec)</th>
<th>Mach Number</th>
<th>Area of c/s (in²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>82.91</td>
<td>2189.9</td>
<td>6683.4</td>
<td>1096.08</td>
<td>0.219</td>
<td>23.469</td>
</tr>
</tbody>
</table>

Table 2. Subsonic Injector

<table>
<thead>
<tr>
<th>Mass Flow Rate (lb/sec)</th>
<th>Pressure (Psia)</th>
<th>Temperature (°R)</th>
<th>Velocity (ft/sec)</th>
<th>Mach Number</th>
<th>Area of c/s (in²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.254</td>
<td>59.13</td>
<td>523.9</td>
<td>876.9</td>
<td>0.205</td>
<td>1.98</td>
</tr>
</tbody>
</table>

Table 3. Sonic and Supersonic Injectors

<table>
<thead>
<tr>
<th>Mass Flow Rate (lb/sec)</th>
<th>Pressure (Psia)</th>
<th>Temperature (°R)</th>
<th>Velocity (ft/sec)</th>
<th>Mach Number</th>
<th>Area of c/s (in²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.51</td>
<td>60.9</td>
<td>568.53</td>
<td>4456.0</td>
<td>1.0</td>
<td>2.448</td>
</tr>
<tr>
<td>1.51</td>
<td>48.0</td>
<td>377.14</td>
<td>5277.0</td>
<td>1.454</td>
<td>1.740</td>
</tr>
<tr>
<td>1.22</td>
<td>57.6</td>
<td>568.53</td>
<td>4456.0</td>
<td>1.0</td>
<td>2.092</td>
</tr>
<tr>
<td>1.22</td>
<td>46.1</td>
<td>377.14</td>
<td>5277.0</td>
<td>1.454</td>
<td>1.464</td>
</tr>
<tr>
<td>1.82</td>
<td>73.47</td>
<td>568.53</td>
<td>4456.0</td>
<td>1.0</td>
<td>2.448</td>
</tr>
<tr>
<td>1.82</td>
<td>53.98</td>
<td>372.74</td>
<td>5423.0</td>
<td>1.503</td>
<td>1.794</td>
</tr>
<tr>
<td>Case</td>
<td>Chemistry</td>
<td>Coolant Flow</td>
<td>Coolant Exit Condition</td>
<td></td>
<td></td>
</tr>
<tr>
<td>------</td>
<td>-----------</td>
<td>--------------</td>
<td>------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td>1.</td>
<td>Frozen Composition</td>
<td>Nominal</td>
<td>Sonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>2.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>Sonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>Supersonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.</td>
<td>Equilibrium Model</td>
<td>Minimum</td>
<td>Supersonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.</td>
<td>Equilibrium Model</td>
<td>Maximum</td>
<td>Sonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.</td>
<td>Equilibrium Model</td>
<td>Maximum</td>
<td>Sonic</td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>(no primary injector flow)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>8.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>(no subsonic flow)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>9.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>(T_W=1060^oR)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>(T_W=1060^oR)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11.</td>
<td>Equilibrium Model</td>
<td>Nominal</td>
<td>(T_W=1060^oR)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Temperature Distribution (°K) - Frozen Composition Model

XY PLANE  1
TEMP CONTOURS
FMIN  2.878E+02
FMAX  3.711E+03
CONTOUR LEVELS
1  2.878E+02
2  4.694E+02
3  6.511E+02
4  8.328E+02
5  1.015E+03
6  1.196E+03
7  1.378E+03
8  1.560E+03
9  1.741E+03
10 1.923E+03
11 2.105E+03
12 2.286E+03
13 2.468E+03
14 2.650E+03
15 2.831E+03
16 3.013E+03
17 3.195E+03
18 3.376E+03
19 3.558E+03
20 3.740E+03
OK>
SUBSCALE NOZZLE CALCULATIONS

Temperature Distribution (°K) - Chemical Equilibrium Model

XY PLANE 1
TEMP CONTOURS
FMIN 2.918E+02
FMAX 3.712E+03
CONTOUR LEVELS
1 2.918E+02
2 4.726E+02
3 6.534E+02
4 8.342E+02
5 1.015E+03
6 1.196E+03
7 1.377E+03
8 1.557E+03
9 1.738E+03
10 1.919E+03
11 2.100E+03
12 2.280E+03
13 2.461E+03
14 2.642E+03
15 2.823E+03
16 3.004E+03
17 3.184E+03
18 3.365E+03
19 3.546E+03
20 3.727E+03
**SUBSCALE NOZZLE CALCULATIONS**

Pressure Distribution (Pa) - Chemical Equilibrium Model

XY PLANE 1
PRES CONTOURS
FMIN 1.860E+04
FMAX 1.462E+07
CONTOUR LEVELS
1 2.711E+04
3 4.533E+04
5 6.356E+04
7 8.178E+04
9 1.000E+05
11 2.000E+05
13 4.000E+05
15 6.000E+05
17 8.000E+05
19 1.000E+06
21 4.425E+06
23 1.128E+07

OK >
Pressure Distribution (Pa) - Chemical Equilibrium Model

XY PLANE 1
PRES CONTOURS
FMIN 2.583E+05
FMAX 6.213E+05
CONTOUR LEVELS
1 2.922E+05
2 3.344E+05
3 3.767E+05
4 4.189E+05
5 4.611E+05
6 5.033E+05
7 5.456E+05
8 5.878E+05
9 6.300E+05
OK>
Mach Number - Chemical Equilibrium Model

XY PLANE 1
MACH CONTOURS
FMIN 1.127E-01
FMAX 4.181E+00
CONTOUR LEVELS
1 2.201E-01
2 4.401E-01
3 6.602E-01
4 8.803E-01
5 1.100E+00
6 1.320E+00
7 1.540E+00
8 1.761E+00
9 1.981E+00
10 2.201E+00
11 2.421E+00
12 2.641E+00
13 2.861E+00
14 3.081E+00
15 3.301E+00
16 3.521E+00
17 3.741E+00
18 3.961E+00
19 4.181E+00
OK>
## COMPUTATIONAL RESULTS FOR THE SUB-SCALE NOZZLE FILM COOLING

<table>
<thead>
<tr>
<th>Model</th>
<th>Primary Injector Type</th>
<th>Mass Flow Rate (lbm/sec)</th>
<th>ISP</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frozen Composition</td>
<td>Nominal, Sonic</td>
<td>84.67</td>
<td>407.7</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Nominal, Sonic</td>
<td>84.67</td>
<td>434.2</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Nominal, Supersonic</td>
<td>84.67</td>
<td>433.8</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Minimum, Sonic</td>
<td>84.38</td>
<td>434.6</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Minimum, Supersonic</td>
<td>84.38</td>
<td>434.3</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Maximum, Sonic</td>
<td>84.98</td>
<td>433.7</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Maximum, Supersonic</td>
<td>84.98</td>
<td>433.3</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Nominal, Sonic</td>
<td>84.42</td>
<td>434.3</td>
</tr>
<tr>
<td>Equilibrium Model</td>
<td>Subsonic Flow</td>
<td>83.16</td>
<td>435.5</td>
</tr>
<tr>
<td>Equilibrium Model Wall at 1060°R</td>
<td>Nominal, Sonic</td>
<td>84.67</td>
<td>432.3</td>
</tr>
<tr>
<td>Equilibrium Model Wall at 1060°R</td>
<td>Nominal, Supersonic</td>
<td>84.67</td>
<td>431.9</td>
</tr>
</tbody>
</table>
HEAT FLUX VS. X FOR $T_{\text{WALL}} = 1060^\circ R$

Nominal, Sonic

Nominal, Supersonic

○○○○ - P&W Analysis for Primary Injector
□□□□ - P&W Analysis for Secondary Injector
CONCLUSIONS

Computations were Performed for the Subscale STME Film Cooling. The Main Conclusions of this Analysis were:

1. Chemistry Needs to be Considered to Predict the ISP Correctly

2. The Coolant Film is Successful in Maintaining the Nozzle Wall Well Below the Stagnation Temperature of the Core Flow

3. Computed Heat Fluxes at the Wall were in Good Agreement with (P&W) Empirical Data

4. The Nozzle Performance is Relatively Insensitive to the Coolant Injection
NLS Nozzle Base Flow Characteristics

J. J. Erhart
pratt & Whitney
West Palm Beach, FL

ABSTRACT

The flow characteristics of the NLS nozzle base area need to be determined in order for heat transfer rates to be estimated. The objective of this work is to calculate these flow characteristics using CFD. A Full Navier-Stokes code in an axisymmetric mode using a k-ε turbulent model with wall functions is applied. Calculations were completed at an altitude of 3,250 and 80,000 feet in the flight trajectory. The results show flow features which can affect vehicle design. Calibration of a 3-D case with data is underway.
NOZZLE BASE CFD ANALYSIS

John J. Erhart

Wednesday, April 29, 1992

MSFC CFD Workshop
OUTLINE

- Motivation
- Approach
- Geometry and Flow Boundary Conditions
- Results
- Summary
MOTIVATION

Understand Base Flow Phenomena

- Parameters Influencing Transport Of Hydrogen Into Base Of Nozzle.
- Define Typical Base Flow Environment
  - Streaklines
  - H₂
  - Mach Number
  - Pressure
APPROACH

- Simplify Geometry – Single Axisymmetric Nozzle With Largest Base Height.

- Two Altitudes – Different Plume Shapes.
GEOMETRY & FLOW BOUNDARY CONDITIONS

CASE | 1  | 2  |
-----|----|----|
P (PSIA) | 14.5 | 0.41 |
T °R | 517 | 398 |
M | .32 | 2.6 |

AIR
R=165.0"

R=6.5"

1 2 3

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
</tr>
</thead>
<tbody>
<tr>
<td>P</td>
<td>65.1</td>
<td>56.2</td>
<td>8.2</td>
</tr>
<tr>
<td>T</td>
<td>1141</td>
<td>835</td>
<td>811</td>
</tr>
<tr>
<td>M</td>
<td>0.4</td>
<td>1.5</td>
<td>2.0</td>
</tr>
<tr>
<td>H₂O</td>
<td>.53</td>
<td>.53</td>
<td>.53</td>
</tr>
<tr>
<td>H₂</td>
<td>.47</td>
<td>.47</td>
<td>.47</td>
</tr>
</tbody>
</table>

P = 1300 PSIA  P = 4.9 PSIA
T = 6336°R  M = 4.04
H₂O = .986  AR = 45
R = 43.6"
HIGH & LOW ALTITUDE – STREAKLINES

Base H₂ Concentration Function Of Nozzle Plume
And Bluff Body Recirculation Interaction

High Altitude

Low Altitude
RESULTS

Mach Number

Mach number at nozzle base region

CONTOUR LEVELS

0.000000
0.100000
0.200000
0.300000
0.400000
0.500000
0.600000
0.700000
0.800000
0.900000
1.000000
1.100000
1.200000
1.300000
1.400000
1.500000
1.600000
1.700000
1.800000
1.900000
2.000000
RESULTS

Pressure

CONTOUR LEVELS
0.00020  0.014
0.00060  0.069
0.00100  0.100
0.00140  0.140
0.00180  0.180
0.00220  0.220
0.00260  0.260
0.00300  0.300
0.00340  0.340
0.00380  0.380
0.00420  0.420
0.00460  0.460
0.00500  0.500
0.00540  0.540
0.00580  0.580

PSF
$10^{-4}$

PSI
RESULTS

$H_2$ – Low Altitude

CONTOUR LEVELS

0.00000
0.00300
0.00600
0.00900
0.01200
0.01500
0.01800
0.02100
0.02400
0.02700
0.03000
RESULTS

Mach Number – Low Altitude

CONTOUR LEVELS
0.00000
0.10000
0.20000
0.30000
0.40000
0.50000
0.60000
0.70000
0.80000
0.90000
1.00000
1.10000
1.20000
1.30000
1.40000
1.50000
1.60000
1.70000
1.80000
1.90000
2.00000
RESULTS

Pressure – Low Altitude

CONTOUR LEVELS
0.19000
0.19100
0.19200
0.19300
0.19400
0.19500
0.19600
0.19700
0.19800
0.19900
0.20000
0.20100
0.20200
0.20300
0.20400
0.20500
0.20600
0.20700
0.20800
0.20900
0.21000
SUMMARY

- $\text{H}_2$ Transport Into The Base Is A Function Of Configuration And Flight Conditions.

<table>
<thead>
<tr>
<th></th>
<th>Low Altitude</th>
<th>High Altitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recirculation Region</td>
<td>Stronger</td>
<td>Weaker</td>
</tr>
<tr>
<td>$\text{H}_2$ Transport Into Base</td>
<td>Trace</td>
<td>$\sim 1 - 1.5%$</td>
</tr>
<tr>
<td>Pressure</td>
<td>$\sim 13.9$ PSIA</td>
<td>$\sim 0.2$ PSIA</td>
</tr>
</tbody>
</table>
HEAT TRANSFER IN ROCKET ENGINE
COMBUSTION CHAMBERS AND NOZZLES

P. G. Anderson*, Y.S. Chen†, and R.C. Farmer*

Abstract

Complexities of liquid rocket engine heat transfer which involve the injector faceplate and regeneratively and film cooled walls are being investigated by computational analysis. A conjugate heat transfer analysis will be used to describe localized heating phenomena associated with particular injector configurations and coolant channels and film coolant dumps. These components are being analyzed, and the analyses verified with appropriate test data. Finally, the component analyses will be synthesized into an overall flowfield/heat transfer model. The FDNS code is being used to make the component analyses. Particular attention is being given to the representation of the thermodynamic properties of the fluid streams and to the method of combining the detailed models to represent overall heating. Unit flow models of specific coaxial injector elements have been developed and will be described. Film cooling simulations of film coolant flows typical of the subscale STME being experimentally studied by Pratt and Whitney have been made, and these results will be presented. Other film coolant experiments have also been simulated to verify the CFD heat transfer model being developed by SECA. The status of this entire study will be presented, and its relevance as a new design tool will be demonstrated.

* SECA, Inc., 3313 Bob Wallace Avenue, Suite 202, Huntsville, AL
† Engineering Sciences, Inc., 4920 Corporate Drive, Suite K, Huntsville, AL
HEAT TRANSFER IN ROCKET ENGINE
COMBUSTION CHAMBERS AND NOZZLES

P. G. Anderson, Y. S. Chen, and R. C. Farmer

SECA, Inc.
OUTLINE OF STUDY

- NAVIER-STOKES FLOW SOLVER

- TWO-EQUATION TURBULENCE MODELS WITH COMPRESSIBILITY CORRECTIONS AND WALL FUNCTION APPROACH

- HOLDEN’S TEST CASE #45
  - Slot Jet Nozzle Flowfield
  - Film Cooling Analysis

- GASL TEST CASE #41
\[ \frac{\partial \rho k}{\partial t} + \frac{\partial \rho k U_j}{\partial X_j} = \frac{\partial}{\partial X_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial X_j} \right] + \mu_t \left[ \frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial x_i} \right] \frac{\partial U_i}{\partial X_j} - \rho \varepsilon (1 + \alpha M_t^2) \]

\[ \frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial \rho \varepsilon U_j}{\partial X_j} = \frac{\partial}{\partial X_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial X_j} \right] + \frac{C_{\varepsilon_2}^* \varepsilon \mu_t}{k} \left[ \frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial x_i} \right] \frac{\partial U_i}{\partial X_j} - \rho C_{\varepsilon_2}^* \frac{\varepsilon^2}{k} \]

where

\[ \alpha = 1, \quad M_t = \frac{k}{a^2} \]

\[ C_{\varepsilon_1}^* = 1.44 \left(1 + 0.08 M^{0.25}\right), \quad C_{\varepsilon_2} = 1.92, \quad M = \frac{V_{\text{total}}}{a} \]
HEAT TRANSFER WALL FUNCTION:

(Integration of Near Wall Energy Balance-- Viegas et al. 1985)

Heat Flux Source Term:

\[ q_w = [h_w - h - Pr_t (u - u_w)^2 / 2] (\tau_w / Pr_t u) \]

\[ = [h_w - h - Pr_t (u - u_w)^2 / 2] (\rho u_r / Pr_t u^+) \]

where

\[ u^+ = \ln \left[ \frac{(y^+ + 11)^{4.02}}{(y^+)^2 - 7.37 y^+ + 83.3}^{0.79} \right] + 0.563 \tan^{-1}(0.12 y^+ - 0.441) - 3.81 \]

\[ Pr_t = 0.9 \]

For Adiabatic Wall Boundary Condition,

\[ h_w = h + Pr_t (u - u_w)^2 / 2 \]
HOLDEN'S TEST CASE #45

- SLOT JET NOZZLE \((H = 0.12)\)
  
  \[ Re = 4.51 \times 10^4 \text{ per inch} \]
  
  \[ \lambda = 0.0945 \]
  
  Mesh: \(61 \times 11 \times 31\)

- FILM COOLING ANALYSIS
  
  \[ Re = 7.00 \times 10^5 / \text{ in} \]
  
  Mesh: \(101 \times 81\)
HOLDEN TEST CASE 45 COOLING JET NOZZLE EXIT MACH NUMBER PROFILE COMPARISONS
HOLDEN FILM COOLING TEST CASE 45 VECTOR & HE-COUPON

<table>
<thead>
<tr>
<th>ID</th>
<th>VALUES</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0 0000E+00</td>
</tr>
<tr>
<td>B</td>
<td>1 0000E-01</td>
</tr>
<tr>
<td>C</td>
<td>2 0999E-01</td>
</tr>
<tr>
<td>D</td>
<td>3 0999E-01</td>
</tr>
<tr>
<td>E</td>
<td>4 0999E-01</td>
</tr>
<tr>
<td>F</td>
<td>5 0999E-01</td>
</tr>
<tr>
<td>G</td>
<td>6 0999E-01</td>
</tr>
<tr>
<td>H</td>
<td>7 0999E-01</td>
</tr>
<tr>
<td>I</td>
<td>8 0999E-01</td>
</tr>
<tr>
<td>J</td>
<td>9 0999E-01</td>
</tr>
</tbody>
</table>

XMIN = -3 3952E+00
XMAX = 7 5834E+00
YMIN = -1 5811E+00
YMAX = 7 5671E+00

FM-IN = 0 0000E+00
FM-AX = 1 0000E+00
DELF = 1 0000E-01

CONTOUR LEVELS
HOLDEN FILM COOLING TEST CASE 45 WITH k-CORRECTED MODEL
(HEAT FLUX UNIT: Btu/ft²/sec)
HOLDEN FILM COOLING TEST CASE 45 WITH E-CORRECTED MODEL
(HEAT FLUX UNIT = BTU/IN²-SEC)
HOLDEN FILM COOLING TEST CASE 45 WITH LAMINAR INLET DATA
- MODEL COMPARISON (HEAT FLUX UNIT = BTU/FT²/SEC)
HOLDEN FILM COOLING TEST CASE 45 WITH TURBO INLET DATA MODEL COMPARISONS (HEAT FLUX UNIT: BTU/FT^2/SEC)
GASL TEST CASE #41

Re = $1.74 \times 10^5$ per inch

$M_{\text{air}} = 3.84$

$P_{\text{air}} = 5.7$ psia

$T_{\text{air}} = 2000 \degree R$

$M_{H_2} = 2.50$ (Fully Developed Laminar Profile)

$Re_{H_2} = 2.557 \times 10^4$

$P_{H_2} = 16.6$ psia

$T_{H_2} = 233 \degree R$

Mesh: $101 \times 81$
Heat flux for case run #41 (frozen chemistry, high RE K-E)
HEAT FLOW FOR GASL RUN#1: (HIGH E UNFL C-CORRECTED)
CONCLUSIONS

- COOLING JET EXIT BOUNDARY CONDITIONS ARE IMPORTANT FOR FILM COOLING ANALYSIS

- 3-D COMPUTATION OF COOLING JET NOZZLE FLOWFIELD PROVIDES THE NEEDED JET EXIT BOUNDARY CONDITIONS

- COMPRESSIBILITY CORRECTIONS FOR TWO-EQUATION TURBULENCE MODELS ARE EFFECTIVE FOR FILM COOLING ANALYSIS
APPLICATION TO STME NOZZLE FLOW

- Mesh Size: 18,000 Grid Points
  40,000 Grid Points
  (2nd case in progress)

- Two-Equation Turbulence Model With Compressibility Correction

- Finite-Rate Chemistry
  (6 Species, 9 Reaction)

Subscale Nozzle Operating Conditions:

Nozzle: \( Re = 2.95 \times 10^5 \) \((Vin)\)
- Mass Flow Rate = 82.6 \((lbm/sec)\)
- \( P_c = 2190 \) \(\text{psia}\)
- \( T_c = 6700 \) \(\text{°R}\)
- \( M_{in} = 0.225 \)
- \( T_{in} = 1.1925 \) (Equilibrium Chem.)

Subsonic Jet: \( \dot{m} = 0.254 \) \(\text{lbm/sec} \) ; \( P = 59.13 \) \(\text{psia}\)
Sonic Jet: \( \dot{m} = 1.51 \) \(\text{lbm/sec} \) ; \( P = 60.90 \) \(\text{psia}\)
- \( T_{jet} = 530 \) \(\text{°R}\)
STME NOZZLE FLOW  H2 MASS FRACTION CONTOUR (FINITE-RATE)

XMIN: 4.278E+00
XMAX: 7.035E+00
YMIN: 3.2624E+00
YMAX: 5.794E+00

FMIN: 1.9835E-02
FMAX: 1.0000E+00
DELF: 4.9000E-02

CONTOUR LEVELS:

<table>
<thead>
<tr>
<th>ID</th>
<th>VALUES</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>2.0000E-02</td>
</tr>
<tr>
<td>B</td>
<td>6.9999E-02</td>
</tr>
<tr>
<td>C</td>
<td>1.7999E-01</td>
</tr>
<tr>
<td>D</td>
<td>1.6999E-01</td>
</tr>
<tr>
<td>E</td>
<td>2.1599E-01</td>
</tr>
<tr>
<td>F</td>
<td>2.6499E-01</td>
</tr>
<tr>
<td>G</td>
<td>3.1399E-01</td>
</tr>
<tr>
<td>H</td>
<td>3.6299E-01</td>
</tr>
<tr>
<td>I</td>
<td>4.1199E-01</td>
</tr>
<tr>
<td>J</td>
<td>4.6099E-01</td>
</tr>
<tr>
<td>K</td>
<td>5.0999E-01</td>
</tr>
<tr>
<td>L</td>
<td>5.5899E-01</td>
</tr>
<tr>
<td>M</td>
<td>6.0799E-01</td>
</tr>
<tr>
<td>N</td>
<td>6.5699E-01</td>
</tr>
<tr>
<td>O</td>
<td>7.0599E-01</td>
</tr>
<tr>
<td>P</td>
<td>7.5499E-01</td>
</tr>
<tr>
<td>Q</td>
<td>8.0399E-01</td>
</tr>
<tr>
<td>R</td>
<td>8.5299E-01</td>
</tr>
<tr>
<td>S</td>
<td>9.0199E-01</td>
</tr>
<tr>
<td>T</td>
<td>9.5099E-01</td>
</tr>
<tr>
<td>U</td>
<td>9.9999E-01</td>
</tr>
</tbody>
</table>
SSME REGENERATIVE COOLING SYSTEM MODEL

SSME MCC and Nozzle Wall Heating
Case 2 Cooling Jets Mesh System:
SUBSCALE STME WALL BLOCK REGION H2-MASS-FRACTION

XMIN: 3.9464E-01
XMAX: 5.6852E-01
YMIN: 3.2013E-01
YMAX: 4.6353E-01

FMIN: 1.1185E-02
FMAX: 1.0000E+00
DELF: 4.9400E-02

CONTOUR LEVELS:

<table>
<thead>
<tr>
<th>ID</th>
<th>VALUES</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.1999E-02</td>
</tr>
<tr>
<td>B</td>
<td>6.1400E-02</td>
</tr>
<tr>
<td>C</td>
<td>1.1079E-01</td>
</tr>
<tr>
<td>D</td>
<td>1.6055E-01</td>
</tr>
<tr>
<td>E</td>
<td>2.0559E-01</td>
</tr>
<tr>
<td>F</td>
<td>2.5899E-01</td>
</tr>
<tr>
<td>G</td>
<td>3.0839E-01</td>
</tr>
<tr>
<td>H</td>
<td>3.5779E-01</td>
</tr>
<tr>
<td>I</td>
<td>4.0719E-01</td>
</tr>
<tr>
<td>J</td>
<td>4.5659E-01</td>
</tr>
<tr>
<td>K</td>
<td>5.0599E-01</td>
</tr>
<tr>
<td>P</td>
<td>7.0299E-01</td>
</tr>
<tr>
<td>Q</td>
<td>8.0239E-01</td>
</tr>
<tr>
<td>R</td>
<td>8.5179E-01</td>
</tr>
<tr>
<td>S</td>
<td>9.0119E-01</td>
</tr>
<tr>
<td>T</td>
<td>9.5059E-01</td>
</tr>
<tr>
<td>U</td>
<td>1.0000E+00</td>
</tr>
</tbody>
</table>
APPLICATION OF COMPUTATIONAL FLUID DYNAMICS
TO THE DESIGN OF THE FILM COOLED STME SUBSCALE NOZZLE
FOR THE NATIONAL LAUNCH SYSTEM.

By

Joseph L. Garrett
Government Engine and Space Propulsion
Pratt and Whitney
West Palm Beach, Florida

ABSTRACT

A status of CFD calculations for the STME film/dump cooled nozzle design will be
presented, with an emphasis on the timely impact of CFD in the design of the sub-scale nozzle
coolant system. The following aspects of the sub-scale coolant delivery system were analyzed
with CFD:

1. Design trade study of a mechanical flow splitting device for uniform distribution of the
subsonic cavity flow.

2. Design trade study of the subsonic cavity lip to achieve coolant film integrity.

3. Analysis of the primary flow interaction with the core/secondary coolant streams.

All design calculations were performed with the Generalized Aerodynamic Simulation Program
(GASP), a 3-D, multi-block, generalized Navier-Stokes code capable of solving with frozen,
finite-rate or equilibrium chemical kinetics.

The initial design of the subsonic cavity flow used square posts to distribute the sonic orifice
jets into a uniform flow. Calculations for this design indicated that an unacceptable
mal-distribution of film occurred. Design modifications involving curved and slotted posts were
computed in an effort to uniformly distribute the secondary coolant flow. Analysis of these
configurations showed that although the flowfield improved in uniformity, it was still
unacceptable, especially at higher feed pressures. Results from these studies were then
incorporated into a design that resulted in the insertion of a porous metal ring into the subsonic
cavity. Subsequent water flow model studies showed that this concept was successful in
uniformly distributing flow exiting the cavity.

In addition to the design of the subsonic cavity, CFD was also used to analyze the
secondary coolant lip and the primary flow interaction with the core/secondary coolant streams.
A series of calculations were first performed to modify the subsonic cavity lip contour. The flow
over the modified lip was then computed simultaneously with the primary injectors to determine
the impact of the subsonic coolant stream on the primary slot jets.

Pressure, temperature, velocity and coolant mass fraction contours will be presented for
these configurations.
APPLICATION OF CFD TO THE DESIGN OF THE FILM COOLED STME SUBSCALE NOZZLE

Joseph L. Garrett
Pratt & Whitney, GESP
West Palm Beach, Florida

Presented At 10th Annual Workshop For CFD Applications In Rocket Propulsion – NASA MSFC
April 29, 1992
ACKNOWLEDGEMENTS

- NASA MSFC Consortium For CFD In Combustion Driven Flows.

- National Aerodynamic Simulation Facility For CRAY YMP Computing Resources.

- CRAY Research For CRAY YMP Computing Resources.
OUTLINE

- Analysis Of Secondary Coolant Cavity
- Support Of Secondary Subsonic Lip Design
- Analysis Of Core/Secondary/Primary Flow Interactions
- Summary
CFD CODE USED

General Aerodynamic Simulation Program

- 3-D, 2-D or Axisymmetric
- Parabolized & Full Navier-Stokes
- Explicit & Implicit Time Integration
- Finite-Rate, Frozen & Equilibrium Chemistry
- Algebraic & 2-Equation Turbulence Models
- Memory Management Techniques
ANALYSIS OF SECONDARY COOLANT CAVITY

Geometry Orientation
Subscale Thrust Chamber Assembly

36 Flow metering holes

Annulus

EDM material removal

72 Vanes
0.055 High

View A
ANALYSIS OF SECONDARY COOLANT CAVITY

Curve Post Design, Low P
ANALYSIS OF SECONDARY COOLANT CAVITY

Slotted Post Design, Low P
ANALYSIS OF SECONDARY COOLANT CAVITY

Comparison of Exit Solutions

Square Post, Low P

Curved Post, Low P

Curved Post, High P

Slotted Post, Low P

Slotted Post, High P
ANALYSIS OF SECONDARY COOLANT CAVITY

Design Solution: Porous Stainless Steel Filter

- H₂ Supply Holes
- 36 Subsonic Orifices
- Annular Groove
- 135 Primary Injector Slots
- Subsonic Cavity
- 40K Chamber
Previous Assumptions

- Algebraic Turbulence Model
- Equilibrium Chemistry in the Chamber
- Frozen Chemistry in the Interaction Region
- Chamber $P_0 = 2250$ psi, $T_0 = 6500$°R
- Injector $P_0 = 70$ psi, $T_0 = 530$°R
- Chamber Walls Fixed at $T = 1440$°R
- Injector Walls are Adiabatic
CORE/SECONDARY COOLANT FLOW INTERACTION

$H_2$ Mass Fraction Contours Indicate Vortex and Diffusion Mixing
CORE/SECONDARY COOLANT FLOW INTERACTION

Velocity Vectors / $H_2$ Mass Fraction Contours Show Vortex Mixing
# COMPARISON OF OLD & NEW SECONDARY SLOT

*Pertinent Geometry And Inflow Conditions*

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Old Slot</th>
<th>New Slot</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_{lip}$</td>
<td>0.005”</td>
<td>0.070”</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>0.210 lbm/sec</td>
<td>0.352 lbm/sec</td>
</tr>
<tr>
<td>$P_0$</td>
<td>70 psi</td>
<td>94.26 psi</td>
</tr>
<tr>
<td>$T_0$</td>
<td>530 R</td>
<td>535.5 R</td>
</tr>
<tr>
<td>$M$</td>
<td>0.20</td>
<td>0.05</td>
</tr>
<tr>
<td>$P$</td>
<td>68.06 psi</td>
<td>94.1 psi</td>
</tr>
<tr>
<td>$T$</td>
<td>525.8 R</td>
<td>535.25 R</td>
</tr>
<tr>
<td>$\varrho$</td>
<td>$7.536 \times 10^{-4}$ slug/ft$^3$</td>
<td>$1.023 \times 10^{-3}$ slug/ft$^3$</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>1.4</td>
<td>1.386</td>
</tr>
</tbody>
</table>
COMPARISON OF OLD & NEW SUBSONIC SLOT

New Conditions Result In Substantial Improvement in Film Effectiveness

H2 Mass Fraction Contours

Note: Increasing Lip Radius and Flow Rate Result in Diminishing the Impact of Vortex Mixing
COMPARISON OF OLD & NEW SUBSONIC SLOT

New Conditions Indicate Dramatic Improvement In Film Cooling

Temperature Contours (R)

Note: Backside Cooling Over The Last 1/3 of Injector Ring Would Further Reduce Temperature Load
COMPARISON OF OLD & NEW SUBSONIC SLOT

Penalty For High Mass Flow Rate Is Increased Shock Strength

Pressure Contours (psi)

Note: Increased Flow Rate has Small Impact on Pressure Sensed by the Primary Lip
<table>
<thead>
<tr>
<th>Property</th>
<th>Core Inflow</th>
<th>Subsonic Inflow</th>
<th>Primary Inflow</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_0$</td>
<td>2250 psi</td>
<td>94.26 psi</td>
<td></td>
</tr>
<tr>
<td>$T_0$</td>
<td>6780 R</td>
<td>535.5 R</td>
<td></td>
</tr>
<tr>
<td>$M$</td>
<td>0.2233</td>
<td>0.05</td>
<td>1.456</td>
</tr>
<tr>
<td>$P$</td>
<td>2187.38 psi</td>
<td>94.1 psi</td>
<td>55.5 psi</td>
</tr>
<tr>
<td>$T$</td>
<td>6735.0 R</td>
<td>535.25 R</td>
<td>372.1 R</td>
</tr>
<tr>
<td>$\rho$</td>
<td>$1.448 \times 10^{-2}$ slug/ft$^3$</td>
<td>$1.023 \times 10^{-3}$ slug/ft$^3$</td>
<td>$8.677 \times 10^{-4}$ slug/ft$^3$</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>1.134</td>
<td>1.386</td>
<td>1.386</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>84.21 lbm/sec</td>
<td>0.352 lbm/sec</td>
<td>1.773 lbm/sec</td>
</tr>
</tbody>
</table>

MESH SIZE: 75x61, 25x69, 59x81, 36x118, 36x118; 19,575 Grid Points
SUBSCALE CORE/FILM COOLANT INTERACTION

New Conditions Produce Less Mixing Of Secondary Film

H2 Mass Fraction Contours
Secondary Film Layer Provides Adequate Cooling for Injector Ring
SUBSCALE CORE/FILM COOLANT INTERACTION

Normal & Tangential Injection Produce Complex Shock & Expansion Waves

Pressure Contours (psi)
SUBSCALE CORE/FILM COOLANT INTERACTION

Integrity of Primary Jet Maintained

H2 Mass Fraction Contours

Note: Mixing of Jet with the core Indicates Exhaust Film is Less than 50% H2
SUBSCALE CORE/FILM COOLANT INTERACTION

Nozzle Geometry & Film Injection Produce Shock Losses

Pressure Contours (psi)

5.00000
9.00000
13.00000
17.00000
21.00000
25.00000
29.00000
33.00000
37.00000
41.00000
45.00000
49.00000
53.00000

Note: Performance Losses to be Quantified
SUMMARY

- CFD Impacted The Design Of The Secondary Cavity Coolant Distribution System.

- CFD Used To Minimize Temperature Loads On The Injector Ring.
FUTURE WORK

- Validate GASP With Subscale Nozzle Test Data.

- Evaluate / Enhance Fullscale Design (Scaling Methodology).
Computational fluid dynamics (CFD) analysis is providing verification of Space Shuttle flight performance details and is being applied to Space Shuttle main engine multiple plume interaction flow field definition. Advancements in real-gas CFD methodology described herein have allowed definition of exhaust plume flow details at Mach 3.5 and 107,000 ft. The specific objective of the study includes the estimate of flow properties at oblique shocks between plumes and plume recirculation into the Orbiter base so that base heating and base pressure can be model accurately. The approach utilizes the Rockwell USA Real Gas Three-Dimensional Navier-Stokes (USARG3D) Code for the analysis. The code has multi-zonal capability to detail the geometry of the plumes and base region and utilizes finite-rate chemistry to compute the plume expansion angle and relevant flow properties at altitude correctly. Through an improved definition of the base recirculation flow properties, heating and aerodynamic design environments of the Space Shuttle Vehicle can be further updated.

Results of IRAD work in progress indicate that at this altitude the plumes intersect and produce oblique shocks. At the hottest spot of the oblique shock between Engines 2 and 3, the recovery pressure and temperature were found to be 216 psfa and 5000 °F. There the flow resembles a location of a source flow with a discriminating streamline driving hot gas back into the base. Considerable exhaust gas is forced back toward the thermal shield of the Orbiter base between Engines 2 and 3, although Engine 1 flow is aspirated. The highest base temperature at the lower center of the heat shield reaches 2500 °F. Future work is planned to integrate the base flow solution to the integrated vehicle forebody flow for a total 'nose-to-plume' solution. With the vertical tail and OMS pods effects included later, it is expected that there may be recirculation of Engine 1 flow, also.
Computational Fluid Dynamics Analysis of Space Shuttle Vehicle and Exhaust Plume Flows at High Altitude Flight Conditions

by

N.S. Dougherty, J.B. Holt, B.L. Liu, and S.L. Johnson
Rockwell International
Space Systems Division
Huntsville, Alabama 35806

April 21, 1992

Work Sponsored in part by NASA Johnson Space Center
OBJECTIVE

Rockwell International
Space Systems Division

Huntsville Operations

Refined definition of Space Shuttle Vehicle airloading and base recirculation plume effects to reduce small uncertainties that remain at high altitude conditions to improve payload managers' margin for

- the vehicle as it flies today

- will fly after 1994 with ASRM's
Space Shuttle ascent computational fluid dynamics (CFD) simulations are being extended to 'nose-to-plume'.

Present effort:

at Mach 3.5 and 107,000 ft
ET FOREBODY PRESSURE

ET SURFACE ADJACENT TO ASRM

Graph showing pressure distribution with labeled axes and data points.
ORBITER FOREBODY PRESSURE

ORBITER CANOPY AND CARGO BAY DOORS

Graph showing computed and wind tunnel data for the pressure distribution along the x/Lo axis.
CONDITIONS:
FREE-STREAM MACH NUMBER = 3.5
ALTITUDE = 107,000 FT, PRESSURE = 17.5 PSFA, TEMPERATURE = -20°F

ORBITER BASE HEATING MECHANISM

BASE HEAT SHIELD
ENGINE 2

2000°F
600°F

BODY FLAP

BASE TEMPERATURES

1400°F

ENGINE 3

2500°F

-20°F

JET 1

OBlique Shock

NONVISCous JET BOUNDARY

JET 2

DISCRIMINATING STREAMLINE:

VELOCITY = U_d
MACH NUMBER = M_d

P_s = P_i (M_d)

NORMAL SHOCK FOR M_d
CONCLUSIONS

• Space Shuttle forebody pressures in the CFD simulation confirm the wind tunnel data

• Space Shuttle CFD simulation provides much greater detail on forebody pressures and temperatures and shows no adverse effects with ASRM's

• Space Shuttle Main Engine exhaust plumes CFD simulation shows base recirculation flow details and good agreement with Orbiter base heat shield pressure/temperature data
Direct Numerical Simulation of a Combusting Droplet with Convection

P. Y. Liang
CFD Technology Center,
Rocketdyne Division, Rockwell International
Canoga Park, California

The evaporation and combustion of a single droplet under forced and natural convection has been studied numerically from first principles using a numerical scheme that solves the time-dependent multi-phase and multi-species Navier-Stokes equations and tracks the sharp gas-liquid interface cutting across an arbitrary Eulerian grid. The flow fields both inside and outside of the droplet are resolved in a unified fashion. Additional governing equations model the inter-phase mass, energy and momentum exchange. Test cases involving iso-octane, n-hexane and n-propanol droplets show reasonable comparison with experimental data regarding parameters such as breakup mode, evaporation rate and flame stand-off distance. The partially validated code is thus readied to be applied to more demanding droplet combustion situations where substantial drop deformation render classical models inadequate.
DIRECT NUMERICAL SIMULATION OF A COMBUSTION DROPLET WITH CONVECTION

PAK-YAN LIANG

ROCKWELL INTERNATIONAL CORPORATION
ROCKETDYNE DIVISION
OBJECTIVE

STUDY THE EVAPORATION AND COMBUSTION OF A SINGLE DROPLET UNDER FORCED AND NATURAL CONVECTION FROM FIRST PRINCIPLES.
SIGNIFICANCE

DIRECT MODELING OF BOTH DROPLET AND AMBIENT IN UNIFIED CFD MODEL REMOVES CONSTRAINTS DUE TO ASSUMPTIONS OF

- SPHERICAL SYMMETRY

- SMALL CONVECTIVE EFFECTS

- LOW EVAPORATION RATE (NO BLOWING)

- SHARP FLAME FRONT (INSTANTANEOUS AND COMPLETE COMBUSTION)
Control Volume Next to Droplet Surface
Governing Equations

- Continuity equation integrated twice to yield

\[
\dot{N}_1 (\Delta z) = c \propto \ln \left( \frac{1 - X_{1\text{gas}}}{1 - X_{1\text{surface}}} \right)
\]

or

\[
\dot{m}_{\text{vap}} = \frac{c \propto \dot{M}_1 A_s}{\Delta z} \ln \left( \frac{1 - X_{1\text{gas}}}{1 - X_{1\text{surface}}} \right)
\]

- Assuming \( N_{\text{total}} \approx N_1 \)
Governing Equations

- Final heat-up rate equation for multi-species

\[ V_\ell \left( c_\ell \dot{M}_1 \right) \left( \frac{c_{p_{1u}}}{\dot{M}_1} \right) \frac{dT_\ell}{dt} = \frac{k \xi A_s \left( T_g - T_\ell \right)}{(\Delta z) \left( e^\xi - 1 \right)} - \dot{m}_{vap} \left( \frac{\Delta H_{vap}}{\dot{M}_1} \right) \]

where

\[ \xi \equiv \frac{\dot{N}_1 c_{p_{1u}}(\Delta z)}{k} = \frac{\dot{m}_{vap}}{A_s} \left( \frac{c_{p_{1u}}}{\dot{M}_1} \right) \left( \frac{\Delta z}{k} \right) \]

- Equation for single species

\[ \dot{m}_{vap} = A_s E_p \left( p_{vap} - p \right) \left( \frac{\dot{M}_1}{2\pi RT_\ell} \right)^{1/2} \]
Governning Equations

- Energy equation integrated twice to yield

\[ \dot{q}_e \left[ e^{\dot{N}_1 \frac{c_{p_{1v}}(\Delta z)}{k}} - 1 \right] = \dot{N}_1 c_{p_{1v}} (T_g - T_e) \]

where \( \dot{q}_e \) is in turn given by

\[ \dot{q}_e = \frac{V_e}{A_s} c_e c_{p_{1e}} \frac{dT_e}{dt} + \dot{N}_1 \left( \Delta H_{\text{vap}} \right) \]
ARICC-ST MODELS

- FULL NAVIER-STOKES IN BOTH GAS AND LIQUID PHASES

- TWO PHASE, VOLUME-OF-FLUID, WITH SURFACE TRACKING

- VARIABLE TEMPERATURE IN BOTH PHASES

- INTER-PHASE MASS AND HEAT TRANSFER (EVAPORATION) AND SURFACE TENSION
Fig. 1 Schematic of Partitioned Computational Volume with Evaporating Free Surface Segments
Fig. 2 Bag Mode Breakup of Simulated (a & c) and Experimental (b) Droplet Due to Shock Wave Passage. (c) was done with evaporation and ignition.

(a) 

(b) 

(c)
Table 1. Highlight of Problem Parameters for Combusting Droplet With Convection Simulations

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Forcet Convection</th>
<th>Natural Convection</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Liquid</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MW</td>
<td>n-Hexane (C₆H₁₄)</td>
<td>n-Propanol (C₃H₇OH)</td>
</tr>
<tr>
<td>Surface tension (dynes/cm)</td>
<td>86.178</td>
<td>60.096</td>
</tr>
<tr>
<td>Density (g/cc)</td>
<td>14.4 (at 333 K)</td>
<td>23.04 (at 293 K)</td>
</tr>
<tr>
<td>Viscosity (poise)</td>
<td>.21554 x 10⁻² (at 423 K)</td>
<td>.521 (at 423 K)</td>
</tr>
<tr>
<td>Conductivity (ergs/cm-s-K)</td>
<td>1.1076 x 10⁶ (at 342 K)</td>
<td>1.925 x 10⁻⁴ (at 470 K)</td>
</tr>
<tr>
<td>Vapor pressure (mmHg)</td>
<td>exp(15.84-2698/(T-48.78))</td>
<td>exp(17.54-3166/(T-83.15))</td>
</tr>
<tr>
<td>Latent heat (ergs/g)</td>
<td>3.165 x 10⁹((1-T_r)/(1-T_r,ref)) ³⁷⁵</td>
<td>6.952 x 10⁹((1-T_r)/(1-T_r,ref)) ³⁷⁵</td>
</tr>
<tr>
<td>Initial drop diam. (cm)</td>
<td>.1815</td>
<td>.210</td>
</tr>
<tr>
<td>Initial drop temp (K)</td>
<td>355.0</td>
<td>293.0</td>
</tr>
<tr>
<td>Ambient</td>
<td>N₂</td>
<td>Air</td>
</tr>
<tr>
<td>Initial temp. (K)</td>
<td>470.0</td>
<td>293.0</td>
</tr>
<tr>
<td>Pressure (atm.)</td>
<td>13.6</td>
<td>3</td>
</tr>
<tr>
<td>Prandtl number</td>
<td>.714</td>
<td>.75</td>
</tr>
<tr>
<td>Viscosity (poise)</td>
<td>1.925 x 10⁻⁴ (at 470 K)</td>
<td>1.813 x 10⁻⁴ (at 293 K)</td>
</tr>
<tr>
<td>Grid size</td>
<td>30 x 100</td>
<td>35 x 105</td>
</tr>
</tbody>
</table>
Fig. 3 Numerical vs Experimental Drop Diameter and Drop Center Temperature Histories in Forced Convection
Fig. 4 Numerical vs Experimental Drop Diameter Histories in Natural Convection. Dashed line is reference D²-law
Fig. 5 Flame Stand-off Distance and Drop Center Temperature vs Time in Natural Convection
Fig. 6 Sequence of Simulated Temperature Contours in Grey Scale
Showing Initial Transient of n-Propanol Droplet Combusting
with Natural Convection. (white Tmax=2500 K, black Tmin=300 K)
Fig. 7 Comparison of Simulated Density Gradient Contours (a) with Experimental Schlieren Photograph (b) of Combusting Droplet from Rush et al (Ref. 11)
Fig. 8 Close-up View of Velocity Vector Plot from n-Propanol Drop Combustion Under Natural Convection Simulation
Fig. 9 $\log_{10}$ Species Concentration Contours from n-Propanol Drop Combustion Under Natural Convection Simulation
CONCLUSIONS

- SIMULATIONS OF DEFORMING, EVAPORATING AND COMBUSTING DROPS PERFORMED WITHOUT EMPIRICAL INPUTS

- REASONABLE COMBUSTION RATE, EVAPORATION RATE AND TEMPERATURE DISTRIBUTION OBTAINED

- EFFECT OF EVAPORATION ON BREAKUP MODE REQUIRE MANY MORE PARAMETRIC STUDIES

- DETAILS OF INTERNAL RECIRCULATING FLOWFIELD REQUIRE LARGE GRID CONCENTRATION AROUND BOUNDARY LAYER
A NUMERICAL MODEL FOR ATOMIZATION-SPRAY
COUPLING IN LIQUID ROCKET THRUST CHAMBERS

by

M. G. Giridharan, A. Krishnan, J. J. Lee and A. J. Przekwas
CFD Research Corporation, Hunstville, AL 35802
and

K. Gross
NASA Marshall Space Flight Center, Hunstville, AL 35802

The physical process of atomization is an important consideration in the stable operation of liquid rocket engines. Many spray combustion CFD codes do not include an atomization sub-model but assume arbitrary drop size distributions, drop initial locations and velocities. It has been shown that the results of spray combustion models are extremely sensitive to the assumed droplet initial conditions. Furthermore, the atomization process itself is a strong function of the local conditions of the liquid and gas flow. Thus it is important to account for the strong mutual coupling between the liquid phase, the spray dynamics and the gas flow. A method of coupling an atomization model with the spray model in a REFLEQS CFD code will be presented. This method is based on a novel Jet-Embedding technique in which the equations governing the liquid jet core are solved separately using the surrounding gas phase conditions. The droplet initial conditions are calculated using a stability analysis appropriate for the atomization regime of liquid jet break-up.

This novel coupling model is used to analyze the SSME fuel preburner single injector flow. Results of the diffusion flame characteristics in a single injection element will be presented. The effect of relative velocity, mixture ratio and droplet initial conditions will be shown. The predictions of present atomization model is compared with that of the widely used CICM correlation. The results are also compared with the predictions of volume-of-fluid method.
A NUMERICAL MODEL FOR ATOMIZATION-SPRAY COUPLING IN LIQUID ROCKET THRUST CHAMBERS

by
M.G. Giridharan, J.J. Lee, A. Krishnan and A.J. Przekwas
CFD Research Corporation

and
K. Gross
Marshall Space Flight Center

10th Annual CFD Workshop
NASA - MSFC
April 30, 1992
OUTLINE

- Introduction
- Atomization-Spray Coupling Model
- Validation
- Demonstration Results
- Conclusions
INTRODUCTION

- Importance of an Atomization-Spray Coupling Model

- Two Approaches:
  - Interface Capturing - VOF Methodology
  - Interface Fitting - Jet-Embedding

- Atomization Models
  - Meyer's Model
  - Linear Dispersion Equation

- Spray Model
  - Eulerian/Lagrangian Particle Tracking
Temperature Distribution in the Thrust Chamber

$U_d = V_d = 25.0 \text{ m/s}, \ D_d = 25 \text{ Microns}$

$lsp = 318.6 \text{ seconds}$
Temperature Distribution in the Thrust Chamber

\[ Ud = Vd = 25.0 \text{ m/s}, \quad Dd = 100 \text{ Microns} \]

\[ Isp = 313.8 \text{ seconds} \]
ATOMIZATION MODELS

Meyers Model

- Energy Balance at the Interface

Linear Theory

- Dispersion Relationship: Growth Rate $= F(\lambda, r, s, Re, We)$

$Wavenumber$ $Wavy-wall$ $Analogy$
SPRAY MODEL

- Improved Version of PSI-CELL Model
- Deterministic Droplet Tracking
- Coupled Droplet Source/Sink Terms for the Gas Phase
- Droplet Boundary Conditions
  - Wall - Zero Normal Momentum
  - Symmetry - Reflection
LIQUID PHASE CALCULATION:

- Obtain Shape and Velocity of Intact Liquid Core

- Space-Marching Technique
GAS PHASE CALCULATION:

- REFLEQS CFD Code
- Gas Phase Grid is Adapted to the Shape of the Core
- Interface Modeled as Sliding Wall
- Combustion Model:
  - Instantaneous Chemistry Model
FLOW CHART OF COUPLING PROCEDURE

1. Initial guess for gas velocity and density at the interface
2. Solve 1-D jet core equations
3. Obtain local diameter of the core and its velocity
4. Prepare initial conditions for the spray model, i.e., the breakup rate and droplet size distribution
5. Adapt the gas phase grid to the liquid jet core shape
6. Impose the sliding wall boundary condition at the interface
7. New gas velocity and density at the interface
8. Obtain local diameter of the core and its velocity
9. Obtain gas phase solution using REFLEQS
10. If converged → Yes: Stop
11. No

Flow chart diagram:
- Initial guess for gas velocity and density at the interface
- Solve 1-D jet core equations
- Obtain local diameter of the core and its velocity
- Prepare initial conditions for the spray model, i.e., the breakup rate and droplet size distribution
- Adapt the gas phase grid to the liquid jet core shape
- Impose the sliding wall boundary condition at the interface
- New gas velocity and density at the interface
- Obtain gas phase solution using REFLEQS
- If converged → Yes: Stop
- No

Diagram notes:
- 4041-09/91 prog.
Experimental Data

- Low Speed Water Jet Experiments by Chigier (1990)
- Reynolds Number: 1460-9300
- Weber Number: 50-200
- Jet Intact Core Length from Photographs
- Drop Size (PDPA)
Comparison of Predictions with Experimental Data

isValidated (Continued)

<table>
<thead>
<tr>
<th>Prediction</th>
<th>Measurements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re=1456</td>
<td>L/D, Re = 1456</td>
</tr>
<tr>
<td>Re=4370</td>
<td>L/D, Re = 4370</td>
</tr>
<tr>
<td>Re=9328</td>
<td>L/D, Re = 9328</td>
</tr>
</tbody>
</table>

Intact Core Length

Drop Size

Weber Number

Drop Size (Micron)
## Operating Condition

<table>
<thead>
<tr>
<th></th>
<th>Oxidizer (LOX)</th>
<th>Fuel (GH₂)</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\dot{m}), kg/sec</td>
<td>0.1498</td>
<td>0.1805</td>
</tr>
<tr>
<td>(u_0), m/sec</td>
<td>25</td>
<td>213</td>
</tr>
<tr>
<td>(\rho_0), kg/m³</td>
<td>1122</td>
<td>91</td>
</tr>
<tr>
<td>T, °K</td>
<td>106</td>
<td>70</td>
</tr>
</tbody>
</table>

## Coaxial Injection Element
40 x 20
Eulerian Gas Grid

Boundary Conditions
RESULTS (CONTINUED)

○ Liang (1986)

--- Present Prediction ---

% of mass atomized

axial distance

4041/1 fig. 7.11
Ug = 200.0 m/s, Ul = 25.0 m/s, Tg = 200 K
(CICM Correlation)

Temperature Distribution

Oxygen Mass Source (gms/sec) Distribution
\[ \text{Ug} = 200.0 \text{ m/s, U1} = 25.0 \text{ m/s, Tg} = 200 \text{ K} \]

(Linear Surface Wave Model)

Temperature Distribution

Oxygen Mass Source (gms/sec) Distribution
Ug = 150.0 m/s, Ul = 25.0 m/s, Tg = 200 K

( CICM Model )

Temperature Distribution

Passive Scalar (Hydrogen) Concentration Distribution
CONCLUSIONS

- Developed a New Jet-Embedding Technique to Couple Atomization and Spray Models
- Predictions have been Validated with Water Jet Data
- Coupling Model has been Successfully Employed to SSME Single Injector Flow and Could be Extended to Multi-Injector Flow
- Computationally Efficient Tool for Rocket Injector Flow Analysis
Numerical Modeling for Dilute and Dense Sprays

C.P. Chen, Y.M. Kim, H.M. Shang, and J.P. Ziebarth
University of Alabama in Huntsville

T.S. Wang
NASA Marshall Space Flight Center

Abstract

Numerical modelings of fuel-droplet spray combustion finds useful applications for the assessment of the engine performance & stability characteristics. With our ongoing studies on turbulent reacting flows, we have successfully implemented a numerical model for spray-combustion calculations. In this model, the governing gas-phase equations in Eulerian coordinate are solved by a time-marching multiple pressure correction procedure based on the operator-splitting technique. The droplet-phase equations in Lagrangian coordinate are solved by a stochastic discrete particle technique. In order to simplify the calculation procedure for the circulating droplets, the effective conductivity model is utilized. This vaporization model includes the effects of variable thermophysical properties, non-unitary Lewis number in the gas-film, the Stefan flow effect, and the effect of internal circulation and transient liquid heating. The $k - \varepsilon$ models are utilized to characterize the time and length scales of the gas-phase in conjunction with turbulent modulation by droplets and droplet dispersion by turbulence. This method entails random sampling of instantaneous gas flow properties and the stochastic process requires a large number of computational parcels to produce the satisfactory dispersion distributions even for rather dilute sprays.

The present study has made two major improvements in spray combustion modelings. Firstly, we have developed a probability density function approach in multidimensional space to represent a specific computational particle. Advantages of a parcel PDF tracking method is to reduce the number of computational parcels representing the spray dynamics as well as to obtain grid-independent solutions for two-phase flows. Secondly, we incorporate the TAB Taylor Analogy Breakup model for handling the dense spray effects. These breakup models is based on the reasonable assumption that atomization and drop breakup are indistinguishable processes within a dense spray near the nozzle exit. Accordingly, atomization is prescribed by injecting drops which have a characteristic size equal to the nozzle exit diameter.

Example problems include the nearly homogeneous and inhomogeneous turbulent particle dispersion, and the non-evaporating, evaporating, and burning dense sprays. Comparison with experimental data will be discussed in detail.
NUMERICAL MODELING FOR DILUTE AND DENSE SPRAYS

Y.M. Kim, H.M. Shang, C.P. Chen, and J.P. Ziebarth
University of Alabama in Huntsville
and
T.S. Wang
NASA/Marshall Space Flight Center

10th Workshop for CFD Applications in Rocket Propulsion
April 28-30, 1992
NASA/Marshall Space Flight Center
MOTIVATION

- To attain a prediction capability to assess the performance and the stability characteristics of liquid-fueled engines.
- To appraise the physical submodels as well as to evaluate numerical procedures for prediction of spray-combusting flows.
- To gain fundamental understanding of the effects of vaporization, swirl, initial size distribution, and droplet dispersion.
- To provide reliable distribution of drop size and velocity.
- To develop an efficient, accurate, and stable numerical model spray combustion.
APPROACH

- Stochastic Particle Tracking Technique
  - Delta function stochastic separated flow (SSF) model
  - Deterministic dispersion width transport (DDWT) model
  - Stochastic dispersion width transport (SDWT) model

- Incorporation of Dense Spray Effects
  - Taylor analogy breakup (TAB) model.
  - Drop collision and coalescence model

- Eulerian-Lagrangian Formulation
  - Multiple pressure correction
  - Non-iterative for transient calculation
  - Applicable to all-speed flows
ISSUES

- Turbulene effects on droplets and turbulence modulation by droplets.
- Incorporation of dense spray effects and primary atomization model.
- Vaporization at subcritical and supercritical condition.
- Numerical accuracy, stability, and efficiency for the fast transient spray-combusting flows.
Turbulence-induced displacement and velocity:

\[
\frac{dv'_{ki}}{dt} = \frac{u'_{ki} - v'_{ki}}{\tau_{ki}}
\]

\[
\frac{dx'_{ki}}{dt} = v'_{ki}
\]

Particle fluctuating locations and velocities:

\[
x'_{ki} = u'_{k_{rms}} \Delta t_{ki} + (v'_{k(i-1)} - u'_{k_{rms}})\tau_{k(i-1)} \left(1 - e^{-\frac{-\Delta t_{ki}}{\tau_{k(i-1)}}}\right)
\]

\[
v'_{ki} = u'_{k_{rms}} + (v'_{k(i-1)} - u'_{k_{rms}})e^{\frac{-\Delta t_{ki}}{\tau_{k(i-1)}}}
\]
Time step to interact with $k^{th}$ eddy:

$$\sum_{i=1}^{m} \Delta t_{ki} = \Delta t_k$$

Variance of a computational particle pdf within the $k^{th}$ eddy:

$$\sigma_k^2 = \sigma_{k-1}^2 + \left( \sum_{i=1}^{m} x'_{ki} \right)^2$$

Normalized particle variance:

$$\hat{\sigma}_{yk} = K \frac{\sigma_{yk}}{\sqrt{N_t}}$$

$\frac{\sigma_{yk}}{\sqrt{N_t}}$ = statistical uncertainty in the mean particle position

$K$ = correction factor to account for undersampling

$N_t$ = total number of computational particles

$\sigma_{k-1}$ = existing variance of the particle pdf
Cumulative pdf distribution at any point in coordinate y:

\[ P(y) = \int_{-\infty}^{y} \frac{1}{\sqrt{2\pi} \sigma_{yk}} e^{-\frac{(y-y_p)^2}{2\sigma_{yk}^2}} \, dy \]

Symmetric cumulative distribution function:

\[ P(y) = 0.5 [ \text{erf} \left( \frac{y-y_p}{\sqrt{2\sigma_{yk}}} \right) + \text{erf} \left( \frac{y+y_p}{\sqrt{2\sigma_{yk}}} \right) ] \]

Two ways to calculate the mean position of parcel:

- Deterministic Dispersion Width Transport (DDWT) model
- Stochastic Dispersion Width Transport (SDWT) model
Numerical Implementations of Collision Model

- Mean collision rate between each pair of "parcels" coexisting in a given numerical cell:

\[ \nu = \frac{N_1}{Vol} \pi (r_1 + r_2)^2 \mid v_1 - v_2 \mid / N_2 \]

- Probability (Poisson distribution) occurring n collisions in time \( \Delta t \):

\[ P_n = e^{-\bar{n}} \frac{\bar{n}^n}{n!} \quad (\bar{n} = \nu \Delta t, \quad P_0 = e^{-\bar{n}} : \text{probability of no collision}) \]

- Stochastic collision sampling procedure:

\[ XX < P_0 \Rightarrow \text{ no collision} \]

\[ XX > P_0 \Rightarrow \text{ collision (collision parameter, } b = \sqrt{YY(r_1 + r_2)} \ ) \]

\[ b < b_{cr} \Rightarrow \text{ coalescence} , \quad b \geq b_{cr} \Rightarrow \text{ grazing collision} \]

\[ b_{cr} = f(\text{drop radii, surface tension, relative velocity}) \]

\[ XX, YY : 1st \text{ and } 2nd \text{ random number} \]
BREAKUP MODEL

- Taylor Analogy Breakup Model
  - analogy between an oscillating & distorting drop and spring-mass system

- Two Extra Equations \((y_p, y'_p)\)
  - \(y_p \rightarrow \text{deformation and } y'_p \rightarrow \text{oscillation}\)
  - functions of We, viscous damping time, and oscillation frequency

- Oscillation amplitude, frequency, \(y_p\), and \(y'_p \rightarrow t_{br}\)

\[
t^n < t_{br} < t^{n+1} : \text{breakup}
\]

Energy conservation \(\rightarrow r_{32} = \frac{r}{1 + \frac{8K}{20} + \frac{\rho_i r_i^3 y''^2 (6K - 5)}{120}}\)

chi - squared distribution for product drops \((r_{br})\)

Mass conservation \(\rightarrow N^{n+1} = N^n \left( \frac{r^n}{r_{br}} \right)^3\)

random velocity components normal to the relative velocity
VALIDATION CASES

- Particle Turbulent Dispersion (Snyder et. al.)
- Particle Laden Turbulent Round Jet (Yuu et. al.)
- Non-Evaporating Solid-Cone Spray (Hiroyasu et. al.)
- Non-Evaporating Hollow-Cone Spray (Shearer et. al.)
- Evaporating and Burning Spray (Yokota et. al.)
Figure 1. Particle dispersion of a nearly-homogeneous flow for SSF model (5000 particles) and DDWT model.
Figure 2. Normalized particle concentration distribution of particle laden round jet for SSF model (10,000 particles) and SDWT model (50 parcels) with various correction factors.
Figure 3. Normalized particle concentration distribution of particle-laden round jet for SDWT model (200 parcels) with various correction factors.
Figure 4. Normalized particle concentration distribution of particle laden round jet for SSF model (10,000 particles) and SDWT model (200 parcels).
Figure 4.22 Spray parcel distribution in a solid-cone spray ($t = 3.0\text{ms}$)
Figure 4.23 Spray tip penetration versus time in a solid-cone spray
Figure 4.24: Sauter mean diameter versus distance from the injector.
Figure 4.25 Spray parcel distribution and velocity vectors in a hollow-cone spray
Figure 4.26 Spray tip penetration versus time in a hollow-cone spray
Figure 4.27 Spray parcel distribution and contours of fuel mass fraction in an evaporating spray
Figure 4.28 Spray tip penetration versus time in an evaporating spray
Figure 4.29 Spray parcel distribution and contours of fuel mass fraction in a burning spray
Figure 4.30 Contours of temperature and oxygen mass fraction in a burning spray
SUMMARIES

- Efficient Particle Dispersion Modeling by DDWT and SDWT.
- Good Agreement with Experiment for Dense Spray Cases.
- Extension of SDWT to Evaporating and Burning Dense sprays.
- Implementation of Volume of Fluid (VOF) method.
- Incorporation of Supercritical Vaporization Model.
Modeling of SSME Fuel Preburner ASI

P. Y. Liang
CFD Technology Center
Rocketdyne Division, Rockwell International Corporation
Canoga Park, California

The Augmented Spark Ignitor (ASI) is a LOX/H2/electrical spark system that functions as an ignition source and sustainer for stable combustion. It is used in the SSME preburner combustor, the SSME main combustion chamber, the J-1 and J-2 engines as well as proposed designs of the Space Transportation Main Engine (STME) main combustor and gas generators. In the SSME it is a long circular cylindrical chamber located along the Main Combustor centerline with a truncated conical dome at the top, which contains two oblique LOX injection ports and two spark plugs offset at 90 degrees. Hydrogen injection is through a number of nearly tangential slots downstream which creates a swirl flow intended to cool the ASI chamber walls. Past incidents of erosion of the ASI spark plugs have often led to the need for replacement of these very costly devices. Thus it is desirable to understand the complex reactive flow field within the ASI both during the initial ignition transient and during the main stage steady state combustion (no sparking).

While it is impossible to perform direct optical diagnostics to measure the internal flow field of the ASI under hot-fire conditions, recent advances in CFD-based combustion modeling have made it feasible to characterize the flow through time-accurate simulations. This paper documents an undertaking to characterize the flow of the ASI. The code consists of a marriage of the Implicit-Continuous-Eulerian/Arbitrary-Lagrangian-Eulerian (ICE-ALE) Navier-Stokes solver with the Volume-of-Fluid (VOF) methodology for tracking of two immiscible fluids with sharp discontinuities. Spray droplets are represented by discrete numerical parcels tracked in a Lagrangian fashion. Numerous physical sub-models are also incorporated to describe the processes of atomization, droplet collision, droplet breakup, evaporation, and droplet and gas phase turbulence. An equilibrium chemistry model accounting for 8 active gaseous species is also used. Taking advantage of the symmetry plane, half of the actual ASI is modeled with a 3-dimensional grid that geometrically resolves the LOX ports, the spark plug locations, and the hydrogen injection slots. The pertinent features and formulations of these submodels will be briefly described in the paper.
OBJECTIVES

- APPLY NEWLY COMPLETED ARICC -3D CODE TO REAL LIFE COMBUSTION HARDWARE

- VALIDATE ATOMIZATION MODEL (PREVIOUSLY ANCHORED FOR COAXIAL INJECTORS) IN TRANSVERSE (NOMINALLY IMPINGING) INJECTION MODE
SIGNIFICANCE

- EROSION AND REPLACEMENT OF ASI SPARK PLUGS EXTREMELY COSTLY

- DESIRABLE TO UNDERSTAND REACTIVE FLOWFIELD INSIDE ASI DURING IGNITION TRANSIENT AND MAIN STAGE

- DIAGNOSTICS WITHIN ASI VERY DIFFICULT

- MAY REPLACE ONE SPARK PLUG WITH SEEDING DEVICE FOR PLUME MEASUREMENTS IN TESTS
MODEL PARAMETERS

- COARSE GRID (11 x 13 x 26) REPRESENTS 180° SECTOR

- CYCLIC BOUNDARIES AT J = 1 & J = 13

- CONSTANT PRESSURE OUTFLOW (5501 PSI) AT 4.84 IN. FROM TOP

- MULTI-PHASE MODEL RESOLVES LOX JET, 8 SPECIES (O, H, O₂, H₂, HO₂, H₂O₂, H₂O, OH) EQUILIBRIUM CHEMISTRY AND ABOUT 5000 DROPLETS

- K-ε TURBULENCE MODEL

- SIMULATION TIME = 1 MS COLD FLOW + 1 MS SPARK IGNITION + 2 MS HOT FIRE TRANSIENT TO STEADY-STATE
Fig. 5  Plots of droplet parcels for  
(a) case 2  (b) case 2  (c) case 4

1018
Fig. 1 Comparision of computed vs experimental drop penetration distances for different transverse gas velocities.
Fig. 2 Variation of penetration with orifice diameter; $V_g = 100 \text{ m/s}$, $V_1 = 15 \text{ m/s}$.
Fig. 3 Streamwise SMD profiles for various orifice diameters \( (d_{jo}) \); \( V_g = 100 \text{ m/s}, V_l = 15 \text{ m/s} \).
Fig. 4 Influence of airstream velocity on mean drop size taken at 12 cm station; $V_g = 15$ m/s, $d_j = 1.3$ mm (comp), = 1.5 mm (exp).
Fig. 1 Schematic of SSME Preburner ASI. (LOX ports are actually 90 deg. apart from spark plugs rather than being on same meridian plane as shown)
TYPICAL MACHINING BURR
OPB, FPB AND MCC ASI

Fig. 2 Close-up Side and Top Views of ASI Dome Region
Fig. 3 Computational Grid (Partial) Across a J-plane (shown reflected across central axis to represent full chamber)
Fig. 4  Velocity Profiles In Vertical and Horizontal Cross Sectional Planes Before Ignition (t=.737 msec)
Fig. 5 Transient Temperature and Velocity Fields During and Right After Ignition by Spark Plugs
Fig. 6 Velocity, Temperature and Log O₂ Concentration Profiles Across j=7 Plane at Pseudo-steady State (t=4.10 msec)
Fig. 7 Velocity, Temperature and Log O₂ Concentration Profiles Across k=6 Plane at Pseudo-steady State (t=4.10 msec)
Fig. 8 Log $H_2O$ Concentration Profiles at $t=4.10$ msec
SUMMARY OF OBSERVATIONS ON AS1

- HYDROGEN SWIRL DOMINATES COLD FLOWFIELD

- SUBSTANTIAL BLOCKAGE BY LOX JET AND SPRAY IMPLIES EFFECTIVELY TRANSVERSE SHEARING ATOMIZATION MODE

- SPARK ENERGY CAUSES VIOLENT MINI-EXPLOSIONS

- EVEN IN STEADY-STATE, INJECTED UNBURNED HYDROGEN FIRST SPIRALS UP TOWARD DOME, THEN DOWNWARD AFTER COMBUSTION. THUS WALL REMAINS COOL BELOW HYDROGEN PORTS
CFD MODELING OF TURBULENT FLOWS AROUND THE SSME MAIN INJECTOR ASSEMBLY USING POROSITY FORMULATION

Gary C. Cheng*, Y.S. Chen†, and Joseph H. Ruf‡

Abstract

Hot gas turbulent flow distributions around the main injector assembly of the Space Shuttle Main Engine (SSME) and LOX flow distributions through the LOX posts have great effect on the combustion phenomenon inside the main combustion chamber. An advanced computational fluid dynamics (CFD) analysis will help to provide more accurate and efficient characterization of this type of flow field. In order to design a CFD model to be an effective engineering analysis tool with good computational turn-around time (especially for 3-D flow problems) and still maintain good accuracy in describing the flow features, the concept of porosity is employed to describe the effects of blockage and drag force due to the presence of the LOX posts in the turbulent flow field around the main injector assembly of the SSME. A validated non-isotropic porosity model is developed and incorporated into an existing Navier-Stokes flow solver (FDNS). Volume and surface porosity parameters, which are based on the configurations of local LOX post clustering, will be introduced in to the governing equations, which can be written as

\[
\frac{1}{J} \left( \nu_s \frac{\partial \rho q}{\partial t} \right) = - \frac{\partial \nu_s F_i}{\partial \xi} + \nu_s S_q + R_q
\]

where \( J \) is the Jacobian, \( F_i \) is the sum of the convective flux and the viscous flux, \( \nu_s \) is the surface porosity, \( \nu_v \) is the volume porosity, \( S_q \) and \( R_q \) are the source and residual terms of the flow variable \( q \), respectively. The drag force and the heat flux source due to the presence of LOX posts will be added to the residual term. 2-D numerical studies have been conducted to identify the drag coefficients of the flows both through tube banks and around the shielded posts with a wide range of Reynolds numbers. A verified model of the drag coefficients is incorporated into the FDNS flow solver. A 2-D flow study of the main injector assembly is performed to verify the proposed porosity model. A reasonable O/F ratio distribution was obtained, therefore, a 3-D CFD analysis is conducted with confidence. The 3-D CFD analysis of the SSME main injector assembly is divided into three parts, LOX dome, LOX post assembly torus, and hydrogen cavity. A 62 x 91 x 16 mesh system is constructed for the LOX dome, where a 37 x 91 x 25 grid system is employed for the torus region, and the hydrogen cavity is discretized into a 29 x 91 x 14 mesh system. The numerical study of the turbulent flow in the SSME Phase II+ power head is analyzed based on 104% power balance level, and the result will be presented in the coming CFD workshop meeting.

* SECA, Inc., 3313 Bob Wallace Ave., Suite 202, Huntsville, AL
† Engineering Sciences, Inc., 4920 Corporate Dr., Suite K, Huntsville, AL
‡ ED 32, NASA/ Marshall Space Flight Center, Huntsville, AL
CFD MODELING OF TURBULENT FLOWS WITHIN SSME MAIN INJECTOR ASSEMBLY USING A POROSITY MODEL

By

Gary C. Cheng, SECA, Inc.

Y.S. Chen, ESI

AND

Joe Ruf
NASA/ Marshall Space Flight Center

NASA Contract No. NAS8-38871

TENTH ANNUAL CFD WORKSHOP MEETING, APRIL, 1992
PRESENTATION OVERVIEW

- OBJECTIVE
- NUMERICAL APPROACH
- PROPOSED POROSITY MODEL
- 3-D POROSITY/CFD ANALYSIS OF PHASE II+ POWER HEAD
- CONCLUSIONS AND RECOMMENDATIONS
OBJECTIVE

- DEVELOPMENT OF ROBUST CFD METHODOLOGY FOR TURBULENT FLOWS IN THE ENGINE 0209

- VALIDATION OF A POROSITY MODEL FOR NAVIER-STOKES FLOW SOLVER

- PREDICTION OF LOCAL MASS FLOW RATE AND O/F RATIO DISTRIBUTIONS DOWNSTREAM OF THE PRIMARY FACE PLATE
NUMERICAL APPROACH

- Flow around LOX post elements in simulated as flow through tube bank
- Mass flow rate through post elements and through porous plates are calculated based on porosity model

- Benchmark the models for porous media
  - Flow through tube banks
  - Flow through shielded posts with and without holes

- 3-D porosity model/CFD analysis of three components of the power head (LOX dome, LOX post assembly, hydrogen cavity)
PROPOSED POROSITY MODELS

● DRAG COEFFICIENTS FOR FLOW THROUGH TUBE BANK

○ NON-SHIELDED ELEMENTS

► Re (Local Flow Reynolds no.) < 4 x 10^3
   \[ C_d = 0.417 \exp(4.932 \, Re^{-0.296}) \]

► 4 x 10^3 < Re < 6 x 10^4
   \[ C_d = 0.647 - 0.5 \times 10^{-6} \, Re \]

► 6 x 10^4 < Re < 10^6
   \[ C_d = 0.618 + 0.491 \times 10^{-6} \, Re - 6.303 \times 10^{-12} \, Re^2 \\
   + 10.694 \times 10^{-18} \, Re^3 - 5.2 \times 10^{-24} \, Re^4 \]

► Re > 10^6
   \[ C_d = 0.2735 \]
- **SHIELDED ELEMENTS**
  - With Holes: \( C_d = 4 \).
  - Without Hole: \( C_d = 48 \).

- **LOX DOME POROSITY MODEL (104% RPL)**

\[
K \text{ (Loss Coeff.)} = \sqrt{\frac{\rho \Delta P}{m^2}} \quad ; \quad \Delta P = P_{\text{exit}} - P_{\text{chamber}} \quad \text{or} \quad \Delta P = P_{\text{exit}} - P_{\text{baffle}}
\]

<table>
<thead>
<tr>
<th></th>
<th>( \dot{m} ) (lb/sec)</th>
<th>( \Delta P ) (psi)</th>
<th>( K ) (ft(^4))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-Baffle Elements</td>
<td>665.105</td>
<td>575.07</td>
<td>( 4.14 \times 10^2 )</td>
</tr>
<tr>
<td>Baffle Elements</td>
<td>105.65</td>
<td>625.49</td>
<td>( 1.785 \times 10^4 )</td>
</tr>
<tr>
<td>First Three Rows</td>
<td>56.154</td>
<td>575.07</td>
<td>( 5.809 \times 10^4 )</td>
</tr>
</tbody>
</table>
### LOX POST ASSEMBLY POROSITY MODEL (104% RPL)

<table>
<thead>
<tr>
<th></th>
<th>Non-Baffle Elements</th>
<th>Baffle Elements</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Row #13</td>
<td>Row #12</td>
</tr>
<tr>
<td>K (in⁻⁴)</td>
<td>135</td>
<td>156</td>
</tr>
</tbody>
</table>

### HYDROGEN CAVITY POROSITY MODEL (104% RPL)

<table>
<thead>
<tr>
<th></th>
<th>( \dot{m} ) (lb/sec)</th>
<th>( \Delta P ) (psi)</th>
<th>K (ft⁻⁴)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary Face Plate</td>
<td>6.77</td>
<td>251</td>
<td>3.578 x 10⁴</td>
</tr>
<tr>
<td>Secondary Face Plate</td>
<td>3.41</td>
<td>98</td>
<td>5.506 x 10⁴</td>
</tr>
<tr>
<td>Baffle Elements</td>
<td>15.25</td>
<td>301</td>
<td>8.467 x 10³</td>
</tr>
<tr>
<td>Non-Baffle Elements</td>
<td>0</td>
<td>251</td>
<td>∞</td>
</tr>
<tr>
<td>BLC Holes</td>
<td>3.67</td>
<td>251</td>
<td>1.16 x 10⁵</td>
</tr>
</tbody>
</table>
3-D POROSITY/CFD ANALYSIS OF PHASE II+ POWER HEAD

- LOX DOME
  - GRID SIZE: 62 x 91 x 16

- INLET FLOW CONDITIONS
  - Static Pressure: 3670 psi
  - Static Temperature: 197 °R
  - Reynolds no.: 1.28 x 10^8 ft\(^1\)
  - Mass Flow Rate: 826.7 lb/sec

- INCOMPRESSIBLE, SINGLE SPECIES
LOX DOME, VELOCITY VECTORS AT -90 DEG (FUEL SIDE) PLANE

XMIN = -1.166E-01
XMAX = 6.3961E-01
YMIN = -7.6695E-01
YMAX = -1.4087E-01
MAIN INJECTOR ASSEMBLY WITH TRANSFER DUCTS

- THREE ZONES
  - Zone #1: 37 x 91 x 25 (Main Injector Assembly)
  - Zone #2: 10 x 21 x 17 (Fuel Transfer Duct)
  - Zone #3: 10 x 15 x 15 (Oxidizer Transfer Duct)

- INLET FLOW CONDITIONS:
  - Fuel Side (Reference Conditions)
    - Static Pressure: 3351 psi
    - Static Temperature: 1666 °R
    - Reynolds no.: 3.17 x 10⁷ ft⁻¹
    - Mass Flow Rate: 77.55 lb/sec (symmetrical)
    - O/F Ratio: 0.8685
Oxidizer Side

Static Pressure: 3353 psi

Static Temperature: 1254 °R

Mass Flow Rate: 33.375 lb/sec (symmetrical)

O/F Ratio: 0.599

U/U_{ref}: 0.693

- INCOMPRESSIBLE, ISOTHERMAL, TWO SPECIES, NON-REACTING FLOW
Geometry of Hot Gas Injector Assembly
LOX Post Assembly, Hot Gas Exit Velocity Contours (ft/sec)

CONTOUR LEVELS
-64.0
-63.5
-63.0
-62.5
-62.0
-61.5
-61.0
-60.5
-60.0
-59.5
-59.0
-58.5
-58.0
-57.5
-57.0
-56.5
-56.0
-55.5
-55.0
-54.5
-54.0
-53.5
-53.0
-52.5
-52.0
-51.5
-51.0
-50.5
-50.0
-49.5
-49.0
HYDROGEN CAVITY

GRID SIZE: 29 x 91 x 14

INLET FLOW CONDITIONS

Static Pressure: 3395 psi
Static Temperature: 448.66 °R
Reynolds no.: 2.52 x 10^7 ft^1
Mass Flow Rate: 14.55 lb/sec (symmetrical)

INCOMPRESSIBLE, SINGLE SPECIES
GRID SYSTEM OF THE HYDROGEN CAVITY (29 X 91 X 14)
HYDROGEN CAVITY, VELOCITY VECTORS AT SYMMETRY PLANE

Oxidizer Side

Fuel Side

XMIN: -1.0202E+00
XMAX: 8.3457E-01
YMIN: -7.7284E-01
YMAX: 7.7284E-01
The O/F ratio distribution along the outer edge of the injector face (no BLC coolant added)
THE O/F RATIO DISTRIBUTION ALONG THE OUTER EDGE OF THE INJECTOR FACE (WITH BL C COOLANT ADDED)
INJECTOR FACE (WITH BLD COOLANT ADDED) THE O/F RATIO DISTRIBUTION AT -90 DEG (FUEL SIDE) OF THE

R (FT)

00+3000 0
00+3000 1
00+3000 2
00+3000 3
00+3000 4
00+3000 5
00+3000 6
00+3000 7
00+3000 8
00+3000 9
THE O/F RATIO DISTRIBUTION AT 90 DEG (OXIDIZER SIDE) OF THE INJECTOR FACE (WITH BLN COOLANT ADDED)
CONCLUSIONS AND RECOMMENDATIONS

• THE PREDICTED O/F RATIO IS CLOSE TO STOICHIOMETRIC AROUND BAFFLE ELEMENTS

• THE RESULTS OF THIS STUDY SHOULD BE USED TO PREDICT ENGINE PERFORMANCE AND HEAT LOADS

• LOCAL MASS FLOW RATE DISTRIBUTION IS DEPENDENT ON PRESSURE AND LOSS COEFFICIENT DISTRIBUTION

• THE 3-D POROSITY/CFD ANALYSIS OF THE POWER HEAD CAN BE IMPROVED BY

  o KNOWING THE DISTRIBUTION OF CHAMBER PRESSURE AND OF BAFFLE ELEMENT DISCHARGE PRESSURE

  o THE MEASUREMENT OF LOSS COEFFICIENTS FOR EACH COMPONENTS

  o USING PROPER INLET FLOW PROFILES TO THE LOX DOME
Computational Fluid Dynamics Analysis of SSME Phase II and Phase II+ Preburner Injector Element Hydrogen Flow Paths

Joseph H. Ruf
Computational Fluid Dynamics Branch
Marshall Space Flight Center, NASA

Phase II+ Space Shuttle Main Engine powerheads E0209 and E0215 degraded their Main Combustion Chamber (MCC) liners at a faster rate than is normal for phase II powerheads. One possible cause of the accelerated degradation was a reduction of coolant flow through the MCC. Hardware changes were made to the preburner fuel leg which may have reduced the resistance and, therefore, pulled some of the hydrogen from the MCC coolant leg.

The preburner injector element's hydrogen flow path changed significantly from the phase II to the II+ design. The hydrogen inlet area was reduced by 42 percent, the annulus length was shortened, and the geometry of the annulus convergence section changed. With the large reduction of inlet area, an increased resistance would normally be expected. However, a 10 percent decrease in fuel flow resistance was quoted for phase II+ preburner injector elements.

To resolve this discrepancy, a Computational Fluid Dynamics analysis was performed to determine hydrogen flow path resistances of the phase II+ fuel preburner injector elements relative to the phase II element. The analysis was performed for 104 percent RPL conditions. FDNS was implemented on axisymmetric grids with the hydrogen assumed incompressible. The analysis was performed in two steps. The first isolated the effect of the different inlet areas and the second modeled the entire injector element hydrogen flow path.

The isolated effect of the reduced inlet area was a 3. percent increased resistance for phase II+ elements. However, the entire flow path model showed no difference between phase II and II+ injector element resistances. Phase II+ annulus geometry changes compensated for the reduced inlet area such that there was no net effect on hydrogen flow path resistance.
Computational Fluid Dynamic Analysis of SSME Phase II and Phase II+ Preburner Injector Element Hydrogen Flow Paths

Joseph H. Ruf
Computational Fluid Dynamics Branch
Marshall Space Flight Center
Computational Fluid Dynamic Analysis of SSME Phase II and Phase II+ Preburner Injector Element Hydrogen Flow Paths

- OBJECTIVE
- BACKGROUND
- APPROACH
- RESULTS
- CONCLUSION
OBJECTIVE

- Analytically determine the hydrogen flow path resistance of the phase II+ preburner injector elements.
BACKGROUND

- A number engine hardware changes were made from phase II to II+ to improve the operating environment in the SSME.

- Hot fire testing of E0209 and E0215 phase II+ engines resulted in an increased rate of main combustion chamber (MCC) liner degradation.

- It was thought the phase II+ hardware changes altered the resistances in the fuel flow circuit such that the coolant flow to the MCC was reduced.

- One area of uncertainty was the resistance of the preburner injector elements. Phase II+ preburner injector element has a reduced hydrogen inlet area and a shorter hydrogen annulus. Rocketdyne quoted a 10% decrease in hydrogen flow resistance for the phase II+ element with respect to phase II.
PHASE II+ INVESTIGATION

SIMPLIFIED RESISTANCE/FLOW MODEL SCHEMATIC:
PREBURNER INJECTOR ELEMENTS

Inlet area ↓ 42%

Phase II
(excluding LOX post support pins)

Phase II+

100 holes

200 holes

1.050

0.500
APPROACH

- Quantify the change in resistance/delta P with a comparative analysis of phase II and II+ elements.
  - phase II preburner injector element hydrogen flow delta P is 400 psi.

- Modeled the fuel preburner injector elements; oxidizer and fuel preburner injector elements are similar in both phase II and II+.

- Performed axisymmetric CFD analysis (FDNS) of the hydrogen flow path of phase II and II+ elements at 104% RPL conditions
  - Assumptions:
    o incompressible flow
    o rows of inlet holes modeled as equivalent area circumferential channels
    o LOX flow not modeled
APPROACH, cont.

- Analysis done in two steps
  - quantify the effect of reduced inlet area
  - quantify effect of entire hydrogen flow path

- Specified mass flow rate and inlet pressure; resistance $\propto$ delta P.

- Performed grid independency study
Phase II
RESULTS

- Effect of inlet area
  - phase II  delta P = 529 psi
  - phase II+ delta P = 545 psi  \( \Delta = +3\% \)

- Full hardware geometry
  - phase II  delta P = 406 psi
  - phase II+ delta P = 405 psi  \( \Delta = -0.25\% \)

- Phase II calculated pressure drop matched well with accepted value (406 vs. 400 psi).

- Majority of the flow passes through the holes closest to the exit.
Hydrogen Flow Distribution Through the Rows of Inlet Holes

Fraction of Total Hydrogen Flow

Row Number
CONCLUSION

- No net effect of phase II+ hardware changes on preburner injector element hydrogen resistance/delta P.
The hydrogen mixer for the STME is used to mix cold hydrogen bypass flow with warm hydrogen coolant chamber gas, which is then fed to the injectors. It is very important to have a uniform fuel temperature at the injectors in order to minimize mixture ratio problems due to the fuel density variations. In addition, the fuel at the injector has certain total pressure requirements. In order to achieve these objectives, the hydrogen mixer must provide a thoroughly mixed fluid with a minimum pressure loss. The AEROVISC CFD code was used to analyze the STME hydrogen mixer, and proved to be an effective tool in optimizing the mixer design. AEROVISC, which solves the Reynolds Stress-Averaged Navier-Stokes equations in primitive variable form, was used to assess the effectiveness of different mixer designs. Through a parametric study of mixer design variables, an optimal design was selected which minimized mixed fuel temperature variation and fuel mixer pressure loss. The use of CFD in the design process of the STME hydrogen mixer was effective in achieving an optimal mixer design while reducing the amount of hardware testing.
STME HYDROGEN MIXER STUDY

TENTH ANNUAL WORKSHOP FOR CFD APPLICATIONS IN ROCKET PROPULSION

NASA MARSHALL SPACE FLIGHT CENTER

Robert F. Blumenthal
Dongmoon Kim
George Bache'
April 30, 1992
STME HYDROGEN MIXER PROVIDES UNIFORM TEMPERATURE HYDROGEN TO INJECTORS

LOX INLET

HYDROGEN MIXER

HYDROGEN CHAMBER BYPASS MANIFOLD

INJECTOR FACE

COMBUSTION CHAMBER COOLANT CHANNELS
STME HYDROGEN MIXER

155° BEND

DIFFUSER SECTION

MIXING CHANNEL

$\dot{m}_H = 44.3 \text{ LB SEC}$

$T_{HS} = 503 \text{ DEG R}$

$\dot{m}_C = 118.9 \text{ LB SEC}$

$T_{CS} = 81.6 \text{ DEG R}$

1096
WHY IS MIXING IMPORTANT?

- Injector Elements All Designed With Identical Metering Orifice Areas and Equal $\Delta P$ Across Each Injector Element Which Therefore Require Uniform Hydrogen Density in Order to Have Equal $H_2$ Flow Rate to Each Element

- A Uniform Mixture Ratio Injector Core Delivers Highest ISP Performance

- Uniform $H_2$ Density (Mixture Ratio) Is Dependent on the Performance of the Hydrogen Mixer

- Uniform Temperature Implies Uniform Density
MODELING ISSUES

Propulsion Division

- DISCRETE COLD INLET HOLES
- HOT GAS INLET ASSUMES UNIFORM FLOW ACROSS PASSAGE
- 3-D WEDGE
- COMPRESSIBLE FLOW
- GAS PROPERTIES BASED ON MIXED TEMPERATURE
- STANDARD K-ε TURBULENCE MODEL
- ADIABATIC WALLS
- EXIT PLANE AT BEGINNING OF DIFFUSER SECTION
- GRID SIZE 97 X 19 X 16
STME HYDROGEN MIXER

- Cold Hydrogen Inlet Hole Size Varied to Determine Effect on Mixing

<table>
<thead>
<tr>
<th>$N_{C\text{NOM}}$</th>
<th>$N_C$</th>
<th>$D_C$ [in]</th>
<th>$A_{CTOT}$ [in$^2$]</th>
<th>Wedge Angle [Deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>495</td>
<td>0.091</td>
<td>3.22</td>
<td>1.45</td>
</tr>
<tr>
<td>750</td>
<td>749</td>
<td>0.074</td>
<td>3.22</td>
<td>0.96</td>
</tr>
<tr>
<td>1000</td>
<td>1033</td>
<td>0.063</td>
<td>3.22</td>
<td>0.70</td>
</tr>
</tbody>
</table>

- Cold Hydrogen Inlet Holes Are Staggered With Respect to the Mixing Channel Centerline
STATIC TEMPERATURE CORRESPONDING TO DIFFERENT L/Dh LOCATIONS

Propulsion Division

<table>
<thead>
<tr>
<th>L/Dh</th>
<th>Temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>3.25E+02</td>
</tr>
<tr>
<td>0.76</td>
<td>3.15E+02</td>
</tr>
<tr>
<td>1.82</td>
<td>3.05E+02</td>
</tr>
<tr>
<td>3.88</td>
<td>2.95E+02</td>
</tr>
<tr>
<td>8.0</td>
<td>2.85E+02</td>
</tr>
<tr>
<td>5.98</td>
<td>2.75E+02</td>
</tr>
<tr>
<td></td>
<td>2.65E+02</td>
</tr>
<tr>
<td></td>
<td>2.55E+02</td>
</tr>
<tr>
<td></td>
<td>2.45E+02</td>
</tr>
<tr>
<td></td>
<td>2.35E+02</td>
</tr>
<tr>
<td></td>
<td>2.25E+02</td>
</tr>
<tr>
<td></td>
<td>2.15E+02</td>
</tr>
<tr>
<td></td>
<td>2.05E+02</td>
</tr>
<tr>
<td></td>
<td>1.95E+02</td>
</tr>
<tr>
<td></td>
<td>1.85E+02</td>
</tr>
<tr>
<td></td>
<td>1.75E+02</td>
</tr>
<tr>
<td></td>
<td>1.65E+02</td>
</tr>
<tr>
<td></td>
<td>1.55E+02</td>
</tr>
<tr>
<td></td>
<td>1.45E+02</td>
</tr>
<tr>
<td></td>
<td>1.35E+02</td>
</tr>
<tr>
<td></td>
<td>1.25E+02</td>
</tr>
<tr>
<td>Plane Between Holes</td>
<td>Plane Through Top Hole</td>
</tr>
<tr>
<td>---------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>5.000E+02</td>
<td>4.790E+02</td>
</tr>
<tr>
<td>4.580E+02</td>
<td>4.370E+02</td>
</tr>
<tr>
<td>4.160E+02</td>
<td>3.950E+02</td>
</tr>
<tr>
<td>3.740E+02</td>
<td>3.530E+02</td>
</tr>
<tr>
<td>3.520E+02</td>
<td>3.320E+02</td>
</tr>
<tr>
<td>3.110E+02</td>
<td>2.900E+02</td>
</tr>
<tr>
<td>2.990E+02</td>
<td>2.890E+02</td>
</tr>
<tr>
<td>2.700E+02</td>
<td>2.660E+02</td>
</tr>
<tr>
<td>2.270E+02</td>
<td>2.060E+02</td>
</tr>
<tr>
<td>1.850E+02</td>
<td>1.650E+02</td>
</tr>
<tr>
<td>1.640E+02</td>
<td>1.430E+02</td>
</tr>
<tr>
<td>1.220E+02</td>
<td>9.80E+01</td>
</tr>
<tr>
<td>0.90E+02</td>
<td>0.000E+01</td>
</tr>
</tbody>
</table>
**GenCorp Aerojet**

**NC - 750 HOLES**

- **Plane Between Inlet Holes**
- **Plane Passing Through Top Inlet Hole**

**Propulsion Division**

| SPEED   | 1.020E+04 | 1.030E+04 | 1.740E+04 | 1.650E+04 | 1.560E+04 | 1.470E+04 | 1.380E+04 | 1.290E+04 | 1.207E+04 | 1.117E+04 | 1.027E+04 | 9.374E+03 | 8.472E+03 | 7.571E+03 | 6.660E+03 | 5.766E+03 | 4.905E+03 | 3.969E+03 | 3.865E+03 | 2.162E+03 | 1.260E+03 |
|---------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|-----------|
| 1.050E+04 | 1.060E+04 | 1.750E+04 | 1.660E+04 | 1.570E+04 | 1.480E+04 | 1.390E+04 | 1.300E+04 | 1.210E+04 | 1.120E+04 | 1.030E+04 | 1.040E+04 | 9.505E+03 | 8.580E+03 | 7.680E+03 | 6.780E+03 | 5.880E+03 | 4.920E+03 | 4.020E+03 | 3.120E+03 | 2.220E+03 | 1.320E+03 |

**Notes:**

- Plane passing through the top inlet hole.
- Plane between inlet holes.
EXIT PLANE TEMPERATURE DEPENDENT ON COLD HYDROGEN INLET HOLE SIZE

NC-500 HOLES

NC-750 HOLES

NC-1000 HOLES
TOTAL TEMPERATURE VARIATION VS. L/DH
NC=500, 750, 1000 HOLES

500 HOLES
750 HOLES
1000 HOLES
TOTAL PRESSURE VARIATION VS. L/DH
NC=500, 750, 1000 HOLES

L/DH

TOTAL PRESSURE [PSI]

--- 500 HOLES
----- 750 HOLES
------ 1000 HOLES
EXIT PLANE MASSFLOW VS. TOTAL TEMPERATURE
NC=500, 750, 1000 HOLES

TOTAL TEMPERATURE [DEG R]

SUM MDOT/(MDOT TOT)

--- 500 HOLES
----- 750 HOLES
----- 1000 HOLES
# COLD HYDROGEN INLET HOLE SIZE RESULTS

<table>
<thead>
<tr>
<th>$N_C$</th>
<th>$T_{\text{AVE}}$ [°R]</th>
<th>$\sigma_T$ [°R]</th>
<th>$\Delta T_T$ [°R]</th>
<th>$\Delta P_T$ [Psi]</th>
</tr>
</thead>
<tbody>
<tr>
<td>500</td>
<td>204.0</td>
<td>56.9</td>
<td>174.5</td>
<td>116.9</td>
</tr>
<tr>
<td>750</td>
<td>203.6</td>
<td>33.1</td>
<td>98.7</td>
<td>95.3</td>
</tr>
<tr>
<td>1000</td>
<td>205.5</td>
<td>44.2</td>
<td>126.2</td>
<td>93.3</td>
</tr>
</tbody>
</table>

$T_{\text{AVE}}$ = Mass-Averaged Value of Total Temperature at Exit Plane

$\Delta T_T$ = Total Temperature Range at Model Exit Plane

$\Delta P_T$ = Net Total Pressure Recovery ($P_{\text{EXIT}} - P_{\text{THINLET}}$)

$\sigma_T$ = Standard Deviation of Temperature at Exit Plane
COLD HYDROGEN INLET FLOW ANGLE WAS VARIED TO DETERMINE EFFECTS ON TOTAL TEMPERATURE AND PRESSURE
EXIT PLANE TEMPERATURE DEPENDENT ON COLD HYDROGEN FLOW ANGLE
NC= 750 HOLES

THETA=0 DEG

THETA=26.5 DEG

THETA=33.7 DEG

THETA=45 DEG

Propulsion Division

TOTAL
2.500E+02
2.450E+02
2.400E+02
2.350E+02
2.300E+02
2.250E+02
2.200E+02
2.150E+02
2.100E+02
2.050E+02
2.000E+02
1.950E+02
1.900E+02
1.850E+02
1.800E+02
1.750E+02
1.700E+02
1.650E+02
1.600E+02
1.550E+02
1.500E+02
TOTAL PRESSURE VARIATION VS. L/DH
FLOW ANGLE = 0.0, 26.5, 33.7, 45 DEG

TOTAL PRESSURE [PSI]

--- ANG = 0.0 DEG
----- ANG = 26.5 DEG
-------- ANG = 33.7 DEG
---------- ANG = 45.0 DEG

L/DH

0.0 1.00 2.00 3.00 4.00 5.00 6.00 7.00 8.00
TOTAL TEMPERATURE VARIATION VS. L/DH
FLOW ANGLE = 0, 26.5, 33.7, 45 DEG

△ TOTAL TEMPERATURE [DEG R]

- ANG = 0.0 DEG
- ANG = 26.5 DEG
- ANG = 33.7 DEG
- ANG = 45.0 DEG

L/DH
EXIT PLANE MASS FLOW VS. TOTAL TEMPERATURE
FLOW ANGLE=0, 26.5, 33.7, 45 DEG

TOTAL TEMPERATURE (DEG R)
## COLD HYDROGEN INLET FLOW ANGLE RESULTS

<table>
<thead>
<tr>
<th>ANG [DEG]</th>
<th>$T_{TAVE}$ [°R]</th>
<th>$\sigma_T$ [°R]</th>
<th>$\Delta T_T$ [°R]</th>
<th>$\Delta P_T$ [Psi]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>203.6</td>
<td>33.1</td>
<td>98.7</td>
<td>95.3</td>
</tr>
<tr>
<td>26.5</td>
<td>199.2</td>
<td>17.8</td>
<td>58.5</td>
<td>11.4</td>
</tr>
<tr>
<td>33.7</td>
<td>199.3</td>
<td>11.8</td>
<td>42.4</td>
<td>-9.5</td>
</tr>
<tr>
<td>45.0</td>
<td>198.8</td>
<td>8.1</td>
<td>24.3</td>
<td>-58.3</td>
</tr>
</tbody>
</table>

- $T_{TAVE}$ = Mass-Averaged Value of Total Temperature at Exit Plane
- $\Delta T_T$ = Total Temperature Range at Model Exit Plane
- $\Delta P_T$ = Net Total Pressure Recovery ($P_{EXIT} - P_{THINLET}$)
- $\Delta_T$ = Standard Deviation of Temperature at Exit Plane
FUTURE WORK

- Examine Other Configurations
  - Swirled Injection
  - Smaller Mixing Channel Area
  - Inline Cold Hydrogen Inlet Holes
  - Modifying Position of Cold Hydrogen Inlet Holes With Respect to the Mixing Channel Centerline
- Provide Design Requirements for Experimental Cold Flow Hardware
- Support Cold Flow Testing
- Analyze Cold Flow Data and Validate Aerovisc Predictive Capability
- Use Validated Model to Design Flight Mixer
AN EXPERIMENTAL STUDY OF THE FLUID MECHANICS ASSOCIATED WITH POROUS WALLS
(AIAA 92-0769)

N. Ramachandran,
Universities Space Research Association,
J. Heaman,
A. Smith,
NASA Marshall Space Flight Center,
Huntsville, AL 35812

ABSTRACT

The fluid mechanics of air exiting from a porous material is investigated. The experiments are filter rating dependent, as porous walls with filter ratings differing by about three orders of magnitude are studied. The flow behavior is investigated for its spatial and temporal stability. The results from the investigation are related to jet behavior in at least one of the following categories: (1) Jet coalescence effects with increasing flow rate, (2) Jet field decay with increasing distance from the porous wall, (3) Jet field temporal turbulence characteristics and (4) Single jet turbulence characteristics. The measurements show that coalescence effects cause jet development and this development stage can be traced by measuring the pseudoturbulence (spatial velocity variations) at any flow rate. The pseudoturbulence variation with increasing mass flow reveals an initial increasing trend followed by a leveling trend, both of which are directly proportional to the filter rating. A critical velocity begins this leveling trend and represents the onset of fully developed jetting action in the flow field. A correlation is developed to predict the onset of fully developed jets in the flow emerging from a porous wall. The data further show, that the fully developed jet dimensions are independent of the filter rating, thus providing a length scale for this type of flow field (1 mm). Individual jet characteristics provide another unifying trend with similar velocity decay behavior with distance; however, the respective turbulence magnitudes show vast differences between jets from the same sample. Measurement of the flow decay with distance from the porous wall show that the higher spatial frequency components of the jet field dissipate faster than the low frequency components. Flow turbulence intensity measurements show an out of phase behavior with the velocity field and are generally found to increase as the distance from the wall is increased.
REFERENCES


Jets

1. Interaction (coalescence)

2. Size (filter rating dependent?)
Figure 3 (plate 1). Unstable flow through a square grid. $U_{fink} = 1000$. The large-scale pattern is steady in time.
Filter rating $\neq$ pore size of material

$\neq$ particle size of material
An Experimental Study of the Fluid Mechanics Associated with Porous Walls

DATA ACQUISITION VIA A/D BOARD ON PERSONAL COMPUTER

HOT WIRE TRANSUDER

PRESSURE REGULATOR

FLOW

POROUS MATERIAL SAMPLES

POROUS MATERIAL EXPERIMENTAL TEST EQUIPMENT
MARSHALL SPACE FLIGHT CENTER/EXPERIMENTAL BRANCH (ED?)
BUILDING 4777C

AIR SOURCE 2400 PSI PORTABLE TANKS
Fig. 1  Velocity distribution above roughness rig section.

Fig. 2  Velocity distribution above smooth porous surface.
Pseudoturbulence (P71)

Velocity fluctuation levels on a length scale instead of a time scale.
PSEUDOTURBULENCE

MEAN VELOCITY (CMS)

PSEUDOTURBULENCE INTENSITY (%)
JET DIAMETER HISTOGRAM

OCCURRENCES

JET DIAMETER (M.M.)

0 1 2 3
**PSEUDOTURBULENCE (cont.)**

<table>
<thead>
<tr>
<th>Filter Rating $\mu$</th>
<th>Porosity $\bar{\phi}$</th>
<th>$V_{cn}$ m/s</th>
<th>$Re_{cn}$</th>
<th>$Re_{cn} \cdot \bar{\phi}^2$</th>
<th>$V_{cn} \bar{\phi}^2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.2838</td>
<td>0.78</td>
<td>49.09</td>
<td>3.95</td>
<td>0.0628</td>
</tr>
<tr>
<td>10</td>
<td>0.3445</td>
<td>0.53</td>
<td>33.35</td>
<td>3.96</td>
<td>0.0629</td>
</tr>
<tr>
<td>30</td>
<td>0.4173</td>
<td>0.399</td>
<td>25.11</td>
<td>4.37</td>
<td>0.0695</td>
</tr>
<tr>
<td>100</td>
<td>0.4780</td>
<td>0.284</td>
<td>17.87</td>
<td>4.00</td>
<td>0.0649</td>
</tr>
</tbody>
</table>
OTHER CATEGORIES OF RESULTS:

1. Jet field decay with distance from the surface.
2. Jet field turbulence characteristics.
3. Single jet turbulence characteristics.
A series of rocket-related studies intended to develop a suitable data base for validation of CFD models of characteristic combustion-driven flows has been undertaken at the Propulsion Engineering Research Center at Penn State. Included are studies of coaxial and impinging jet injectors as well as chamber wall heat transfer effects. The objective of these studies is to provide fundamental understanding and benchmark quality data for phenomena important to rocket combustion under well-characterized conditions. Diagnostic techniques utilized in these studies emphasize determinations of velocity, temperature, spray and droplet characteristics and combustion zone distribution. Since laser diagnostic approaches are favored, the development of an optically accessible rocket chamber has been a high priority in the initial phase of the project. During the design phase for this chamber, the advice and input of the CFD modeling community were actively sought through presentations and written surveys. Based on this procedure, a suitable uni-element rocket chamber has been fabricated and is presently under preliminary testing. Results of these tests, as well as the survey findings leading to the chamber design, was presented.

In particular, laser-induced fluorescence imaging results for hydroxyl radicals have been obtained. These experiments were conducted using gaseous hydrogen/gaseous oxygen propellants in the optically accessible uni-element rocket chamber. Heat transfer studies demonstrated the effectiveness of the curtain window purge needed to protect the optical surfaces. These results also demonstrated the capability to determine wall heat transfer rates in the present rocket test chamber. Measurements indicated that heat transfer rates were approximately an order of magnitude smaller than observed in actual rocket chambers. If larger propellant flow rates are utilized, this difference should be further narrowed.

Related results from impinging jet studies under non-combusting conditions revealed several trends regarding drop size and liquid sheet breakup. The general experimental observation that breakup length increases with increasing jet velocity and decreasing impingement angle is in agreement with previous studies. However, this trend is opposite to predictions derived from linear stability-based analysis of liquid sheet atomization. However, drop size predictions for linear stability-based analysis reproduced the observed trend of decreasing drop size with increasing jet velocity and increasing impingement angle.
Experimental Studies of Characteristic Combustion-Driven Flows for CFD Validation


The Propulsion Engineering Research Center
The Pennsylvania State University
University Park, PA 16802

Workshop for Computational Fluid Dynamic (CFD) Applications in Rocket Propulsion

April 28-30, 1992
OUTLINE

- CFD Validation - Optically Accessible Rocket
- Heat Transfer Measurements
- Impinging Jet Studies
- Summary
- Future Work
CFD VALIDATION - OPTICALLY ACCESSIBLE ROCKET
M. Moser, S. Pal, R. Santoro, C. Merkle

OBJECTIVE

To Provide Benchmark-Quality Data for CFD Code Validation for Combustion-Driven Flows

Obtain Fundamental Data Under Realistic and Well Characterized Conditions
APPROACH

Emphasize Uni-element Coaxial Injectors

Obtain Fundamental Data Under Well Characterized and Realistic Conditions

- Pressure
- Combustion zone
- Mean Velocity
- Turbulence Intensity
- Species
- Droplet Size and Velocity

Employ Non-Intrusive Advanced Diagnostics

- Laser Induced Fluorescence
- Raman Spectroscopy
- Laser Velocimetry
- Flow Visualization
- Phase Doppler Particle Analyzer
- Polarization Ratio
- Raman Spectroscopy

Closely Integrate CFD Objectives Into Experimental Program

- Experiment Design
- Measured Quantities
- Boundary Condition Specification
SUMMARY OF SURVEY RESULTS

• Square Geometry Is Acceptable
  Initially Axisymmetric Approximation Suitable Eventually 3-d Verification Required

• Chamber Mach Number Need Not Be Matched to Actual Rocket Conditions

• Recirculation Effect In Uni-Element Chamber Can Be Accommodated If Suitable Measurements Are Available

• Measurements And Boundary Conditions - Nearly Everything Is Important
  Typically All Parameters Rate 1 Or 2 On Scale Of 5

Responses From: MSFC
Aerojet
Rocketdyne
CFDRC
OPTICALLY ACCESSIBLE ROCKET

- Two Inch Square Cross-section
- Variable Length 6-12 Inches
- Two Inch Diameter Quartz Windows For Viewing
- Slot Windows For Laser Access
- Injector Design Is Modular
RESULTS

- Rocket Chamber Has Been Checked Out To 400 psi Chamber Pressure
- Window Purges Effectiveness Tested With Heat Flux Gages
- Quartz Windows Tested To 200 psi For Four Second Firings
- Shadowgraph Photograph Obtained
- 2-D Laser Induced Fluorescence Image Of OH Obtained
I and additional data systems

Data acquisition computer

Sequencer and abort monitor

Control panel

Instrumentation room

Laser system

Test cell

flowcontrol panel

Test stand and rocket
HEAT TRANSFER MEASUREMENTS
S. Pal, C. Merkle

Objective:

- Demonstrate Capability
- Test Effectiveness Of Window Purge
OBJECTIVES

- Characterize Spray Phenomena Associated With Impinging Jet Injectors
  - Breakup Length
  - Drop Size
  - Periodic Structures

- Investigate Sensitivity of Spray Characteristics to Geometric and Operational Parameters
  - Jet Velocity
  - Orifice Diameter
  - Impingement Angle
  - Pre-Impingement Length
  - Orifice Length

- Compare Experimental Results with Theoretical Models of Sheet Breakup and Drop Formation
  - Linear Growth of Surface Waves Due to Aerodynamic Forces
Impinging Jet Injector System

Impingement Effects
- Liquid Inertia
- Oscillations

Orifice Effects
- Vortex Shedding
- Cavitation
- Reattachment

Upstream Effects
- Turbulence
- Flow Patterns

Ambient Phase Effects
- Pressure
- Friction
- Oscillations

Re = $10^5 - 10^6$
ΔP = 5 - 30 atm
L/d = 1 - 5
$U_j = 6.4 \text{ m/s}$

$2\theta = 60^\circ$

$U_j = 18.5 \text{ m/s}$

$2\theta = 60^\circ$

$U_j = 6.4 \text{ m/s}$

$2\theta = 80^\circ$

$U_j = 18.5 \text{ m/s}$

$2\theta = 80^\circ$
NOTE:

$\bar{x}_b + x'_b$

$\bar{x}_b - x'_b$

$\bar{W} + W'$

$\bar{W}$

$\bar{W} - W'$

$D = 0.64 \text{ mm}$
$L/D = 80$
$l_j = 25 \text{ mm}$

Heidmann et al. Data ($2\theta = 90^\circ$)
ANALYTICAL RESULTS
(AT ORDER REGRESSION)

\( x = 16 \text{ mm} \)
\( y = 0 \text{ mm} \)
\( D = 0.64 \text{ mm} \)
\( L/D = 80 \)
\( l_j = 25 \text{ mm} \)

EXPERIMENTAL DATA

\( x = 41 \text{ mm} \)
\( y = 0 \text{ mm} \)
\( D = 0.64 \text{ mm} \)
\( L/D = 80 \)
\( l_j = 25 \text{ mm} \)
SUMMARY

- Experimental Results Were Compared with Predictions From Linear Aerodynamic Instability Derived Model
  - Breakup Length Predictions Oppose Observed Trends
  - Drop Size Predictions Reproduce Observed Trend and Agree Within Factor of 2

- General Appearance Favors Impact Wave Breakup Mechanism

- Orifice L/D Has No Significant Effect on Spray Characteristics

- Pre-Impingement Length Has Measurable Effect on Breakup Length and Drop Size
SUMMARY

- Successful Firings of Optically Accessible Rocket Test Chamber Achieved

- Shadowgraph of Combustion Zone Obtained

- Preliminary Two-Dimensional OH Imaging Results Obtained

- Wall Heat Transfer Rate Measurement Capability Demonstrated

- Fundamental Studies of Impinging Sprays Are Elucidating Basic Atomization Mechanisms
FUTURE WORK

- Semi-Quantitative Measurement of Relative OH Concentration

- Velocimetry (initially single point measurements, eventually 2-D velocimetry measurements)

- Raman Spectroscopy
ACKNOWLEDGEMENTS

The CFD Validation Studies Are Supported By The Marshall Space Flight Center Under Contract NAS8-38862

the Impinging Jet Studies Are Supported by The Air Force Office Of Scientific Research Under Grant AFOSR-91-0336
Experiments were conducted to define the nature of the aerodynamics and heat transfer for the flow within the disk cavities and blade attachments of a large-scale model, simulating the SSME turbopump drive turbines. These experiments of the aerodynamic driving mechanisms explored the following: (1) flow between the main gas path and the disk cavities, (2) coolant flow injected into the disk cavities, (3) coolant density, (4) leakage flows through the seal between blades, and (5) the role that each of these various flows has in determining the adiabatic recovery temperature at all of the critical locations within the cavities. The model and the test apparatus provide close geometrical and aerodynamic simulation of all the two-stage cavity flow regions for the SSME High Pressure Fuel Turbopump and the ability to simulate the sources and sinks for each cavity flow.

Carbon dioxide was used as a trace gas for constant density experiments or as the simulated "heavy gas" coolant. Gas samples were withdrawn at selected locations on the rotating and stationary surfaces in the fore and aft cavity and the interstage seal regions of the two stage system. The gas samples were used to determine the fraction of gas at a location which originates from each of three coolant injection locations or four gas path locations. Samples were also withdrawn at selected locations in the blade shank regions.

A parametric series of experiments was conducted with constant density fluids and an exploratory series of experiments was conducted with CO₂ as the simulated coolant. Experimental results showed (1) the variation of coolant distribution on the cavity and disk surfaces as a function of coolant flow ratio, (2) the effects on the coolant distribution for changes in the coolant inlet distributions, and (3) increased mixing of coolant with the ingested gas when a heavy gas (density ratio equal 1.5) was used as the coolant.
TURBINE DISK CAVITY AERODYNAMICS AND HEAT TRANSFER

Contract NAS8-37462

B.V. Johnson
W.A. Daniels

Tenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion

April 30, 1992
ACTUAL AND MODEL DISK/CAVITY SYSTEMS
MODEL INSTRUMENTATION

- Thermocouples
- Pressure/CO₂ taps in passages
- Pressure/CO₂ taps on rotating components
- Pressure/CO₂ taps on stationary components
MODEL SEAL REGION AND GAS SOURCE/EXIT LOCATIONS

REGION VI
Rotor 2 Blade Shanks

REGION V
Rotor 1 Blade Shanks

REGION IV
Aft Cavity & Rotor 2

REGION III
Center Cavity & Rotor 2

REGION II
Center Cavity & Rotor 1

REGION I
Forward Cavity & Rotor 1
COOLANT CONCENTRATION ON ROTOR AND STATIONARY WALLS

Variables: Radius
Coolant flow rate

Region IV: Aft Cavity & Rotor 2
Coolant: Air

Rotor Wall

Stationary Wall

Dimensionless coolant flow rate, $\frac{\dot{m}_c/2\pi\mu_a R_0}{(\rho_a \Omega R_0^2/\mu_a)^{0.8}}$
COOLANT CONCENTRATION ON ROTOR AND STATIONARY WALLS

Variables:
Radius
Coolant flow rate

Region IV: Aft Cavity & Rotor 2
Coolant: CO₂

Rotor Wall

Stationary Wall

\[ \phi_{14} \]

Dimensionless coolant flow rate, \[ \frac{m_c/2 \pi \mu_a R_0}{\rho_a \Omega R_0^2 \mu_a^{0.8}} \]

R/RO
0.411
0.620
0.706
0.815
0.903

R/RO
0.473
0.576
0.650
0.775
0.925
EFFECT OF COOLANT DENSITY ON DISTRIBUTION

Variables:
- Radius
- Coolant flow rate
- Coolant density

Region IV: Aft Cavity & Rotor 2

Rotors Wall Stationary Wall

1 - m

Coolant v

0.815 W

0.620

Coolant C02

v

Air

Dimensionless coolant flow rate, \( \frac{\dot{m}_c}{2\pi\mu a R_0}/(\rho a \Omega R_0^2/\mu a)^{0.8} \)
COOLANT DISTRIBUTION ON ROTOR

Region IV: Aft Cavity & Rotor 2
Coolant: Air

Local free disk entrainment flow rate

Dimensionless coolant flow rate, \( \frac{m_c}{2 \pi \mu \alpha R_0^2} \left( \frac{\rho \alpha \Omega R_0^2}{\mu \alpha} \right)^{0.8} \)
COOLANT DISTRIBUTION ON STATIONARY WALL

Region IV: Aft Cavity & Rotor 2
Coolant: Air

Dimensionless coolant flow rate, \( \left( \frac{\dot{m}_c}{2\pi\mu_a R_0} \right) / \left( \rho_a \Omega R_0^2 / \mu_a \right)^{0.8} \)
COOLANT DISTRIBUTION ON ROTOR

Region IV: Aft Cavity & Rotor 2
Coolant: CO₂

Local free disk entrainment flow rate

Dimensionless coolant flow rate, \( \frac{m_c}{2 \pi \mu_a R_o} \| \frac{\rho_a \Omega R_o^2}{\mu_a} \)
COOLANT DISTRIBUTION ON STATIONARY WALL

Region IV: Aft Cavity & Rotor 2
Coolant: CO₂

Dimensionless coolant flow rate, $\frac{m_c}{2\pi \mu a R_0^2} / (\frac{\rho_a \Omega R_0^2}{\mu_a})^{0.8}$

Local free disk entrainment flow rate
RESULTS/CONCLUSIONS

Constant Density

- Coolant flows approximately one-half free disk entrainment rate provide full purge of cavity ($\phi > 80\%$ below blade shanks)

- Coolant concentration on rotor surface high ($\phi > 90\%$) for coolant flows 1/4 design flow rate

- Cavity walls have largest variation of $\phi$ with coolant flow rate
Variable Density (Exploratory Experiments with CO₂)

- Density ratio has strong effect
  - Coolant concentration on rotor decreased from constant density results at comparable weight flow or volume flow rates.
  - Coolant concentration on aft cavity wall decrease significantly from constant density results at comparable flow rates.

- Decreased coolant concentration attributed to increased mixing and probable instability of rotating flow with higher gas densities at low radii.
A Numerical Study of Two-Dimensional Vortex Shedding From Rectangular Cylinders

A. H. Hadid, M. M. Sindir
CFD Technology Center
Rockwell International/Rocketdyne Division
Canoga Park, California
R. I. Issa
Department of Mineral Resources Engineering
Imperial College of Science
Technology and Medicine,
London, SW7, 2BP, England

An efficient time-marching, non iterative calculation method is used to analyze time-dependent flows around rectangular cylinders. The turbulent flow in the wake region of a square section cylinder is analyzed using an anisotropic k-ε model. Initiation and subsequent development of the vortex shedding phenomenon is naturally captured once a perturbation is introduced in the flow. Transient calculations using standard eddy-viscosity and an anisotropic k-ε models averaged over an integral number of cycles to get the fluctuating energy (organized and turbulent) are compared with experimental data. It is shown that the anisotropic k-ε model resolves the anisotropy of the Reynolds stresses and give mean energy distribution closer to the experiment than the standard k-ε model.
A NUMERICAL STUDY OF TWO-DIMENSIONAL VORTEX SHEDDING FROM RECTANGULAR CYLINDERS

A. H. HADID AND M. M. SINDIR

ROCKWELL INTERNATIONAL/ROCKETDYNE DIVISION
CANOGA PARK, CA 91303

10TH WORKSHOP FOR CFD APPLICATIONS IN ROCKET PROPULSION
APRIL 28 - 30, 1992
OVERVIEW

- AN EFFICIENT TIME-ACCURATE CALCULATIONAL METHOD IS USED TO PREDICT THE 2D TRANSIENT VORTEX SHEDDING MOTION BEHIND A SQUARE OBSTACLE
- THE SEPARATED TURBULENT FLOW BEHIND THE OBSTACLE IS MODELED USING AN ANISOTROPIC $k-\varepsilon$ TURBULENCE MODEL
- COMPARISONS WITH EXPERIMENTAL RESULTS SHOW REASONABLE AGREEMENT AND RIGHT TRENDS
INTRODUCTION

- VORTEX SHEDDING IS A PERIODIC UNSTEADY FLOW PHENOMENON OF PRACTICAL IMPORTANCE

- ATTEMPTS TO CALCULATE TWO-DIMENSIONAL VORTEX SHEDDING FLOW PAST SQUARE CYLINDERS WERE SUCCESSFUL AT LOW REYNOLDS NUMBERS WHERE FLOW IS LAMINAR AND THE PERIODIC FLUCTUATIONS ARE RESOLVED BY THE UNSTEADY N-S EQUATION

- AT HIGH REYNOLDS NUMBERS, TURBULENT FLUCTUATIONS ARE SUPERIMPOSED ON THE PERIODIC UNSTEADY MOTION WHICH MUST BE MODELED

- PREVIOUS ANALYSIS OF VORTEX SHEDDING FLOW AT HIGH REYNOLDS NUMBERS WERE NOT SUCCESSFUL DUE TO THE INADEQUACY OF THE STANDARD $k-\varepsilon$ MODELS TO ACCOUNT FOR THE ANISOTROPY OF THE TURBULENT INTENSITIES
INTRODUCTION (CONTD.)

- STANDARD $k$-$\varepsilon$ MODELS TEND TO DAMP PERIODIC SHELDDING MOTION UNDERPREDICTING THE STROUHAL NUMBER

- FOR A SUCCESSFUL SIMULATION OF TRANSIENT TURBULENT FLOWS A RELIABLE TIME ACCURATE NUMERICAL PROCEDURE AND A GOOD TURBULENCE MODELS ARE NEEDED

- AN EFFICIENT NON-ITERATIVE TIME ACCURATE NUMERICAL SCHEME BASED ON THE "PISO" METHODOLOGY IS EMPLOYED TO ANALYZE THE TRANSIENT VORTEX SHEDDING FLOW

- TURBULENCE IS MODELLED BY USING THE ANISOTROPIC $k$-$\varepsilon$ TURBULENCE MODEL
"PISO" NUMERICAL PROCEDURE

- THE INCOMPRESSIBLE TIME-DEPENDENT N-S EQUATIONS ARE IMPLICITLY DISCRETIZED ON A NON-STAGGERED GRID USING THE FINITE-VOLUME METHODOLOGY

  \[ u_p = H_p(u) + D^u P_\xi + E^u P_\eta + C^u_p \]

  \[ v_p = H_p(v) + D^v P_\xi + E^v P_\eta + C^v_p \]

- A PRESSURE CORRECTION EQUATION IS ASSEMBLED UTILIZING THE CONTINUITY EQUATION

- AT EACH TIME STEP, THE VELOCITY FIELD IS UPDATED ACCORDING TO

  \[ u_{p_{\lambda+2}} = H_p(u^\lambda) + D^u P_{\xi}^{\lambda+2} + E^u P_\eta^{\lambda+1} \]

  \[ v_{p_{\lambda+2}} = H_p(v^\lambda) + D^v P_{\xi}^{\lambda+1} + E^v P_\eta^{\lambda+2} \]

WHERE \( \lambda \) REPRESENTS THE CORRECTOR STAGE LEVEL
"PISO" NUMERICAL PROCEDURE (CONTD.)

- A MINIMUM OF TWO-CORRECTOR STAGES ARE NECESSARY FOR MANY PRACTICAL PURPOSES
- METHOD IS ESSENTIALLY NON-ITERATIVE WHERE THE SOLUTION PROCESS IS SPLIT INTO A SERIES OF STEPS WHEREBY OPERATIONS ON PRESSURE ARE DECOUPLED FROM THOSE ON VELOCITY AT EACH TIME STEP IN A SERIES OF CORRECTOR STEPS
MODEL EQUATIONS

- IN TRANSIENT PERIODIC FLOWS, THE INSTANTANEOUS VELOCITY \( (u_i) \)
  IS DECOMPOSED (TRIPLE DECOMPOSITION) INTO:

\[
u_i = U_i + u_i' = \overline{u_i} + \tilde{u}_i + u_i'\]

WHERE, \( U_i \) IS THE PHASE AVERAGED VELOCITY

\[
U_i (x_i, t) = \frac{1}{N} \sum_{n=0}^{N} u_i(x_i, t+nT)
\]

\( \overline{u_i} \) IS THE TIME MEAN VELOCITY COMPONENT
\( \tilde{u}_i \) IS THE PERIODIC ORGANIZED FLUCTUATING COMPONENT
\( u_i' \) IS THE RANDOM TURBULENT FLUCTUATING COMPONENT

- PHASE AVERAGED MOMENTUM EQUATIONS

\[
\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\nu \frac{\partial U_i}{\partial x_j} + R_{ij})
\]

WHERE \( R_{ij} = -\langle u_i u_j \rangle \) IS THE PHASE-AVERAGED REYNOLDS STRESS
TENSOR AND \( \nu \) IS THE KINEMATIC VISCOSITY
MODEL EQUATIONS (CONT'D.)

STANDARD $k$-$\varepsilon$ MODEL

- EDDY VISCOSITY CONCEPT

$$R_{ij} = -\frac{2}{3}k \delta_{ij} + \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$

WHERE $k$ IS THE PHASE AVERAGED TURBULENT KINETIC ENERGY
AND $\nu_t = C_\mu \frac{k^2}{\varepsilon}$, $\varepsilon$ IS THE PHASE AVERAGED ENERGY DISSIPATION RATE

- $k$ AND $\varepsilon$ ARE DETERMINED FROM

$$\frac{\partial k}{\partial t} + \vec{U} \cdot \nabla k = \frac{\partial}{\partial x_i} \left[ (\nu + \frac{\nu_t}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + G - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \vec{U} \cdot \nabla \varepsilon = \frac{\partial}{\partial x_i} \left[ (\nu + \frac{\nu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i} \right] + \frac{\varepsilon}{k} (C_1G - C_2\varepsilon)$$

WHERE $G = R_{ij} \frac{\partial U_i}{\partial x_j}$ IS THE TURBULENT GENERATION TERM
ANISOTROPIC $k$-$\varepsilon$ MODEL

MODEL EQUATIONS (CONT'D.)

- NONLINEAR CORRECTIONS ARE ADDED TO IMPROVE THE EDDY-VISCOSITY REPRESENTATION ON THE REYNOLDS-STRESSES AS DEVELOPED BY YOSHIZAWA USING "TSDIA" AND BY SPEZIALE

\[ R_{ij} = -\frac{2}{3} \kappa \delta_{ij} + \frac{2}{3} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) + \frac{1}{3} \left( \sum_{m=1}^{3} \sum_{n=1}^{3} \tau_{mn} \delta_{ij} \delta_{mn} \sum_{m=1}^{3} \tau_{mn} \delta_{ij} \right) \]

WHERE $\tau_{mn} = C_{mn} e^{2}$

\[ S_{ij} = \frac{\partial U_i}{\partial x_k} \frac{\partial U_j}{\partial x_k} \]

\[ S_{2ij} = \frac{1}{2} \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \]

\[ S_{3ij} = \frac{\partial U_k}{\partial x_i} \frac{\partial U_k}{\partial x_j} \]

$C_{mn}$ (m=1,2,3) ARE MODEL CONSTANTS

Rockwell International Rocketdyne Division

-6
MODEL EQUATIONS (CONT'D.)

- NISIZIMA USED THIS MODEL TO STUDY FULLY DEVELOPED TURBULENT SQUARE DUCT FLOW WHERE DEVIATIONS OF THE REYNOLDS STRESSES FROM ITS ISOTROPIC EDDY-VISCOSITY PLAYS A CENTRAL ROLE

- YOSHIZAWA SHOWED FOR THE SIMPLE ASYMMETRIC CHANNEL FLOW BETWEEN A ROUGH AND A SMOOTH WALLS THE NONCOINCIDENCE OF THE LOCATION OF ZERO SHEAR AND MAXIMUM VELOCITY

- MYONG AND KASAGI USED THE MODEL FOR FULLY DEVELOPED TURBULENT CHANNEL FLOW WITH CT1 AND CT2 OPTIMIZED TO REPRODUCE THE ANISOTROPY OF TURBULENT INTENSITIES

- IN THE PRESENT STUDY THE MODEL CONSTANTS CT1, CT2 AND CT3 WERE OPTIMIZED TO 0.01, 0.01 AND 0.001 RESPECTIVELY TO SATISFY THE REALIZABILITY CONSTRAINT

- WALL FUNCTIONS WERE USED TO BRIDGE THE NEAR OBSTACLE WALL REGION
RESULTS AND DISCUSSIONS

- EXPERIMENTAL RESULTS OF DURAO ET AL. FOR TURBULENT FLOW AROUND A SQUARE CYLINDER MOUNTED IN A WATER CHANNEL AT RE=14000 WERE USED TO COMPARE WITH THE COMPUTATIONS.

- IN THEIR MEASUREMENTS THEY USED SPECTRAL ANALYSIS AND DIGITAL FILTERING OF THE LDV DATA TO SEPARATE AND QUANTIFY THE TURBULENT AND NON TURBULENT PERIODIC MOTIONS.

- THEY MEASURED THE INSTANTANEOUS VALUES OF THE VELOCITY COMPONENTS AND EVALUATED THE TIME MEAN VALUES OF THE STRESSES AND KINETIC ENERGY.

- CALCULATIONS ARE PERFORMED FOR TURBULENT FLOW AROUND A SQUARE OBSTACLE (STEP HEIGHT H=20 mm) IN A DOMAIN EXTENDING ABOUT 16H DOWNSTREAM AND 2.5H UPSTREAM OF OBSTACLE.
RESULTS AND DISCUSSIONS (CONTD.)

- CALCULATIONS CAPTURED THE VORTEX SHEDDING PHENOMENON AFTER EXPLICITLY PERTURBING THE FLOW AT THE INLET

- COMPUTATIONAL DOMAIN WAS RESOLVED BY 75X40 CELLS WITH CLUSTERING AT THE OBSTACLE WALLS

- AN OPTIMIZED TIME STEP OF 0.001 sec. WAS CHOSEN IN THE CALCULATIONS
NORMAL VELOCITY HISTORY AT CENTERLINE 5 STEP HEIGHTS
DOWNSTREAM AT RE=14000
POWER SPECTRUM OF THE NORMAL VELOCITY FLUCTUATIONS

- FIGURE CONFIRMS OSCILLATORY NATURE OF THE FLOW WITH A SINGLE PREDOMINANT FREQUENCY AT ABOUT 4.7 Hz IN AGREEMENT WITH EXPERIMENT
STREAKLINE PLOT AT RE = 14000
RESULTS AND DISCUSSIONS (CONTD.)

- TO CALCULATE THE TIME-MEAN KINETIC ENERGY OF THE VELOCITY FLUCTUATIONS (ORGANIZED + TURBULENT), LET
  \[ \hat{u}_i = \tilde{u}_i + u'_i = u_i - \overline{u_i} \]

  THE TIME-MEAN KINETIC ENERGY CAN BE WRITTEN AS;

  \[ \overline{E} = \frac{3}{4} (\overline{u'_1^2} + \overline{u'_2^2}) \]

  WHERE \[ \overline{u'_i^2} = \overline{u_i^2} - \overline{u_i}'^2 + \overline{u_i''}^2 \] (i=1,2)

- THE FIRST TWO TERMS ON THE RIGHT HAND SIDE REPRESENT THE ORGANIZED PERIODIC ENERGY CONTRIBUTION

- THE LAST TERM REPRESENT THE TURBULENT ENERGY CONTRIBUTION
MEAN AXIAL VELOCITY ALONG THE CENTERLINE

Centreline Distribution of Mean Axial Velocity
- Durão et al. [8]
- Standard k-ε Model
- Anisotropic k-ε Model
MEAN KINETIC ENERGY ALONG THE CENTERLINE
MEAN AXIAL VELOCITY ALONG THE CENTERLINE

Graph showing the distribution of mean axial velocity with respect to $0\Omega/\Omega$. The graph includes data points and lines representing different models. The x-axis is labeled $0\Omega/\Omega$, and the y-axis is labeled as $X/H$.

Anisotropic $k-\varepsilon$ Model
Anisotropic $k-\varepsilon$ Model With Zero Production of $k$
Upstream of Obstacle

---

Rockwell International
Rocketdyne Division

1200
NORMAL STRESS CONTOURS

$\frac{\gamma v}{U_b^2}$ contours at T=3 sec., Using the Standard k-ε Model

$\frac{\gamma v}{U_b^2}$ contours at T=3 sec., Using the Anisotropic k-ε Model
DISCUSSION

- RESULTS OF MEAN KINETIC ENERGY SHOW THAT THE ANISOTROPIC k-ε MODEL PREDICTS A BETTER TREND THAN THE STANDARD ISOTROPIC MODEL.

- THE IMPROVED TREND IS MAINLY DUE TO AN INCREASE IN THE TURBULENT ENERGY CONTRIBUTION AS SHOWN FROM THE NORMAL STRESSES CONTOURS.

- LENGTH OF RECIRCULATION ZONE BEHIND OBSTACLE AND THE LOCATION OF THE MAXIMUM FLUCTUATING ENERGY ARE IMPROVED USING THE ANISOTROPIC MODEL.
MEASUREMENTS INDICATE THAT IN FRONT OF OBSTACLE THE FLOW REMAINED VIRTUALLY LAMINAR WITH NEGLIGIBLE FLUCTUATIONS IN THE k-e MODEL VELOCITY GRADIENTS ARE LARGE AT THE STAGNATION POINT PRODUCING EXCESSIVELY LARGE TURBULENT KINETIC ENERGY.

WHEN THIS UNREALISTIC PRODUCTION OF ENERGY IS SUPRESSED IN FRONT OF THE OBSTACLE, THE TURBULENT KINETIC ENERGY IS DECREASED EVERYWHERE.

HOWEVER, THE ENERGY OF THE PERIODIC FLUCTUATIONS IS INCREASED TO GIVE A NET INCREASE IN THE ENERGY OF THE TOTAL FLUCTUATIONS.

THERE STILL SEEMS TO BE SOME FINITE KINETIC ENERGY IN FRONT OF THE OBSTACLE WHICH CAN ONLY BE DUE TO THE PERIODIC FLUCTUATIONS WHICH IS NOT INDICATED IN THE EXPERIMENTAL RESULTS.
A Status of the activities of the Turbine Technology Team of the Consortium for Computational Fluid Dynamics (CFD) Application in Propulsion Technology is presented. The team consists of members from the government, industry, and universities. The goal of this team is to demonstrate the benefits to the turbine design process attainable through the application of CFD. This goal is to be achieved by enhancing and validating turbine design tools for improved loading and flowfield definition and loss prediction, and transferring the advanced technology to the turbine design process.

In order to demonstrate the advantages of using CFD early in the design phase, the Space Transportation Main Engine (STME) turbines for the National Launch System (NLS) were chosen on which to focus the team's efforts. The Turbine Team activities run parallel to the STME design work.

Work during the past year has centered on transferring technology obtained through the team's Generic Gas Generator Turbine Program (reported on in the 1990 workshop) to the STME LOX turbine design point. A preliminary baseline design was analyzed through CFD, and areas requiring refinement to eliminate local overspeeds and separations were found. An improved baseline design has been finalized. The team is currently comparing results from five team members (Aerojet, NASA/Ames Research Center, NASA/Lewis Research Center, Pratt & Whitney, and Scientific Research Associates). These solutions will be compared to data to be obtained in the Marshall Space Flight Center Turbine Airflow Facility. Interrogation of these solutions are also in progress to determine high loss locations and to provide guidance for developing and/or implementing concepts to control these losses.
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

LISA GRIFFIN
MAY 1, 1992
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Overview

- Structure/Objectives

- Turbine Team Participants

- Program Overview
  - Code Development/Enhancement
  - Validation Experiments
  - Advanced Hardware Development

- Subsonic Turbine Development Program
  - Background
  - Gas Generator Oxidizer Turbine (GGOT)

- New Programs
  - Volute Development
  - Supersonic Turbine Development

- Summary
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Structure/Objectives

The consortium for CFD application in propulsion technology

Objectives:
- Development/enhancement of CFD models and codes
- Validation of state-of-the-art CFD codes
- Application of CFD in design of advanced hardware concepts
- Verification testing of advanced hardware concepts

Turbine Team

Objectives:
- Coordinate/focus MSFC turbine technology activities
- Provide a forum for interaction/technology transfer
- Provide peer review for turbine technology activities
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Participants

- NASA Marshall Space Flight Center (MSFC)
- NASA Ames Research Center (ARC)
- NASA Lewis Research Center (LeRC)
- Aerojet
- Pratt and Whitney (P&W)
- Rocketdyne (RKDN)
- Calspan - University of Buffalo Research Center (CUBRC)
- Rotodata
- Scientific Research Associates (SRA)
- SECA
- United Technologies Research Center (UTRC)
- Carnegie Mellon University (CMU)
- Pennsylvania State University (PSU)
- The University of Alabama (UA)
- The University of Alabama in Huntsville (UAH)
- Virginia Polytechnic Institute (VPI)
Program Overview

- Code Development/Enhancement
  - Rotor/Stator Interaction (NASA/ARC)
  - 3D Navier-Stokes Code for Volutes (NASA/LeRC)
  - Validation of 3D Unsteady Rotor/Stator Interaction Code ROTOR3 (P&W)
  - Enhancement and Validation of ROTOR3 for Supersonic Turbines (RKDN)
  - Turbulence Modeling (PSU)

- Validation Experiments
  - 3D Rotor Heat Transfer (UTRC)
  - Unsteady Interrow Aerodynamics (P&W)
  - SSME HPFTP Fuel Turbine
    -- Baseline Aerodynamics (MSFC)
    -- Baseline Unsteady Aerodynamics/Heat Transfer (CUBRC)
    -- Smooth Blades (MSFC)
    -- Circumferential Exit ΔP (MSFC)
    -- ATD (MSFC)
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Program Overview

- Advanced Hardware Development
  - Subsonic Turbine Development
  - Volute Development
  - Supersonic Turbine Development
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Subsonic Turbine Development Program
Background

- Focused flow analysis tools on the STME fuel turbine design point resulting in Generic Gas Generator (G³ T) design
  - Advanced Turbine concept developed
    - Reduced parts
    - Increased efficiency
  - Analyses in overlapping regions with different codes showed consistent results
  - Axial gap selection modified
    - Improved efficiency (1 pt.)
    - Reduced blade loadings/stresses

- Design concept incorporated into STME LOX turbine

- Focused flow analysis tools on the STME LOX turbine design point resulting in Gas Generator Oxidizer Turbine (GGOT)
**Background**

**Aerodynamic Design Approach**

<table>
<thead>
<tr>
<th>General Description</th>
<th>Previous State-of-the-Art</th>
<th>G³ T Design</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>70/30 Work Split</td>
<td>50/50 Work Split</td>
</tr>
<tr>
<td></td>
<td>Nominal Annulus Height</td>
<td>Increased Annulus Height</td>
</tr>
<tr>
<td>Blade Turning</td>
<td>135°</td>
<td>160°</td>
</tr>
<tr>
<td>Fluid Acceleration</td>
<td>0.9</td>
<td>1.6</td>
</tr>
<tr>
<td>Max Blade Mach Number</td>
<td>1.32</td>
<td>0.87</td>
</tr>
<tr>
<td>Efficiency</td>
<td>Base</td>
<td>+10 percent</td>
</tr>
<tr>
<td>Airfoil Count</td>
<td>Base</td>
<td>-55 percent</td>
</tr>
</tbody>
</table>
BASELINE GGGT AERODYNAMIC DESIGN

Calculated Streamlines - Midspan

1st Vane

1st Blade

2nd Vane

2nd Blade
CONSORTIUM FOR CFD APPLICATION IN PROPULSION TECHNOLOGY

Turbine Stage Technology Team-Baseline GGGT aerodynamic analyses

LeRC
Inverse design code

SRA
3D N/S, tip lkg.

MSFC
3D N/S

UTRC
2D N/S

ARC
Unsteady 2D N/S

P&W
Multi-stage 3D Euler

AVB344072 902711
ANALYSIS OF A NEW TURBINE DESIGN

Instantaneous Temperature Contours

Preliminary

gap = 0.20 inch

Baseline

gap = 0.35 inch
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Subsonic Turbine Development Program
GGOT

• Phase I Task Description
  - Development of preliminary baseline design (P&W)
  - Analyses of preliminary design (P&W, NASA/ARC, and NASA/LeRC)
  - Modification of preliminary design (P&W)
  - Analysis of modified design (P&W, NASA/ARC, NASA/LeRC, SRA, and Aerojet)
  - Development of final baseline design (P&W)
  - Comparisons of CFD solutions
  - Experimental evaluation of baseline design (MSFC)

• Phase II Task Description
  - Interrogate baseline results for regions of losses
  - Development of concepts to control losses
    -- Parametric studies with CFD
  - Development of advanced design
  - Detailed analyses of advanced design
  - Experimental evaluation of advanced design
CONSORTIUM FOR CFD APPLICATION IN PROPULSION TECHNOLOGY

Turbine Technology Team - Baseline GGOT Aerodynamic Analyses

- Pratt & Whitney - Steady 3D Euler
- ARC - Unsteady 2D Navier/Stokes
- LeRC - Steady 3D Navier/Stokes
- SRA - Steady 3D N/S, Tip Lkg
- Aerojet - Steady 3D N/S, Tip Lkg
GAS GENERATOR OXIDIZER TURBINE (GGOT) DESIGN

Streaklines Downstream of Rotor

Preliminary

3D Flow Analysis Shows Separation Downstream of Rotor

Baseline

3D Flow Analysis Shows Downstream Separation Eliminated
OXIDIZER TECHNOLOGY TURBINE BASELINE DESIGN

Concerns / Potential Weaknesses

Vane Losses

- Secondary
- Profile

Addressed in Baseline Design Philosophy

Blade Losses

- Secondary
- Profile
- Leakage
- Tip Leakage Control Studies
- 3D Navier-Stokes Analyses, Steady & Unsteady

Addressed in Baseline Design Philosophy
GGOT ROTOR - PRESSURE LOSS COEFFICIENT CONTOURS

15% AXIAL CHORD DOWNSTREAM OF ROTOR

NO TIP LEAKAGE

WITH TIP LEAKAGE

\[
C_{pt} = \frac{P_{t_r \text{ (max inlet)}} - P_{t_r}}{\frac{1}{2} \rho_0 V_o^2}
\]
GGOT ROTOR - PRESSURE LOSS COEFFICIENT CONTOURS

**99% AXIAL CHORD DOWNSTREAM OF ROTOR**

<table>
<thead>
<tr>
<th>Cpt</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.37E+00</td>
</tr>
<tr>
<td>2.26E+00</td>
</tr>
<tr>
<td>2.15E+00</td>
</tr>
<tr>
<td>2.05E+00</td>
</tr>
<tr>
<td>1.94E+00</td>
</tr>
<tr>
<td>1.84E+00</td>
</tr>
<tr>
<td>1.73E+00</td>
</tr>
<tr>
<td>1.63E+00</td>
</tr>
<tr>
<td>1.52E+00</td>
</tr>
<tr>
<td>1.42E+00</td>
</tr>
<tr>
<td>1.32E+00</td>
</tr>
<tr>
<td>1.21E+00</td>
</tr>
<tr>
<td>1.10E+00</td>
</tr>
<tr>
<td>1.00E-01</td>
</tr>
<tr>
<td>9.00E-02</td>
</tr>
<tr>
<td>7.85E-01</td>
</tr>
<tr>
<td>6.20E-01</td>
</tr>
<tr>
<td>4.71E-01</td>
</tr>
<tr>
<td>3.65E-01</td>
</tr>
<tr>
<td>2.60E-01</td>
</tr>
</tbody>
</table>

**NO TIP LEAKAGE**

**WITH TIP LEAKAGE**

\[
Cpt = \frac{P_{t} (\text{max inlet}) - P_{t}}{1/2 \rho_o V_o^2}
\]
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

New Program - Volute Development

• Issues
  - Volumes traditionally designed using existing database and limited analysis
  - STME LOX turbine volumes out of experience base
    -- High Mach number
    -- High swirl (discharge diffuser/volute)
    -- Vaneless diffuser design considered for cost and weight
  - Benchmark quality volute dataset non-existent

• Task Descriptions
  - Design of three baseline volutes with existing design methodology
  - Analysis of baseline volutes for design and off-design conditions
  - Experimental evaluation of baseline volutes on GGOT for code validation
  - Development of loss control and side load reduction concepts
    -- Parametric studies with CFD
    -- Detailed analysis of improved designs
    -- Experimental evaluation of improved volutes on advanced concepts GGOT
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

New Program - Supersonic Turbine Development

- Issues
  - High losses and blade excitations found in supersonic turbines
    -- Durability requirements require special attention paid to unsteady loadings and shock interactions
    -- Low confidence in the understanding of unsteady loadings
  - Supersonic turbine database from a rotating environment virtually non-existent

- Task Descriptions
  - Design baseline turbine with design requirements consistent with STME fuel turbine
  - Analysis of baseline STME fuel turbine
  - Experimental evaluation of baseline design for design verification, code validation, and database expansion
  - Development of advanced concepts
    -- Parametric studies of candidate concepts to control losses and interactions
    -- Detailed analyses of advanced design
    -- Experimental evaluation of advanced design
A Summary of the Activities of the NASA/MSFC Turbine Technology Team

Summary

- Team of government, industry, and university experts in place and focused on relevant turbine design and analysis issues

- Codes and models are being developed and enhanced to address complex flows in turbines

- Unique experimental data is being taken to provide information for thorough calibration/validation of turbine analysis tools

- Team activities have been coordinated and focused on demonstration of CFD tools for turbine design
Three-dimensional flow phenomena in a supersonic turbine blade row have been studied numerically to evaluate CFD as a tool to design supersonic turbine stages.

The details of the three-dimensional flow structure inside the supersonic turbine blade row and the overall aerodynamic performance at design and off-design conditions are analyzed and the results are compared between the experimental data and the numerical results.
A CRITICAL EVALUATION OF A THREE-DIMENSIONAL NAVIER-STOKES CFD AS A TOOL TO DESIGN SUPersonic TURBine STAGEs

C. HAH, O. KWON, AND M. SHOEMAKER
NASA LEWIS RESEARCH CENTER
21000 BROOKPARK ROAD, CLEVELAND, OHIO
SUPERSONIC TURBINE STAGE

COMMONLY USED IN SPACE ENGINES

DESIGN IS BASED ON 2-D METHOD

VERY LOW EFFICIENCY
FLOWS INSIDE SUPersonic TURBINE

Mach. Number

2.51

0.00
OBJECTIVES

UNDERSTAND DETAILED FLOW STRUCTURE

INVESTIGATE LOSS MECHANISM

DEVELOP 3-D DESIGN METHOD
FLOWS INSIDE SUPersonic TURBINE

UNSTARTED FLOW

STARTED FLOW
OVERALL APPROACHES

APPLY 3-D NAVIER-STOKES CODE

VERIFY FOR STARTING MACH #, FLOW SEPARATION

EVALUATE CURRENT CAPABILITY
COMPARISON FOR UNSTARTED FLOW
COMPARISON FOR STARTED FLOW
SHOCK-INDUCED VORTEX

MERIDIONAL VELOCITY VECTORS
CONCLUSIONS

3-D VISCOUS CFD CAPTURES MOST FLOW STRUCTURE
CAN BE USED FOR DESIGN APPLICATION
NEEDS HIGH QUALITY DATA FOR VERIFICATION
ABSTRACT

The Gas Generator Oxidizer Turbine (GGOT) Blade is being analyzed by various investigators under the NASA MSFC sponsored Turbine Stage Technology Team design effort. The present work concentrates on the tip clearance region flow and associated losses; however, flow details for the passage region are also obtained in the simulations. The present calculations simulate the rotor blade row in a rotating reference frame with the appropriate coriolis and centrifugal acceleration terms included in the momentum equations. The upstream computational boundary is located about one axial chord from the blade leading edge. The boundary conditions at this location have been determined by Pratt & Whitney using an Euler analysis without the vanes to obtain approximately the same flow profiles at the rotor as were obtained with the Euler stage analysis including the vanes. Inflow boundary layer profiles are then constructed assuming the skin friction coefficient at both the hub and the casing. The downstream computational boundary is located about one axial chord from the blade trailing edge, and the circumferentially averaged static pressure at this location was also obtained from the P&W Euler analysis.

Results have been obtained for the 3-D baseline GGOT geometry at the full scale design Reynolds number. Details of the clearance region flow behavior and blade pressure distributions have been computed. The spanwise variation in blade loading distributions are shown, and circumferentially averaged spanwise distributions of total pressure, total temperature, Mach number and flow angle are shown at several axial stations. The spanwise variation of relative total pressure loss shows a region of high loss in the region near the casing. Particle traces in the near tip region show vortical behavior of the fluid which passes through the clearance region and exits at the downstream edge of the gap. Future work will include collaboration with the P&W design team to identify design changes which may reduce clearance flow losses.

† Work supported under NASA MSFC Contract NAS8-3865.
* Currently at United Technologies Research Center, East Hartford, CT.
NAVIER-STOKES ANALYSIS OF OXIDIZER TURBINE BLADE (GGOT) WITH TIP LEAKAGE

WORKSHOP FOR CFD APPLICATIONS IN ROCKET PROPULSION
APRIL 30, 1992

HOWARD J. GIBELING
JAYANT S. SABNIS

SCIENTIFIC RESEARCH ASSOCIATES, INC.
GLASTONBURY, CT 06033
SUPPORTED BY

NASA MARSHALL SPACE FLIGHT CENTER

CONTRACT NAS8-38865

Scientific Research Associates
OBJECTIVE

APPLY CFD TECHNOLOGY TO ADVANCE TURBINE DESIGN FOR SPACE PROPULSION APPLICATIONS.
GGOT SIMULATION SUMMARY

- UTILIZE SRA GMINT CODE
- GENERAL NON-RECTANGULAR BLOCK STRUCTURE
- SINGLE GRID
- FULL NAVIER-STOKES EQUATIONS
- NO-SLIP WALL BOUNDARY EQUATIONS WITH SUBLAYER RESOLUTION
- IMPLICIT LINEARIZED BLOCK SOLVER (ADI)
GGOT SIMULATION SUMMARY (Continued)

- GRID GENERATION
  - "FALSE CORNER" GRID STRUCTURE
  - 2-D ELLIPTIC GRIDS GENERATED WITH EAGLE
    100 x 160 POINTS IN CROSS-SECTIONAL PLANE
  - 3-D GRID CONSTRUCTION
    21 BLADE CROSS-SECTIONAL PLANES
    REDISTRIBUTION IN SPANWISE DIRECTION
    40 POINTS FROM HUB TO TIP
    25 POINTS IN CLEARANCE REGION
GGOT SIMULATION FLOW PARAMETERS

- SUPPLIED BY P&W DESIGN TEAM
- CIRCUMFERENTIALLY - AVERAGED SPANWISE DISTRIBUTIONS FROM EULER CODE
  - UPSTREAM AXIAL MASS FLUX
  - UPSTREAM TOTAL TEMPERATURE
  - UPSTREAM FLOW ANGLES
  - DOWNSTREAM STATIC PRESSURE
- HUB AND CASING ENDWALL BOUNDARY LAYER PROFILES CONSTRUCTED WITH ASSUMED $C_f = 0.002$
OXIDIZER TURBINE BASELINE DESIGN
FULL SCALE TURBINE FLOWPATH CLOSE-UP

1. INFLOW STATION @ x = 0.6"
2. OUTFLOW STATION @ x = 3.8"

VANE (20)
BLADE (42)

AXIAL LOCATION

11.437R
11.072R
11.087R

30°
BASELINE GGOT HUB SECTION GRID
(100 x 60)
BASELINE GGOT WITH TIP CLEARANCE
MACH NUMBER CONTOURS AT 9.2% SPAN
BASELINE GGOT WITH TIP CLEARANCE
MACH NUMBER CONTOURS AT 53.5% SPAN
BASELINE GGOT WITH TIP CLEARANCE
PRESSURE CONTOURS AT 9.2% SPAN
BASELINE GGOT WITH TIP CLEARANCE PRESSURE CONTOURS AT 90% SPAN
Baseline GGOT with Clearance Blade Surface Pressure (9.2% span)
Baseline GGOT with Clearance
Blade Surface Pressure (53.5% span)
Baseline GGOT with Clearance
Blade Surface Pressure (90% span)
Circumferentially Averaged Variables
Total Pressure Loss Across Blade

Relative Mach Number

2.9 inches
1.4 inches

Relative Total Pressure Loss

Scientific Research Associates

Percent Blade Span

1264
PARTICLE TRACES

BASELINE GGT WITH TIP CLEARANCE
TRACES ORIGINATING AT 100% SPAN - I-FLOW VIEW

0.005
1.37 DEG
2.63 x 10**6
1.46 x 10**3
100 x 160 x 40

MACH
ALPHA
Re
TIME
GRID

Scientific Research Associates
CONCLUSIONS AND FUTURE WORK

- CLEARANCE FLOW DETAILS PREDICTED
- BLADE PRESSURE DISTRIBUTIONS - BLOCKAGE EFFECTS
- SPANWISE VARIATION OF TOTAL PRESSURE LOSS
- FURTHER ANALYSIS OF DATA NECESSARY
- TIP CLEARANCE FLOW BEHAVIOR
- SECONDARY FLOW BEHAVIOR
- COORDINATION WITH PRATT AND WHITNEY DESIGN EFFORT
NUMERICAL SIMULATION OF TURBOMACHINERY FLOWS WITH ADVANCED TURBULENCE MODELS

B. Lakshminarayana, R. Kunz, J. Luo, S. Fan
The Pennsylvania State University

A three dimensional full Navier-Stokes (FNS) code is used to simulate complex turbomachinery flows. The code incorporates an explicit multistep scheme and solves a conservative form of the density averaged continuity, momentum and energy equations in body fitted coordinates. Rotation terms are included for the computation of rotor flows. A compressible low-Reynolds number form of the $k$-$\varepsilon$ turbulence model, and a $q$-$\omega$ model and an algebraic Reynolds stress model have been incorporated in a fully coupled manner to approximate Reynolds stresses.

The code is used to predict viscous flow field in a backswept transonic centrifugal compressor for which laser two focus data is available. The tip clearance flow, and curvature and rotation induced secondary flows are captured to good accuracy. Solutions which incorporate Reynolds stress model show significant, though not dramatic, differences in predicted secondary flows, wall shear stress and performance parameters, when compared to the $k$-$\varepsilon$ solution.

The code is also utilized to simulate the tip clearance flow in a cascade. An embedded H-grid topology was utilized to resolve the flow physics in the gap. The data and predictions were performed with and without tip clearance, endwall, wall motion. Additionally, both Euler and Navier-Stokes computations were performed. The results indicate that the Navier-Stokes code captures the flow physics accurately, including tip vortex strength, trajectory, loading and interaction of tip clearance flow with the secondary flow.

The code has also been used to predict the two dimensional viscous and thermal flow field in a transonic turbine nozzle with $75^\circ$ turning, $M_1 = 0.15$, $M_2 = 0.7$ to 1.11, $Re = 0.5 - 2 \times 10^6$, and $T_o = 415^\circ$K. Good agreement is obtained for pressure distribution, wake and surface heat transfer.

The code has been extended to include unsteady Euler solution for predicting the unsteady flow through a cascade due to incoming wakes, simulating rotor-stator interaction. Unsteady characteristic boundary conditions are specified at inlet and exit. The predicted unsteady surface pressure distribution, unsteady lift and moment, pressure wave propagation in a flat plate due to an incoming gust compares well with the analytical theories and earlier computations. This code will be integrated with a boundary layer code to capture the unsteady viscous flow and heat transfer.
NUMERICAL SIMULATION OF TURBOMACHINERY FLOWS WITH ADVANCED TURBULENCE MODELS

B. Lakshminarayana, R. Kunz, J. Luo, S. Fan
The Pennsylvania State University

1. Introduction
2. Technique and Turbulence Models
3. Computation of Transonic Centrifugal Compressor Flow Field
4. Computation of Tip Clearance Flows in Cascades
5. Computation of Aerothermal Field in a Transonic Nozzle
6. Simulation of Rotor/Stator Interaction
7. Conclusions
GOVERNING EQUATIONS
(Cartesian)

\[ \frac{\partial Q}{\partial t} = -\left( \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} \right) + \left( \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} + \frac{\partial G_v}{\partial z} \right) + S \]

\[
Q = \begin{pmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
\rho e_0 \\
\rho k
\end{pmatrix}, \quad
E = \begin{pmatrix}
\rho u \\
\rho u u + p \\
\rho u v \\
(\rho e_0 + p) u \\
\rho u e
\end{pmatrix}, \quad
F = \begin{pmatrix}
\rho v \\
\rho v u \\
\rho v v + p \\
(\rho e_0 + p)v \\
\rho v e
\end{pmatrix}, \quad
G = \begin{pmatrix}
\rho w \\
\rho u w \\
\rho v w \\
\rho w w + p \\
\rho w e
\end{pmatrix},
\]

\[
E_v = \begin{pmatrix}
0 \\
\tau_{xx} \\
\tau_{xy} \\
\tau_{xz} \\
\frac{\mu_1}{\frac{\partial k}{\partial x}} + \frac{\mu_1}{Pr_k \frac{\partial k}{\partial x}}
\end{pmatrix}, \quad
F_v = \begin{pmatrix}
0 \\
\tau_{yx} \\
\tau_{yy} \\
\tau_{yz} \\
\frac{\mu_1}{Pr_k \frac{\partial e}{\partial y}} + \frac{\mu_1}{Pr_e \frac{\partial e}{\partial y}}
\end{pmatrix}, \quad
G_v = \begin{pmatrix}
0 \\
\tau_{zx} \\
\tau_{zy} \\
\tau_{zz} \\
\frac{\mu_1}{Pr_k \frac{\partial e}{\partial z}} + \frac{\mu_1}{Pr_e \frac{\partial e}{\partial z}}
\end{pmatrix}, \quad
S = \begin{pmatrix}
\rho(\omega^2 y + 2\omega w) \\
0 \\
\rho(\omega^2 z - 2\omega v) \\
0 \\
C_1 P - C_2 \rho e + D + \mathcal{E}
\end{pmatrix}
\]

- Relative velocities, constant rotation rate about x axis, \( \omega \). Averaged quantities.
- Energy, \( e_0 = \varepsilon + \frac{q^2}{2} - \frac{\omega^2}{2} \), rothalpy constant along streamlines for inviscid steady state.
Governing Equations - Turbulence Model

- Density-averaged low-Re number k-ε, conservative form:

\[
\frac{\partial \tilde{Q}}{\partial \bar{t}} + \left( \frac{\partial \tilde{E}_c}{\partial \bar{\xi}} + \frac{\partial \tilde{F}_c}{\partial \bar{\eta}} \right) = \left( \frac{\partial \tilde{E}_v}{\partial \bar{\xi}} + \frac{\partial \tilde{F}_v}{\partial \bar{\eta}} \right) + \tilde{S}
\]

\[
\tilde{k} = 2 \frac{\bar{\rho} \bar{u}_i \bar{u}_i}{\bar{\rho}}, \quad \tilde{\varepsilon} = \frac{v_p \partial \bar{u}_i}{\partial \bar{x}_j} \frac{\partial \bar{u}_i}{\partial \bar{x}_j}, \quad \mu_t = \frac{C_{u_k} \bar{\rho} \tilde{k}^2}{\tilde{\varepsilon}}.
\]

\[
\tilde{Q} = \frac{1}{J} \left( \frac{\bar{\rho} \tilde{k}}{\bar{\rho} \varepsilon} \right), \quad \tilde{E}_c = \frac{1}{J} \left( \frac{\bar{\rho} \tilde{k} U}{\bar{\rho} \varepsilon U} \right), \quad \tilde{F}_c = \frac{1}{J} \left( \frac{\bar{\rho} \tilde{k} V}{\bar{\rho} \varepsilon V} \right), \quad \tilde{S} = \frac{1}{J} \left( \frac{P - \bar{\rho} \varepsilon + D}{C_{f1} \rho - C_{f2} \bar{\rho} \varepsilon \tilde{k} + \varepsilon} \right).
\]

\[
\tilde{E}_v = \frac{1}{J} \left[ \left( \mu_t + \frac{\mu_t}{Pr_k} \right) \left( \nabla \bar{\xi} \cdot \nabla \bar{\eta} \frac{\partial \tilde{k}}{\partial \bar{\eta}} + (\nabla \bar{\xi} \cdot \nabla \bar{\eta}) \frac{\partial \tilde{k}}{\partial \bar{\eta}} \right) \right], \quad \tilde{F}_v = \frac{1}{J} \left[ \left( \mu_t + \frac{\mu_t}{Pr_k} \right) \left( \nabla \bar{\xi} \cdot \nabla \bar{\eta} \frac{\partial \tilde{e}}{\partial \bar{\eta}} + (\nabla \bar{\xi} \cdot \nabla \bar{\eta}) \frac{\partial \tilde{e}}{\partial \bar{\eta}} \right) \right].
\]
Coakley's q-ω model

- \( q = \sqrt{k}, \ \omega = \varepsilon/k \)
- \( \mu_t = c_\mu f_\mu \rho q^2/\omega \)
  \( c_\mu = 0.09 \)
  \( f_\mu = 1 - \exp(-\alpha R) \)
  \( \alpha = 0.02 \)
  \( R = \frac{qy}{\nu} \)

- q-equation

\[
\frac{\partial (pq)}{\partial t} + \frac{\partial (pq u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t/P_{q} \right) \frac{\partial q}{\partial x_j} \right]
\]

\[
= \frac{1}{2} \left( c_\mu f_\mu \frac{S}{\omega^2} - 2 \frac{f_\mu}{3 \omega} - 1 \right) \rho \omega q
\]

\[
S = \frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} - 2 \frac{\partial u_k}{\partial x_j} \frac{\partial u_k}{\partial x_j} - 2 \frac{\partial u_k}{\partial x_k} \frac{\partial u_k}{\partial x_k}
\]

\( P_{q} = 2.0 \)
The above equations are in Favre-averaged form.
In the fully turbulent region \([Y^+ = 0(10^2)]\), a compressible extension to the high Reynolds number form of the ARSM due to Galmes and Lakshminarayana (1984) is adopted:

\[
-\rho u_i''u_j'' = -\frac{2}{3} \delta_{ij} \rho k - \rho k T_{ij}
\]

where

\[
T_{ij} = \frac{R_{ij}(2-C_2)/2 + (P_{ij} - 2P\delta_{ij}/3)(1-C_2)}{P + \rho \epsilon(C_1-1)},
\]

\[
R_{uk} = -2\omega_p (e_{ipj}u_k''u_j'' - e_{kpj}u_i''u_j''),
\]

\[
P_{ij} = (-\rho u_i''u_k'' \frac{\partial u_j}{\partial x_k} - \rho u_j''u_k'' \frac{\partial u_i}{\partial x_k}), \quad 2P = P_{ii}
\]

where \(C_1 = 1.5, \ C_2 = 0.6\).
In the fully turbulent region \([Y^+ = 0(10^2)]\), a compressible extension to the high Reynolds number form of the ARSM due to Galmes and Lakshminarayana (1984) is adopted:

\[
-\rho u_i'' u_j'' = - \frac{2}{3} \delta_{ij} \rho k - \rho k T_{ij}
\]

where

\[
T_{ij} = \frac{R_{ij}(2-C_2)/2 + (P_{ij} - 2P\delta_{ij}/3)(1-C_2)}{P + \rho e(C_1-1)},
\]

\[
R_{ik} = -2 \omega_p \left( e_{ij} \rho u_k'' u_j'' - e_{kj} \rho u'_i u_j'' \right)
\]

\[
P_{ij} = (-\rho u_i'' u_k'' \frac{\partial u_j}{\partial x_k} - \rho u_j'' u_k'' \frac{\partial u_i}{\partial x_k}), \quad 2P = P_{ii}
\]

where \(C_1 = 1.5\), \(C_2 = 0.6\).
RKCC Code

Explicit Multistage, Conservative, Compressible Formulation

Generalized Coordinates; 2-D, 3-D, Unsteady; --> Turbomachinery (Rotation Periodic B.C.)

Coupled Compressible Low-Reynolds-Number K-ω Model, q-ω Model, ARSM Model

FNS + Full Turbulent KE Production Term

IRS for 2-D

Finite Diff (Flux Evaluation = Cell Vertex Finite Volume)

Characteristic B.C.'s, Embedded H Mesh, Tip Clearance Topologies for Turbomachinery Application

Eigenvalue & Local Velocity Scaling of Artificial Dissipation
\[ M = 4.0 \text{ kg/s} \]
\[ M_{\text{rel}} = 0.8 \]
\[ N = 22,363 \text{ RPM} \]
\[ P_{\text{in}} / P_{\text{out}} = 4 \]

**Figure 1**  a) Meridional view of 59 x 27 x 27 computational grid for Krain impeller computation. Streamwise grid indexing and location of experimental laser planes are indicated for reference. b) Cross stream views of 59 x 27 x 27 computational grid for Krain impeller computation.
Figure 2. L2F data acquisition Plane IV. a) Computed normalized relative helicity contours. b) Meridional velocity profiles. Experimental measurements (symbols). c) Meridional velocity contours. Experimental contours (top), computed contours (bottom).
Figure 3 Qualitative representation of physical phenomena contributing to the formation of the wake flow region. a) Representative sketch adapted from Eckardt (1976) and b) computed normalized relative helicity at 68% chord.
Figure 4 Meridional velocity profiles at L2F data acquisition Plane IV. Comparison of solutions using three turbulence models: low Reynolds number $k-\epsilon$ model (solid), $k-\epsilon$/ARS model, $R_{ij} = 0$ (long dash), $k-\epsilon$/ARS model, $R_{ij} \neq 0$ (short dash).
Figure 5 Near wall relative radial and crossflow velocity profiles at 90% chord. a) midspan on suction surface, b) midspan on pressure surface, c) mid passage on hub.
Comparison of solutions using three turbulence models: low Reynolds number k-ε model (square), k-ε/ARS model, $R_{ij} = 0$ (circle), k-ε/ARS model, $R_{ij} \neq 0$ (triangle). $W_3$ is the radial component (+ towards tip), $W_2$ is the relative circumferential component (+ towards pressure surface)
Figure 6. Schematic of experimental configurations.
(Note: $z = 0$ is located .04 chord inboard of the cascade symmetry plane [k=nk]. For the cases with clearance, this corresponds to the blade tip [$t/c = .04$]).

Figure 7. Views of computational nomenclature a) Detail of leading edge.
three-dimensional grid.
Figure 8. Comparison of computed and measured blade static pressure coefficient at three spanwise locations. Measured values (symbols), Navier-Stokes (solid line), Euler (short dash line).
Figure 9  Contours of dynamic pressure ($pq^{1/2}$, in percent of inlet dynamic head) at outlet measurement plane. a) Experimental. b) Navier-Stokes. c) Euler.
Figure 10  Comparison of computed and measured spanwise distribution of mass-weighted, pitchwise averaged outlet flow angle. Measured values (symbols), Navier-Stokes (solid line), Euler (short dash line).

Figure 11  Comparison of computed and measured spanwise distribution of blade normal force coefficient. Measured values (symbols), Navier-Stokes, embedded H-grid (solid line), Navier-Stokes, "pinched" standard H-grid (long dash line), Euler (short dash line).
Figure 12: Comparison of computed and measured spanwise distribution of mass-weighted, pitchwise averaged outlet flow angle. Measured values (symbols), Navier-Stokes (line). a) Experiment B, \( \tau/c = 0.00 \). b) Experiment B, \( \tau/c = 0.04 \).

Figure 13: Comparison of computed and measured spanwise distribution of blade normal force coefficient. Measured values (symbols), Navier-Stokes (line). a) Experiment B, \( \tau/c = 0.00 \). b) Experiment B, \( \tau/c = 0.04 \).
Figure 14 Contours of flow angle at outlet measurement plane for Experiment C, \( \tau/c = 0.04 \).
a) Measured. b) Predicted. c) Comparison of computed and measured spanwise distribution of mass-weighted, pitchwise averaged outlet flow angle for Experiment C, \( \tau/c = 0.04 \). Measured values (symbols), Navier-Stokes (line).
VKI CASCADE

- Highly loaded transonic turbine nozzle guide vane cascade

- Geometrical data:

  \[ C = 67.65 \text{ mm} \]
  \[ S/C = 0.850 \]
  \[ \gamma = 55.0 \text{ deg} \quad \text{(stagger angle)} \]
  \[ \alpha \approx 75.0 \text{ deg} \quad \text{(flow turning angle)} \]
Figure 17  Blade isotropic Mach no. distribution
\( M_{2is}=0.84, P_{o1}=1.435 \text{ Bar}, T_{o1}=412 \text{K}, \)
\( M_1=0.15, \ Re_{2}=1000,000, Tu_{\infty}=1\% \),
Figure 18  The computed and measured wakes for above case
(0.433 chord length downstream)
Figure 19. Mach no. contour for above case
Figure 20  Blade heat transfer
(M2is=0.93, P01=0.915 Bar, M 1 = 0.15
Tu∞=1%, T01=403K, Re,2= 600,000)
Figure 21  Computation Grid for a Flat Plate Cascade
Figure 22  Unsteady Blade Surface Pressure Jump  
  a) Amplitude,  b) Phase Angle
Figure 23  Unsteady Blade Pressure for a Flat Plate Cascade (at zero steady incidence) with an Incoming Moving Wake (Runge-Kutta Explicit Technique, M=0.7)
Figure 24  Vector plot of unsteady velocities (mean components is subtracted)
CONCLUSIONS
RKCC

● 3D CODE WITH K-ε/ARSM MODEL, GOOD ACCELERATION PROPERTIES AND OPTIMUM ARTIFICIAL DISSIPATION HAS BEEN CODED, VALIDATED AND USED EXTENSIVELY TO PREDICT VISCOUS FLOW FIELD IN SUPERSONIC CASCADE, SUBSONIC CASCADES, LOW SPEED COMPRESSOR, EVOLUTION OF TIP CLEARANCE FLOW, HIGH SPEED CENTRIFUGAL COMPRESSOR FLOW FIELD.

● 3D CODE IS PRESENTLY EXTENDED TO INCLUDE TIME ACCURATE SOLUTION ROTOR/STATOR INTERACTION EFFECTS.

● TURBULENCE MODEL SOURCE TERMS DO NOT AFFECT THE STABILITY OF THE NUMERICAL SCHEME - CONTRARY TO THAT GENERALLY PERCEIVED.

● IMPLICIT TREATMENT OF TURBULENCE SOURCE TERM DID NOT IMPROVE CONVERGENCE RATE.

● NO ADVANTAGE IS NUMERICALLY COUPLING THE GOVERNING EQUATION AND K-ε MODEL.

● NEAR WALL TURBULENCE PHYSICS AND THE EFFECT OF ROTATION AND COMPRESSIBILITY PREDICTED ACCURATELY.
THE EFFECT OF ROTATION AND ENDWALL SECONDARY FLOW IN THE NEAR WAKE, AND EFFECT OF ROTATION ON 3D BLADE BOUNDARY LAYERS ARE CAPTURED ACCURATELY.

THE SPANWISE DISTRIBUTION OF LOSSES IS PREDICTED WELL AWAY FROM THE ENDWALLS, BUT ONLY FAIR PREDICTION IS ACHIEVED IN THE ENDWALL REGION.

STAGE PRESSURE RISE, MERIODIONAL VELOCITIES IN A CENTRIFUGAL COMPRESSOR IS PREDICTED TO ENGINEERING ACCURACY. THE CRITICAL ELEMENT IS GRID, FOLLOWED BY TURBULENCE MODELLING.

THE "PINCHED" H GRID IS THE MOST APPROPRIATE GRID TO USE IN THE TIP CLEARANCE REGION.

THE CODE ACCURATELY CAPTURES SHOCK WAVE/BOUNDARY LAYER INTERACTION IN A CASCADE, WITH THE EXCEPTION OF REFLECTED WAVES FROM THE SUCTION SURFACE; SHOCK LOSSES ARE PREDICTED WELL.
ABSTRACT

DEVELOPMENT OF A CFD CODE FOR INTERNAL FLOWS IN LIQUID FUELED ENGINES

SUBMITTED TO

WORKSHOP FOR COMPUTATIONAL FLUID DYNAMIC APPLICATIONS IN ROCKET PROPULSION

To support the design efforts of engines in which liquid propellants are used, one is often required to analyze incompressible, two-dimensional or axisymmetric flows within ducts and cavities with rotating walls and complicated geometries. The steady-state solution is of interest in most cases.

This code is intended to provide a tool for efficient CFD analysis of such flow problems, taking advantage of the artificial compressibility concept and the Beam-Warming numerical scheme modified for second order accurate, implicit boundary conditions. These concepts ensure a stable, robust, and accurate algorithm due to the reduced speed of sound and the accuracy of the boundary conditions.

The code is dedicated only to two-dimensional or axisymmetric flows with or without swirl. Three-dimensional computation is excluded to increase efficiency and speed.

This paper briefly presents the theory of the code, as well as several benchmark applications with comparison to well known analytical solutions. In all these test cases, the code produced remarkably accurate results.

Youssef Dakhoul, Sverdrup Technology
620 Discovery Drive
Huntsville, AL 35806
DEVELOPMENT OF A CFD CODE
FOR INTERNAL FLOWS
IN LIQUID FUELED ENGINES

By

Youssef Dakhoul  Sverdrup Technology

Presented to

The Tenth Workshop for CFD Applications in Rocket Propulsion
MSFC April 1992
MOTIVATIONS

- CFD Analysis of cavity flows required to support design effort

- Desire to use the most recent and advanced methods
  Artificial compressibility  Reduces speed of sound  Improves stability
  Beam-Warming method  Stable  Robust

- Desire to develop a specific-purpose code with minimum overhead
  2D, Planer, Axisymmetric, with or without swirl, Incompressible
  No temperature computation  No chemistry
  No compressible flow  No 3D computation
TECHNICAL OBJECTIVES

- Develop a CFD code for internal flows in liquid fueled engines
- Valid for 2D planar and axisymmetric flows with or without swirl
- Use the Artificial Compressibility Concept
- Use the Beam-Warming scheme with implicit boundary conditions
- Test the laminar capability of the code
- Add a suitable turbulence model
- Test the turbulent capability of the code
Governing Equations

\[ Q_t + E_x + F_y + \alpha H = 0 \]

\[ Q = \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix} \quad E = \begin{bmatrix} \rho \beta u \\ uu + p/\rho - \nu (2u_x) \\ uv - \nu (u_y + v_x) \\ uw - \nu (w_x) \end{bmatrix} \]

\[ F = \begin{bmatrix} \rho \beta v \\ uv - \nu (u_y + v_x) \\ vv + p/\rho - \nu (2v_y) \\ vw - \nu (w_y - w/y) \end{bmatrix} \quad H = \frac{1}{y} \begin{bmatrix} \rho \beta v \\ uv - \nu (u_y + v_x) \\ vv - wv - \nu (2v_y - 2v/y) \\ 2vw - \nu (2w_y - 2w/y) \end{bmatrix} \]
Nondimensionalization

\[ Q^* = \begin{bmatrix} p^* \\ u^* \\ v^* \\ w^* \end{bmatrix} \quad E^* = \begin{bmatrix} \beta^* u^* \\ u^* u^* + p^* - \frac{\nu^*}{R_e} (2u^*_x) \\ u^* v^* - \frac{\nu^*}{R_e} (u^*_y + v^*_x) \\ u^* w^* - \frac{\nu^*}{R_e} (w^*_x) \end{bmatrix} \]

\[ F^* = \begin{bmatrix} \beta^* v^* \\ u^* v^* - \frac{\nu^*}{R_e} (u^*_y + v^*_x) \\ v^* v^* + p^* - \frac{\nu^*}{R_e} (2v^*_y) \\ v^* w^* - \frac{\nu^*}{R_e} (w^*_y - w^*/y^*) \end{bmatrix} \quad H^* = \frac{1}{y^*} \begin{bmatrix} \beta^* v^* \\ u^* v^* - \frac{\nu^*}{R_e} (u^*_y + v^*_x) \\ v^* v^* - w^* w^* - \frac{\nu^*}{R_e} (2v^*_y - 2v^*/y^*) \\ 2v^* w^* - \frac{\nu^*}{R_e} (2w^*_y - 2w^*/y^*) \end{bmatrix} \]

\[ u^* = \frac{u}{v_{ref}} \quad t^* = \frac{t}{t_{ref}} \quad p^* = \frac{p}{\rho_{ref} v_{ref}^2} \quad \beta^* = \frac{\beta}{v_{ref}^2} \quad \nu^* = \frac{\nu}{\nu_{ref}} \quad \rho^* = \frac{\rho}{\rho_{ref}} = 1 \]

\[ \rho_{ref} = \rho = \text{fluid density} \quad t_{ref} = \frac{x_{ref}}{v_{ref}} \quad R_e = \frac{x_{ref} v_{ref}}{\nu_{ref}} \]
Transformation

\[ Q_i + E_i + F_i + \alpha H = 0 \]

\[ Q = \frac{1}{J} \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix}, \quad E = \frac{1}{J} \begin{bmatrix} \beta (\xi_x u + \xi_y v) \\ \xi_x (u^2 + p) + \xi_y uv - \frac{\nu}{K_e} (a_1 u_x + a_2 u_\eta + a_3 v_x + a_4 v_\eta) \\ \xi_x uv + \xi_y (v^2 + p) - \frac{\nu}{K_e} (a_5 u_x + a_6 u_\eta + a_7 v_x + a_8 v_\eta) \\ \xi_x uw + \xi_y vw - \frac{\nu}{K_e} (a_9 w_x + a_{10} w_\eta + a_{11} w) \end{bmatrix} \]

\[ F = \frac{1}{J} \begin{bmatrix} \beta (\eta_x u + \eta_y v) \\ \eta_x (u^2 + p) + \eta_y uv - \frac{\nu}{K_e} (b_1 u_x + b_2 u_\eta + b_3 v_x + b_4 v_\eta) \\ \eta_x uv + \eta_y (v^2 + p) - \frac{\nu}{K_e} (b_5 u_x + b_6 u_\eta + b_7 v_x + b_8 v_\eta) \\ \eta_x uw + \eta_y vw - \frac{\nu}{K_e} (b_9 w_x + b_{10} w_\eta + b_{11} w) \end{bmatrix} \]

\[ H = \frac{1}{J y} \begin{bmatrix} \beta v \\ uv - \frac{\nu}{K_e} (c_1 u_x + c_2 u_\eta + c_3 v_x + c_4 v_\eta) \\ v^2 - w^2 - \frac{\nu}{K_e} (c_5 v_x + c_6 v_\eta + c_7 v) \\ 2vw - \frac{\nu}{K_e} (c_8 w_x + c_{10} w_\eta + c_{11} w) \end{bmatrix} \]
Flux Jacobians

\[ \frac{\partial \mathcal{E}}{\partial \mathcal{Q}} = \begin{bmatrix} 0 & \xi_x \beta & \xi_y \beta & 0 \\ \xi_x & 2\xi_x u + \xi_y v - \frac{\xi_x}{\mathcal{H}}(a_1 J_x + a_2 J_y) & \xi_y u - \frac{\xi_y}{\mathcal{H}}(a_3 J_x + a_4 J_y) & 0 \\ \xi_y & \xi_x v - \xi_y \frac{\xi_y}{\mathcal{H}}(a_3 J_x + a_6 J_y) & \xi_x u + 2\xi_y v - \frac{\xi_x}{\mathcal{H}}(a_7 J_x + a_8 J_y) & 0 \\ 0 & \xi_x w & \xi_y w & \xi_x u + \xi_y v - \frac{\xi_x}{\mathcal{H}}(a_9 J_x + a_{10} J_y + a_{11} J) \end{bmatrix} \]

\[ \frac{\partial \mathcal{F}}{\partial \mathcal{Q}} = \begin{bmatrix} 0 & \eta_x \beta & \eta_y \beta & 0 \\ \eta_x & 2\eta_x u + \eta_y v - \frac{\eta_x}{\mathcal{H}}(b_1 J_x + b_2 J_y) & \eta_y u - \frac{\eta_y}{\mathcal{H}}(b_3 J_x + b_4 J_y) & 0 \\ \eta_y & \eta_x v - \eta_y \frac{\eta_y}{\mathcal{H}}(b_3 J_x + b_6 J_y) & \eta_x u + 2\eta_y v - \frac{\eta_x}{\mathcal{H}}(b_7 J_x + b_8 J_y) & 0 \\ 0 & \eta_x w & \eta_y w & \eta_x u + \eta_y v - \frac{\eta_x}{\mathcal{H}}(b_9 J_x + b_{10} J_y + b_{11} J) \end{bmatrix} \]

\[ \frac{\partial \mathcal{H}}{\partial \mathcal{Q}} = \frac{1}{y} \begin{bmatrix} 0 & 0 & \beta & 0 \\ 0 & v - \frac{v}{\mathcal{H}}(c_1 J_x + c_2 J_y) & u - \frac{u}{\mathcal{H}}(c_3 J_x + c_4 J_y) & 0 \\ 0 & 0 & 2v - \frac{2v}{\mathcal{H}}(c_5 J_x + c_6 J_y + c_7 J) & -2w \\ 0 & 0 & 2w & \frac{2w}{\mathcal{H}}(c_5 J_x + c_6 J_y + c_7 J) \end{bmatrix} \]
The Beam-Warming Method

\[
\left[ I + \Delta t \left( \frac{\partial}{\partial \xi} A^n + \frac{\partial}{\partial \eta} B^n + \alpha C^n \right) - I \epsilon_{\text{imp}}^2 J^{-1} (\delta_\xi^2 + \delta_\eta^2) J \right] \delta Q^{n+1} =
\]

\[
\Delta Q^n - \epsilon_{\text{exp}}^4 J^{-1} (\delta_\xi^4 + \delta_\eta^4) J Q^n
\]

where \( I \) is the \( 4 \times 4 \) unit matrix, and:

\[
\delta Q^{n+1} = Q^{n+1} - Q^n
\]

\[
\Delta Q^n = -\Delta t \left[ \mathcal{E}_\xi^n + \mathcal{F}_\eta^n + \alpha \mathcal{H}^n \right]
\]

\[
\delta_\xi^2 f = f_{i-1} - 2f_i + f_{i+1}
\]

\[
\delta_\xi^4 f = f_{i-2} - 4f_{i-1} + 6f_i - 4f_{i+1} + f_{i+2}
\]
The first step is:

\[
\left[ I + \Delta t \left( \frac{\partial}{\partial \xi} A^n \right) - I \epsilon_{\text{imp}}^2 J^{-1} \delta^2 \right] \delta Q^* = \Delta Q^n - \epsilon_{\text{exp}}^4 J^{-1} (\delta_\xi^4 + \delta^4_\eta) J Q^n
\]

which is solved for \( \delta Q^* \)

and the second step is:

\[
\left[ I + \Delta t \left( \frac{\partial}{\partial \eta} B^n + \alpha C^n \right) - I \epsilon_{\text{imp}}^2 J^{-1} \delta^2 \right] \delta Q^{n+1} = \delta Q^*
\]

which is solved for \( \delta Q^{n+1} \) at all nodes.
Treatment of the Boundary Conditions

Writing the first step at each node from $i = 1$ to $i = i_{\text{max}}$ (for a certain constant $j$) produces a set of equations that can be expressed in a block matrix form:

$$
\begin{bmatrix}
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\cdot & \cdot & \cdot \\
\end{bmatrix}
\begin{bmatrix}
\delta \mathbf{Q}_{i} \\
\delta \mathbf{Q}_{i} \\
\delta \mathbf{Q}_{3} \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\delta \mathbf{Q}_{i}^* \\
\end{bmatrix}
= 
\begin{bmatrix}
d_{1} \\
d_{2} \\
d_{3} \\
d_{i} \\
d_{i} \\
d_{i} \\
d_{i} \\
d_{i} \\
d_{i} \\
d_{i} \\
\end{bmatrix}
$$

Note that the existence of the 'o' matrices prevents the block matrix from being tridiagonal.

These 'o' matrices are associated with domain edges as well as east and west edges of blocked-out areas.

They appear because one-sided, second-order-accurate differencing was used at these edges.

To restore the tridiagonal nature of the block matrix, lag the third set of unknowns in the first set of equations, $\delta \mathbf{Q}_{3}^*$, by one iteration. Now, the product $o \delta \mathbf{Q}_{3}^*$ can be subtracted from $d_{1}$ and the 'o' in the first row dissapears. Of course, similar treatment for the last row and for edges of blocked-out areas is required.
PIPE FLOW

GRID 81 x 31 NODES

Costant Velocity \( j \) \( \rightarrow \) \( i \) Monitor Point (41,16) Length = 0.06 m
Diameter = 0.004 m

Constant Pressure
SLIDE BLOCK FLOW

GRID 111 x 21

Monitor point (56,11)
Initial Pressure Field = 0.0
Initial Velocity Field = 0.0

\[ \begin{align*}
  h_1 &= 3 \times 10^{-4} \text{ m} \\
  h_2 &= 9 \times 10^{-5} \text{ m}
\end{align*} \]

15.0 m/sec

L = 0.1 m

Stationary Wall

\[ h_1 \]

Constant Pressure

\[ y \]

\[ x \]
SLIDE BLOCK FLOW
PRESSURE CONTOURS (Pascal)

CONTOUR LEVELS
0.0
50000.0
100000.0
150000.0
200000.0
250000.0
300000.0
350000.0
400000.0
450000.0
500000.0
550000.0
600000.0
650000.0
700000.0
750000.0
800000.0
850000.0
900000.0
950000.0
1000000.0
1050000.0
1100000.0
1150000.0
1200000.0
1250000.0
1300000.0
1350000.0
1400000.0
1450000.0
1500000.0
1550000.0
1600000.0
1650000.0
1700000.0
1750000.0
Wedge Flow

Normalized Distance from Bottom

U Velocity (m/sec) at x/L=0

Computed
Analytical
TWO CONCENTRIC CYLINDERS
GRID 61 x 21 NODES

Outer Cylinder  Radius = 0.03 m  Constant Pressure at Zero

Periodic Boundary

Inner Cylinder  Radius = 0.005 m  Omega = 0.121648 rad/sec
Length = 0.1 m  Impulsively Started

Initial Pressure Field = 0.0
Initial Velocity Field = 0.0
Two Concentric Cylinders

Distance from Inner Cylinder (m)

- Computed

- Analytical

Swirl Velocity (m/sec)
Two Concentric Cylinders

Iteration (thousands)

RMS of Pressure Residual

$10^{-7}$ $10^{-6}$ $10^{-5}$ $10^{-4}$
TAYLOR VORTICES
GRID 101 x 31 NODES

Outer Cylinder  Radius = 0.04035 m

Inner Cylinder  Radius = 0.038 m
Length = 0.009306 m
Omega = 17.8628 rad/sec

Point of Constant Pressure at Zero
Monitor Point (51,16)
TAYLOR VORICES
PRESSURE CONTOURS (Pascal)

CONTOUR LEVELS
-0.5
0.0
0.5

1333
DRIVEN CAVITY FLOW
GRID 51 x 51

Initial Pressure Field = 0.0
Initial Velocity Field = 0.0
Driven Cavity Flow

RMS of Pressure Residual

Iteration (thousands)
TWO ECCENTRIC CYLINDERS
GRID $81 \times 31$

Outer Cylinder
Radius = 2.0 m

Inner Cylinder
Radius = 1.0 m

Omega = $18.5e-6$ rad/sec
Impulsively Started

Initial Pressure Field = 0.0
Initial Velocity Field = 0.0
TWO ECCENTRIC CYLINDERS
RELATIVE PRESSURE CONTOURS (1.e-7 Pascal)
Preburner Pump Cavity

$p = 0.0$ Top exit

$0.3^\circ$

$-4.57 \text{ ft/sec}$

$1791 \text{ rad/sec}$

$43.86 \text{ ft/sec}$

$1152 \text{ rad/sec}$

$0.3915^\circ$

$-31.66 \text{ ft/sec}$

Axis of symmetry
ATD PREBURNER DISCHARGE CAVITY
Grid 138 x 120
ATD PREBURNER DISCHARGE CAVITY
Swirl Velocity Contours (fps)
Development of the KIVA-II CFD Code for Rocket Propulsion Applications

by Robert V. Shannon Jr.

and

Alvin L. Murray

Aerotherm Corporation
1500 Perimeter Parkway, Suite 225
Huntsville, AL 35806

Due to the existence of liquid, solid, and hybrid rocket motors a need exists for a fluid dynamics code which can solve for both the Navier-Stokes equations in multi-dimensions as well for liquid and solid particle motion within the gas flow domain. This type of CFD code must couple the gas and particle motion so that the effects of one upon the other can begin to be understood. Disciplines such as the evaluation of solid motor performance as well as nozzle erosion predictions for solid motors have a great need for the type of information that this type of computer code can provide. In addition, it is difficult to accurately simulate liquid motor combustion chamber flows without solving for the liquid oxidizer droplet motion, breakup, and evaporation coupled with the reacting gas flow.

The KIVA-II code, originally developed at Los Alamos National Laboratories to solve fluid dynamics problems in internal combustion engines, has been developed to solve rocket propulsion type flows. This work was supported by the NASA Solid Propulsion Integrity Program. The objective of the work performed was to develop this CFD code so that both liquid and solid particle motion could be simulated for arbitrary geometry and for high speed as well as low speed reacting flows.

Modification of the KIVA-II gas flow algorithm involved: incorporating independently specifiable supersonic and subsonic inflows and outflows, symmetric as well as periodic boundary conditions, the capability to use generalized single or multi-specie thermodynamic data and transport coefficients and allowing the user to specify arbitrary wall temperature/heat flux distributions. Major modifications to the algorithms involving both convection and diffusion were successfully performed and results have been compared with some available experimental data on nozzle flows to verify these modifications.

Modification of the algorithms governing particle motion involved: incorporation of multi-specie particle types, generalized liquid property thermodynamic data, options which prevent or allow particle evaporation, a generalized particle injection algorithm, simulation of particle elastic or inelastic collisions with solid boundaries, periodic or symmetric particle boundary conditions at boundaries which are not inflow, outflow, or wall boundaries, and incorporation of a particle drag model which allows for both Reynolds and Mach number effects on particle drag coefficients.

This new CFD code has been shown to successfully solve rocket propulsion flows as well as rocket propulsion flows with entrained particles for several different rocket nozzles, submerged and otherwise. Verification of this new CFD code continues by comparing results to experimental data as well as to predictions of other CFD codes. This program is proving to be a valuable tool in the prediction of rocket propulsion flows.
Development of the KIVA-II CFD Code for Rocket Propulsion Applications

by

R.V. Shannon Jr.
and
A.L. Murray

Aerotherm Corporation
Huntsville, Alabama
Development of the KIVA-II CFD Code for Rocket Propulsion Applications

Original KIVA-II Code - General Information

- KIVA-II code originally issued in May 1989
- Approx. 14000 source lines including routines for plotting
- Authored by A.A. Amsden, P.J. O'Rourke, T.D. Butler
- Two-equation turbulence k-e model
- Kinetic and/or equilibrium chemical reaction modeling
- Stochastic particle technique for liquid sprays
- Initial and boundary conditions written specifically for IC engine calculations
Original KIVA-II Code - General Information

- Some boundary conditions inappropriate for high-speed flow problems
- Subsonic inflow boundary conditions not coupled with the overall pressure field solution
- Fuel spray was single component but modeling for droplet collisions and aerodynamic breakup was incorporated.
KIVA-IIG Code - General Changes Gas Flows

- Approx. 30000 source lines not including routines for plotting
- Multiple inflow/outflows allowed
- Symmetric as well as periodic boundary conditions allowed
- Advection routines were recast in a form which directly computes the fluxes before using them
- Thermodynamic data input will allow for multiple species thermodynamic data. Alternatively, an equilibrium gas mixture can be modeled by a single Mollier table.
- Species transport property data is independently specified allowing for fewer points in regions where these quantities do not vary much. Viscosity, specific heats, and conductivity may all vary with pressure as well as temperature.
KIVA-IIIG Code - General Changes Gas Flows

- Multiple wall types may be specified allowing the user to have a wall temperature distribution in his/her flowfield calculation.
- Conjugate Residual Method used for calculating the pressure and temperature fields now includes boundary condition specifications.
KIVA-IIG Code - General Changes Parcel Motion

- Multiple particle species available
- Generalized liquid property thermodynamic data
- Option to prevent or allow particle evaporation for each particle type
- Generalized particle injection algorithm which makes each simulated injector independent of all other injectors as far as injector characteristics
- Simulation of particle elastic or inelastic collisions with solid boundaries
- Periodic or symmetric particle boundary conditions at fluid boundaries
- 3 different particle drag models available
JPL Nozzle Test Case

- Chamber temperature 1100 Kelvin
- Chamber pressure $4.9 \times 10^6$ dynes/cm$^2 = 4.8$ atm
- Air was modeled by a single Mollier table representing thermodynamic data for equilibrium air.
VELOCITY VECTORS FOR PLANAR NOZZLE
PLANAR NOZZLE TEMPERATURE CONTOURS
PLANAR NOZZLE MACH NUMBER CONTOURS
Two Inch Motor Test Case

- Chamber temperature 3564 Kelvin
- Chamber pressure $5.9 \times 10^7$ dynes/cm$^2 = 58$ atm
- The gas mixture of many species was modeled by a single Mollier table representing thermodynamic data for the equilibrium mixture.
- Nozzle geometry is unusual.
TWO INCH MOTOR NOZZLE TEST CASE

FLOW ANALYSIS GRID

CALCULATED VELOCITY VECTORS

Aerotherm Corporation
A Subsidiary of DynCorp
TWO INCH MOTOR NOZZLE TEST CASE

CALCULATED PRESSURES

\[
\begin{align*}
\Delta & Q_{\text{MAX}} = 5.64 \times 10^7 \text{ dynes/cm}^2 \\
\Delta & Q_{\text{MIN}} = 5.64 \times 10^6 \text{ dynes/cm}^2 \\
\Delta & DQ = 2.98 \times 10^6 \text{ dynes/cm}^2
\end{align*}
\]

CALCULATED TEMPERATURES

\[
\begin{align*}
\Delta & Q_{\text{MAX}} = 3493 ^\circ K \\
\Delta & Q_{\text{MIN}} = 2254 ^\circ K \\
\Delta & DQ = 73 ^\circ K
\end{align*}
\]

Aerotherm Corporation
A Subsidiary of DynCorp
TWO INCH MOTOR NOZZLE TEST CASE

QMAX = 2.42
QMIN = 0.143
DQ = 0.134

CALCULATED MACH CONTOURS
MNASA Motor Test Nozzle Flowfield Analysis

- Chamber temperature 3564 Kelvin
- Chamber pressure $5.1 \times 10^7$ dynes/cm$^2 = 50$ atm
- The gas mixture of many species was modeled by a single Mollier table representing thermodynamic data for the equilibrium mixture.
- 10 micron particles "injected" into the flowfield. Injected particle density was 3.94 grams/cm$^3$.
- Crowe-Hermsen modified drag model used.
- Gas-particle momentum and energy coupled
MNASA MOTOR TEST CONFIGURATION

Reference Thiokol Report TWR-60115

Aerotherm Corporation
A Subsidiary of DynCorp
PARCEL VELOCITY CHRONOLOGY

$T = 7.5(10)^{-5}$ sec

$T = 2.8(10)^{-3}$ sec

$T = 4.0(10)^{-3}$ sec
PARCEL VELOCITY CHRONOLOGY

\[ T = 4.4(10)^{-3} \text{ sec} \]

\[ T = 4.7(10)^{-3} \text{ sec} \]

\[ T = 4.8(10)^{-3} \text{ sec} \]
PARCEL VELOCITY CHRONOLOGY

$T = 5.7 \times 10^{-3}$ sec

$T = 5.72 \times 10^{-3}$ sec

$T = 5.73 \times 10^{-3}$ sec
MNASA MOTOR 10 MICRON PARTICLE TRAJECTORIES
Summary

- A new finite volume 2D/3D Navier-Stokes CFD code has been developed by using the KIVA-II code as the starting point.
- Demonstration of this new code has been shown with promising results and verification with experimental data will continue.
- Particle motion has been verified for low particle Reynolds numbers and demonstrated for typical solid rocket motor flowfields.
Effective application of computational fluid dynamics (CFD) in the engineering environment requires that key design and analysis tools be integrated to the greatest extent possible. A computational design system (CDS) is described in which these tools are integrated in a modular fashion. This CDS ties together four key areas of computational analysis: description of geometry, grid generation, computational codes, and postprocessing. Common input and output formats and necessary translators are established to facilitate data transfer between the four key areas and to enhance system modularity. Advances made in three key areas are described.

Significant progress has been made toward integration of geometry definition systems with CFD grid generation tools. Geometric data have successfully been passed from the Catia CAD and Patran CAE systems to the Rockwell Automated Grid Generation System (RAGGS). The IGES standard was employed in each case. The CFD Pump Consortium impeller geometry is used to illustrate that a reasonable level of integration is achievable for complex geometries.

While many CFD grid generators are available in the public domain, their capabilities vary widely. An effort is underway to systematically review available codes, identify those most applicable, and integrate them into the CDS described. Seven grid generation codes are currently being reviewed according to previously defined evaluation criteria described herein.

Postprocessing tools are being developed, reviewed, and integrated into the CDS as well. Engineering data extraction tools are being developed to be completely consistent with existing code methodologies and to remain completely modular. Tools for data visualization have advanced noticeably over the last few years. These are being reviewed and integrated into the CDS.

Integration of improved CFD analysis tools through integration with the Rocketdyne CDS has made a significant positive impact in the use of CFD for engineering design problems. Complex geometries are now analyzed on a frequent basis and with far greater ease.
A COMPUTATIONAL DESIGN SYSTEM FOR RAPID CFD ANALYSIS

E. P. Ascoli, S. L. Barson, M. E. DeCroix, and M. M. Sindi
Rockwell International, Rocketdyne Division

Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion
April 28-30, 1992
NASA Marshall Space Flight Center
INTEGRATION OF CFD TOOLS NEEDED FOR EFFICIENT APPLICATION IN ENGINEERING ENVIRONMENT

- APPLICATION OF CFD IN ENGINEERING CYCLE REQUIRES:
  - INTEGRATION WITH EXISTING TOOLS AND PROCEDURES
  - USE FOR MULTIPLE LEVELS OF ANALYSIS
    - RAPID TURNAROUND PARAMETRICS
    - FINAL DETAIL DESIGN ANALYSIS

- PAST EMPHASIS ON FLOW SOLVER DEVELOPMENT OUTPACED THAT OF "PERIPHERAL" TOOLS
  - PRE- AND POST-PROCESSING CAN TAKE MORE THAN 50% OF TOTAL CYCLE TIME
  - TOOL INTERFACES OFTEN POOR OR COMPLETELY LACKING
COMPUTATIONAL DESIGN SYSTEM INTEGRATES
ANALYTICAL TOOLS FOR EFFICIENT USE

• TIES TOGETHER FOUR KEY AREAS OF COMPUTATIONAL ANALYSIS
  • GEOMETRIC DESCRIPTION - MATHEMATICAL REPRESENTATION
    OF HARDWARE GEOMETRY
  • GRID GENERATION - DISCRETIZATION OF FLOW DOMAIN
  • COMPUTATIONAL CODES - ANALYTICAL SOLUTION
  • POST-PROCESSING - REDUCTION, ORGANIZATION, AND
    PRESENTATION OF ANALYTICAL DATA

• ESTABLISHES COMMON INPUT/OUTPUT FORMATS AND
  NECESSARY TRANSLATORS

• MODULAR APPROACH ALLOWS USE OF MOST APPROPRIATE
  TOOL FOR EACH APPLICATION
ROCKETDYNE ADVANCED COMPUTATIONAL ENGINEERING SYSTEM (RACES)

GEOMETRY / SURFACE DEFINITION
- EXISTING CAD SYSTEMS (e.g. CATIA)
- CAE SYSTEMS (e.g. PATRAN)
- USER-DEFINED (e.g. ALGEBRAIC)

GRID GENERATION OPTIONS
(SURFACE AND VOLUME GRIDS)
- INTEGRATED (e.g. PATRAN)
- SEPARATE (e.g. RAGGS)

COMPUTATIONAL CODES
- ALL EXISTING MAJOR RD CODES
- ADD OTHERS AS NEEDED

POST-PROCESSING
- ENGINEERING DATA REDUCTION
- DATA VISUALIZATION
- DATABASE OF CFD SOLUTIONS
ADVANCES MADE TOWARD IMPROVED AND UNIFIED SET OF CFD TOOLS

- INTEGRATION OF GEOMETRY DEFINITION SYSTEMS WITH GRID GENERATORS

- SYSTEMATIC REVIEW AND INTEGRATION OF AVAILABLE GRID GENERATION TOOLS

- DEVELOPMENT AND INTEGRATION OF NEW POSTPROCESSORS
CAD / CAE INTERFACES ESTABLISHED

- IGES INTERFACE IDENTIFIED AS PREFERRED STANDARD
  - INDUSTRY STANDARD
  - SUPPORTED BY MOST CAD AND CAE SOFTWARE

- CATIA (CAD) GEOMETRY FILES PASSED TO ROCKWELL AUTOMATED GRID GENERATION (RAGGS) CODE
  - COMPLEX GEOMETRY - CONSORTIUM IMPELLER
  - TRANSLATOR UPGRADES REQUIRED

- PATRAN (CAE) GEOMETRY FILES PASSED TO RAGGS
  - SEVERAL CASES RANGING FROM SIMPLE TO COMPLEX
  - EASY GEOMETRY CREATION METHOD

- TRANSLATOR CHARACTERIZATION AND VALIDATION REQUIRED
CATIA IMPELLER GEOMETRY PASSED TO RAGGS

3-D CFD GRID GENERATED

CATIA → IGES → RAGGS
GRID GENERATORS REVIEWED

• BACKGROUND
  • MANY GRID GENERATION CODES CURRENTLY AVAILABLE
  • DIFFERENT APPROACHES AND CODE FEATURES
  • NEED EXISTS TO CONDUCT SYSTEMATIC REVIEW OF
    GRID GENERATION CODES CURRENTLY AVAILABLE

• OBJECTIVE
  • IDENTIFY MOST APPROPRIATE CODE(S)
  • INTEGRATE WITH RACES

• APPROACH
  • DEVELOP EVALUATION CRITERIA / SCORING GUIDELINES
  • IDENTIFY CANDIDATE CODES
  • THREE PHASES TO FINAL SELECTION
    • PRE-SCREEN
    • PRELIMINARY EVALUATION
    • FINAL EVALUATION
# Seven Grid Generation Codes Reviewed

<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ICEM (CDC)</td>
<td>Extensive CAD capability, Third party source</td>
</tr>
<tr>
<td>IGB (NASA Lewis)</td>
<td>Template type for turbine blades, not a general purpose code</td>
</tr>
<tr>
<td>EAGLEVIEW (MSU)</td>
<td>Eagle based grid generation, new interactive interface</td>
</tr>
<tr>
<td>GENIE (MSU)</td>
<td>First generation interactive code, best suited for 2-D and limited 3-D</td>
</tr>
<tr>
<td>GRIDGEN (GENERAL DYNAMICS)</td>
<td>Interactive general purpose code, highly rated in industry</td>
</tr>
<tr>
<td>PATRAN (PDA Engineering)</td>
<td>Limited as CFD grid generator, third party source</td>
</tr>
<tr>
<td>RAGGS (ROCKWELL, NAA)</td>
<td>Interactive general purpose code, developing product</td>
</tr>
</tbody>
</table>
EVALUATION CRITERIA ESTABLISHED
FIVE MAIN CATEGORIES

• GEOMETRY DEFINITION
  • INTERFACES (INPUT AND OUTPUT)
  • CREATION CAPABILITY
  • SURFACE ACCURACY
    • AS TRANSLATED TO GG
    • AS CREATED IN GG

• SURFACE AND VOLUME GRID CAPABILITIES
  • ACCURACY WITH SURFACE DEFINITION
  • METHODOLOGIES AVAILABLE
    (e.g., ALGEBRAIC, ELLIPTIC, HYPERBOLIC)

• GRID TYPES SUPPORTED
  • MULTIZONE
  • PERIODIC
  • H-, C-, O-TYPES
  • FAN (DEGENERATE)
EVALUATION CRITERIA (CONT'D)

- GRID CONTROL
  - CLUSTERING OPTIONS
  - LOCAL REFINEMENT
  - ORTHOGONALITY
  - SMOOTHNESS

- USABILITY
  - TIME TO LEARN
  - SPEED(TOTAL CYCLE TIME)
  - MODIFY EXISTING GRID
  - SCRIPTING / TEMPLATE CAPABILITY
  - GRID DIAGNOSTICS
  - ERROR HANDLING
  - SIZE LIMITATIONS
  - DOCUMENTATION AND SUPPORT

- OTHERS
  - AVAILABILITY (SOURCE CODE, THIRD PARTY)
  - PORTABILITY
  - COST
DEVELOPMENT AND INTEGRATION OF POSTPROCESSORS

- POSTPROCESSING MUST ACCOUNT FOR
  - VARIOUS FLOW SOLVER METHODOLOGIES
  - ENGINEERING DATA EXTRACTION
  - DATA VISUALIZATION

- OBJECTIVE: PROVIDE / DEVELOP CONSISTENT SET OF POSTPROCESSING TOOLS

- APPROACH
  - DEVELOP GENERAL ENGINEERING DATA EXTRACTION CAPABILITY
  - REVIEW, SELECT, AND INTEGRATE DATA VISUALIZATION CODES
DATA VISUALIZATION

- OBSERVATIONS
  - SIGNIFICANT ADVANCES OVER RECENT YEARS
  - NUMEROUS CODES AVAILABLE
  - TRUE ENGINEERING TOOLS THAT AID IN UNDERSTANDING COMPLEX SOLUTIONS

- ONGOING REVIEW OF AVAILABLE CODES

- FAST (NASA Ames) IS PRIMARY VISUALIZATION TOOL
  - ATTRACTIVE USER INTERFACE
  - PLOT3D STANDARDS PLUS MANY ENHANCEMENTS
  - CURRENTLY IN BETA RELEASE (2.0)
  - BETA RELEASE (3.0) DUE FOR NEAR TERM RELEASE
THREE DIMENSIONAL GRAPHICS POST-PROCESSORS YIELD MORE INFORMATION ABOUT COMPLICATED GEOMETRIES AND CFD SOLUTIONS
COMPUTATIONAL DESIGN SYSTEM SIGNIFICANTLY ENHANCES CFD ANALYSIS CAPABILITY

- GOOD PROGRESS IN INITIAL PHASE
  - BASIC CAPABILITY IN PLACE
  - IMMEDIATE POSITIVE IMPACT

- FURTHER INTEGRATION EFFORTS REQUIRED
  - GEOMETRY INTERFACES
  - INTERNAL TRANSLATORS
  - INTERDISCIPLINARY TRANSLATORS

- EXTENDED DATABASE CAPABILITY ESSENTIAL
  - DATA STORAGE, ORGANIZATION, AND RETRIEVAL
  - SIMPLIFY TRANSLATOR ISSUES
OPTIMUM DESIGN OF NINETY DEGREE BENDS

Vijay Modi
Department of Mechanical Engineering
Columbia University

Abstract

An algorithm for the optimum design of an internal flow component to obtain the maximum pressure rise is presented. Maximum pressure rise in a duct with simultaneous turning and diffusion is shown to be related to the control of flow separation on the passage walls. Such a flow is usually associated with downstream conditions that are desirable in turbomachinery and propulsion applications to ensure low loss and stable performance. The algorithm requires the solution of an "adjoint" problem in addition to the "direct" equations governing the flow in a body, which in the present analysis are assumed to be the laminar Navier-Stokes equations. Earlier studies have usually addressed such problems for the case of inviscid and/or irrotational flow. These assumptions may not be valid in flows that undergo sharp turning resulting in strong secondary flows and possibly separating and recirculating regions. The theoretical framework and computational algorithms presented in this study are for the steady Navier-Stokes equations.

A novel procedure is developed for the numerical solution of the adjoint equations. This procedure is coupled with a direct solver in a design iteration loop, that provides a new shape with a higher pressure rise. This procedure is first validated for the design of optimum plane diffusers in two-dimensional flow. The direct Navier-Stokes and the "adjoint" equations are solved using a finite volume formulation for spatial discretization in an artificial compressibility framework. The discretized equations are integrated using explicit Runge-Kutta time steps to obtain steady-state solutions. It is found that the computational work required to solve the "adjoint" problem is of the same order as that required to solve the direct problem. It is also found that the procedure converges within about ten iterations, and in addition, the number of design iterations are not sensitive to the grid used for the calculations. This is a significant computational advantage over heuristic design procedures based on point by point sensitivity analysis where the work increases with the refinement of the grid.

A simplified version of the above approach is then utilized to design ninety degree diffusing bends. The bend inlet is square with intermediate and exit cross-sections constrained to be rectangular. The location of bend walls is then determined in order to obtain the maximum pressure rise through the bend. Calculations were carried out for a mean radius ratio at inlet of 2.5 and Reynolds numbers varying from 100 to 500. While at this stage laminar flow is assumed it is shown that a similar approach can be conceived for turbulent flows.
OPTIMUM DESIGN OF NINETY DEGREE BENDS

Vijay Modi, Assistant Professor
Hayri Cabuk, Post Doctoral Research Associate
Jian-Chun Huan, Graduate Student
Richard Quadracci, Graduate Student

Department of Mechanical Engineering,
Columbia University,
New York, New York 10027
MOTIVATION

# How to shape internal flow passages
# Combined turning and diffusing flow
# Maximize pressure rise
# Can not assume inviscid or 2-D flow

OBJECTIVES

# Develop theoretical framework – laminar 3-D
# Develop Navier-Stokes and Adjoint solvers
# Validate Navier-Stokes solver
  (Laminar 90 degree bend, Taylor et al. 1982)
# Validate Optimization Approach on
  2-D straight diffusers
# Apply to the design of ninety degree bends
$u_{i,i} = 0$

$u_j u_{i,j} = -p_i^* + \nu u_{i,ij}, \quad p^* = \frac{p}{\rho}$

# No slip BC on $\Gamma_M$

# Dirichlet BC for $u_i$ at $\Gamma_I$ and $\Gamma_O$

$$J(\Gamma_M) = \int_{\Gamma_I} p^* u_i n_i ds + \int_{\Gamma_O} p^* u_i n_i ds$$
\[ [u^\varepsilon, p^\varepsilon] \equiv \text{Solution to NS in } \Omega_\varepsilon \]

\[ u_i^\varepsilon = u_i + \varepsilon \phi_i \]

\[ p^\varepsilon = p^* + \varepsilon \pi \]

\[ \phi_{i,i} = 0 \]

\[ u_j \phi_{i,j} + \phi_j u_{i,j} = -\pi_{i,i} + \nu \phi_{i,j,j} \]

\[ \phi_i = 0 \quad \text{on } (\Gamma - \Gamma_M) \]
\( u_i^\varepsilon \big|_{P_\varepsilon} = u_i^\varepsilon \big|_{P} + \varepsilon \rho \left( \frac{\partial u_i^\varepsilon}{\partial n} \right)_{P} + O(\varepsilon^2) \)

\[ = u_i \big|_{P} + \varepsilon \phi_i \big|_{P} + \varepsilon \rho \left( \frac{\partial u_i}{\partial n} \right)_{P} + O(\varepsilon^2) \]

\( \phi_i = -\rho \left( \frac{\partial u_i}{\partial n} \right) \) on \( \Gamma_M \).
\[ J(\Gamma_{M_{e}}) - J(\Gamma_{M}) = \varepsilon \delta J + O(\varepsilon^2) \]

\[ \delta J = \int_{\Gamma_{I}} \pi u_{i} n_{i} ds + \int_{\Gamma_{O}} \pi u_{i} n_{i} ds \]

\[ z_{i,i} = 0 \quad \text{in } \Omega \]

\[ \nu z_{i,jj} + u_{j}(z_{i,j} + z_{j,i}) - r_{i} = 0 \quad \text{in } \Omega \]

\[ z_{i} = u_{i} \quad \text{on } \Gamma \]

\[ \delta J = \nu \int_{\Gamma_{M}} \rho(s) \left( \frac{\partial u_{i}}{\partial n} \right) \left( \frac{\partial z_{i}}{\partial n} \right) ds \]

\[ \rho(s) = \omega(s) \left( \frac{\partial u_{i}}{\partial n} \right) \left( \frac{\partial z_{i}}{\partial n} \right) \]
Our "Adjoint" equation:

\[\nu z_{i,jj} + u_j (z_{i,j} + z_{j,i}) - r_i = 0 \quad \text{in } \Omega\]

\[w_i = \frac{1}{2} (z_i - u_i)\]

\[q = \frac{1}{2} (r - p^* + (1/2)u_j^2 - 2u_j w_j)\]

Pironneau's "Adjoint" equation:

\[w_{i,i} = 0 \quad \text{in } \Omega\]

\[\nu w_{i,jj} + u_j w_{i,j} + w_j u_{i,i} - q_i = -u_j u_{i,j} \quad \text{in } \Omega\]

\[w_i = 0 \quad \text{on } \Gamma\]

\[\delta J = \nu \int_{\Gamma_M} \rho(s) \left( \frac{\partial u_i}{\partial n} \right) \left( \frac{\partial u_i}{\partial n} + 2 \frac{\partial w_i}{\partial n} \right) ds\]

\[\frac{d\omega}{dt} = \omega(\text{const}) + \cdots\]
NAVIER-STOKES SOLVER

\[ p_t = -\beta^2 u_{i,i} \]
\[ u_{i,t} + u_j u_{i,j} = -p_i + \nu u_{i,jj} \]

a) Artificial Compressibility
b) Runge-Kutta Time Integration
c) Finite Volume Discretization
d) Artificial Dissipation
e) Local Time Stepping
e) Implicit Residual Smoothing
Figure 4: Geometry of a circular bend with square cross section.

Figure 5: Streamwise velocities in a bend: a) $\Theta = 30$ degrees, b) $\Theta = 60$ degrees, c) $\Theta = 77.5$ degrees.
ADJOINT EQUATION SOLVER

\[ r^*_t = -\beta^2 z_{i,i} \]

\[ z_{i,t} = \nu z_{i,j} + u_j (z_{i,j} + z_{j,i}) - \frac{1}{2} (z_k z_k)_{,i} - r^*_i \]

MESH GENERATION

# Thompson et al.

\[ x_{\xi\xi} + x_{\eta\eta} = 0 \]

\[ y_{\xi\xi} + y_{\eta\eta} = 0 \]
Boundary Conditions

# No-slip bc on the walls
# Zero normal derivatives at exit
# Streamwise velocity component is specified at entrance
# Zero normal derivatives for remaining velocities at entrance
# Typical bc's at symmetry planes
# Zero second derivatives for pressure (a computational bc)
Diffuser Profile History

Re=200, 61 by 31 grid

○ : N=1
□ : N=2
△ : N=5
* : N=10
Skin Friction History

\[ \text{Re}=200, \text{ 61 by 31 grid} \]

\[ \begin{align*}
\bigcirc &: N=1 \\
\square &: N=2 \\
\triangle &: N=5 \\
\ast &: N=10 
\end{align*} \]
Pressure Rise History

Re=200, 61 by 31 grid

○ : Area-averaged pressure rise
△ : Flow-averaged pressure rise
\[ C_p = \frac{\text{Actual Pressure Rise}}{\text{Ideal Pressure Rise}} \]

- \( \bigcirc \): Optimal diffusers
- \( \triangle \): Straight diverging diffusers
Separating Initial Profile

Re=100, 31 by 11 grid

○ : N=1
□ : N=2
△ : N=3
◊ : N=4
* : N=11
Grid Study

Re=200

○ : 31 by 16
□ : 61 by 31
* : 121 by 61
The error in the location of the optimal diffuser profile corresponding to a 2 percent error in the total pressure rise. The optimum shape lies between the high and low $y$-values shown in the graph ($Re=500$, grid size is 61 by 21, and $L/W_i=3$).
Figure 3: A representative section of a three-dimensional diffuser. Flow enters at upstream boundary $\Gamma_I$ and exits at downstream boundary $\Gamma_O$. The walls to be shaped are $\Gamma_M$. 

Sketch of the inlet header
ISSUES

# Geometry Constraints

In general: Move walls by $\varepsilon \rho(s)$ along the normal direction, everywhere

New shape may not satisfy

- specified mean passage location
- specified cross-sectional shape
- overall system geometry

# Present Work

- No correction on side walls ($z = 0, z = z_{\text{max}}$)
- Apply mid-plane ($z = z_{\text{max}}/2$) correction to all $z$ locations
- Hence all cross-sections are rectangular

# Laminar flow results ($Re < 500$)
Governing Equations:

\[ u_{i,i} = 0 \]
\[ u_j u_{i,j} = -p_{,i} + \nu u_{i,ij} \]

Design Objective:

Maximize Static Pressure Rise

\[ J = \int_{\Gamma_I} p \, ds + \int_{\Gamma_O} p \, ds \]

Cabuk and Modi, 1990

\[ \left( \frac{\partial u}{\partial n} \right)_{\text{wall}} = 0 \]
\[ \left( \frac{\partial u}{\partial n} \right)_{\text{wall}} = \epsilon \]
DESIGN ALGORITHM

1: Choose an initial shape.
2: Generate the computational grid.
3: Solve the N-S equations.
4: Compute shear stress on the walls.
5: Compare wall shear stress to target distribution and determine the amount of boundary movement \( \rho(s) \).
6: Update the shape.
7: Go to step (2)

\[
\rho(s) = \omega(s) \left[ \left( \frac{\partial u}{\partial n} \right)_{\text{wall}} - \left( \frac{\partial u}{\partial n} \right)_{\text{target}} \right]
\]
Iteration History (Re=200)

Dashed Curve : N=1

○ : N=8
△ : N=2
◇ : N=5
Wall shear stress along the outer wall, $\text{Re}=100$

- $\bigcirc$ : Optimum diffusing bend
- $\triangle$ : Elliptic diffusing bend

Dashed Curve : Target distribution
Wall shear stress along the inner wall, \( \text{Re} = 100 \)

- ○ : Optimum diffusing bend
- △ : Elliptic diffusing bend

Dashed Curve : Target distribution
Pressure rise along the header (Re=100)

\[ \frac{(P - P_{\text{ent}})}{(0.5 \rho U^2)} \]

Arclength along the bend

○ : Optimum header
△ : Elliptic header
Optimum Shapes

Re = 100

Re = 200

Re = 300

Re = 400

Re = 500
# Performance = f(shape)

# Possible Applications:

- 90 Degree Bend
- Turn Around Ducts
- Transition Ducts
- S-shaped Ducts
- Straight or Curved Diffusers
- Turbine Blades
- Engine Inlets
- Turning Vanes
CONCLUSIONS

Theory

# Theoretical Framework for Design with Navier-Stokes equations
# Determine $\rho(s)$ from Direct+Adjoint or from Direct alone

Computational

# Direct and Adjoint Solvers Validated for Plane Diffusers
# Design of 90 degree Bend with Specified Cross-section, Max. $\Delta p$
# Number of Design Cycles < 10
# “Flow” Interpretation of Adjoint Problem

Future Plan

# Apply to 3-D turbulent flow
# Specify mean line, vary cross-section
# Other objectives: Min. Distortion
A Multi-Domain Method for Subsonic Viscous Flows
Daniel C. Chan and Munir M. Sindir
CFD Technology Center
Rocketdyne Division, Rockwell International Corporation
Canoga Park, California

We have developed a Schwarz type domain decomposition method for a pressure base, two- and three-dimensional Navier-Stokes solver. This technique allows one to partition a flow path, which can be characterized by complex geometry and/or complicated flow physics, into smaller sub-domains according to the local geometric simplicity or estimated flow scales. We can, then, sweep the sub-domains in some order and solve the Navier-Stokes equations using as boundary conditions, along the domain interfaces, the Dirichlet conditions which are taken from the most recent update of the solution in the adjacent neighboring domains. With this technique, one can minimize the adverse effects caused by grid skewness and the stiffness problem caused by disparate flow scales.

This code has been successfully applied to complicated engineering problems and the results are presented as separate papers in this conference. Here, we report the results of a few fundamental flow cases to demonstrate that a judicious use of the multi-domain method can offer a significant convergence acceleration over the traditional one-domain method. This method can be extended to exploit the architecture of a parallel computer to further improve the speed.
A MULTI-DOMAIN METHOD FOR SUBSONIC VISCOUS FLOWS

BY
DANIEL C. CHAN AND MUNIR M. SINDIR
CFD TECHNOLOGY CENTER
ROCKETDYNE DIVISION, ROCKWELL INTERNATIONAL

PRESENTED AT THE NASA MARSHALL SPACE FLIGHT CENTER
TENTH WORKSHOP FOR COMPUTATIONAL FLUID DYNAMIC APPLICATIONS IN ROCKET PROPULSION
APRIL 28-30, 1992
MOTIVATION

• OPTIMIZE THE DESIGN OF ROCKET ENGINE COMPONENTS
  • CONSISTENCY AND ACCURACY
  • RAPID TURN AROUND

• CHALLENGES
  • COMPLEX FLOW PATH GEOMETRY
  • COMPLICATED FLOW PHYSICS
  • DISPARATE FLOW SCALES
  • STRONG COMPONENTS INTERACTIONS

• OPTIMIZE THE DESIGN OF A ROCKET PUMP
  • FLANGE-TO-FLANGE ANALYSIS
APPROACH

- PARTITION A COMPLEX FLOW PATH ACCORDING TO
  - GEOMETRIC CONSTRAINTS
    - SIMPLIFY GRID GENERATION PROCESS
  - INTERACTIVE COMPONENTS
    - LACK OF WELL-DEFINED BOUNDARY CONDITIONS
  - FLOW SCALES
    - LOCALIZED POCKET OF RECIRCULATING FLOW

- NUMERICAL ISSUES
  - TRANSFER OF INFORMATION ACROSS INTERFACE
    - ACCURACY AND STABILITY
  - GRID TOPOLOGY
    - SIMPLY CONNECTED
    - MULTIPLY CONNECTED
  - DATA MANAGEMENT
    - NEAREST NEIGHBORS
    - INTERFACIAL CONNECTIVITIES

- HARDWARE ISSUES
  - VECTOR LENGTH
  - CORE MEMORY
  - PARALLELISM
MULTI-DOMAIN PATCHING ALGORITHM

• SCHWARZ APPROACH
  • SWEEP THE SUB-DOMAINS IN A SEQUENTIAL ORDER
  • APPLY DIRICHLET CONDITION ALONG DOMAIN INTERFACES
    • USE MOST RECENT SOLUTION FROM THE NEIGHBORING DOMAINS
  • NO MAJOR MODIFICATION TO BASIC NAVIER-STOKES SOLVER
  • NOT TIME ACCURATE
  • STABILITY COULD DEPEND ON INITIAL CONDITION

• GREEN'S FUNCTION APPROACH
  • FOR LINEAR DIFFERENTIAL OPERATORS ONLY
  • STRONG DOMAIN COUPLING
  • MORE WORK PER TIME STEP
  • TIME ACCURATE
  • FULLY PARALLEL
FLOW OVER A HALF CIRCLE
SINGLE AND FOUR DOMAIN GRID TOPOLOGY
DIFFERENT GRID CONNECTIVITY IN COMPUTATIONAL DOMAIN
FLOW OVER A HALF CIRCLE
SINGLE AND FOUR DOMAIN FLOW SOLUTION
(Re=100)

SINGLE DOMAIN

FOUR DOMAINS

Rockwell International
Rocketdyne Division
GREEN'S FUNCTION MULTI-DOMAIN ALGORITHM
ONE-DIMENSIONAL HELMHOLTZ EQUATION FORMULATION
PULICANI, 1988

• GOVERNING EQUATION AND BOUNDARY CONDITIONS
  \[ U''(x) - \sigma U(x) = f(x), \quad x \in \Omega = [-1;1[ \]
  \[ U(-1) = g^- \text{ and } U(1) = g^+ \]

• ANALYTICAL SOLUTION
  For \( f(x) = -2AC^2(\tanh Cx)(1 - \tanh^2 Cx) - \sigma U_e(x) \)
  \[ U_e(x) = A \tanh Cx + B \]
  where, \( A = 0.5, B = 0.5, C = 20, \sigma = 100 \)
GREEN'S FUNCTION MULTI-DOMAIN ALGORITHM
ONE-DIMENSIONAL HELMHOLTZ EQUATION FORMULATION
PULICANI, 1988

- PARTITION INTO TWO DOMAINS
- CENTRAL DIFFERENCING FOR INTERIOR NODES
- SECOND ORDER BACKWARD DIFFERENCING FOR BOUNDARY NODES
GREEN'S FUNCTION MULTI-DOMAIN ALGORITHM

• SEEK THE SOLUTION IN A FORM OF

\[ U^j = \overline{U}^j + \lambda_1 U_1^j + \lambda_2 U_2^j \text{ where, } j = 1, \ldots, J \text{ number of subdomains} \]

\[ (\overline{U}^j)^{(2)} - \sigma \overline{U}^j = f \text{ with } \overline{U}^j(x^{j-1}) = 0 \text{ and } \overline{U}^j(x^{j}) = 0 \]

\[ (\overline{U}_i^j)^{(2)} - \sigma \overline{U}_i^j = 0 \text{ with } \overline{U}_i^j(x^{j-1}) = -i + 2 \text{ and } \overline{U}_i^j(x^{j}) = i - 1 \]

\[ i = 1, 2 \]

• CONTINUITY AT DOMAIN INTERFACE REQUIRE

\[ U^j(x^j) = U^{j+1}(x^j) \]

\[ \frac{d}{dx} U^j(x^j) = \frac{d}{dx} U^{j+1}(x^j) \]
ONE-DIMENSIONAL HELMHOLTZ EQUATION
WITH 31 POINTS IN EACH DOMAIN, INTERFACE AT X=0
ONE-DIMENSIONAL HELMHOLTZ EQUATION
WITH 5 POINTS IN FIRST DOMAIN, 51 POINTS IN SECOND INTERFACE AT X=-0.1
ONE-DIMENSIONAL HELMHOLTZ EQUATION
WITH 5 POINTS IN FIRST DOMAIN, 61 POINTS IN SECOND INTERFACE AT X=-0.4
FUNDAMENTAL TEST CASES

- LAMINAR BACKWARD FACING STEP
  - ARMALY, DURST, PEREIRA AND SCHONUNG, 1983
  - TWO-DIMENSIONAL ANALYSIS FOR REYNOLDS NUMBER RANGING FROM 50 TO 1,500
  - THREE-DIMENSIONAL ANALYSIS FOR REYNOLDS NUMBER OF 1,000
  - TWO COMPUTATIONAL DOMAINS
  - PARABOLIC VELOCITY PROFILE IMPOSED AT INLET PLANE LOCATED AT FOUR STEP HEIGHTS UPSTREAM OF EXPANSION
  - PREDICTED REATTACHMENT LENGTHS COMPARED WITH EXPERIMENTAL MEASUREMENTS

- LAMINAR DRIVEN CAVITY FLOW
  - GHIA, GHIA AND SHIN, 1982
  - TWO-DIMENSIONAL ANALYSIS FOR REYNOLDS NUMBER RANGING FROM 1 TO 10,000
  - TWO AND FOUR COMPUTATIONAL DOMAINS
  - INITIALIZE WITH STAGNATING CONDITION
2-D LAMINAR BACKWARD FACING STEP
PREDICTED STREAMLINE AT DIFFERENT REYNOLDS NUMBERS

Re=100

Re=389

Re=1,000
2-D LAMINAR BACKWARD FACING STEP
REATTACHMENT LENGTHS AT DIFFERENT REYNOLDS NUMBERS

Rockwell International
Rocketdyne Division
2-D DRIVEN CAVITY
FLOW TOPOLOGY AT DIFFERENT REYNOLDS NUMBERS

Re=10

Re=1,000
2-D DRIVEN CAVITY
PREDICTED STREAMLINES FOR FOUR DOMAIN COMPUTATIONS
USING 81X81 POINTS
(Re=10,000)

Present Result (81x81 Grid)

Ghia's Result (257x257 Grid)
2-D DRIVEN CAVITY
CPU TIME REQUIREMENT FOR FOUR AND SINGLE DOMAIN COMPUTATIONS USING 161X161 POINTS

![Bar Chart]

- **SINGLE DOMAIN**
- **FOUR DOMAINS**

CRAY CPU TIME (SEC.)

REYNOLDS NUMBER

@Ah

Rockwell

Rockwell International

Rockwell International

Rock*ldyn*

Division
2-D CAVITY WITH AN ORIFICE
GRID SYSTEM
2-D CAVITY WITH AN ORIFICE
CONVERGENCE HISTORY AND STREAMLINES FOR TWO-DOMAIN COMPUTATIONS USING 1500 POINTS

LEGEND
- MAIN CAVITY
- ORIFICE

Rockwell International
Rocketdyne Division
2-D CAVITY WITH AN ORIFICE

CPU TIME REQUIREMENT FOR TWO AND SINGLE DOMAIN COMPUTATIONS USING 1500 POINTS

![Bar chart showing CPU time for laminar and turbulent flows in single and two domains.](chart.png)
SUMMARY

- MODELING OF COMPLEX FLOW PATHS
  - GEOMETRIC COMPLEXITY
  - POTENTIAL FOR COMPONENTS COUPLING

- OFFER CONVERGENCE ACCELERATION
  - GROUP DISPARATE FLOW SCALE
  - USE CORE MEMORY
  - USE APPROPRIATE VECTOR LENGTH

EXPLOIT PARALLEL COMPUTER ARCHITECTURE
- CONCURRENT COMPUTATION OF SUB-DOMAINS ON DIFFERENT CPU'S
Statement of Problem: The development of turbulence within rocket propulsion chamber flows remains a difficult problem to predict. Within solid propellant rockets, for example, the flow can exhibit multiple regions of transition to turbulence, and is susceptible to various modes of aeroacoustic interaction, potentially associated with instability of the combustion/flowfield process.

Objective: To formulate, develop and validate a large eddy simulation (LES) method for compressible channel flows. Additionally, to assess the potential and limitations of the method with regard to predicting flows of interest in realistic systems.

Approach: The LES method separates the resolvable scale motions from the unresolvable (subgrid) scales by applying a spatial filter to the compressible Navier-Stokes equations. The subgrid-scale Reynolds terms are modeled using a compressible extension of an existing incompressible model for wall bounded flows. The equations are solved numerically using a modified four-step Runge-Kutta procedure in time and second or fourth-order finite differences in space.

Results and Conclusions: The method has been validated by simulating a low Reynolds number ($Re_b = 5400$), low Mach number ($M_c = 0.3$) turbulent Poiseuille flow. Various statistical comparisons are made with incompressible experimental and direct simulation data at similar Reynolds numbers, including higher-order statistics and spatial correlations. The results compare favorably with the incompressible data.

A high subsonic Mach number ($M_c = 0.3$) turbulent Poiseuille flow is also simulated for comparison with the low Mach number results at nominally constant Reynolds number. The mean velocity profile is seen to depart from the low Mach number profile, corresponding to an expected dependence of the mean density and temperature profiles on Mach number. The turbulence velocity statistics are found to be reasonably independent of Mach number. Pressure fluctuation statistics are also found to scale with the wall shear stress independently of Mach number. Pressure fluctuations, although small in magnitude, are observed not to be isobarically related.

The current simulations have validated the algorithm in the incompressible limit and have demonstrated the ability of the method to simulate high subsonic Mach number flows. The principal impediment in application of the method to practical, high Reynolds number chamber flow problems is the large CPU time requirement of the calculations. At present, this bottleneck is caused in large part by resolution requirements for the turbulence-producing "streaks" in the viscous wall layer. Improvements in the subgrid-scale modeling could greatly reduce CPU requirements, leading to more rapid engineering use.
LARGE EDDY SIMULATION OF COMPRESSIBLE CHANNEL FLOWS*

Robert A. Beddini
Jeffrey P. Ridder

Aerothermal Simulations Laboratory
Department of Aeronautical and Astronautical Engineering
University of Illinois at Urbana-Champaign

CFD Applications in Rocket Propulsion
NASA / MSFC
Huntsville, AL
April 28-30, 1992

*Research sponsored by NASA under grant NGT-50383, and by the National Center for Supercomputing Applications, University of Illinois at Urbana-Champaign
MOTIVATION

- SOLID PROPULSION INTERNAL FLOWS UNDERGO TRANSITION TO TURBULENCE WITHIN THE CHAMBER.

- TRANSITIONAL FLOW AND NEAR-SURFACE TURBULENCE HAVE BEEN SHOWN IN PRIOR STUDIES TO AFFECT PROPELLANT COMBUSTION (EROSIVE BURNING & ACOUSTIC RESPONSE).

- PRIOR STUDIES HAVE SHOWN INADEQUACIES IN TURBULENCE MODELS IN PREDICTING TRANSITIONAL FLOW DEVELOPMENT WITH LARGE SURFACE INJECTION RATES.
  - BOTH \( k-\varepsilon \) AND FULL REYNOLDS STRESS MODELS REQUIRED SUBSTANTIAL PARAMETER ADJUSTMENT FOR SIMPLEST INTERNAL FLOWS.
  - EFFECTS OF NEAR WALL "PSEUDOTURBULENCE" DIFFICULT TO MODEL.
OBJECTIVES

- FORMULATE AND DEVELOP LARGE-EDDY SIMULATION (LES) METHOD FOR COMPRESSIBLE CHAMBER FLOWS.
  
  - NO PRIOR SIMULATIONS OF WALL BOUNDED COMPRESSIBLE TURBULENT FLOWS HAVE BEEN REPORTED.

- VALIDATE MODEL & METHOD BY COMPARISON WITH DIRECT SIMULATIONS OF TURBULENT CHANNEL FLOW AT LOW REYNOLDS NUMBERS.

- ASSESS PRACTICALITY AND LIMITATIONS OF LES METHOD FOR PROBLEMATIC FLOWS.
ACTUAL SOLID ROCKET FLOW
APPROACH

• ADVANTAGES
  • LES APPROACH RESOLVES 3-D TIME-DEPENDENT LARGE SCALE MOTIONS WHICH NATURALLY EXIST IN CHAMBERS.
  • CAN INDICATE POTENTIAL VORTEX/ACOUSTIC INTERACTIONS.
  • CAN PROVIDE TURBULENT LENGTH SCALE INFORMATION NEEDED IN OTHER TURBULENCE MODELS.
  • EMPIRICAL ASSUMPTIONS CONFINED TO SMALLER EDDY SCALES – PRESUMED MORE “UNIVERSAL”.

• DISADVANTAGES
  • EXTENSIVE CPU REQUIREMENTS IMPLY NONROUTINE APPLICATION OF SIMULATIONS AT PRESENT.
  • UNRESOLVED (SUBGRID) SCALES OF TURBULENCE MUST BE EMPIRICALLY MODELLING. (THERE IS NO ALTERNATIVE IF VERY LARGE REYNOLDS NUMBERS ARE CONSIDERED.)
APPROACH - METHOD

- EQUATIONS: 3-D TIME-DEPENDENT COMPRESSIBLE NAVIER-STOKES, FILTERED IN SPACE AND FAVRE AVERAGED.
- SUBGRID REYNOLDS STRESSES AND HEAT FLUX MODELLED USING SMAGORINSKY MODEL (cf MOIN & KIM, ‘83).
- NUMERICAL METHOD:
  - METHOD DOES NOT ASSUME PERIODIC (SPECTRAL) DECOMPOSITION IN ANY DIRECTION.
  - SECOND AND FOURTH ORDER CENTRAL DIFFERENCES IN SPACE ASSESSED.
  - MODIFIED FOURTH-ORDER TIME DIFFERENCING SCHEME DEVELOPED FOR CELL $Re > 10$. RESULTS IN 40% DECREASE IN CPU TIME.
  - CODE WELL VECTORIZED – 150 MFLOPS ON CRAY Y/MP.
APPLICATION CONDITIONS

- POISEUILLE FLOW - Re~ 5400 (BASED ON AV. U AND HEIGHT)
- THOUGH NOT REQUIRED, PERIODIC BC'S IN ON LENGTH AND DEPTH BOUNDARIES WERE USED TO COMPARE WITH PRIOR STUDIES.
- Mach No. OF 0.3 USED TO VALIDATE METHOD AND ASSESS SUBGRID PARAMETERS
- Mach No. OF 0.7 USED TO ASSESS COMPRESSIBLE FLOW EFFECTS W/O ADDITIONAL SHOCK CAPTURING DISSIPATION TERMS.
- GRID RESOLUTION EFFECTS ASSESSED – BASELINE GRID WAS 32 L x 90 H x 80 D. (=> 4 MW MEMORY REQUIREMENT).
Turbulent Channel Flow

X-Y CENTER-PLANE

Y-Z CENTER-PLANE

1.45

u/ub

0
RESULTS (LOW Mach No.)

- Subgrid constant $C_{mk}$ minimized to produce least affect on turbulence.
- Mean velocity profile and Reynolds shear stress profile compare favorably with direct N-S studies and experimental data.
- Wall pressure fluctuations compare favorably with direct N-S solutions, but are 1/3 lower than experimental data.
- Macro length-scales compare favorably with direct N-S solutions, but dissipation length scales were not adequately resolved.
RESULTS (HIGH Mach No.)

- AS EXPECTED, MEAN VELOCITY PROFILE AFFECTED BY MEAN DENSITY GRADIENT IN NEAR WALL REGION, BUT followS CROCCO SCALING.
- REYNOLDS STRESSES AND PRESSURE FLUCTUATIONS SCALE WELL WITH MEAN VELOCITY AT THIS SUBSONIC MACH NUMBER.
- RMS TEMPERATURE AND DENSITY FLUCTUATIONS FOUND TO SCALE NEARLY AS $M^2$, BUT ARE NOT ISOBARICALLY RELATED.
MEAN VELOCITY PROFILE COMPARISON

\[ \frac{\bar{u}}{u_*} \]

- \( M \approx 0.7 \)
- \( M \approx 0.3 \)
- Niederschulte et al.

\[ y/\delta \]

University of Illinois at Urbana - Champaign / Aerothermal Simulations Lab
RESOLVED RMS PRESSURE AND TEMPERATURE FLUCTUATIONS

\[ \frac{p}{p_w u^2} \]

\[ \frac{T'}{T^*} \]

\[ y/\delta \]

\[ M = 0.7 \]
\[ M = 0.3 \]
SUMMARY/CONCLUSIONS

- LES METHOD FOR WALL-BOUNDED COMPRESSIBLE FLOWS DEVELOPED AND VALIDATED.
- FEASIBILITY OF LES SIMULATIONS FOR MORE COMPLEX FLOWS INDICATED PROVIDING MEAN FLOW IS NOMINALLY TWO-D.
- FAVORABLE COMPARISON WITH DIRECT SIMULATION RESULTS AND EXPERIMENTAL DATA AT LOW MACH NO.
  - IMPROVEMENTS REQUIRED IN SUBGRID MODELING.
  - SCALE UP TO VERY-LARGE Re OTHERWISE LIMITED BY RESOLUTION OF NEAR WALL STREAKS
SUMMARY/CONCLUSIONS (2)

- Pressure fluctuation spectra obtained - potential feedback to propellant response.
- Why do direct and LES methods underpredict pressure fluctuations?
- Length-scale data obtained - could provide information to turbulence models for other applications.
- Normalized Reynolds stresses and pressure fluctuations nearly independent of Mach no.
- Normalized temperature and density fluctuations vary nearly as $M^2$
  => Are these accounted for in hot wire data reduction??
Treating Convection in Sequential Solvers

Wei Shyy  
Siddharth Thakur

Department of Aerospace Engineering, Mechanics and Engineering Science  
University of Florida, Gainesville, FL

The treatment of the convection terms in the sequential solver, a standard procedure found in virtually all pressure based algorithms, to compute the flow problems with sharp gradients and source terms is investigated. Both scalar model problem and one-dimensional gas dynamics equations have been used to study the various issues involved. Different approaches including the use of nonlinear filtering technique and the adoption of TVD type schemes have been investigated. Special treatments of the source terms such as pressure gradients and heat release have also been devised, yielding insight and improved accuracy of the numerical procedure adopted.
A Proposed Hierarchy of Test Problems

1. Nonlinear scalar wave equation to study capturing of sharp gradient

2. 1-D gas dynamics in a tube

   (a) to study coupling among $u$, $v$, $p$ and $\rho$

   $\rho = 0.445$
   $m = 0.311$
   $E = 8.928$

   __________________

   $\rho = 0.5$
   $m = 0$
   $E = 1.4275$

   open
   end

   Diaphragm

   (b) to study interaction with B.C.s and among waves

   $p = 3.5277$
   $\rho = 0.445$
   $u = 0$

   closed
   end

   $p = 0.571$
   $\rho = 0.5$
   $u = 0$

   closed
   end

   Diaphragm

   (c) to study formation and propagation of acoustic and entropy waves

   $p = p_0 + 0.2 p_0 \cos(2\pi x / L)$

   (initial perturbation)

   closed
   end

   $p_0 = 40$ atm., $T = 2800$ K, $\gamma = 1.236$
3. 1-D Combusting Flow in a Duct

- to study effect of chemical heat release and its impact on acoustic and entropy waves.
- to study propagation and interaction of nonlinear waves in pressure, thermal and convective fields.
- to understand the longitudinal combustion instability and to devise active control strategy

4. Multidimensional Problems
- For a numerical scheme two features control its performance:
  - Amplification factor \(\rightarrow\) numerical damping
  - Phase angle \(\rightarrow\) numerical dispersion

- Problems of convection treatment:
  - First order upwind scheme \(\rightarrow\) excessive damping
    dispersion problem suppressed
  - Higher order schemes \(\rightarrow\) no excessive damping
    dispersion problem appears

- Two approaches investigated
  - Nonlinear filtering
  - TVD type approach with source term treatment and artificial compression
Backward Euler time stepping scheme:

- **First-order upwind scheme**
  - Exact solution
  - $2\pi/\beta = \text{wavelength}/\Delta x$

- **Second-order central difference scheme**
  - Exact solution
  - $2\pi/\beta = \text{wavelength}/\Delta x$

- **Second-order upwind scheme**
  - Exact solution
  - $2\pi/\beta = \text{wavelength}/\Delta x$
Point: A highly dispersive scheme may become a good one if dispersive problems in high wave number can be fixed; or better yet, to make constructive use of these wiggles.

A possibility: Nonlinear Filtering

Key elements:

* maintains conservation laws
* utilizes standard schemes as basis
* attempts to eliminate wiggles a posteriori (a geometric approach)
* effective only for short wavelength oscillations (2\(\Delta\) and 4\(\Delta\)) & hence can check the filtering effectiveness via grid refinement

Define

--- Energy Content

\[ E = \left[ \sum_{j} (\phi_j - \Phi_j)^2 \right]^{1/2} \]

exact solution
numerical solution

--- Area Content

\[ A = \sum_{j} \phi_j \]

--- Goal:

* minimize \(E\)
* maintain \(A\)
Effect of filtering on oscillations with $2\Delta$ wavelength. For this case one application is sufficient to suppress oscillation completely.

$16\Delta$ sine wave is only slightly altered after first filtering. Further application has no effect. $E = 0.999$
Effect of filtering on a 1-D compressible flow solution.

Re = $10^4$  

time step = 600
Euler Equations as a Simultaneous System

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

\[ U = \begin{bmatrix} \rho \\ m \\ E \end{bmatrix}, \quad F = \begin{bmatrix} m \\ m^2/\rho + p \\ (E + p)m/\rho \end{bmatrix} \]

Can also write

\[ \frac{\partial U}{\partial t} + A(U)\frac{\partial U}{\partial x} = 0, \quad A(U) = \frac{\partial F(U)}{\partial U} \]

Speed of sound

\[ c = \sqrt{\frac{\partial p}{\partial \rho}} = \sqrt{\frac{\gamma p}{\rho}} \]

The eigenvalues of Jacobian matrix are

\( (a^1, a^2, a^3) = (u - c, u, u + c) \)
Sequential Approach with Coordinated Characteristics

\[ \frac{\partial p}{\partial t} + \frac{\partial [\rho u]}{\partial x} = 0 \]

\[ \frac{\partial m}{\partial t} + \frac{\partial [mu]}{\partial x} = -\frac{\partial p}{\partial x} \]

\[ \frac{\partial E}{\partial t} + \frac{\partial [Eu]}{\partial x} = -\frac{\partial (pu)}{\partial x} \]

Pressure terms: source terms

The local characteristic speed: convection speed

\[ a_{j+1/2} = \frac{1}{2} (u_j + u_{j+1}) \]

(same for all equations)
Special Source Term Treatment

Conservation law with a source term $\Psi(u)$

$$u_t + f(u)_x = \psi(u)$$

Examples:

Method I - MacCormack's explicit predictor-corrector method

Method II: Operator splitting (Strang's time-splitting)

$$U^{n+1} = S_{\psi}(k/2) \ S_f(k) \ S_{\psi}(k/2) \ U^n$$

where $S_f$ represents the numerical solution operator for the system

$$u_t + f(u)_x = 0$$

and $S_{\psi}$ is the numerical solution operator for the ODE

$$u_t = \psi(u)$$
a) First-order upwind and second-order Lax-Wendroff schemes.

(b) Harten's TVD scheme ($\delta = 0$).

Density profiles using the simultaneous solution approach using different schemes.
The standard shock tube (open-ended) problem: density and energy profiles using sequential approach with source term treatment, for different values of $\delta$. 

\[ \delta = 0.0 \]

\[ \delta = 0.8 \]
THE RESONANT PIPE PROBLEM

SIMULTANEOUS SOLVER

\[ \lambda = 0.01 \quad \delta = 0.0 \]

SEQUENTIAL SOLVER

\[ \lambda = 0.01 \quad \delta = 0.8 \]
Conclusions

- For sequential solvers, coordination of propagation speed among equations requires extra care.

- Can apply modern TVD type schemes with source term treatment to improve accuracy.

- Can utilize nonlinear filtering techniques to eliminate dispersion problems.

- Several test problems have been investigated; results show promise.
Conference publication includes 59 abstracts and presentations and three invited presentations given at the Tenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion held at George C. Marshall Space Flight Center, April 28–30, 1992. The purpose of the workshop is to discuss experimental and computational fluid dynamic activities in rocket propulsion. The workshop is an open meeting for government, industry, and academia. A broad number of topics are discussed including computational fluid dynamic methodology, liquid and solid rocket propulsion, turbomachinery, combustion, heat transfer, and grid generation.