Thirteenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology

R.W. Williams, Compiler

Proceedings of a workshop held at Huntsville, Alabama
April 25-27, 1995

March 1996
Thirteenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology

R.W. Williams, Compiler
Marshall Space Flight Center • MSFC, Alabama
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>X-34 SMALL REUSABLE BOOSTER (J.W. Cole)</td>
<td>1</td>
</tr>
<tr>
<td>REUSABLE LAUNCH VEHICLE (RLV) PROPULSION TECHNOLOGY (J.E. Lee)</td>
<td>13</td>
</tr>
<tr>
<td>OVERVIEW OF MSFC CFD ACTIVITIES (P. McConnaughey)</td>
<td>41</td>
</tr>
<tr>
<td>CFD ANALYSIS OF THE ATD HPFTP INLET GUIDE VANES (R. Garcia and R. Williams)</td>
<td>57</td>
</tr>
<tr>
<td>DYNAMIC WATER FLOW TESTS USING FOUR-BLADED AXIAL FLOW INDUCERS</td>
<td>95</td>
</tr>
<tr>
<td>(T. Nesman, W. Bordelon, and J. Jong)</td>
<td></td>
</tr>
<tr>
<td>INFLUENCE OF TIP CLEARANCE AND REYNOLDS NUMBER ON THE FLOW IN THE ADP INDUCER (J.G. Moore and J. Moore)</td>
<td>119</td>
</tr>
<tr>
<td>CFD ANALYSIS IN THE DESIGN OF A WATER-JET-DRIVE SYSTEM (R. Garcia)</td>
<td>135</td>
</tr>
<tr>
<td>PROCESS DEMONSTRATION AND REACT CODE VALIDATION FOR SHROUDED INDUCER OPERATION AT AND OFF DESIGN (M. Subbaraman and E. Ascoli)</td>
<td>159</td>
</tr>
<tr>
<td>NUMERICAL SIMULATION OF THE UNSTEADY FLOW FIELD IN AN INTEGRATED CENTRIFUGAL IMPELLER-VANED DIFFUSER SYSTEM (L. Walitt)</td>
<td>177</td>
</tr>
<tr>
<td>NUMERICAL SIMULATION OF DIFFUSER FLOW (W. Chen, E. Ascoli and A. Eastland)</td>
<td>211</td>
</tr>
<tr>
<td>COMPUTATIONAL ANALYSIS OF VOLUTES (E.D. Lynch)</td>
<td>281</td>
</tr>
<tr>
<td>3-D COMPUTATIONAL FLUID STRUCTURE INTERACTION (FSI) FOR TURBOMACHINERY; METHODOLOGY DEVELOPMENT FOR CALCULATION OF ROTORDYNAMIC FLOWS (M. Williams, W. Chen, L. Brozowski, A. Eastland, and M. Sindir)</td>
<td>293</td>
</tr>
<tr>
<td>ROCKETFYNE’S TURBOMACHINERY DESIGN PROCESS IMPROVEMENTS FOR RRTT AND FUTURE PROGRAMS (G.H. Prueger, B. Kovac, W. Chen, E. Ascoli, and D.P. Mondkar)</td>
<td>331</td>
</tr>
<tr>
<td>THREE-DIMENSIONAL VISCOUS FLOW ANALYSIS FOR ROCKET PROPULSION SYSTEM WITH STRUCTURED AND UNSTRUCTURED GRID METHODS (J. Loellbach and F. Tsung)</td>
<td>357</td>
</tr>
<tr>
<td>A MESH GENERATION TEMPLATE FOR TURBOMACHINERY BLADE PASSAGES (E.P. Ascoli and G.H. Prueger)</td>
<td>385</td>
</tr>
</tbody>
</table>
## TABLE OF CONTENTS (Continued)

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>AN ANALYTICAL INVESTIGATION OF THE EFFECTS OF STATOR AIRFOIL CLOWING ON TURBINE PERFORMANCE (L.W. Griffin, F.W. Huber, and O.P. Sharma)</td>
<td>403</td>
</tr>
<tr>
<td>TURBULENCE MODELING AND COMPUTATION OF TURBINE AERODYNAMICS AND HEAT TRANSFER (B. Lakshminarayana and J. Luo)</td>
<td>427</td>
</tr>
<tr>
<td>OPTIMIZATION METHODOLOGY FOR UNCONVENTIONAL ROCKET NOZZLE DESIGN (W. Follett)</td>
<td>467</td>
</tr>
<tr>
<td>COMBUSTOR MODELING UNDER GOX-RICH CONDITIONS (M. Deshpande, D. Schwer, H. Tsuei, S. Venkateswaran, and C.L. Merkle)</td>
<td>479</td>
</tr>
<tr>
<td>OXYGEN-RICH COMBUSTION EXPERIMENTS IN A LOX/\text{GH}_2\text{ UNI-ELEMENT ROCKET} (S.A. Rahman, H.M. Ryan, S. Pal, and R.J. Santoro)</td>
<td>515</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMIC ANALYSES OF OXYGEN-RICH PREBURNERS UTILIZING SECONDARY DILUTION (J.M. Grenda and C.L. Merkle)</td>
<td>537</td>
</tr>
<tr>
<td>SPRAY COMBUSTION MODELING WITH VOF AND FINITE-RATE CHEMISTRY (Y. Chen, H. Shang, P. Liaw and T. Wang)</td>
<td>567</td>
</tr>
<tr>
<td>PROGRESS TOWARDS AN EFFICIENT AND GENERAL CFD TOOL FOR PROPULSION DESIGN/ANALYSIS (C.F. Cox, P. Cinnella, and S. Westmoreland)</td>
<td>593</td>
</tr>
<tr>
<td>NUMERICAL MODELING OF SPRAY COMBUSTION WITH AN UNSTRUCTURED-GRID METHOD (H.M. Shang, Y.S. Chen, P. Liaw, M.H. Shih, and T.S. Wang)</td>
<td>613</td>
</tr>
<tr>
<td>NUMERICAL INVESTIGATION OF TWO-PHASE TURBULENT FLOW OF CHARGED DROPLETS IN ELECTROSTATIC FIELD (S. Kim)</td>
<td>641</td>
</tr>
<tr>
<td>GAS/GAS INJECTOR TECHNOLOGY PROGRAM (K. Tucker, B. Palaszewski, B. Santoro, and S. Farhangi)</td>
<td>669</td>
</tr>
</tbody>
</table>
# THIRTEENTH WORKSHOP FOR COMPUTATIONAL FLUID DYNAMIC APPLICATIONS IN ROCKET PROPULSION AND LAUNCH VEHICLE TECHNOLOGY - VOLUME II

## TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>COMBUSTION ZONE CHARACTERIZATION OF GO\textsubscript{2}/GH\textsubscript{2} ROCKET USING LASER-INDUCED FLUORESCENCE OF OH (M. D. Moser and R.J. Santoro)</td>
<td>685</td>
</tr>
<tr>
<td>SPATIALLY RESOLVED SPECIES MEASUREMENTS IN A GO\textsubscript{2}/GH\textsubscript{2} PROPELLANT ROCKET (M.J. Foust, T. Ni and R.J. Santoro)</td>
<td>703</td>
</tr>
<tr>
<td>FLUCTUATING PRESSURE ANALYSIS OF A 2-D SSME NOZZLE AIR FLOW TEST (D. Reed and H. Hidalgo)</td>
<td>723</td>
</tr>
<tr>
<td>EXPERIMENTAL INVESTIGATION OF THE EFFECTS OF ACCELERATION ON HEAT TRANSFER IN THE TURBULENT BOUNDARY LAYER (W.M. Chakroun and R.P. Taylor)</td>
<td>745</td>
</tr>
<tr>
<td>COMPUTATIONAL AND EXPERIMENTAL EFFORTS IN GRAVITY PROBE B MICROTHRUSTER ANALYSIS (A. Droege, A. Smith and J. Carter)</td>
<td>773</td>
</tr>
<tr>
<td>A PARAMETRIC STUDY OF A PLUG NOZZLE USING THE LIQUID PROPELLANT PROGRAM (LPP) CODE (S.S. Dunn and D.E. Coats)</td>
<td>791</td>
</tr>
<tr>
<td>CFD ANALYSIS OF MODULAR THRUSTERS PERFORMANCE (R.J. Ungewitter, J. Beck and A. Ketchum)</td>
<td>813</td>
</tr>
<tr>
<td>PROPELLANT CHEMISTRY FOR CFD APPLICATIONS (R.C. Farmer, P.G. Anderson, and G.C. Cheng)</td>
<td>829</td>
</tr>
<tr>
<td>THERMO-KINETICS CHARACTERIZATION OF KEROSENE/RP-1 COMBUSTION FOR TRIPROPELLANT ENGINE DESIGN CALCULATIONS (T.-S. Wang)</td>
<td>861</td>
</tr>
<tr>
<td>APPLICATION OF OPTIMIZATION TECHNIQUES TO DESIGN OF UNCONVENTIONAL ROCKET NOZZLE CONFIGURATIONS (W. Follett, A. Ketchum, A. Darian, and Y. Hsu)</td>
<td>879</td>
</tr>
<tr>
<td>TRANSONIC AERODYNAMIC CHARACTERISTICS OF A PROPOSED WING BODY REUSABLE LAUNCH VEHICLE CONCEPT (A.M. Springer)</td>
<td>889</td>
</tr>
<tr>
<td>ASCENT AERODYNAMIC PRESSURE DISTRIBUTIONS ON WB001 (B. Vu, J. Ruf, F. Canabal and J. Brunty)</td>
<td>907</td>
</tr>
<tr>
<td>ASSESSMENT OF LIFTING BODY LINEAR AEROSPIKE PLUME EFFECTS ON VEHICLE AERODYNAMICS (J.H. Ruf, A.L. Frost, B. Vu and F. Canabal)</td>
<td>935</td>
</tr>
<tr>
<td>TPS SIZING FOR ACCESS-TO-SPACE VEHICLES (W. Henline, D. Olynick, G. Palmer and Y.-K. Chen)</td>
<td>963</td>
</tr>
<tr>
<td>COMPUTATIONAL ISSUES ASSOCIATED WITH TEMPORALLY DEFORMING GEOMETRIES SUCH AS THRUST VECTORING NOZZLES (K. Boyalakuntla, B.K. Soni, H.J. Thornburg and R. Yu)</td>
<td>977</td>
</tr>
<tr>
<td>Topic</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>HYBRID GRID TECHNIQUES FOR PROPULSION APPLICATIONS</td>
<td>1007</td>
</tr>
<tr>
<td>(R.P. Koomullil, B.K. Soni and H.J. Thornburg)</td>
<td></td>
</tr>
<tr>
<td>A STRUCTURED GRID BASED SOLUTION-ADAPTIVE TECHNIQUE FOR COMPLEX</td>
<td>1037</td>
</tr>
<tr>
<td>SEPARATED FLOWS (H. Thornburg, B.K. Soni, B. Kishore and R. Yu)</td>
<td></td>
</tr>
<tr>
<td>GENIE++ - A MULTI-BLOCK STRUCTURED GRID SYSTEM (T. Williams,</td>
<td>1073</td>
</tr>
<tr>
<td>N. Nadenthiran, H. Thornburg and B.K. Soni)</td>
<td></td>
</tr>
<tr>
<td>SURFACE AND VOLUME GRID GENERATION IN PARAMETRIC FORM</td>
<td>1119</td>
</tr>
<tr>
<td>(T. Yu, B.K. Soni, T. Benjamin, and R. Williams)</td>
<td></td>
</tr>
<tr>
<td>OVERVIEW OF CFD ANALYSES SUPPORTING THE REUSABLE SOLID ROCKET</td>
<td>1153</td>
</tr>
<tr>
<td>MOTOR (RSRM) PROGRAM AT MSFC (E. Stewart, P. McConnaughey,</td>
<td></td>
</tr>
<tr>
<td>J. Lin, E. Reske, D. Doran, R.H. Whitesides, and Y.-S. Chen)</td>
<td></td>
</tr>
<tr>
<td>RSRM CHAMBER PRESSURE OSCILLATIONS: TRANSIT TIME MODELS</td>
<td>1169</td>
</tr>
<tr>
<td>AND UNSTEADY CFD (T. Nesman and E. Stewart)</td>
<td></td>
</tr>
<tr>
<td>A COUPLED CFD/FEM STRUCTURAL ANALYSIS TO DETERMINE DEFORMED</td>
<td>1189</td>
</tr>
<tr>
<td>SHAPES OF THE RSRM INHIBITORS (R.A. Dill and R. H. Whitesides)</td>
<td></td>
</tr>
<tr>
<td>CFD FLOW ANALYSIS AND CODE VALIDATION FOR THE MSFC EIGHT-PERCENT</td>
<td>1227</td>
</tr>
<tr>
<td>ASRM COLD FLOW MODEL PART II (J. Lin and E. Reske)</td>
<td></td>
</tr>
<tr>
<td>APPLICATION OF TWO-PHASE CFD TO THE DESIGN AND ANALYSIS OF A</td>
<td>1245</td>
</tr>
<tr>
<td>SUBSCALE MOTOR EXPERIMENT TO EVALUATE PROPELLANT SLAG PRODUCTION</td>
<td></td>
</tr>
<tr>
<td>(R.H. Whitesides and R.A. Dill)</td>
<td></td>
</tr>
<tr>
<td>NUMERICAL INVESTIGATION OF SLAG BEHAVIOR FOR RSRM</td>
<td>1293</td>
</tr>
<tr>
<td>(P. Liaw, Y.-S. Chen, H. Shang, M. Shih, D. Doran and E. Stewart)</td>
<td></td>
</tr>
<tr>
<td>COMBUSTION PROCESSES IN HYBRID ROCKET ENGINES (S. Venkateswaran</td>
<td>1313</td>
</tr>
<tr>
<td>and C.L. Merkle)</td>
<td></td>
</tr>
<tr>
<td>CFD ANALYSIS OF THE 24-INCH JIRAD HYBRID ROCKET MOTOR</td>
<td>1337</td>
</tr>
<tr>
<td>(P.-Y. Liang, R. Ungewitter and S. Claflin)</td>
<td></td>
</tr>
</tbody>
</table>
Combustion Zone Characterization of GO$_2$/GH$_2$ Rocket Using Laser-Induced Fluorescence of OH.

M. D. Moser and R. J. Santoro

Propulsion Engineering Research Center
and
Department of Mechanical Engineering
The Pennsylvania State University
University Park, PA 16802

With recent interest in gas/gas injectors for use in rocket combustors, there is a critical need for experiments that address this combustion process in terms of detailed flowfield measurements. Such measurements would also serve as a data base for validating computational fluid dynamic (CFD) computer codes. A series of studies have been undertaken at the Propulsion Engineering Research Center (PERC) at the Pennsylvania State University to measure various parameters such as velocity, species concentration, and temperature downstream of a shear coaxial injector in an optically accessible uni-element rocket chamber. Techniques applied to this study to date include the following: laser Doppler velocimetry (LDV) for velocity; laser light scattering (LLS) for flow visualization and estimating mixture fraction and density; laser-induced fluorescence (LIF) of hydroxyl radicals (OH) to determine the characteristics and extent of the reaction zone; and Raman spectroscopy to measure major species concentrations and temperature. The results of the LIF studies are presented here.

The OH molecule is a key intermediate in hydrocarbon and hydrogen combustion. High OH concentration, indicated by high fluorescence intensity, mark the location of the primary reaction zone where the oxidizer to fuel ratio is nearly stoichiometric. Two-dimensional imaging of LIF near the injector face provides a qualitative view of the reaction zone structure. Two-dimensional LIF was limited to qualitative measurements near the injector face due to poor signal to noise ratio with the present experimental setup.

One-dimensional measurements of LIF, which provide a radial profile of relative OH concentration, have been made at several axial locations in the combustion chamber. Results from multiple images, typically 120, have been averaged to yield average OH profiles at each axial location probed. Probability density functions (PDF) of OH peak widths and locations show that the reaction zone is thin near the injector face as expected and remains thin as the flow progresses downstream. Also, the increase in widths of the average OH peaks as the flow progresses downstream is due to movement of the thin reaction zone rather than an increase in individual OH peaks. This analysis indicates the flame is a wrinkled laminar flame front in the region probed.
COMBUSTION ZONE CHARACTERIZATION OF GO₂/ GH₂ ROCKET USING LASER-INDUCED FLUORESCENCE OF OH

Marlow D. Moser and Robert J. Santoro

13th Workshop for CFD Applications in Rocket Propulsion
Marshall Space Flight Center

April 25-27, 1995
MOTIVATION

Need for Low Cost, Reusable, and Reliable Propulsion Systems Require Detailed Understanding of Combustion Phenomena

- Full Scale Tests Are Expensive
- Computer Technology Enables Detailed Modeling
- Advances in Optical Diagnostic Techniques Enhance Measurement Capabilities
OBJECTIVE

- Apply Laser-Based Diagnostics To Study Flowfield Of Gas/Gas Coaxial Injector
  - Laser Doppler Velocimetry
  ✓ Laser-Induced Fluorescence of OH
  - Laser Light Scattering From Tracer Particles
  - Raman Spectroscopy

- Obtain Data Where Boundary Conditions Are Well Specified And Provide To Rocket Research Community
IMPACT

- Gain Insight in Gas/Gas Injector Design

- Obtain Data Base for CFD Code Validation

- Extend Application of Laser Based Diagnostics to High Pressure & Reynold's Number Flows
CRYOGENIC COMBUSTION LABORATORY

Control Room
- Rack #2
- Data Acquisition Systems
- Control Panel

Instrumentation Room
- Laser Systems
- Optical Access

Test Cell
- Test Stand
- Propellant Supply System
- Optical Tables
- LOX Tank

Propellant Storage Area
- $\text{GH}_2$
- $\text{GN}_2$
- $\text{GD}_2$
OPTICALLY-ACCESSIBLE ROCKET CHAMBER
## TEST CONDITIONS

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\text{GH}_2$ Mass Flow Rate</td>
<td>0.010 (0.022) kg/s (lbm/s)</td>
</tr>
<tr>
<td>$\text{GO}_2$ Mass Flow Rate</td>
<td>0.042 (0.093) kg/s (lbm/s)</td>
</tr>
<tr>
<td>$\text{GO}_2/\text{GH}_2$ Mass Flow Ratio</td>
<td>4.2</td>
</tr>
<tr>
<td>Chamber Pressure</td>
<td>1.31 (191) MPa (psia)</td>
</tr>
<tr>
<td>Annulus Reynolds Number</td>
<td>$6 \times 10^4$</td>
</tr>
<tr>
<td>Post Reynolds Number</td>
<td>$3 \times 10^5$</td>
</tr>
</tbody>
</table>
LASER-INDUCED FLUORESCENCE OF OH

- Hydroxyl-Radicals (OH) Are a Key Intermediate
- Relatively Simple Technique
- Indicates Reaction Zone Location and Structure
LASER-INDUCED FLUORESCENCE OF OH

- Excite (1,0) Band of OH
- Collect Fluorescence from (1,1) and (0,0) Band
- Two-Dimensional Images
  - Laser Beam Formed into Sheet
  - Laser Tuned to Q₁(9) and Q₂(8) Lines at 283.92 nm
- One-Dimensional Images
  - Laser Beam Focused
  - Laser Tuned to P₂(8) Line at 285.98 nm or to S₂₁(8) Line at 278.83 nm
LASER-INDUCED FLUORESCENCE
Experimental Setup
PEAK WIDTH PDFs

12.7 mm
25.4 mm
50.8 mm
127 mm

PDF

FWHM (mm)
PEAK POSITION PDFs & LIF PROFILES
PEAK POSITION PDFs & LIF PROFILES

Z = 50.8 mm

Z = 127 mm

PDF

LIF

r (mm)
SUMMARY/CONCLUSIONS

- Gas/Gas Shear Coaxial Injector Exhibits Poor Mixing Characteristics

- Reaction Zone has Characteristics of Thin Wrinkled Laminar Flame Front
  - Reacting Flowfield Dominated by Large Scale Turbulence

- Data Available To Rocket Research Community
  - Average LIF Profiles Useful for General Profile Shape and Location Comparisons
  - Velocity and Light Scattering Data Also Available
Spatially Resolved Species Measurements in a GO$_2$/GH$_2$ Propellant Rocket

M.J. Foust, T. Ni and R.J. Santoro

Propulsion Engineering Research Center and
Department of Mechanical Engineering
The Pennsylvania State University
University Park, PA 16802

Statement of Problem

Local species concentration and temperature are among the most important parameters to characterize in a combustion system. However, the harsh environment in rocket chambers under hot-fire conditions limits the applicability of conventional probing methods for the acquisition of this information. Laser-based diagnostic methods show great promise for achieving this goal by providing instantaneous images of species concentrations and temperature.

Objective of Work

The objective of the current work is to develop a non-intrusive technique to experimentally determine the major species and temperature field in the combustion chamber of an uni-element rocket for a GO$_2$/GH$_2$ propellant combination.

Approach

The experiments were conducted at the Cryogenic Combustion Laboratory at Penn State University. The uni-element rocket chamber used is modular in design and is easily configured to provide optical access along the chamber length. A shear coaxial injector was used to introduce GO$_2$ and GH$_2$ into the combustion chamber. The nominal flow rates of GO$_2$ and GH$_2$ are 0.1 lb/s and 0.025 lb/s, respectively, resulting in an O/F ratio of four. The experiments were for a chamber pressure of 190 psia.

One-dimensional profiles of species concentrations and temperature were measured by using laser-induced spontaneous Raman spectroscopy. The Raman system consists of a flash pumped dye laser operating at 10 Hz and an intensified CCD camera. The dye laser has a typical pulse energy of 2 J at 511 nm and a pulse duration of 5 µs. The Raman emission was detected at a right angle to the laser beam. A narrow band interference filter was placed in front of the camera to selectively measure the number density of the species. For each rocket firing, 50 single-shot Raman images and 50 background images were captured. By using different optical filters, Raman images of oxygen, hydrogen, water and nitrogen (used for window cooling) were obtained. Measurements were conducted at 1, 2, and 5 inch downstream of the injector face. The ratio of the signal to background level for hydrogen and oxygen Raman images at 1 inch downstream is about 10. Further downstream, the background luminosity increases significantly. Thus, the species concentration can only be determined from averaged Raman images. The temperature profiles were calculated from averaged data of total species number density using the ideal gas law. Since the Raman signal is stronger in lower temperature regions, the averaged temperature generally underestimates the temperature in regions where temperature fluctuates highly.

Conclusions

Single-shot and averaged profiles of species concentration have been measured under combusting conditions. These results demonstrate that the laser-based technique can be effectively applied for in-situ measurements in a rocket chamber. Experiments with an improved detection system for obtaining images of instantaneous and simultaneous multi-species concentration and temperature are underway.
SPATIALLY RESOLVED SPECIES MEASUREMENTS IN A GO$_2$/GH$_2$ PROPELLANT ROCKET

M. J. Foust, T. Ni and R. J. Santoro

13th Workshop for CFD Applications in Rocket Propulsion
NASA/ Marshall Space Flight Center

April 25-27, 1995

PENN STATE
OVERVIEW

- Demonstrate Application of Raman Spectroscopy
- Present Measurements Using 2 Techniques
  - Optical Bandpass Filters
    - Good Light Collection for Obtaining Single Species
    - Cannot Assess Flame Luminosity Contribution
    - Simultaneous Species Measurements Require Multiple Detectors
  - Spectrometer
    - Provides Simultaneous Species Measurements Using Single Detector
    - Assessment of Flame Background
    - Needs Wide Dynamic Range
- Summarize Raman Experiments

PENNSTATE

Propulsion Engineering Research Center
MOTIVATION

- Improved Fundamental Understanding of Combustion-Driven Flows
- Combusting Flowfield Measurements used for CFD Code Validation
- Performance Assessment for Candidate Gas/Gas Injectors in Reusable Launch Vehicle Program
APPRAOCHE

• Experiments in a Uni-Element Rocket

• Apply Laser-Based Diagnostic Techniques
  - Laser Doppler Velocimetry
  - Laser-Induced Fluorescence of OH
  - Laser Light Scattering from Tracer Particles
  ✔ Laser-Induced Raman Spectroscopy
OPTICALLY-ACCESSIBLE ROCKET CHAMBER

Nitrogen Purge

Gaseous Hydrogen

Slot Window

Igniter

Cooling Water In

Gaseous Oxygen

Viewing Window

Cooling Water Out

PENNSTATE

Propulsion Engineering Research Center
**GO₂/GH₂ ROCKET TEST CONDITIONS**

Shear Coaxial Injector

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>GH₂ Mass Flow Rate</td>
<td>0.010 (0.023) kg/s (lbm/s)</td>
</tr>
<tr>
<td>GO₂ Mass Flow Rate</td>
<td>0.042 (0.092) kg/s (lbm/s)</td>
</tr>
<tr>
<td>GN₂ Mass Flow Rate</td>
<td>0.010 (0.022) kg/s (lbm/s)</td>
</tr>
<tr>
<td>GO₂/GH₂ Mixture Ratio</td>
<td>4.0</td>
</tr>
<tr>
<td>Chamber Pressure</td>
<td>1.29 (187) MPa (psia)</td>
</tr>
</tbody>
</table>
RAMAN SPECTROSCOPY
Stokes Vibrational Shift

- Raman Signal Specific to Each Species
- Linearly Proportional to Species Number Density
- $1/\lambda^4$ Wavelength Dependence

PENN STATE
Propulsion Engineering Research Center
EXPERIMENTAL SETUP

- Optimized System for Collecting Weak Raman Signal
SPECIES MEASUREMENTS IN
\( \text{GO}_2/\text{GH}_2 \) Rocket

- Obtained 35 Instantaneous Line Images Individually for Each Species (\( \text{GO}_2, \text{GH}_2, \text{H}_2\text{O} \) and \( \text{GN}_2 \) - Purge) During 4 sec. Rocket Firings

- Calibrated Measurements for Each Species Using Pure Gas Concentration

- Extracted Average Species Mole Fractions from 100 Images
INSTANTaneous RAMAN IMAGES

Shear Coaxial Injector
25.4 mm from Injector Face

O₂  H₂  H₂O

MIN  False Color Scale  MAX

Propulsion Engineering Research Center
**MEAN RAMAN IMAGES**

Shear Coaxial Injector

\[ Z = 25.4 \text{ mm} \quad Z = 50.8 \text{ mm} \quad Z = 127 \text{ mm} \]

\[ \text{O}_2 \quad \text{H}_2 \quad \text{H}_2\text{O} \]

\[ \text{MIN} \quad \text{False Color Scale} \quad \text{MAX} \]

PENNSTATE

Propulsion Engineering Research Center
SPECS MEASUREMENTS

Radial Profiles of Species Mole Fractions

$\text{N}_2$

$\text{H}_2$

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

$\text{O}_2$

$z = 127 \text{ mm}$

$z = 50.8 \text{ mm}$

$z = 25.4 \text{ mm}$

Mole Fraction

r (mm)
EXPERIMENTAL MEASUREMENTS
Mean Density Comparisons @ 25.4 mm from Injector Face

![Graph showing mean density comparisons between Raman and Light Scattering methods at a distance of 25.4 mm from the injector face. The x-axis represents the radial distance (r) in mm, and the y-axis represents the mean density in kg/m³.]
GO$_2$/GH$_2$ ROCKET
Oxygen Mole Fraction Comparisons

- Raman
- Numerical

$Z = 25.4$ mm  $Z = 50.8$ mm  $Z = 127$ mm

- Computation Predicts Faster Combustion Rates

PENNSTATE
Propulsion Engineering Research Center
SINGLE-SHOT MULTI-SPECIES DETECTION (Recent Development)
SIMULTANEOUS SPECIES MEASUREMENTS
25.4 mm from Injector Face

Log Intensity Scale

Linear Intensity Scale

position

wavelength

H₂
648 nm
H₂O
628 nm
N₂
580 nm
O₂
555 nm
Laser/Rayleigh
511 nm

PENN STATE

Propulsion Engineering Research Center
SUMMARY/CONCLUSIONS

- 2 Techniques for Raman Species Detection
  - Optical Bandpass Filters
    - Single-Shot Measurements for a Single Species
    - Spatially Resolved Average Species Mole Fractions
    - Feasible in Relatively Low Flame Background
  - Spectrometer
    - Single-Shot Multiple Species Measurements
    - Possible Method for Subtracting Flame Background Luminosity

- Shear Coaxial Injector Exhibits Poor Mixing
  - $\text{GO}_2$ Core Region Extends Beyond Mid-Chamber
  - $\text{H}_2\text{O}$ Mole Fraction Levels Low
ACKNOWLEDGEMENTS

• Funding by NASA/Marshall Space Flight Center Under Contract NAS-8-38862 is Gratefully Acknowledged
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Darren Reed
Induced Environments Branch
NASA Marshall Space Flight Center, Huntsville, Alabama 35812

Homero Hidalgo
SSME Assurance Office
NASA Marshall Space Flight Center, Huntsville, Alabama 35812

Abstract

To better understand the SSME startup/shutdown transients, an airflow test of a 2-D nozzle was conducted at MSFC's Trisonic wind tunnel. Photographic and other instrumentation show during a SSME start large nozzle shell distortions occur as the Mach disk is passing through the nozzle. During the earlier development of the SSME, this startup transient resulted in a low cycle fatigue failure of one of the LH2 feedlines. The 2-D SSME nozzle test was designed to measure the static and fluctuating pressure environment and color schlieren video during the startup and shutdown phases of the run profile.

The model consisted of two identical blocks having the same inner contour of the SSME nozzle. The sides of the nozzle were made of glass for schlieren photography. The upper block was instrumented for static pressure measurements. The lower block was instrumented with thirteen Entron fluctuating pressure transducers. Steady state and slow sweep flows were tested for three back pressure conditions (0.5-2.0 psi, 7 psi, 14 psi.) The static pressure data was acquired by a scanning pressure system. The fluctuating pressure data was recorded onto a VHS analog tape recorder. The video, static pressure, and fluctuating pressure data were time synchronized for data correlation.

The schlieren video clearly shows a lambda (λ) shock foot moving down the throat during the slow sweep. The fluctuating pressure RMS time histories show the levels increase as the downstream foot of the lambda shock approaches. When the shock foot is directly above the transducer, levels decrease about 50%. When the upstream leg of the lambda shock approaches the transducer the level quickly jumps up to twice the downstream leg values. After the upstream leg of the lambda shock passes the transducer, the level falls down to the noise floor of the measurement.

Schlieren video, model configuration, fluctuating pressure time histories, power spectrum densities of the test will be shown. Future 2-D nozzle tests and plans for a 3-D nozzle facility will be addressed.
Fluctuating Pressure Analysis of a 2-D Space Shuttle Main Engine (SSME) Nozzle Air Flow Test

Darren Reed
Homero Hidalgo
NASA / MSFC

Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology
Huntsville, Alabama
26 April 1995
Introduction / History

- SSME Nozzles are subjected to significant unsteady aerodynamic forces during engine start and shutdown transients
- High loads are associated with the start / shutdown nozzle transients
  - High Stress in Nozzle Aft Region (Excitation of nozzle flexural modes)
  - Actuator Sideloads
- These transients were severe enough to cause two major test failures of the large coolant supply tubes, downcomers (steerhorn failures)
  - First failure: Test 750-041 (14 May 1979) Engine E0201
    - fatigue load failure
    - resolved by increasing steerhorn thickness
  - Second failure: Test SF6-03 (4 Nov 1979) Engine E2002
    - incorrect weld material
    - resolved by adding nickel plating to tee weld joints, added steam loop to coolant line
SSME COMBUSTION DEVICES

Actuator Connection
Points on SSME => Sideloads

Low Stress
Random Excitation

Moderate Stress

High Stress
Nozzle shell modes

Added Steam Loop
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Test Objectives

- To better understand the unsteady nozzle flows, a wind tunnel experiment using a scaled 2-D (planar) contour model of the SSME nozzle was run
- Tests were conducted at MSFC’s 14 inch Trisonic Wind Tunnel facility
- Model was instrumented to measure static, fluctuating pressures, and coloured schlieren videotapes
  - Recording schlieren video of the shock structure as it move out of the nozzle during startup and back in during shutdown was one of the main objectives
  - The static pressure ports would help define the relative strength of the shocks
  - The fluctuating pressure transducers were used to measure the unsteady levels and the show the spectrum shape
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Model and Test Descriptions

- 2-D SSME contour shape
  - Area Ratio = 8.8:1
  - Nozzle Length = 11 inches (6.8 inches from throat to exit)
  - Nozzle Width = 5.0 inches
  - Nozzle Exit Height = 5.0 inches
  - Throat Height = 0.568 inches

- Model Instrumentation
  - 18 Ports - 12 Fluctuating Pressure Transducers Recorded (Lower Block)
  - 18 Static Pressure Ports (Top Block)

- Facility Measurements
  - Total Pressure, Total Temperature, and Static Pressure at Nozzle Exit
  - Schlieren Video

- Test Conditions
  - 3 Nozzle exit pressure conditions (2 psia, 7 psia, and atmospheric)
  - Slow sweep runs
  - 5 steady state runs at predetermined shock locations
NOZZLE BLOCK MODEL AND PRESSURE PORT LOCATIONS

TOP VIEW OF NOZZLE BLOCK

PRESSURE PORT LOCATIONS

PORTS IN BLACK INDICATE PORTS AVAILABLE FOR DYNAMIC DATA ACQUISITION

SIDE VIEW OF NOZZLE BLOCK

NOZZLE THROAT

PRESSURE PORT LOCATIONS
Test Section With 2-D SSME Nozzle Contour Model Installed

- WINDOWS
- STRUTS
- Po, To
- SSME NOZZLE CONTOUR MODEL
- SCREENS
- Pinf
- HIGH PRESSURE AIR SUPPLY
- HAND CRANK (FOR OUTER WALL ADJUSTMENT)

NOTE: NOT TO SCALE
Conclusions

- Fluctuating Pressures are highest at the upstream edge of the lambda shock
- Fluctuating pressure levels decrease “inside” the shock foot
- Spectrum shapes show mostly low frequency energy - this is consistent with similar flow conditions (external bow shock impingement)
- Nondimensional amplitude, $\Delta C_p$, levels are similar to external flow conditions
- The plane flow nozzle with side windows is a good method to observe the shock wave patterns
- Data from this experiment have helped describe the unsteady aerodynamic forces a nozzle experiences during startup and shutdown
Future Fluctuating Pressure Analyses of Nozzle Transients

- The Fluid Dynamics Division has developed plans for 3-D subscale nozzles with the following capabilities:
  - Maximum test pressure: 350 psi (nitrogen)
  - Maximum flow rate primary: 12 lb/s @ 810 °
    secondary: 50 lb/s
  - Minimum back pressure: 0.05 psia
  - Maximum run duration: 360 sec. @ 12 lb/s
  - Maximum supply temperature: 350 °F
  - Maximum testable area ratio: 230
  - Test Cabin Size: 3 ft diameter x 5 ft

- Two different nozzle contours are to be tested in the Trisonic wind tunnel special test section this July

- The new nozzles will be instrumented and tested similar to the SSME nozzle
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Fluid Dynamics Division's 3-D Nozzle Test Facility
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Rocketdyne Tripropellant Nozzle Contour

![Diagram of Rocketdyne Tripropellant Nozzle Contour]
Fluctuating Pressure Analysis of a 2-D SSME Nozzle Air Flow Test

Dual Bell Concept Nozzle Contour
EXPERIMENTAL INVESTIGATION OF THE EFFECTS OF ACCELERATION ON 
HEAT TRANSFER IN THE TURBULENT BOUNDARY LAYER

Walid M. Chakroun
and
Robert P. Taylor

Department of Mechanical Engineering
Mississippi State University
Mississippi State, MS 39762

Abstract

Rocket propulsion system components such as turbines/pumps and nozzles often have aerodynamically-rough surfaces or surfaces which become rough during operation. Also, these surfaces are often in regions of accelerating flow. The interaction between surface roughness and acceleration is complicated and not predicted by a simple superposition of flat-plate rough-wall correlations and smooth-wall acceleration effects.

For the smooth wall, acceleration causes a decrease in the Stanton number when compared with equivalent unaccelerated flow. When the acceleration is strong enough, the turbulent boundary-layer heat transfer rates will approach those of a laminar flow and the boundary layer is said to have relaminarized. Under proper conditions, rough-wall accelerated flow can have the opposite behavior with increasing Stanton numbers and hence much larger heat transfer rates.

The objective of this research was to experimentally investigate the combined effects of freestream acceleration and surface roughness on heat transfer and fluid flow in the turbulent boundary layer. The experiments included a variety of flow conditions ranging from aerodynamically-smooth to transitionally-rough to fully-rough boundary layers with accelerations ranging from moderate to moderately strong. The test surfaces used were a smooth-wall test surface and two rough-wall test surfaces which were roughened with 1.27 mm diameter hemispheres spaced 2 and 4 base diameters apart in a staggered array. The experiments were conducted in the Turbulent Heat Transfer Test Facility in the mechanical engineering laboratories at Mississippi State University. The measurements consisted of Stanton number distributions, mean-temperature profiles, skin-friction distributions, mean-velocity profiles, turbulence-intensity profiles, and Reynolds-stress profiles.

The Stanton numbers for the rough-wall experiments increased with acceleration. For aerodynamically-smooth and transitionally-rough boundary layers, the effect of the roughness is not seen immediately at the beginning of the accelerated region as it is for fully-rough boundary layers; however, as the boundary layer thins under acceleration, the surface becomes relatively rougher resulting in a sharp increase in Stanton number.
EXPERIMENTAL INVESTIGATION OF THE EFFECTS OF ACCELERATION ON HEAT TRANSFER IN THE TURBULENT BOUNDARY LAYER

by
Walid M. Chakroun
and
Robert P. Taylor

Department of Mechanical Engineering
Mississippi State University

April 26, 1995
BACKGROUND

- Acceleration effects on smooth-wall boundary layers--strong accelerations cause sharp decrease in the Stanton number

\[ K = \frac{v}{U_e^2} \frac{dU_e}{dx} \]

\( K > 3.0 \times 10^{-6} \) relaminarization

- Fully-rough boundary layers react to acceleration in the opposite way--Stanton number increases under acceleration.
Effects of acceleration on smooth-wall turbulent Stanton numbers, from Kays and Moffat (1975)
Effects of acceleration on full-rough turbulent Stanton numbers, from Coleman (1976)
In a smooth-wall boundary layer, acceleration stretches the eddies reducing the trans-boundary-layer diffusion, decreases the boundary-layer thickness, increases the viscous sublayer thickness, and reduces the Stanton number.

In a fully-rough boundary layer, acceleration stretches the eddies reducing the trans-boundary-layer diffusion, decreases the boundary-layer thickness, increases the nondimensional size of the roughness elements, and increases the Stanton number.
OBJECTIVES OF THIS WORK

- Experimentally investigate the effects of freestream acceleration on heat transfer in the turbulent rough-wall boundary layer.

- Investigate a variety of flow conditions ranging from aerodynamically-smooth to fully-rough boundary layers.
EXPERIMENTS

Closed-loop boundary-layer wind tunnel with 2.5-m long test section.

Flexible top wall to adjust edge velocity.

Roughness made with 1.27-mm diameter hemispheres spaced 2 and 4 diameters apart.

Stanton numbers determined form energy balance on individual plates.

Velocity and turbulence profiles measured with hot-wire anemometry.
Turbulent heat transfer test facility
Surface roughness description
**HEAT TRANSFER DATA**

<table>
<thead>
<tr>
<th>$K \times 10^6$</th>
<th>$U_\infty$ m/s</th>
<th>L/$d_0$ = 2</th>
<th>L/$d_0$ = 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.3</td>
<td>28</td>
<td>fully rough</td>
<td>fully rough</td>
</tr>
<tr>
<td>0.6</td>
<td>12</td>
<td>fully rough</td>
<td>tran'ly rough</td>
</tr>
<tr>
<td>0.9</td>
<td>8</td>
<td>tran'ly rough</td>
<td>tran'ly rough</td>
</tr>
<tr>
<td>1.4</td>
<td>5</td>
<td>aero'ly smooth</td>
<td>aero'ly smooth</td>
</tr>
</tbody>
</table>

Roughness state based on the boundary layer conditions just upstream of the accelerated region.
\[ K = 1.4 \times 10^{-6}, \ L/d_o = 2, \text{ aerodynamically-smooth approaching flow} \]
$K = 0.95 \times 10^{-6}, \ L/d_0 = 2, \ \text{transitionally-rough approaching flow}$
$K = 0.59 \times 10^{-6}$, $L/d_o = 2$, fully-rough approaching flow
\[ K = 0.34 \times 10^{-6}, \ L/d_0 = 2, \text{ fully-rough approaching flow} \]
$K = 1.4 \times 10^{-6}, \ \frac{L}{d_0} = 4$, aerodynamically-smooth approaching flow
\[ K = 0.85 \times 10^{-6}, \quad L/d_o = 4, \quad \text{transitionally-rough approaching flow} \]
$K = 0.6 \times 10^{-6}$, $L/d_0 = 4$, transitionally-rough approaching flow
K = 0.31 \times 10^{-6}, \quad L/d_0 = 4, \quad \text{fully-rough approaching flow}
<table>
<thead>
<tr>
<th>FLUID-FLOW DATA</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>K x 10^6</td>
<td>U_∞ m/s</td>
<td>L/d_o</td>
</tr>
<tr>
<td>0.3</td>
<td>28</td>
<td>4</td>
</tr>
<tr>
<td>1.4</td>
<td>5</td>
<td>fully rough</td>
</tr>
</tbody>
</table>

Roughness state based on the boundary layer conditions just upstream of the accelerated region.
$K = 1.4 \times 10^{-6}$, $L/d_0 = 2$, aerodynamically-smooth approaching boundary layer
\[ K = 1.4 \times 10^{-6}, \quad L/d_0 = 2, \quad \text{aerodynamically-smooth approaching boundary layer} \]
K = 1.4 \times 10^{-6}, L/d_o = 2, aerodynamically-smooth approaching boundary layer
K = 1.4 \times 10^{-6}, L/d_o = 2, aerodynamically-smooth approaching boundary layer
CONCLUSIONS

- Stanton numbers for fully-rough boundary layers increase with acceleration compared to zero-pressure gradient boundary layers.

- For the flows considered, roughness effects increased in the region of acceleration, indicating flow regime trend toward a rougher state when accelerated.

- For aerodynamically-smooth and transitionally-rough flows the effect of acceleration is not seen immediately at the beginning of the accelerated region as it is for fully-rough flows; however, as the boundary layer thins under acceleration, the surface becomes relatively rougher resulting in a sharp increase in Stanton number.
CONCLUSIONS (Continued)

- After the acceleration, Stanton numbers return to the $K=0$ baseline case only for the fully-rough boundary layers. For the others, the Stanton numbers show a distinct shift indicating different roughness states upstream and downstream of the acceleration.

- Acceleration decreases the turbulent kinetic energy throughout the boundary layer for both the smooth and rough walls.
REFERENCES


Computational and Experimental Efforts
in Gravity Probe B Microthruster Analysis

Gravity Probe B, an experiment to test the theory of relativity, will be launched near the turn of the millennium. Due to the precise pointing requirements needed to successfully carry out this experiment, the satellite will use sixteen proportionally controlled microthrusters as a main component of the attitude control system. These microthrusters use the helium boil-off from the on-board dewar as propellant.

Marshall Space Flight Center, overseeing the project, verified the design of the thruster flow path by both computational and experimental methods. The flow performance of the thruster has been adequately characterized. Graphs show specific impulse, thrust coefficient, discharge coefficient, and mass flow rate trends. Value was added to the program through gained confidence in the design of the thruster and through evaluation of some design trade-offs.

This work may be valuable in the future due to the possible need of small thrusters on spacecraft that have precise pointing requirements.
Computational and Experimental Efforts in GPB Microthruster Analysis

Alan Droge and Andrew Smith
NASA/MSFC

James Carter
Dynamic Engineering, Inc.
April 26, 1995

Presented at Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology
Background

- Gravity Probe B (GPB)
  - A satellite borne relativity experiment
  - Requires precise pointing control and acceleration free environment to be provided by attitude control system

- Microthrusters
  - Helium gas from dewar boiloff used as propellant
  - Sixteen microthrusters on spacecraft; used for orbit trim, spin-up, spin-down, and attitude control
  - Concerns about mission lifetime and control saturation
Objectives

- Verify design of microthruster
  - Thrust
  - Specific Impulse

- Gain knowledge about the physics of rarefied thruster flows
  - In the future, this size of thruster may become more common on spacecraft, due to:
    » extremely tight spacecraft attitude control requirements
    » use of large liquid helium dewars
Method

Direct Simulation Monte Carlo (DSMC):

Limits on areas of application

Slow - not useful for large parametric studies

Works well for low Reynolds number flows, costly to use for higher Reynolds number flows

Gives good characterization of the flowfield

Experiment:

Covers all of the flowfield

Very fast once hardware is in place useful for parametric studies, useful for assessment of configuration change

Possible data scatter at low Reynolds numbers, but works well for higher Reynolds number flows
Results

• Requirements:

  - Thrust ≥ 8 mN at $P_{\text{inlet}} \geq 9.7$ torr and mass flow $\leq 1.52E-05$ lbm/s
  - Thrust $\leq 0.05$ mN at $P_{\text{inlet}} \leq 12.5$ torr and mass flow $\leq 9.48E-07$ lbm/s
  - Thrust $\geq 2.55$ mN at $P_{\text{inlet}} \geq 4.2$ torr and mass flow $\leq 4.85E-06$ lbm/s

• Microthruster Characterization

  - Thrust
  - $I_{sp}$

• Added Value

  - Conical Nozzle vs. Sharp Edged Orifice
  - Analysis of change in piston and valve seat design
Laval Nozzle-Modified Piston:

Thrust = f(Mass Flow)

Meet Requirements

Fails to meet Requirements

DSMC

12.15 M  * 9.7 M  ▲ 6.0 M  + 4.0 M  ○ 3.1 M
Nozzle - Modified Piston:

\[ I_{sp} = f(\text{Mass Flow}) \]

Supply Pressures (torr):
- 12.15 M
- 9.7 M
- 6.0 M
- 4.0 M
- 3.1 M
DSMC Calculation of Specific Impulse

![Graph showing the relationship between specific impulse (Isp) and mass flow rate. The x-axis represents mass flow rate in units of 10^-7 kg/s, and the y-axis represents Isp in seconds. The graph shows a trend of increasing Isp with increasing mass flow rate.]
Computational and Experimental Efforts in GPB Microthruster Analysis

Note: Grids are not on the same scale
GP-B MODEL INSTRUMENTATION
Contributions of the Nozzle Bell and the Cylindrical Throat to the MSFC A2 Configuration Total Thrust

Throat Reynolds Number Approx. = 30

Mass Flow Rate x 10^4 (grams/sec)
Nozzle vs. Sharp Edge Orifice Comparison: Discharge Coefficient

Supply Pressure = 9.7 torr
Laval Nozzle - Original vs Modified Piston: Mass Flow vs Piston Position

Mass Flow Rate (lbm/sec)

Supply Pressure
12.15 torr

Piston Position (microns)
Nozzle vs. Sharp Edge Orifice Comparison: Thrust Coefficient

[Graph showing the comparison between nozzle and sharp edge orifice for thrust coefficient as a function of Reynolds number. The graph includes data points for nozzle and nozzle with DSMC, with supply pressure set at 9.7 torr.]
Conclusions

- Thruster meets or exceeds requirements
- Specific Impulse varies over operating range
- Current nozzle design is adequate throughout operating range
- Change in piston/valve seat increased flow resistance through the thruster
Future Work

- DSMC
  - Plume characterization and comparison with experiment
  - Plume impingement on spacecraft

- Experimental
  - Possible re-run of experiment with highly sensitive force balance in order to eliminate data scatter at low Reynolds numbers

- Both
  - Write detailed NASA Technical Memos or Technical Papers
Subject: A parametric study of a plug nozzle, using the Liquid Propellant Program (LPP) Code
By: Stuart S Dunn, Douglas E Coats, Software and Engineering Associates, Inc.

Abstract

The Liquid Propellant Program (LPP) computer code is a super-set of the industry standard Two Dimensional Kinetics (TDK) computer code, which has been developed by Software and Engineering Associates, Inc. (SEA, Inc.) over the past twelve years. The TDK code uses a Two-Dimensional Method of Characteristics solution with fully coupled finite rate kinetics for axially symmetric nozzles. The chemical reactions are modeled with a generalized reaction package that includes 3rd body efficiencies and four reaction rate forms. The code performs optional solutions for frozen or equilibrium flow. TDK evaluates discrete shocks, both attached or induced. The Transonic module models variable mixture ratio profiles from the combustion chamber injector. The Mass Addition Boundary Layer module (MABL) calculates the boundary parameters with the same chemistry options, and includes transpiration or tangential slot injection of gas at the wall.

The LPP upgrades include: planar nozzles, scarfed nozzles, plug nozzles, and scramjet nozzle configurations. The code evaluates both upper and lower wall flow simulation, and includes the interaction with the external flow. The MABL module evaluates equilibrium radiation heat transfer for both upper and lower walls. In addition, the LPP code models combustion effects due to injector inefficiencies with the Spray Combustion Analysis Program (SCAP) module. The LPP package provides extensive post plotting capabilities for flow visualization. The LPP is sufficiently fast and robust to provide performance predictions for extensive parametric studies and sufficiently accurate to provide flow field and performance solutions for detailed studies.

The evaluation of a planar or axially symmetric plug nozzle has received recent interest due to the SSTO studies. The LPP code allows easy modeling of a plug nozzle configuration, since the user is allowed to input an arbitrary inner and outer wall geometry (referred to as the plug and the cowl). The transonic analysis models both planar or axially symmetric annular flow, including straited and variable mixture ratio profiles. When the internal flow reaches the exit of the outer wall, a Prandtl-Meyer fan allows the flow to expand to the external pressure. At this point, a pressure boundary condition is applied for either quiescent sub-sonic, or super-sonic external flow. The MABL analyses is subsequently performed to evaluate the boundary layer losses for both the inner and outer walls. Following JANNAF standard procedures, the characteristic analysis is automatically repeated with the boundary layer compensated wall geometry.

The above procedure was employed to parametrically evaluate the performance of several plug nozzle configurations at different flight conditions. The altitude compensating effects are evaluated and related to ideal conventional nozzle performance. An optimization technique is presented, which includes chemistry, divergence, and boundary layer effects. Graphical output includes flow field contours, and wall property profiles.
Plug Nozzle Parametric Study
Using The Liquid Performance Program (LPP)

Stuart S. Dunn and Douglas E. Coats

Software and Engineering Associates, Inc.
333 S. Carson Meadows, Suite 44
Carson City, NV 89701
702-882-1966
OUTLINE

- WHAT IS LPP
- PLUG NOZZLE DESCRIPTION
- LPP PLUG NOZZLE CAPABILITY
- EFFECT OF INLET AND COWL GEOMETRY ON PERFORMANCE
WHAT IS LPP?

The Liquid Performance Program (LPP) is a Super Set of the JANNAF TDK Code.

- Liquid Rocket Engine Performance
- Scramjets
- Plug and Aerospike Nozzles
- Efficient MOC Solver, Can Model Actual or Boundary Layer Displaced Walls Automatically
- Finite Rate and Equilibrium Chemistry
  - 3 different types of reactions including a global first order type
  - generalize symbolic reactions
WHAT IS LPP? (Continued)

- Mass Addition Boundary Layer
  - Models Wall Equilibrium Radiation Heat Transfer
  - Calculates Boundary Layers On Both Upper and Lower Walls
  - Tangential Slot Injection or Transpiration Cooling

- Planar and Axisymmetric Flow
  - Handles External Flow Interactions
WHAT IS LPP? (Continued)

- Spray Combustion Module
- Standard Plume Flowfield (SPF) and Rao Optimum Nozzle Linkage
- Pre and Post Processors
Plug Nozzles

- Characteristics
  - Center Body
  - Short Nozzle Configuration
  - Automatic Altitude Compensating
  - Complex Structure
  - Difficult To Cool

Annular Chamber

Expansion Fans
Flow Characteristics

Free Stream

Nozzle Exit

Kernel

Cowl

Fan

Dividing Streamline

Last DRC

Base Flow Region
LPP Plug Nozzle Capability

- 2D or Axisymmetric Flow
- External Flow Modeled With Newtonian Pressure Boundary
- Boundary Layer Computed On Both Upper and Lower Walls
- Base Flow Region Not Modeled
SSTO Base Line Plug Nozzle

- 1,000,000 lbf Thrust Class
- 2630 psia Chamber Pressure
- Annular Nozzle With 50 psia Exit Pressure and Mach 3 Flow
- L H2 - L O2 Propellants at an O/F=6
Isp Variation With Inlet Angle

![Graph showing Isp variation with inlet angle.]

- **Vacuum**: ■
- **Sea Level**: ●
- **8 kft, M=1**: ★
Isp Variation With Cowl Length

![Graph showing Isp variation with cowl length. The graph compares Isp in Vacuum, Sea Level, and 8 kft, M=1 conditions. The x-axis represents cowl length, and the y-axis represents Isp.]
SST0 H2/O2 PLUG NOZZLE ... STILL AIR AT 8 KFT : PRESSURE

LEGEND

1) 6.117
2) 7.454
3) 9.084
4) 11.07
5) 13.49
6) 16.44
7) 20.04
8) 24.42
9) 29.75
10) 36.26
11) 44.19
12) 53.85
X) 65.62

21 APR 95 16:43:24
STILL AIR AT 8 KFT
MACH 1 AIR AT 8KFT
Isp vs. Area Ratio For an SSME Type Engine (Pc=2630psia)
Isp Variation With Pressure Ratio

Pressure Ratio (Pc/P Ambient)

Isp

Plug @ Static P
Conv. 77.5:1
Plug in Flight
CONCLUSIONS

- Demonstrated The Ability Of LPP To Compute Performance of Plug Nozzles
- Performance Is Best With Short Cowls
- Performance Is Improved When The Flow Is Channeled Parallel To The Plug
- Showed That The Altitude Compensation Of Plug Nozzles Is Highly Dependent On The Flight Trajectory
RECOMMENDATIONS

- That An Automatic Trajectory Option Be Added To LPP
- That A Simple Base Flow Model Be Added To LPP
The effective performance of modular thrusters in an aerospike configuration is difficult to determine. Standard analytical tools are applicable to conventional nozzle shapes, but are limited when applied to an aerospike nozzle (An aerospike nozzle is an altitude compensating external nozzle). Three baseline nozzle shapes are derived using standard analytical procedures. The baseline nozzle sizes are restricted to fill a volume envelope. The three shapes are an axi-symmetric round nozzle, a 2D planar square exit nozzle, and a super elliptic round to nearly square nozzle. The integrated (thruster /aerospike) performance of the three nozzles is determined through the use of 3-D viscous CFD calculations where complex features of the flowfield can be accurately captured. The resulting installed performance is then used to evaluate the efficiency of these nozzle shapes for aerospike applications.

The determination of effective performance of a thruster nozzle integrated into an aerospike nozzle requires the solution of the three dimensional turbulent Navier-Stokes equations. The model used in this study consisted of two zones; one of the upstream thruster cowl surface so freestream conditions can be accurately predicted, and two, the aerospike surface beginning with with thruster outflow and extending to the end of the aerospike surface. The numerical grid consisted of over 120,000 nodes and used symmetry on the thruster centerline and edge. A two species non-reacting chemistry model was used to capture the variation of fluid properties between the hot plume gas and freestream air.

From the results of the three baseline nozzle aerospike calculations, the effective performance of the nozzle was determined. The flowfield of these calculations do show some variation between the cases. Recirculation zones on the cowl surface is predicted for the 2D planar nozzle and a smaller one for the super elliptic nozzle. The recirculation is caused by the strong pressure gradient between the plume and freestream flows. The axi-symmetric nozzle results indicates recirculation zones on the thruster face. These recirculation zones smooth the pressure gradient between the plume and freestream flow limiting the formation of recirculation on the cowl surface. Thruster to thruster interaction is evident for the axi-symmetric and super elliptic calculation while the 2D planar nozzle did not have any lateral expansion in the nozzle so thruster to thruster interaction is limited. The integrated performance results, at the altitude choosen, shows very little variation between the three thruster shapes. This result allows for nozzle shape determination to based on additional considerations (thermal, structural, weight) besides performance.
CFD ANALYSIS OF MODULAR THRUSTERS PERFORMANCE

CFD WORKSHOP
MARSHALL SPACE FLIGHT CENTER
HUNTSVILLE, AL

APRIL 25-27, 1995

Ronald J. Ungewitter
CFD Technology Center

James Beck and Andrew Ketchum
Performance Analysis and Applied Fluid Dynamics
BASELINE COMPARISONS

• GOAL
  - TO EVALUATE THE PERFORMANCE OF THREE BASELINE NOZZLE SHAPES INDIVIDUALLY AND INTEGRATED INTO AEROSPIKE

• APPROACH
  - USE MOC AND CFD CODES TO DETERMINE THE Isp OF THE INDIVIDUAL BASELINE NOZZLES
  - COMPARE THE MOC AND CFD RESULTS FOR CONSISTENCY
  - USE 3D CFD MODEL TO DETERMINE THE INSTALLED BASELINE NOZZLE / AEROSPIKE PERFORMANCE
BASELINE NOZZLE DEFINITIONS

- 3 UNIQUE SHAPES
  - AXISYMMETRIC
  - 2-D PLANAR
  - 3-D SUPER-ELLIPSE

- CONSTRAINTS
  - SAME NOZZLE LENGTH
  - SQUARE EXIT
  - SAME MASS FLOW (THROAT AREA)

- EACH SHAPE OPTIMIZED FOR Isp
## Baseline Nozzle Designs

<table>
<thead>
<tr>
<th>Baseline Thrust Cell Nozzle</th>
<th>Schematic</th>
<th>Throat Area (in²)</th>
<th>Nozzle Length (in)</th>
<th>Exit Dimension (in)</th>
<th>Nozzle Area Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axisymmetric</td>
<td><img src="image" alt="Axisymmetric Schematic" /></td>
<td>3.7688</td>
<td>11.585</td>
<td>$D = 7.519$</td>
<td>11.8</td>
</tr>
<tr>
<td>2-D Planar</td>
<td><img src="image" alt="2-D Planar Schematic" /></td>
<td>3.7688</td>
<td>11.585</td>
<td>$H = W = 7.519$</td>
<td>15.0</td>
</tr>
<tr>
<td>3-D Super-Elliptic</td>
<td><img src="image" alt="3-D Super-Elliptic Schematic" /></td>
<td>3.7688</td>
<td>11.585</td>
<td>$H = W = 7.519$</td>
<td>14.8</td>
</tr>
</tbody>
</table>
**BASELINE COMPARISONS**
**BASELINE NOZZLE AND ANALYTICAL METHOD COMPARISON**

- MOC AND CFD CALCULATIONS WERE MADE OF EACH BASELINE NOZZLE SHAPE

<table>
<thead>
<tr>
<th>NOZZLE SHAPE</th>
<th>EXIT AREA RATIO</th>
<th>MOC INVISCID</th>
<th>CFD INVISCID</th>
<th>CFD VISCIOUS</th>
</tr>
</thead>
<tbody>
<tr>
<td>AXISYMMETRIC</td>
<td>11.8:1</td>
<td>409.0</td>
<td>410.6</td>
<td>406.1</td>
</tr>
<tr>
<td>2-D PLANAR</td>
<td>15.0:1</td>
<td>414.2</td>
<td>417.0</td>
<td>411.4</td>
</tr>
<tr>
<td>3-D SUPER ELLIPTIC</td>
<td>14.8:1</td>
<td>412.9</td>
<td>414.6</td>
<td>409.1*</td>
</tr>
</tbody>
</table>

*VALUE BASED ON LAMINAR CFD PREDICTION WITH SKIN FRICTION ESTIMATED BASED ON PREVIOUS CALCULATIONS AND WETTED SURFACE AREA

**CONCLUSIONS:**
- MOC AND CFD PREDICT CONSISTENT RESULTS
- MOC CODES PROVIDE RAPID ANALYSIS CAPABILITY
- CFD CODE PROVIDES RANGE OF ANALYSIS OPTIONS
BASELINE COMPARISONS
3-D CFD MODEL

- FULL NAVIER-STOKES SOLUTIONS
- BALDWIN-LOMAX TURBULENCE MODEL
- TWO SPECIES (FREESTREAM, PLUME) NONREACTING CHEMISTRY MODEL
- TWO ZONE, 125,350 NODE GRID
  - ZONE ONE INCLUDES FLOW OVER COWL
  - ZONE TWO SIMULATES INFINITE ARRAY OF THRUSTERS AND AEROSPIKE SURFACE
- FREESTREAM INLET CONDITIONS AT 50,000 FT (MACH NUMBER = 1.83), REPRESENTATIVE OF MIDPOINT OF FLIGHT ENVELOPE
GEOMETRY OF INTEGRATED THRUST CELL MODEL

COWL

FREESTREAM INFLOW PLANE

COMPUTATIONAL GRID AT CONSTANT Z PLANE

THRUST CELL NOZZLE EXIT

AEROSPIKE CONTOUR
BASELINE COMPARISONS
FLOW FEATURES COMMON TO ALL SOLUTIONS

- NORMAL SHOCK UPSTREAM OF THRUSTERS ON COWL SURFACE, DECREASING IN STRENGTH FROM COWL SURFACE

- MODULE TO MODULE INTERACTION CAUSES THREE DIMENSIONAL PLUME SHAPE

- RECIRCULATION REGIONS ON COWL SURFACE AND/OR ON THRUSTER FACE

- MODULE TO MODULE INTERACTIONS ON AEROSPIKE SURFACE

- AEROSPIKE EXPANDS FLOW TO SIMILAR PRESSURE VALUES
PARTICLE TRACES

2D PLANAR THRUSTERS

AXISYMMETRIC THRUSTERS

RED: PLUME FLOW
LIGHT BLUE: FREESTREAM FLOW
: RECIRCULATION FLOW
THRUSTR CELL TECHNOLOGIES: INTEGRATED THRUST CELL / AEROSPIKE ANALYSIS
BASELINE NOZZLE COMPARISON

PRESSURE CONTOURS ALONG AEROSPIKE SURFACE

2D PLANAR THRUSTERS

AXISYMMETRIC THRUSTERS

PRESSURE (psia)

0.0  16.25  30.0
PRESSURE PROFILES ALONG AEROSPIKE SURFACE

2D PLANAR THRUSTERS

triangle = nozzle center
square = nozzle wall

AXISYMMETRIC THRUSTERS

triangle = nozzle center
square = nozzle wall
### Baseline Comparisons

**Installed Baseline Nozzle / Aerospike Performance**

<table>
<thead>
<tr>
<th>Baseline Shape</th>
<th>Nozzle Thrust*</th>
<th>Face PdA*</th>
<th>Aerospike Thrust*</th>
<th>Aerospike Friction*</th>
<th>Total Thrust*</th>
<th>Isp (SEC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axisymmetric</td>
<td>12403</td>
<td>15</td>
<td>3748</td>
<td>129</td>
<td>16037</td>
<td>430.2</td>
</tr>
<tr>
<td>2-D Planar</td>
<td>12566</td>
<td>--</td>
<td>3554</td>
<td>125</td>
<td>15995</td>
<td>429.0</td>
</tr>
<tr>
<td>3-D Super Elliptic</td>
<td>12493</td>
<td>&lt; 1.0</td>
<td>3660</td>
<td>129</td>
<td>16024</td>
<td>429.8</td>
</tr>
</tbody>
</table>

* ALL VALUES LBS

### Conclusions:

- Predicted values of installed performance are effectively equivalent.
- Similarity of performance allows for other design aspects (e.g., thermal, structural) to be considered in nozzle shape selection.
BASELINE COMPARISONS
TASK CONCLUSIONS

- CFD AND MOC PREDICT CONSISTENT RESULTS
- MOC CODES PROVIDE RAPID ANALYSIS CAPABILITY
- CFD CODE PROVIDE RANGE OF ANALYSIS OPTIONS
- INSTALLED BASELINE NOZZLE / AEROSPIKE PERFORMANCE PREDICTIONS FOR THREE NOZZLE SHAPES EFFECTIVELY THE SAME
- SIMILARITY OF PERFORMANCE ALLOWS FOR OTHER DESIGN ASPECTS (EG. THERMAL, STRUCTURAL) TO BE CONSIDERED IN NOZZLE SHAPE SELECTION
ABSTRACT

Current concepts for Reusable Launch Vehicle design have created renewed interest in the use of RP-1 fuels for high pressure and tri-propellant propulsion systems. Such designs require the use of analytical methodology which accurately accounts for the effects of real fluid properties, combustion of large hydrocarbon fuel molecules, and the possibility of soot formation. These effects are inadequately treated in current computational fluid dynamics (CFD) codes which are used for propulsion system analyses.

The objective of this investigation is to provide an accurate analytical description of hydrocarbon combustion thermodynamics and kinetics which is sufficiently computationally efficient to be practical design tool when used with CFD codes such as the FDNS code.

A rigorous description of real fluid properties for RP-1 and its combustion products will be derived from the literature and from experiments conducted in this investigation. Upon the establishment of such a description, the fluid description will be simplified by using the minimum of empiricism necessary to maintain accurate combustion analyses and including such empirical models into an appropriate CFD code. An additional benefit from this approach is that the real fluid properties analysis simplifies the introduction of the effects of droplet sprays into the combustion model.

Typical species compositions of RP-1 have been identified, surrogate fuels have been established for analyses, and combustion and sooting reaction kinetics models have been developed. Methods for predicting the necessary real fluid properties have been developed and essential experiments have been designed. Verification studies are in progress, and preliminary results from these studies will be presented. The approach has been determined to be feasible, and upon its completion the required methodology for accurate performance and heat transfer CFD analyses for high pressure, tripropellant propulsion systems will be available.
PROPELLANT CHEMISTRY FOR CFD APPLICATIONS

R.C. Farmer, P.G. Anderson, Gary C. Cheng

SECA, Inc.
Huntsville, AL
RP-1 COMBUSTION CHEMISTRY

- RP-1 combustion model w/soot formation
- Verify model w/data from literature
- Verify model w/new test data
- Describe real fluid thermodynamic properties
- Add real fluid HC properties to CICM
- Real fluid single element model
  - Real fluid HC, \( \text{H}_2 \), and \( \text{O}_2 \)
  - Real fluid tri-propellant
  - Additional turbulence models
- Account for radiation
- Assemble elements to make injector model
RP-1 is a straight run kerosene fraction which is subjected to acid washing and sulfur dioxide extraction. The mean molecular weight is about 174, and the H/C ratio is about 1.9. This implies that RP-1 is a multicomponent hydrocarbon fuel without a specified species distribution.

<table>
<thead>
<tr>
<th>Composition by Hydrocarbon Type</th>
<th>Volume %</th>
</tr>
</thead>
<tbody>
<tr>
<td>dinuclear aromatics</td>
<td>2.3</td>
</tr>
<tr>
<td>mononuclear aromatics</td>
<td>15.1</td>
</tr>
<tr>
<td>dicyclo-paraffins</td>
<td>11.6</td>
</tr>
<tr>
<td>monocyclo-paraffins</td>
<td>33.8</td>
</tr>
<tr>
<td>branched-paraffins</td>
<td>15.1</td>
</tr>
<tr>
<td>normal paraffins</td>
<td>22.1</td>
</tr>
<tr>
<td></td>
<td>100.0</td>
</tr>
</tbody>
</table>

or

<table>
<thead>
<tr>
<th>Composition</th>
<th>Volume %</th>
</tr>
</thead>
<tbody>
<tr>
<td>aromatics</td>
<td>17.4</td>
</tr>
<tr>
<td>cyclo-paraffins</td>
<td>45.4</td>
</tr>
<tr>
<td>paraffins</td>
<td>37.2</td>
</tr>
<tr>
<td></td>
<td>100.0</td>
</tr>
</tbody>
</table>
Distillation Curve for RP-1

![Distillation Curve for RP-1](image-url)
<table>
<thead>
<tr>
<th>Molecular Formula</th>
<th>Name</th>
<th>Type</th>
<th>Normal Boiling Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1,3-Diethylbenzene</td>
<td>Benzene</td>
<td>181.10</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1-Methyl-3-propylbenzene</td>
<td>Benzene</td>
<td>181.80</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>n-Bulylbenzene</td>
<td>Benzene</td>
<td>183.27</td>
</tr>
<tr>
<td>C_{10}\text{H}_{18}</td>
<td>Bicycloparaffin</td>
<td>Bicycloparaffin</td>
<td>183.4</td>
</tr>
<tr>
<td>C_{10}\text{H}_{18}</td>
<td>Bicycloparaffin</td>
<td>Bicycloparaffin</td>
<td>183.7</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1-Methyl-4-propylbenzene</td>
<td>Benzene</td>
<td>183.30</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 2-Diethylbenzene</td>
<td>Benzene</td>
<td>183.42</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1,3-Dimethyl-5-ethylbenzene</td>
<td>Benzene</td>
<td>183.58</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 4-Diethylbenzene</td>
<td>Benzene</td>
<td>183.78</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, Methyl-2-propylbenzene</td>
<td>Benzene</td>
<td>184.80</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 4-Dimethyl-2-ethylbenzene</td>
<td>Benzene</td>
<td>186.83</td>
</tr>
<tr>
<td>C_{10}\text{H}_{18}</td>
<td>trans-Decahydro-naphthalene</td>
<td>Bicycloparaffin</td>
<td>187.25</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 3-Dimethyl-4-ethylbenzene</td>
<td>Benzene</td>
<td>188.20</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 2-Dimethyl-4-ethylbenzene</td>
<td>Benzene</td>
<td>189.48</td>
</tr>
<tr>
<td>C_{10}\text{H}_{16}</td>
<td>Tricyclo (3.3.1.1)- decane</td>
<td>Tricycloparaffin</td>
<td>190</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 3-Dimethyl-2-ethylbenzene</td>
<td>Benzene</td>
<td>190.01</td>
</tr>
<tr>
<td>C_{10}\text{H}_{12}</td>
<td>1-Methyldindan</td>
<td>Aromatic-cycloparaffin</td>
<td>190.6</td>
</tr>
<tr>
<td>C_{10}\text{H}_{12}</td>
<td>2-Methyldindan</td>
<td>Aromatic-cycloparaffin</td>
<td>191.4</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1, 2-Dimethyl-3 ethylbenzene</td>
<td>Benzene</td>
<td>193.91</td>
</tr>
<tr>
<td>C_{10}\text{H}_{18}</td>
<td>cis-Decahydro-naphthalene</td>
<td>Bicycloparaffin</td>
<td>195.69</td>
</tr>
<tr>
<td>C_{11}\text{H}_{24}</td>
<td>n-Undecane</td>
<td>Normal paraffin</td>
<td>195.89</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1,2,4,5-Tetramethyl-benzene</td>
<td>Benzene</td>
<td>196.80</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1,2,3,5-Tetramethyl-benzene</td>
<td>Benzene</td>
<td>198.00</td>
</tr>
<tr>
<td>C_{11}\text{H}_{20}</td>
<td>Bicycloparaffin</td>
<td>Bicycloparaffin</td>
<td>202.5</td>
</tr>
<tr>
<td>C_{11}\text{H}_{16}</td>
<td>1-Methyl-3-n-butylbenzene</td>
<td>Benzene</td>
<td>204.1</td>
</tr>
<tr>
<td>C_{10}\text{H}_{14}</td>
<td>1,2,3,4-Tetramethyl-benzene</td>
<td>Benzene</td>
<td>205.04</td>
</tr>
<tr>
<td>C_{10}\text{H}_{12}</td>
<td>4-Methyldindan</td>
<td>Aromatic-cycloparaffin</td>
<td>205.5</td>
</tr>
<tr>
<td>Molecular Formula</td>
<td>Name</td>
<td>Type</td>
<td>Normal Boiling Point</td>
</tr>
<tr>
<td>-------------------</td>
<td>-------------------------------------</td>
<td>-----------------------</td>
<td>----------------------</td>
</tr>
<tr>
<td>C_{11}H_{16}</td>
<td>1,3-Dimethyl-4-n propylbenzene</td>
<td>Benzene</td>
<td>206.6</td>
</tr>
<tr>
<td>C_{10}H_{12}</td>
<td>1,2,3,4-Tetra-hydronaphthalene</td>
<td>Aromatic-cycloparaffin</td>
<td>207.57</td>
</tr>
<tr>
<td>C_{11}H_{16}</td>
<td>1,2-Dimethyl-4-n-propylbenzene</td>
<td>Benzene</td>
<td>208.5</td>
</tr>
<tr>
<td>C_{12}H_{26}</td>
<td>Trimethylbenzene</td>
<td>Benzene</td>
<td>212.3</td>
</tr>
<tr>
<td>C_{12}H_{26}</td>
<td>n-Dodecane</td>
<td>Normal paraffin</td>
<td>216.28</td>
</tr>
<tr>
<td>C_{10}H_{8}</td>
<td>Naphthalene</td>
<td>Dinuclear aromatic</td>
<td>217.96</td>
</tr>
<tr>
<td>C_{11}H_{14}</td>
<td>2-Methyl-(1,2,3,4-tetra-hydronaphthalene)</td>
<td>Aromatic-cycloparaffin</td>
<td>220.7</td>
</tr>
<tr>
<td>C_{11}H_{14}</td>
<td>6-Methyl-(1,2,3,4-tetra-hydronaphthalene)</td>
<td>Aromatic-cycloparaffin</td>
<td>229.03</td>
</tr>
<tr>
<td>C_{14}H_{30}</td>
<td>2,6,10-Trimethylundecane</td>
<td>Branched paraffin</td>
<td>231</td>
</tr>
<tr>
<td>C_{11}H_{14}</td>
<td>5-Methyl-(1,2,3,4-tetra-hydronaphthalene)</td>
<td>Aromatic-cycloparaffin</td>
<td>234.35</td>
</tr>
<tr>
<td>C_{13}H_{28}</td>
<td>n-Tridecane</td>
<td>Normal paraffin</td>
<td>235.43</td>
</tr>
<tr>
<td>C_{11}H_{10}</td>
<td>2-Methylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>241.05</td>
</tr>
<tr>
<td>C_{11}H_{10}</td>
<td>1-Methylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>244.64</td>
</tr>
<tr>
<td>C_{13}H_{32}</td>
<td>2,6,10-Trimethylundecane</td>
<td>Branched paraffin</td>
<td>249</td>
</tr>
<tr>
<td>C_{14}H_{30}</td>
<td>n-Tetradecane</td>
<td>Normal paraffin</td>
<td>253.52</td>
</tr>
<tr>
<td>C_{12}H_{10}</td>
<td>Biphenyl</td>
<td>Dinuclear aromatic</td>
<td>255.0</td>
</tr>
<tr>
<td>C_{13}H_{12}</td>
<td>2-Methylbiphenyl</td>
<td>Dinuclear aromatic</td>
<td>255.3</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>2-Ethynaphthalene</td>
<td>Dinuclear aromatic</td>
<td>257.9</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>1-Ethynaphthalene</td>
<td>Dinuclear aromatic</td>
<td>258.7</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>2,6-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>262</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>2,7-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>263</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>1,7-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>263</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>1,6-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>263</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>1,3-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>265</td>
</tr>
<tr>
<td>C_{12}H_{12}</td>
<td>1,5-Dimethynylnaphthalene</td>
<td>Dinuclear aromatic</td>
<td>265</td>
</tr>
<tr>
<td>C_{14}H_{14}</td>
<td>2,5-Dimethylbiphenyl</td>
<td>Dinuclear aromatic</td>
<td>267</td>
</tr>
<tr>
<td>C_{15}H_{16}</td>
<td>Trimethylbiphenyl</td>
<td>Dinuclear aromatic</td>
<td>267</td>
</tr>
<tr>
<td>C_{16}H_{18}</td>
<td>Tetramethylbiphenyl</td>
<td>Dinuclear aromatic</td>
<td>267</td>
</tr>
</tbody>
</table>
Note: Critical pressure for RP-1 is 340 psia, and critical temperature is 679°K.
RP-1 COMBUSTION PROPERTIES

HEAT OF COMBUSTION

\[ \text{HOC} = -18640 \text{ (Btu/lbm)} = -10.346 \text{ (kcal/gm)} \]

HEAT OF VAPORIZATION

\[ \text{HOV} = 106 \text{ (Btu/lbm)} \]

HEAT OF FORMATION (HOF)

To determine a HOF for mixtures, an effective molecular formula must be specified and used to evaluate the HOF. Frequently, an arbitrary molecular weight of 100 gms is assumed for a basis for thermodynamic calculations.

<table>
<thead>
<tr>
<th>H/C</th>
<th>HOF (kcal/100gm)</th>
<th>Source</th>
<th>Implied HOC (kcal/gm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.8624</td>
<td>-36.01</td>
<td>Lockheed</td>
<td>-10.287</td>
</tr>
<tr>
<td>1.90</td>
<td>-41.6</td>
<td>SAIC</td>
<td>-10.305</td>
</tr>
<tr>
<td>2.0</td>
<td>-44.36</td>
<td>Aerojet '78</td>
<td>-10.403</td>
</tr>
<tr>
<td>1.9063</td>
<td>-42.0</td>
<td>TMX - 1783</td>
<td>-10.305</td>
</tr>
<tr>
<td>1.9423</td>
<td>-33.068(v)</td>
<td>SP-273</td>
<td>-10.441(v)</td>
</tr>
<tr>
<td>1.9423</td>
<td>-38.946(l)</td>
<td>SP-273</td>
<td>-10.382(1)</td>
</tr>
<tr>
<td>1.922</td>
<td>-39.915</td>
<td>Surrogate Fuel</td>
<td>-10.346</td>
</tr>
</tbody>
</table>
**SPECTROSCOPIC ANALYSIS**
**OF H-1 GAS GENERATOR EXHAUST GASES**

<table>
<thead>
<tr>
<th>Compound</th>
<th>Sample #1</th>
<th>Sample #3</th>
<th>Sample #4</th>
</tr>
</thead>
<tbody>
<tr>
<td>CO</td>
<td>26.15*</td>
<td>36.82</td>
<td>37.93</td>
</tr>
<tr>
<td>CO₂</td>
<td>8.44</td>
<td>8.86</td>
<td>9.96</td>
</tr>
<tr>
<td>H₂O</td>
<td>2.66</td>
<td>1.14</td>
<td>0.85</td>
</tr>
<tr>
<td>H₂</td>
<td>1.97</td>
<td>2.22</td>
<td>2.34</td>
</tr>
<tr>
<td>CH₄</td>
<td>2.92</td>
<td>4.20</td>
<td>4.40</td>
</tr>
<tr>
<td>C₂H₂</td>
<td>4.16</td>
<td>5.62</td>
<td>4.27</td>
</tr>
<tr>
<td>C₂H₄</td>
<td>5.93</td>
<td>6.93</td>
<td>7.70</td>
</tr>
<tr>
<td>C₃H₆</td>
<td>0.72</td>
<td>...</td>
<td>...</td>
</tr>
<tr>
<td>Propyne</td>
<td>0.69</td>
<td>0.30</td>
<td>0.41</td>
</tr>
<tr>
<td>Propene</td>
<td>2.84</td>
<td>3.15</td>
<td>3.95</td>
</tr>
<tr>
<td>Diacetylene</td>
<td>0.12</td>
<td>0.08</td>
<td>0.21</td>
</tr>
<tr>
<td>1,3 Butadiene</td>
<td>1.30</td>
<td>1.32</td>
<td>1.52</td>
</tr>
<tr>
<td>2 Butene</td>
<td>1.57</td>
<td>0.80</td>
<td>1.16</td>
</tr>
<tr>
<td>- Butene</td>
<td>...</td>
<td>1.76</td>
<td>0.54</td>
</tr>
<tr>
<td>1,5 Hexadyne</td>
<td>1.98</td>
<td>1.58</td>
<td>1.73</td>
</tr>
<tr>
<td>3 Methyl Pentene -1</td>
<td>...</td>
<td>0.71</td>
<td>0.57</td>
</tr>
<tr>
<td>Cyclopentene</td>
<td>...</td>
<td>0.74</td>
<td>0.99</td>
</tr>
<tr>
<td>1,2,3 trimethylcyclopantane</td>
<td>...</td>
<td>1.14</td>
<td>0.97</td>
</tr>
<tr>
<td>Benzene</td>
<td>...</td>
<td>1.05</td>
<td>1.20</td>
</tr>
<tr>
<td>Ethyl Benzene</td>
<td>...</td>
<td>0.99</td>
<td>0.36</td>
</tr>
<tr>
<td>Carbon</td>
<td>...</td>
<td>5.22</td>
<td>2.06</td>
</tr>
<tr>
<td>RP-1</td>
<td>38.51</td>
<td>17.58</td>
<td>17.14</td>
</tr>
</tbody>
</table>

* Weight percentage composition of combustion products
Generalized Combustion Kinetics Model

| I. | PURE PYROLYSIS | \( \text{ALIPHATICS} \) | \( \text{AROMATICS} \) | \( \begin{bmatrix} C_2H_2 \\ CH_4 \\ C_2H_4 \\ H_2 \end{bmatrix} \) | = INTERMEDIATES |
| II. | OXIDATIVE PYROLYSIS | \( \text{ALIPHATICS} \) | \( \text{AROMATICS} \) | \( \begin{bmatrix} OH \\ O_2 \end{bmatrix} \) | \( \begin{bmatrix} CH_4 \\ C_2H_2 \\ C_2H_4 \end{bmatrix} \) | \( \begin{bmatrix} H_2 \\ C_xH_yO_z \end{bmatrix} \) |
| III. | PARTIAL OXIDATION | \( \text{ALIPHATICS} \) | \( \text{AROMATICS} \) | \( \begin{bmatrix} CH_4 \\ C_2H_2 \\ C_2H_4 \end{bmatrix} \) | \( \begin{bmatrix} O_2 \\ OH \end{bmatrix} \) | \( \begin{bmatrix} H_2 \\ CO \\ CO_2 \\ H_2O \end{bmatrix} \) |
| IV. | ELEMENTARY STEPS TO COMPLETION | \( \begin{bmatrix} CO \\ H_2 \\ C_xH_yO_z \\ H_2O_2 \end{bmatrix} \) | \( \begin{bmatrix} O \\ OH \\ CHO \end{bmatrix} \) | \( \begin{bmatrix} H_2O \\ CO_2 \end{bmatrix} \) |

V. | SOOT FORMATION |
| VI. | SOOT GASIFICATION |
| VII. | NO\(_x\) FORMATION |

\( \text{FUEL} + \begin{bmatrix} FBN \\ N_2 \end{bmatrix} \rightarrow \begin{bmatrix} HCN \\ NH_1 \end{bmatrix} \) |

BOUND NITROGEN AND/OR FUEL RICH NITROGEN CONVERSION

THERMAL FIXATION
Soot Model

Initial Polycyclic Aromatic Hydrocarbon (PAH) Formation
- Pyrolysis and combustion of fuel to form benzene and acetylene
- Implicit finite-rate chemistry

Planar PAH Growth
- HACA
  - Hydrogen abstraction, carbon addition (through reactions with acetylene)
- Oxidation
  - Implicit finite-rate chemistry using reactions from Frenklach, et al
- Properties for PAH compounds
  - Benson’s group contribution method to obtain $C_p$ for ideal gases
  - Benson data in tabular form for $300K < T < 1500K$
  - Used CEC data for selected species to generate group contributions as functions of temperature for $300K < T < 5000K$
  - Generated needed $C_p$ data in CEC format
  - $S_{298}^o$ corrected for symmetry and optical isomers using Benson’s data.
Soot Model (Cont.)

Aerosol Dynamics Based on Frenklach and Harris’s "Method II"
  • Method of moments
  • Nucleation
    Collisions of planar PAH’s to form 3-D particles
  • Coagulation
    Collisions of 3-D particles to form larger particles

Surface Growth on 3-D Particles
  • Collisions with planar PAH’s
  • HACA
  • Oxidation by reactions with $O_2$ and OH

Model provides soot formation and oxidation rates to be used to develop a quasiglobal reaction rate for soot production.
Cp’s for Diphenyl Compounds
Dotted lines are synthesized data
## Toluene and Iso-Octane Quasiglobal Kinetics Model

<table>
<thead>
<tr>
<th>GLOBAL MECHANISM</th>
<th>A</th>
<th>B</th>
<th>E/R</th>
<th>POWER DEPENDENCIES</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Toluene (aromatic)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[ \text{C}_8\text{H}_8 \rightarrow 3.5 \text{C}_2\text{H}_6 + 0.5 \text{H}_2 ]</td>
<td>1.7982 E10</td>
<td>0</td>
<td>3.5000 E4</td>
<td>([\text{C}_8\text{H}_8]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_8\text{H}_8 + 3.5 \text{O}_2 \rightarrow 7 \text{CO} + 4 \text{H}_2 ]</td>
<td>4.4963 E9</td>
<td>1</td>
<td>2.6785 E4</td>
<td>([\text{C}_8\text{H}_8]^{0.5}[\text{O}_2]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_8\text{H}_8 + \text{OH} \rightarrow 3.25 \text{C}_2\text{H}_6 + 0.5 \text{CO} + 0.5 \text{H}_2\text{O} + 0.75 \text{H}_2 ]</td>
<td>1.4721 E17</td>
<td>0</td>
<td>1.4510 E4</td>
<td>([\text{C}_8\text{H}_8]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td><strong>Iso-Octane (aliphatic)</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[ \text{C}<em>9\text{H}</em>{18} \rightarrow 4 \text{C}_2\text{H}_4 + \text{H}_2 ]</td>
<td>1.0473 E12</td>
<td>0</td>
<td>3.5229 E3</td>
<td>([\text{C}<em>9\text{H}</em>{18}]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}<em>9\text{H}</em>{18} + 4 \text{O}_2 \rightarrow 8 \text{CO} + 9 \text{H}_2 ]</td>
<td>1.2900 E9</td>
<td>1</td>
<td>2.5160 E4</td>
<td>([\text{C}<em>9\text{H}</em>{18}]^{0.5}[\text{O}_2]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}<em>9\text{H}</em>{18} + \text{OH} \rightarrow 3.75 \text{C}_2\text{H}_6 + 0.5 \text{CO} + 0.5 \text{H}_2\text{O} + 1.5 \text{H}_2 ]</td>
<td>2.0000 E17</td>
<td>0</td>
<td>1.4919 E4</td>
<td>([\text{C}<em>9\text{H}</em>{18}]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td><strong>Secondary Fuel</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_2 + 6 \text{OH} \rightarrow 4 \text{H}_2\text{O} + 2 \text{CO} ]</td>
<td>4.7850 E15</td>
<td>0</td>
<td>1.3883 E4</td>
<td>([\text{C}_2\text{H}_2]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_2 + 2 \text{OH} \rightarrow 2 \text{CO} + 2 \text{H}_2 ]</td>
<td>2.8000 E16</td>
<td>0</td>
<td>0</td>
<td>([\text{C}_2\text{H}_2]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_4 + 4 \text{OH} \rightarrow 2 \text{CO} + 2 \text{H}_2\text{O} + 2 \text{H}_2 ]</td>
<td>2.2020 E15</td>
<td>0</td>
<td>1.2079 E4</td>
<td>([\text{C}_2\text{H}_4]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_4 + 2 \text{OH} \rightarrow 2 \text{CO} + 3 \text{H}_2 ]</td>
<td>2.1129 E27</td>
<td>-3.0</td>
<td>6.3602 E3</td>
<td>([\text{C}_2\text{H}_4]^{1.0}[\text{OH}]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_2 + \text{O}_2 \rightarrow 2 \text{CHO} ]</td>
<td>4.0000 E12</td>
<td>0</td>
<td>1.4092 E4</td>
<td>([\text{C}_2\text{H}_2]^{1.0}[\text{O}_2]^{1.0})</td>
</tr>
<tr>
<td>[ \text{C}_2\text{H}_4 + \text{M} = \text{C}_2\text{H}_2 + \text{H}_2 + \text{M} ]</td>
<td>2.0893 E17</td>
<td>0</td>
<td>3.9810 E4</td>
<td>([\text{C}_2\text{H}_4]^{1.0}[\text{M}]^{1.0})</td>
</tr>
<tr>
<td><strong>Soot Parameters</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[ \text{C}_7\text{H}_8 = \text{HC} \rightarrow \text{soot} ]</td>
<td>4.0465 E14</td>
<td>-2.0</td>
<td>1.6110 E4</td>
<td>([\text{HC}]^{1.43}[\text{O}_2]^{0.5})</td>
</tr>
</tbody>
</table>

The values for \( i = A \), \( B \), \( T \), and \( Z \) are as follows:

- \( i = A \): 2.000 E1
- \( B \): 4.4600 E-3
- \( T \): 1.5100 E5
- \( Z \): 2.1300 E1

As indicated by the equation for [soot]
SOOT PARAMETERS

\[
\text{soot} + O_2 \rightarrow CO_2 \quad 12 P_{O_2} A_t \left[ \frac{K_A X}{1 + K_Z P_{O_2}} + K_B (1 - X) \right]
\]

\[
X = \left[ 1 + \frac{K_T}{(K_B P_{O_2})} \right]^{-1}
\]

\[
K_i = A_i \exp \{ -E_i / RT \}, \quad i = A, B, T, Z
\]

where \( A_t = 6 \left[ C_s / (\rho_s \cdot D_s) \right] \) (cm\(^2\) surface/cm\(^3\)),

\( P_{O_2} = \) partial pressure of O\(_2\) (atm),

\( C_s = (g \cdot \text{soot}/cm^3 \text{ of gas}), \quad \rho_s = (g \cdot \text{soot}/cm^3 \text{ of soot}), \)

\( D_s = \) diameter of soot (cm),

\([\text{soot}] = \) mass of soot/volume of gas (g/cm\(^3\)).
Elementary Reactions

<table>
<thead>
<tr>
<th>ELEMENTARY MECHANISM</th>
<th>A</th>
<th>B</th>
<th>E/R</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wet CO Mechanism</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\text{H}_2 + \text{O}_2 = \text{OH} + \text{OH}$</td>
<td>1.7000 E13</td>
<td>0</td>
<td>2.4070 E4</td>
</tr>
<tr>
<td>$\text{OH} + \text{H}_2 = \text{H}_2\text{O} + \text{H}$</td>
<td>2.1900 E13</td>
<td>0</td>
<td>2.5900 E3</td>
</tr>
<tr>
<td>$\text{OH} + \text{OH} = \text{O} + \text{H}_2\text{O}$</td>
<td>6.0230 E12</td>
<td>0</td>
<td>5.5000 E2</td>
</tr>
<tr>
<td>$\text{O} + \text{H}_2 = \text{H} + \text{OH}$</td>
<td>1.8000 E10</td>
<td>1.0</td>
<td>4.4800 E3</td>
</tr>
<tr>
<td>$\text{H} + \text{O}_2 = \text{O} + \text{OH}$</td>
<td>1.2200 E17</td>
<td>-0.91</td>
<td>8.3690 E3</td>
</tr>
<tr>
<td>$\text{M} + \text{O} + \text{H} = \text{OH} + \text{M}$</td>
<td>1.0000 E16</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$\text{M} + \text{O} + \text{O} = \text{O}_2 + \text{M}$</td>
<td>2.5500 E18</td>
<td>-1.0</td>
<td>5.9390 E4</td>
</tr>
<tr>
<td>$\text{M} + \text{H} + \text{H} = \text{H}_2 + \text{M}$</td>
<td>5.0000 E15</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$\text{M} + \text{H} + \text{OH} = \text{H}_2\text{O} + \text{M}$</td>
<td>8.4000 E21</td>
<td>-2.0</td>
<td>0</td>
</tr>
<tr>
<td>$\text{CO} + \text{OH} = \text{H} + \text{CO}_2$</td>
<td>4.0000 E12</td>
<td>0</td>
<td>4.0300 E3</td>
</tr>
<tr>
<td>$\text{CO} + \text{O}_2 = \text{CO}_2 + \text{O}$</td>
<td>3.0000 E12</td>
<td>0</td>
<td>2.5000 E4</td>
</tr>
<tr>
<td>$\text{CO} + \text{O} + \text{M} = \text{CO}_2 + \text{M}$</td>
<td>6.0000 E13</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
RP-1 COMBUSTION VERIFICATION EXPERIMENTS

Criteria: Minimize Effect of Turbulent Mixing

- Single Drop Combustion - LSU
- Well Stirred Reactor - Exxon
- Rocket Test Motor - General Dynamics Corporation (GDC)
High Pressure Apparatus for the Measurement of Single Droplet Combustion
Photograph of Burning Droplets (n-heptane in air).

Effect of Pressure: (a) n-heptane; (b) n-dodecane.
JSC Soot Emissions For Toluene/Methylnaphthalene Blends;
$T = 1900 \, \text{K}; \tau = 3 \, \text{ms}$.
Dependence of soot concentration on equivalence ratio; atomization air = 15 g/min/nozzle; indicated temperature = 1900 ± 35 K.
Photograph of the exit flow at an exit pressure of 2 atm. Region 1 is the undisturbed cone, region 2 is the Prandtl-Meyer expansion zone, and region 3 is the mixing zone. The spectrometer line of sight is indicated by a dotted line.
Rayleigh Theory for Small Particles

\[
K_\lambda = \left( \frac{36\pi nk}{(n^2-k^2+2)^2 + 4n^2k^2} \right) \left( \frac{\rho_c}{\rho_o} \right) \left( \frac{1}{\lambda} \right)
\]

\(K_\lambda (\text{cm}^3) = \kappa_\lambda \rho_c\) = linear absorption coefficient

\(\kappa_\lambda (\text{cm}^2/\text{gm}) =\) mass absorption coefficient

\(\rho_c (\text{gm/cc of volume}) = \) soot density

\(\rho_o (\text{gm/cc of soot}) = \) bulk soot density
CONCLUSIONS:

Real fluid RP-1 and H₂ combustion kinetics models will be incorporated into several element models and evaluated.

The element models will be assembled into an entire injector model and the motor performance evaluated.
Thermo-Kinetics Characterization of Kerosene/RP-1 Combustion for Tripropellant Engine Design Calculations

Ten-See Wang
Computational Fluid Dynamics Branch
NASA/Marshall Space Flight Center

13th Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Session 10: Combustion/Nozzles/Tripropellant
April 26, 1995
NASA/Marshall Space Flight Center, MSFC, Alabama
Acknowledgments

- The development of the thermo-kinetics models for kerosene/RP-1 combustion is performed under the Rocketdyne - Marshall Space Flight Center Cooperative Agreement.

- The multi-phase FDNS CFD code is being developed by Engineering Sciences, Inc. under the sponsorship of SBIR Program.

- Several individuals have contributed to the discussion and collection of kerosene/RP-1 physical properties:
  - Dr. R.C. Farmer; SECA, Inc.
  - Dr. Y.-S. Chen; ESI
  - John Hutt, Hou Trinh, Klaus Gross; NASA - MSFC
  - Dr. C.-P. Chen, UAH.
Objectives

- To develop a simple substitute fuel model representing kerosene/RP-1 and to develop its thermochemical properties according to the available database.
- To develop a simplified combustion kinetics model for the substitute fuel
- To test this thermo-kinetics model on 3 tri-axial, tri-propellant, single element injectors
Introduction

• Kerosene is derived from petroleum and RP-1 is a straight run from kerosene fraction. Both are complex mixtures of many substances and the actual composition depends on the specifications.

• Elemental formulas have been formulated for thermo-equilibrium calculations, but they cannot be used for CFD design applications.

• A molecular substitute fuel model has to be developed to approximate the general thermo-physical properties of kerosene/RP-1.

• While multi-component fuel model can be formulated if the information is available, for which it is not, a one-component model is most efficient for computational purpose.

• This fuel model can be refined to tailor specific fuel specifications if the information is available.
Physical-thermo-chemical properties of kerosene/RP-1.

<table>
<thead>
<tr>
<th></th>
<th>Kerosene</th>
<th>RP-1</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Molecular Formula</strong></td>
<td>175\textsuperscript{S6}</td>
<td>172-175\textsuperscript{C4}</td>
</tr>
<tr>
<td><strong>Molecular Weight</strong></td>
<td></td>
<td>13.97-14.03\textsuperscript{C4}, 13.97\textsuperscript{S7}</td>
</tr>
<tr>
<td><strong>Elemental Formula</strong></td>
<td>CH\textsubscript{1.95}-CH\textsubscript{2.0}\textsuperscript{C4}, CH\textsubscript{1.9423}\textsuperscript{S7}, CH\textsubscript{1.953}\textsuperscript{S6}</td>
<td></td>
</tr>
<tr>
<td><strong>Formula Weight</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>H_c, Btu/lb</strong></td>
<td>-18500\textsuperscript{M4}, -18577\textsuperscript{C3}</td>
<td>-18433\textsuperscript{C4}, -18640\textsuperscript{S7}</td>
</tr>
<tr>
<td><strong>H_f,298K, cal/mole</strong></td>
<td></td>
<td>-5430\textsuperscript{S7}/CH\textsubscript{1.9423}</td>
</tr>
<tr>
<td><strong>C_p,516K,1 atm, cal/mole-K</strong></td>
<td>101\textsuperscript{C4}</td>
<td></td>
</tr>
<tr>
<td><strong>Paraffins (n and iso) %</strong></td>
<td>41\textsuperscript{C4}</td>
<td></td>
</tr>
<tr>
<td><strong>Naphthenes %</strong></td>
<td></td>
<td>56\textsuperscript{C4}</td>
</tr>
<tr>
<td><strong>Aromatics %</strong></td>
<td>5\textsuperscript{M4}</td>
<td>5\textsuperscript{NI}, 3\textsuperscript{C4}</td>
</tr>
<tr>
<td><strong>Olefins %</strong></td>
<td>1\textsuperscript{M4}</td>
<td>0\textsuperscript{C4}</td>
</tr>
</tbody>
</table>
Thermodynamic Consistency Test 1

- $C_nH_m + (n + 0.25m) O_2 = n CO_2 + 0.5m H_2O$

- $H_f(C_nH_m) = n H_f(CO_2) + 0.5m H_f(H_2O) - H_c$
<table>
<thead>
<tr>
<th>Formula</th>
<th>H&lt;sub&gt;f&lt;/sub&gt;298K</th>
<th>M</th>
<th>H/C</th>
<th>Composition</th>
<th>Reference</th>
</tr>
</thead>
<tbody>
<tr>
<td>C&lt;sub&gt;10&lt;/sub&gt;H&lt;sub&gt;19&lt;/sub&gt;</td>
<td>-57.1</td>
<td>139</td>
<td>1.90</td>
<td>-</td>
<td>Harsha, et al., 1982</td>
</tr>
<tr>
<td>C&lt;sub&gt;12&lt;/sub&gt;H&lt;sub&gt;26&lt;/sub&gt;</td>
<td>-69.5</td>
<td>170</td>
<td>2.17</td>
<td>Paraffin</td>
<td>Lawver, 1982</td>
</tr>
<tr>
<td>C&lt;sub&gt;12&lt;/sub&gt;H&lt;sub&gt;23&lt;/sub&gt;</td>
<td>-57.1</td>
<td>167</td>
<td>1.91</td>
<td>-</td>
<td>Amsden, 1993</td>
</tr>
<tr>
<td>C&lt;sub&gt;12&lt;/sub&gt;H&lt;sub&gt;28.8&lt;/sub&gt;</td>
<td>-37.5</td>
<td>175</td>
<td>2.37</td>
<td>-</td>
<td>Chen, et al., 1993</td>
</tr>
</tbody>
</table>
One-Component Kerosene/RP-1 Substitute Fuel: C$_{12}$H$_{24}$
  : Pentyl-Methylcyclohexane
  : Pentyl-Methylcyclohexane/n-Dodecene

- 58.3% Naphthene
- 41.7% Paraffin
Comparison of thermo-chemical characterization of model fuel with reported data.

<table>
<thead>
<tr>
<th></th>
<th>Kerosene</th>
<th>RP-1</th>
<th>Kerosene/RP-1 substitute fuel model</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Molecular Formula</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Molecular Weight</strong></td>
<td>$175^{S6}$</td>
<td>$172-175^{C4}$</td>
<td>$C_{12}H_{24}$</td>
</tr>
<tr>
<td><strong>Elemental Formula</strong></td>
<td>$CH_{1.95}-CH_{2.0}^{C4}$,</td>
<td>$CH_{2.0}$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>$CH_{1.9423}^{S7},CH_{1.953}^{S6}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Formula Weight</strong></td>
<td>$13.97-14.03^{C4},13.97^{S7}$</td>
<td>$14.03^{C4}$</td>
<td></td>
</tr>
<tr>
<td><strong>He, Btu/lb</strong></td>
<td>$-18500^{M4}$, -18577$^{C3}$</td>
<td>$-18433^{C4}$, -18640$^{S7}$</td>
<td>$-18500$</td>
</tr>
<tr>
<td><strong>Hf,298K, cal/mole</strong></td>
<td>$-5430^{S7}/CH_{1.9423}$</td>
<td>$-92200/C_{12}H_{24},-7683/CH_{2.0}$</td>
<td>$-92200/C_{12}H_{24},-7683/CH_{2.0}$</td>
</tr>
<tr>
<td><strong>C_p,516k, 1 atm, cal/mole-K</strong></td>
<td>$101^{C4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Paraffins (n and iso) %</strong></td>
<td>$41^{C4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Naphthenes %</strong></td>
<td>$56^{C4}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Aromatics %</strong></td>
<td>$5^{M4}$</td>
<td>$5^{NI},3^{C4}$</td>
<td>$0$</td>
</tr>
<tr>
<td><strong>Olefins %</strong></td>
<td>$1^{M4}$</td>
<td>$0^{C4}$</td>
<td>$0$</td>
</tr>
</tbody>
</table>
Thermodynamic Consistency Test 2

- The theoretical rocket performance of CnHm and its elemental form CHm/n should be identical
  - RD-170 operating conditions
  - equivalence ratio : 1.2939
  - SUPAR : 36.9
  - chamber pressure: 241.9 atm
Fig. 1 Calculated chamber gas composition.
Fig. 2 Calculated nozzle exit gas composition.
<table>
<thead>
<tr>
<th>Reaction</th>
<th>Paraffin Global Step</th>
<th>Naphthalene Global Step</th>
<th>Ref.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>C$_2$H$_4$ + 6O$_2$ → 12CO + 12H$_2$</td>
<td>C$_2$H$_4$ + 6O$_2$ → 12CO + 12H$_2$</td>
<td></td>
</tr>
<tr>
<td>Wet CO Mechanism</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>H$_2$ + O$_2$ = OH + OH</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OH + H$_2$ = H$_2$O + H</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OH + OH = O + H$_2$O</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>H + O$_2$ = O + OH</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>M + O = O$_2$ + M</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>M + H + H = H$_2$O + M</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>M + H + O = H$_2$O + CO$_2$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CO + O = CO$_2$ + O</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>CO + O + M = CO$_2$ + M</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

* M stands for third-body collision partner.
Kerosene Substitute Fuel Model/Quasi-global Kerosene Kinetics/CFD Test on 3 Shear Triaxial Injectors

- GOX/GRP/GH2, LOX/GRP/GH2, and LOX/LRP/GH2 uni-element injectors
- Mixture ratio of RD-704 is used.
- The injector/combustor setup follows that of Penn-State shear coaxial/optically accessible rocket chamber.
  - equivalence ratio : 1.2317
  - SUPAR : 2.6514
  - chamber pressure: 30.8 atm
Multiphase FDNS Numerical Methodology and Physical Models for Triaxial Tripropellant Injectors

- Multi-species VOF formulation for liquid jets and multi-species particle formulation for droplets
- Jet surface primary atomization - empirical mean particle size and empirical mass stripping rate (experimental data required for modeling)
- Droplet secondary breakup - Taylor Analogy Breakup
- Droplet vaporization - General Evaporization Model
- Lagrangian particle tracking with turbulence dispersion
- Finite-rate chemistry integration - Penalty Function Method
- Kinetics - Quasiglobal kerosene combustion kinetics (9 species and 14 reactions)
- Thermodynamics
  - real gas: kerosene and 8 other species
  - real liquid: LOX (NBS table)
  : kerosene (Lefebvre)
Summary

- A simple kerosene/RP-1 substitute fuel model and its thermochemical properties have been developed according to available database.
- This substitute fuel model and its thermodynamics have passed the thermodynamic consistency test.
- In addition, a simple kerosene/RP-1 quasiglobal combustion kinetics model has been developed based on the composition of the substitute fuel model.
- Preliminary CFD test of the thermo-kinetics model on 3 triaxial, tripropellant injectors have shown reasonable flame temperatures.
Application of Optimization Techniques to Design of Unconventional Rocket Nozzle Configurations

W. Follett, A. Ketchum, A. Darian, Y. Hsu
Rocketdyne Division, Rockwell International, Canoga Park, California

Abstract

Several current rocket engine concepts such as the bell-annular tripropellant engine, and the linear aerospike being proposed for the X-33, require unconventional three-dimensional rocket nozzles which must conform to rectangular or sector shaped envelopes to meet integration constraints. These types of nozzles exist outside the current experience database, therefore, application of efficient design methods for these propulsion concepts is critical to the success of launch vehicle programs.

The objective of this work is to optimize several different nozzle configurations, including 2-D and 3-D geometries. Methodology includes coupling CFD analysis to genetic algorithms and Taguchi methods, as well as implementation of a streamline tracing technique. Results of applications are shown for several geometry classes including: 3-D thruster nozzles with round or superelliptic throats and rectangular exits, 2-D and 3-D thrusters installed within a bell nozzle, and 3-D thrusters with round throats and sector shaped exits.

Due to the novel designs considered for this study, there is little experience base which can be used to guide the effort and limit the design space. With a nearly infinite parameter space to explore, simple parametric design studies cannot possibly search the entire design space within the time frame required to impact the design cycle. For this reason, robust and efficient optimization methods are required to explore and exploit the design space to achieve high performance engine designs. Five case studies which examine the applications of various techniques in the engineering environment are presented in this paper.

The first study uses two-dimensional CFD coupled to Taguchi methods to determine optimal design parameters for the D-1 test engine being built for the SSTO Advanced Propulsion Technology contract. The D-1 engine utilizes a ring of small thrusters within a larger bell nozzle. This study was used to determine the optimal value of four design variables to achieve the best overall performance during both low altitude (thrusters firing) and high altitude (thrusters not firing) operational modes. Two other case studies investigate the problem of using multidisciplinary techniques to optimize a 3-D thruster design with both genetic algorithms and Taguchi methods. The relative strengths and weaknesses of these two methods are apparent when using them to solve this problem using up to 21 design variables. This thruster is also designed using streamline tracing techniques for the fourth case study.

The final study uses Taguchi methods to determine the optimal 3-D thruster module design when installed in a bell nozzle. This requires full 3-D solutions of the thruster and bell nozzles to quantify module-to-module interaction effects.

Software which couples optimization techniques to CFD have tremendous potential as aerodynamic design tools. However, to function effectively in the engineering environment, the optimization algorithms must be robust and efficient. Several optimization techniques have been demonstrated for rocket nozzle design, and their performance on these real world applications has been assessed.
APPLICATION OF OPTIMIZATION TECHNIQUES
TO DESIGN OF UNCONVENTIONAL ROCKET
NOZZLE CONFIGURATIONS

W. Follett
A. Ketchum
A. Darian
Y. Hsu
Rocketdyne Division - Rockwell International

Computational Fluid Dynamics Branch
Fluid Dynamics Division
Structures and Dynamics Laboratory
Science and Engineering Directorate
Marshall Space Flight Center

13th Workshop for CFD
Applications in Rocket Propulsion
Huntsville, Alabama
April 25-28, 1995

Rockwell Aerospace
Rocketdyne

CFD Technology Center
CFC 95-016- 1/8/MAY/95/ed
OPTIMIZATION OF UNCONVENTIONAL ROCKET NOZZLE CONFIGURATIONS

- APPLICATIONS FOR NASA ADVANCED PROPULSION TECHNOLOGIES CONTRACT AND AIR FORCE MODULAR THRUST CELL CONTRACT

- GEOMETRY
  - BELL-ANNULAR OR AEROSPIKE NOZZLES
  - 3-D THRUSTERS WITH RECTANGULAR OR SECTOR SHAPED EXITS

- ANALYSIS METHODS
  - 2-D AND 3-D CFD (EULER AND FNS)
  - 3-D MOC

- OPTIMIZATION METHODS
  - TAGUCHI, GENETIC ALGORITHMS, STREAMLINE TRACING
REPRESENTATIVE NOZZLE CONFIGURATIONS

AEROSPIKE

BELL-ANNULAR


2-D BELL-ANNULAR D-1 TEST ENGINE DESIGN STUDY

- **OBJECTIVE**
  - MAXIMIZE AVERAGE THRUST, ISP FOR THRUSTER ON AND THRUSTER OFF

- **OPTIMIZATION METHOD**
  - TAGUCHI L9 MATRIX
  - 4 DESIGN VARIABLES
  - 20 2-D CFD EVALUATIONS

- **IMPROVEMENT OVER BASELINE**
  - 1.2% IN THRUST & ISP

---

Objective:
Maximize average thrust, ISP for thruster on and thruster off.

Optimization Method:
Taguchi L9 Matrix
- 4 Design Variables
- 20 2-D CFD Evaluations

Improvement over baseline:
1.2% in thrust & ISP

Baseline vs. Final images showing pressure distribution with a color scale from 0.75 to 200.00 psi.

Shaded area represents the computational domain.
3-D THRUST CELL OPTIMIZATION

- OBJECTIVE
  - MAXIMIZE: THRUST(THRUSTER ONLY) / SYSTEM WEIGHT
  - MINIMIZE PEAK HEAT LOAD

- OPTIMIZATION METHODS
  - TAGUCHI L32 & L64 MATRICES
  - GENETIC ALGORITHM
  - 15-21 DESIGN VARIABLES

- 3-D MOC EVALUATIONS
  - 460 FOR TAGUCHI
  - 1000 FOR GENETIC

- IMPROVEMENT OVER BASELINE
  - 4.6% IN THRUST / WEIGHT

**Thrust Cell Taguchi Optimum**
Mach Number Contours

- Thrust Cell Taguchi Optimum
- Mach Number Contours

**Mach Number Minimum** = 1.0
**Mach Number Maximum** = 4.0

**CONVERGENCE HISTORIES**

**TAGUCHI**

- PREDICTION
- CONFIRMATION

**GENETIC**

- Thrust Weight (N)

**CONVERGENCE HISTORIES**

- Generation Number

Rockwell Aerospace
Rocketdyne

CFD Technology Center
3-D THRUST CELL OPTIMIZATION

BASELINE
OPTIMIZED FOR AERO ONLY
T/W = 56.39

GENETIC ALGORITHM OPTIMIZATION
T/W = 58.97

TAGUCHI OPTIMIZATION
T/W = 58.98

TAGUCHI OPTIMIZATION
RESTRICTED TO ROUND THROAT
T/W = 58.92
3-D STREAMLINE TRACING METHOD

• OBJECTIVE
  • MAXIMIZE THRUST, Isp FOR STAND-ALONE THRUSTER

• DESIGN METHOD
  • STREAMLINE TRACING
  • RAO OPTIMUM NOZZLE-MODEL FLOWFIELD
  • THREE 2-D MOC EVALUATIONS

• PERFORMANCE
  • 0.15% LESS Isp THAN TAGUCHI & G.A. OPTIMUMS
3-D BELL-ANNULAR D-1 INSTALLED THRUSTER DESIGN

• OBJECTIVES
  • MAXIMIZE THRUST, ISP
  • MINIMIZE PEAK HEAT LOAD

• OPTIMIZATION METHOD
  • TAGUCHI L9 MATRIX
  • 4 DESIGN VARIABLES
  • 10 3-D CFD EVALUATIONS

• IMPROVEMENTS OVER BASELINE
  • 0.1% IN THRUST
  • 23% REDUCTION IN PEAK HEAT FLUX

![Baseline and optimum diagrams]
OPTIMIZATION OF UNCONVENTIONAL ROCKET NOZZLE CONFIGURATIONS

CONCLUSIONS

• OPTIMIZATIONS METHODS ARE COMBINED WITH 3-D CFD ANALYSIS TO CREATE A VERY POWERFUL AERODYNAMIC DESIGN TOOL
  • ALLOWS DESIGN IF COMPLEX CONFIGURATIONS WHICH WERE PREVIOUSLY INFEASIBLE
  • PROVIDES HIGH FIDELITY ANALYSIS EARLY IN THE DESIGN CYCLE

• MULTIDISCIPLINARY OPTIMIZATION IS CRITICAL FOR ROCKET NOZZLE DESIGNS
  • ROBUST AERO PERFORMANCE ALLOWS DESIGN FLEXIBILITY
  • OTHER CONCERNS (THERMAL, WEIGHT, MANUFACTURING) MAY BE MORE IMPORTANT FOR DELIVERING OPTIMAL “SYSTEM” PERFORMANCE
Transonic Aerodynamic Characteristics of A Proposed Wing Body Reusable Launch Vehicle Concept

Anthony M. Springer
NASA Marshall Space Flight Center

A proposed wing body reusable launch vehicle was tested in the NASA Marshall Space Flight Center 14X14-inch trisonic wind tunnel during the winter of 1994. This test resulted in the vehicle's subsonic and transonic, Mach 0.3 to 1.96, longitudinal and lateral aerodynamic characteristics. The effects of control surface deflections on the basic vehicle's aerodynamics including a body flap, elevons, ailerons, and tip fins are presented.

As an outcome of NASA's 1993 Access to Space study, a more in-depth follow-on study was undertaken. Three candidate reusable launch vehicle configurations which would provide reusable single stage to orbit capability were selected. A wing body configuration was one of these candidate concepts, the other two concepts being a vertical lander and a lifting body. The wing body configuration was a direct outgrowth of the access to space option three reference single stage to orbit rocket vehicle. This vehicle matured during the subsequent reusable launch vehicle (RLV) study into the vehicle which was tested. Initially, the vehicle's aerodynamic characteristics were determined using aerodynamic prediction codes. To obtain a better fidelity in the aerodynamic data, a series of scale models of the proposed wing body vehicle were tested at the NASA Marshall Space Flight Center (MSFC) and the NASA Langley Research Center (LaRC). The vehicle was tested at low subsonic and hypersonic conditions at LaRC and at subsonic, transonic, and supersonic conditions at MSFC. The results of the transonic testing in MSFC's 14-Inch Trisonic Wind Tunnel (TWT) facility are presented herein.

A .004 scale RLV wind tunnel model was tested during the winter of 1994 at the NASA Marshall Space Flight Center 14X14-inch trisonic wind tunnel (TWT). The subsonic and transonic, Mach 0.3 to Mach 2.0, aerodynamic characteristics of the WB001 reference wing-body vehicle were determined. This wind tunnel test provided aerodynamic data for the basic vehicle, wing and body contributions, and control surface increments. The data derived from this test were used to construct an aerodynamic database for flight mechanics and structural loads studies on the wing body vehicle.

The WB001 vehicle is generically a wing-body combination. The body consists of a drooped nose followed by a cylindrical core section 28.55 ft in diameter, full scale, with a total body length of 185.6 ft, full scale. The wing is a NACA-0010 airfoil at the root linearly varying to a NACA-0012 airfoil at the tip with a 54 degree leading edge sweep, 3.5 degrees of dihedral, and an aspect ratio of 1.91. Control surfaces for this configuration consist of ailerons, elevons, and tip fins.

The vehicle is longitudinally stable and can be trimmed at both subsonic and transonic Mach numbers. This assumes a vehicle center of gravity at 68.6% body length or 127.32 feet aft of the nose. At subsonic Mach numbers, the vehicle is stable in trim for all control deflections. The vehicle for the subsonic Mach range can be trimmed at the desired angle-of-attack for entry, approximately 15 degrees. This trim angle is accomplished through various control surface deflections, see figure 84. The vehicle for the transonic Mach range, Mach 0.95 to 2.0, has stable trim points but not at the desired angle-of-attack, approximately 15 degrees angle-of-attack. It can be extrapolated from the current data trends that for a larger elevon deflection between 20 and 30 degrees, the vehicle will be neutrally stable at the desired trim point of 15 degrees.

The WB001 vehicle is laterally unstable for the subsonic and transonic Mach range. The tip fin deflections provide a trim angle range of approximately 1 to 2 degrees, therefore, larger tip fins and deflectable surfaces are desirable. Enlarging the tip fins by an approximate geometric factor of 3 to 4 should result in the vehicle being neutrally stable.
The Transonic Aerodynamic Characteristics Of A Proposed Wing Body Reusable Launch Vehicle Concept

Anthony M. Springer
NASA MSFC ED34
13th CFD Workshop
April 26, 1995
Introduction:

- Vehicle Configuration
- Wind Tunnel Description
- Test Summary
- Results and Conclusions
- Follow-Ons
- Data Availability
Vehicle Configuration:

- Generic Wing Body Configuration
- Length = 185.6 ft
- Drooped Nose
- Core Diameter = 28.55 ft
- Wing Span = 93 ft
- NACA-0010 to NACA-0012 Airfoil
- 54 Degree Leading Edge Sweep
- 3.5 Degree Dihedral
- Aspect Ratio = 1.91
Wind Tunnel Description:

- 14X14-Inch Trisonic Wind Tunnel
- 14X14-Inch Test Section
- Mach Range 0.3 to 5.0
- Intermittent Blow Down Type Tunnel
Test Summary:

- Static Stability Aerodynamic Characteristics
- Mach 0.3 to 1.96
- Basic WB001 Vehicle
- Elevon, Aileron, Body Flap Deflections (-10°)
- Tip Fin Deflections (10°, 20°)
WB001 Tip Fin Control Deflections Mach 1.10

Graph showing the relationship between CBLF and Beta for different control settings:
- bo
- rtf10
- rtf20

The graph plots CBLF on the y-axis against Beta on the x-axis, with data points indicating the deflections at various Beta values.
Wing Body Lift to Drag Ratio

Mach

5 4.5 4 3.5 3 2.5 2 1.5 1 0.5

0 0.2 0.4 0.6 0.8 1 1.2 1.4 1.6 1.8 2
Wing Body Aerodynamic Center

Lref = 185.6 ft.

- WB001 Yaw
- No Tip Fins Yaw
- WB001 Pitch

Note: Under 3% Change in Pitch
Xa.c. for No Tip Fin Configuration
WB001 Trim Angle @ C.G. = 68.6%
Follow-Ons:

- Determine Supersonic Aerodynamic Char.
- Larger Elevon Deflections
- Vertical Tail
- Split V-Tail
- Larger Tip Fins (Factor of 3)
Data Availability

- Wing Body Transonic Aerodynamic Characteristics have been documented in NASA TM-XXXX
- This memo is available through NASA to interested parties
- The follow-on test will be documented in a forthcoming NASA publication
Ascent Aerodynamic Pressure Distributions on WB001

B. Vu, J. Ruf, F. Canabal
CFD Branch

J. Brunty
System Load Branch

To support the reusable launch vehicle concept study, the aerodynamic data and surface pressure for WB001 were predicted using three CFD codes at several flow conditions during the ascent phase. The results have been compared between code to code and code to aerodynamic database as well as available experimental data. A set of particular solutions have been selected and recommended for use in preliminary conceptual designs. These CFD results have also been provided to the structure group for wing loading analyses.
Ascent Aerodynamics Analysis of WB001 Configuration

B.T. Vu
J.H. Ruf
F. Canabal
J. Brunty
OUTLINE

- Objectives
- Codes employed
- Cases considered
- Results and discussions
- Structural analyses
- Conclusions
OBJECTIVES

- Predict the flow field environment during ascent
- Compute the aerodynamic coefficients
- Provide three-dimensional surface pressure for structure analysis
CODES EMPLOYED

• OVERFLOW
  - capable of solving overset grids
  - used at ARC and JSC for orbiter analyses

• GASP
  - finite-volume, density-based
  - used at LaRC and Wright Patterson for NASP vehicle design

• FDNS
  - finite-difference, pressure-based
  - used at MSFC for reacting flow analyses
## CASES CONSIDERED

<table>
<thead>
<tr>
<th>Transonic (M=1.1)</th>
<th>Supersonic (M=5.72)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AOA</strong></td>
<td><strong>AOA</strong></td>
</tr>
<tr>
<td>6°</td>
<td>0°</td>
</tr>
<tr>
<td></td>
<td>6°</td>
</tr>
<tr>
<td><strong>GASP inv/vis.</strong></td>
<td><strong>GASP vis.</strong></td>
</tr>
<tr>
<td><strong>OVERFLOW inv/vis.</strong></td>
<td><strong>OVER FLOW vis.</strong></td>
</tr>
<tr>
<td><strong>FDNS vis.</strong></td>
<td></td>
</tr>
<tr>
<td></td>
<td>8°</td>
</tr>
</tbody>
</table>
RLV Concept Study Review

RESULTS & DISCUSSIONS

- Computational domain for both cases properly generated to capture physics associated flow conditions

- Surface pressure and symmetry Mach contours
  - good agreement for surface pressure
  - good agreement centerline Mach contours

- Pressure coefficients for vehicle nose
  - predicted stagnation $C_p$ agrees with isentropic theory
  - $M = 1.1$  $C_p=1.36$ (CFD / GASP)
  - $M = 1.1$  $C_p=1.34$ (Theory)
Transonic Grid

(159x129x65)
Supersonic Grid

(159x129x43)
COMPARISONS

• Code-to-code comparison
  – good agreement for surface pressure and centerline Mach contours

• 3 aerodynamic coefficients are compared between codes and with APAS database
  – excellent agreement in high supersonic case
  – good agreement in transonic case
Surface Pressure and Centerline Mach Contours

$M = 1.1$ & $AOA = 6 \text{ deg.}$ (LUS soln)
Surface Pressure and Centerline Mach Contours

M=1.1 & AOA=6deg. (GASP Soln)
Surface Pressure and Centerline Mach Contours
Mach=1.1 & AOA=6 deg. (OVERFLOW Soln)
Surface Pressure and Centerline Mach Contours
Mach=1.1 & AOA=6 deg. (OVERFLOW Viscous Soln.)
Cp vs. X (Mach=1.1, AOA=6deg.)

Viscous Turbulent, FDNS solution

- Upper Surface
- Lower Surface

X in ft
Centerline Surface Pressure Distributions

Mach=1.1 & AOA=6deg.

-g22
-Cp

Zone 1–2

Zone 2–3

X in ft.

GASP Soln.
Centerline Surface Pressure Distributions

Mach=1.1 & AOA=6 deg.

\(-C_p\) vs. \(X\) in ft.

OVERFLOW Soln.
Surface Pressure and Centerline Mach Contours

$M=5.72$ & $AOA=6\,\text{deg.}$ (GASP Soln)
Surface Pressure and Centerline Mach Contours

$M=5.72 \& \text{AOA}=6\text{deg. (OVERFLOW Soln)}$
Centerline Surface Pressure Distributions

Mach=5.72 & AOA=6deg.

-Cp

Zone 1–2

Zone 2–3

GASP Soln.
Centerline Surface Pressure Distributions

Mach=5.72 & AOA=6 deg.

$Cp$ vs $X$ in ft.

- Upper Surface
- Lower Surface

OVERFLOW Soln.
### RLV Concept Study Review

- **Predicted aerodynamic coefficients**

<table>
<thead>
<tr>
<th></th>
<th>APAS</th>
<th>OVERFLOW</th>
<th>GASP (inv.)</th>
<th>GASP (vis.)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Transonic</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( M = 1.10 &amp; \alpha = 6^\circ )</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_N )</td>
<td>0.34</td>
<td>0.31</td>
<td>0.32</td>
<td>0.31</td>
</tr>
<tr>
<td>( C_A )</td>
<td>0.12</td>
<td>0.23</td>
<td>0.20</td>
<td>0.22</td>
</tr>
<tr>
<td>( C_M )</td>
<td>-0.049</td>
<td>-0.035</td>
<td>-0.041</td>
<td>-0.04</td>
</tr>
<tr>
<td><strong>Supersonic</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( M = 5.72 &amp; \alpha = 6^\circ )</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>( C_N )</td>
<td>0.07</td>
<td>0.0672</td>
<td>N/A</td>
<td>0.0651</td>
</tr>
<tr>
<td>( C_A )</td>
<td>0.07</td>
<td>0.0714</td>
<td>N/A</td>
<td>0.0735</td>
</tr>
<tr>
<td>( C_M )</td>
<td>-0.002</td>
<td>-0.0028</td>
<td>N/A</td>
<td>-0.0033</td>
</tr>
</tbody>
</table>
Surface Pressure
Ascent Aerodynamics Simulation
(Mach=0.51, Pitch=11 deg., Yaw=-8.6 deg.)
Mach 0.51 T=50 sec. with Trim - Y Shear

Mach 0.51 T=50 sec. with Trim - RZ Moment

X Station (in)
SSTO Winged Body WB001
Von Mises Stresses
Mach 0.51 time=50sec.
CONCLUSIONS

• Predicted aerodynamic data and surface pressure for WB001 using 3 three codes at several flow conditions

• 3D finite element model and CFD pressure distribution provides the visual representation regarding structural deformations, load paths and stress patterns

• Base flow interactions (plume expansions, base recirculations, etc.) could affect the overall solutions; therefore must be considered in future work
The lifting body/linear aerospike is one of three configurations being studied for an SSTO vehicle. A preliminary aerodynamic database existed for the current lifting body configuration, however, this database was developed without considering plume effects. A combined effort by the Computational Fluid Dynamics and Experimental Fluids Dynamics Branches was undertaken to determine first order effects of plume/external flow interactions on vehicle aerodynamics of this lifting body/linear aerospike configuration. Of interest were plume pumping/entrainment at low Mach numbers and plume induced separation of flow over the vehicle at higher altitudes. The CFD analysis included combinations of four Mach numbers, two angles of attack, and four throttle settings. The majority of the CFD was two dimensional centerline analysis of the lifting body/aerospike. Incremental plume effects were derived by comparing the power-on, power-off, and throttled cases and were extrapolated to the preliminary aerodynamic database.

The plume had little effect on the vehicle aerodynamics for supersonic freestream velocities. At subsonic freestream velocities, the plume affected the vehicle aerodynamics through both jet pumping/entrainment and the jet flap effect.
Assessment of Lifting Body Linear Aerospike Plume Effects on Vehicle Aerodynamics

Presented to:
Workshop for CFD Applications in Rocket Propulsion
Marshall Space Flight Center
MSFC, Alabama

Mr. Joseph H. Ruf
Mr. Bruce T. Vu
Mr. Francisco Canabal
Computational Fluid Dynamics Branch
Mr. Alonzo L. Frost
Experimental Fluid Dynamics Branch
Marshall Space Flight Center
April 27, 1995
Assessment of Lifting Body/Linear Aerospike Plume Effects on Vehicle Aerodynamics

Overview

- Introduction
- Objective and Approach
- 2D CFD Results
- Application of CFD
- Conclusions
Introduction

- Lifting body with integrated linear aerospike is one of the three candidate configurations for X33/RLV.

- The then current lifting body configuration was the Lockheed K10.

- A preliminary aerodynamic database w/o plume effects and a preliminary flight trajectory existed.

- Because of vehicle configuration and close proximity of plume to vehicle, it was felt there were potential plume/external flow interactions.
  - Subsonic/low altitude - plume entrainment/jet pumping
  - Supersonic/medium and high altitude - plume induced separated flow on vehicle (à la, Saturn, Shuttle)
CONFIGURATION K-10

Sref = 5,600 sq. ft.
Swet = 13,100 sq. ft.

Total Internal Volume = 67,000 cu. ft.
Main Hyd. Tank Volume = 47,500 cu. ft.
Fwd LOX Tank Volume = 5,200 cu. ft.
Aft LOX Tank Volume = 12,700 cu. ft.
Objective

Determine first order effects of plume/external flow interactions on vehicle pitch plane aerodynamics to generate plume effect increments for the aerodynamic database.

Approach

- Performed series of 2D CFD analyses of K10 centerline to determine vehicle aerodynamics at different flight conditions.
  - Three Mach numbers: 0.6, 1.2, 3.0
  - Power-off, power-on, power-on-throttled
  - Two angles of attack: 0 and 6 degrees

- Two Dimensional analysis was chosen for several reasons.
  - Short deadline for decision on a flight test.
  - 3D geometry quite complex. Long turn around time for above parametrics with 3D CFD.
  - 2D would allow for many more cases to be perform in the parametric study.
  - 2D analysis would exaggerate plume/vehicle flow interaction - conservatism (axisymmetric calculations could also be run as lower bound on interactions).
  - The centerline pressure coefficients could be extrapolated to 3D vehicle.
<table>
<thead>
<tr>
<th>Freestream Mach</th>
<th>0.6</th>
<th>1.2</th>
<th>3.0</th>
<th>5.0</th>
</tr>
</thead>
<tbody>
<tr>
<td>A.O.A.</td>
<td>α=0</td>
<td>α=6</td>
<td>α=0</td>
<td>α=6</td>
</tr>
<tr>
<td>Throttle Setting</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Power - off</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Power - on</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Power -on-throttled</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>70%/130%</td>
<td>x</td>
<td>x</td>
<td></td>
<td></td>
</tr>
<tr>
<td>130%/70%</td>
<td></td>
<td></td>
<td>x</td>
<td></td>
</tr>
</tbody>
</table>
Approach, cont.

- **Grid(s)**
  - Vehicle centerline profile was extracted from 3D surface grid.
  - 2D grid generated with GENIE, refined with GEN2D. Two zone grid, one for forward part of vehicle, one for vehicle base/aerospike. 43000 points total.
  - Different grid wall spacing for each freestream Mach number for acceptable y⁺’s.

- **CFD**
  - GASP v2.3, Baldwin Lomax
  - Frozen flow, two species: air and a hot gas. Hot gas was average properties of exhaust products.
  - Convergence based on vehicle pressure coefficients reaching steady state.
  - CPU hours required varied from 0.5 to 15 hrs. Typical was about 8 hrs.

- **Derivation of plume effect increments**
  - Compared power-on, power-off and throttled cases to generate plume effect increments to preliminary data base.
  - Centerline 2D pressure deltas were applied to limited areas of total vehicle and extrapolated for the actual 3D geometry.
  - Incremental pressure distribution was integrated to determine total aerodynamic force and moment coefficients.
CFD Results

- **Mach 5.0 ~ 124Kft**
  - no significant plume billowing present
  - no significant plume induced separation on the vehicle

- **Mach 3.0 ~ 76Kft**
  - no significant plume billowing
  - no significant plume induced separation on the vehicle

- **Mach 1.2 ~ 31Kft**
  - no significant plume effects

- **Mach 0.6 ~ 12Kft**
  - 0 degree $\alpha$ power-on vs. power-off
    - minor effect seen
  - 6 degree $\alpha$ power-on vs. power-off
    - significant plume effects through plume entrainment and jet flap effect
Mach Number
Mach 5.0 Freestream, 126Kft
K10 2D Centerline, 0 degrees Alpha

CONTOUR LEVELS
0.00
0.20
0.40
0.60
0.80
1.00
1.20
1.40
1.60
1.80
2.00
2.20
2.40
2.60
2.80
3.00
3.20
3.40
3.60
3.80
4.00
4.20
4.40
4.60
4.80
5.00
5.20
5.40
5.60
5.80
6.00
6.20
6.40
6.60
6.80
7.00
7.20
7.40
7.60

q.2.img
Mach Number
Mach 0.6 Freestream, 12Kft
K10 Centerline 20, 0 degree Alpha

CONTOUR LEVELS
K10 2D Centerline Pressure Coefficient
Mach 0.6  Alpha=0 degrees

Data from aircraft munition stores test, Cottrell, 1987.
CFD Results, cont.

- **Mach 0.6 ~ 12Kft**
  - 0 degree $\alpha$ power-on vs. power-on-throttled
    » Deflected plume effects lift consistently over length of vehicle through jet flap effect.
  - 6 degree $\alpha$ power-on vs. power-on-throttled
    » Deflected plume effects lift differently on forward and aft part of vehicle through jet flap effect.
K10 2D Centerline Pressure Coefficients

Mach 0.6 Alpha=0 degrees

P-On vs. P-On-Throttled (70%/130% RPL Upper/Lower Thrusters)

Difference due to deflected plume acting as jet flap over majority of vehicle.

Mach 0.6 Alpha=6 degrees

P-On vs. P-On-Throttled (70%/130% RPL Upper/Lower Thrusters)

Decrease in lift due to deflected plume for vehicle at AOA over majority of vehicle.

Mach 0.6 Alpha=6 degrees

P-On vs. P-On-Throttled (130%/70% RPL Upper/Lower Thrusters)

Increase in lift due to deflected plume for vehicle at AOA over majority of vehicle.
Mach 0.6 Alpha = 0 degrees Aft End

Plume deflected down decreases local negative lift on aft end.

Mach 0.6 Alpha = 6 degrees Aft End

Plume deflected up increases local negative lift on aft end.

Plume deflection increases lift consistently over entire vehicle.
Application of CFD to Preliminary Database

- Increments were only generated for subsonic portion of aero database

- Pressure deltas were applied to database by assuming full effect at aft end and linearly decreasing to no effect at forward end of vehicle
Plume effects on LB pressure distribution @ Mach 0.6, angle-of-attack=0 deg

DTC - differential throttle control; upper/lower @ 130%/70%; Pon = 100%/100%
dCp = difference in pressure coefficient (power on - power off)
Plume effects on LB normal force coeff. @ alpha=0

Plume effects on LB pitching moment coeff. @ alpha=0

S_ref=5600 sq ft; L_ref=1450.0 in.; mom. ref. @ 72%.
DTC - differential throttle control; upper/lower @ 130%/70%; Pon = 100%/100%

dCp = difference in pressure coefficient (power on - power off)
Conclusions

- No significant plume aerodynamic effects existed for supersonic flight regime. Plume induced separation of flow over vehicle was not a significant effect on aerodynamics of K10.
- No significant plume effects seen between power-on and power-off at 0 degree $\alpha$ for Mach numbers analyzed.
- Significant plume/external flow interactions existed for Mach 0.6 at angles of attack through jet flap effect, jet entrainment and change in effective cowl angle.
  - Jet flap effect propagated well forward in 2D analysis.
  - Jet entrainment and increased/decreased effective cowl angle affected the aft end of K10.
- 3D calculations are under way. Initial results indicates less plume effect on the forward part of vehicle than in 2D analysis.
- A methodology has been developed to generate first order plume effect increments for a power-off aerodynamic database using 2D centerline CFD analysis.
TPS Sizing for Access-to-Space Vehicles

William Henline, David Olynick and Grant Palmer
NASA Ames Research Center, MS 230-2,
Moffett Field, CA 94035-1000

Y.-K. Chen
Eloret Institute, MS 234-1, Moffett Field, CA 94035-1000

Abstract

A study was carried out to identify, develop, and benchmark simulation techniques needed for optimum TPS material selection and sizing for reusable launch vehicles. Fully viscous, chemically reacting, Navier-Stokes flow solutions over the Langley wing-body single stage to orbit (SSTO) configuration were generated and coupled with an in-depth conduction code. Results from the study provide detailed thermal protection system (TPS) heat shield materials selection and thickness sizing for the wing-body SSTO. These results are the first ever achieved through the use of a complete, trajectory based hypersonic, Navier-Stokes solution database. TPS designs were obtained for both laminar and turbulent entry trajectories using the Access-to-Space baseline materials such as tailorable advanced blanket insulation (TABI). The TPS design effects (material selection and thicknesses) of coupling material characteristics to the aerothermal environment are illustrated. Finally, a sample validation case using the shuttle flight data base is included.

For the laminar trajectory, the TPS areal mass density is 1.2 lbm/ft², while the turbulent trajectory yields slightly less than 1.3 lbm/ft². An additional conclusion from this study is that the TABI blankets will have to be manufactured in thicknesses greater than 1.5-2.0 inches. Further, if typical turbulent flow conditions are found on these SSTO vehicles during re-entry, some of the baseline materials may experience significant over-temperatures.
TPS Sizing for Access to Space Vehicles

by

William Henline, David Olynick, Grant Palmer and Y.-K. Chen

NASA Ames Research Center

CFD Workshop April 27, 1995
Relationship Between Ames Complementary Analysis Tasks For All Candidate TPS

**TPS System Integration Task**
- Engineering analysis of life-cycle performance (vehicle+operational) performance
- Required Vehicle System Data
- Benchmarks Along Trajectory

**TPS Point Design Task**
- High-fidelity aerothermal environments and TPS sizing
- Environments & TPS Selection
- Accurate Surface B.C.

**Surface Catalysis Task**
- Accurate TPS surface property characterization

Tasks provide quantitative methodology for assessing life-cycle performance (including operations) of all candidate TPS and thus OMB TPS criteria
TPS Sizing for Access to Space Vehicles

Winged Body Configuration

AFRSI on the upper surfaces
Carbon-Carbon on the winglets
Carbon-Carbon on the Wing Leading Edges
Carbon-Carbon on the Nose
TABI on the Lower Surfaces

Lockheed Lifting Body

Use of New TPS Technology Reduces Life Cycle Cost by 20%

McDonnell Douglas DC-3

NASA Ames Research Center

Space Technology Division
Fully Coupled Thermal Analysis for TPS Sizing

Real-GAS
Navier-Stokes
CFD

\[ q \rightarrow T \]

In-depth
Conduction

Experimentally Characterized
Material Properties
(catalysis, emissivity, etc.)
Trajectory (Altitude - Velocity) Plot for the LaRC SSTO Vehicle

Once Around - Abort with 1100 NM Cross-Range

- CFD Anchor Point Solutions

Laminar Flow  Transition and Turbulent Flow

Trajectory Time (sec)

LaRC - Weilmeunster/Wurster 4/94
Material Mapping From
from CFD solution
at 1000 s
fully catalytic

Red - ACC (T>2000°F)
Green - TABI (1200°F-2000°F)
Blue - AFRSI (T<1200°F)

Surface Temperature Contours using Material
Dependent Surface Kinetics (Stewart/NRA)
and Emissivity at
Peak Heating (1300 s)

Winged Body Configuration 1300 s
Material Mapping
Orange-TABI (Temperature Limit Exceeded)
2000 °F < T < 2300 °F

Red - ACC (T > 2000°F)
Green - TABI (1200°F - 2000°F)
Blue - AFRSI (T < 1200°F)

Heat Transfer Contours (W/cm²) using Material Dependent Surface Kinetics (Stewart/NRA) and Emissivity at Peak Heating

Winged Body Configuration 1300 S
Surface TPS Thickness (in.)

3.71973
3.47242
3.2251
2.97779
2.73048
2.48317
2.23585
1.98854
1.74123
1.49391
1.2466
0.999288
0.751975
0.504662
0.257349

Top Layer TPS Thickness (in.) for the LaRC Winged Body SSTO Vehicle (Total Heating Time, 6200 sec)

(TURBULENT FLOW SOLUTION)
Effect of TPS Material Properties on Surface Temperatures

- $\gamma = 1, \varepsilon = 0.85$
- Material Dependent $\gamma$ and $\varepsilon$
- Material Dependent $\gamma$ and $\varepsilon = 0.85$
- ACC $\gamma = 0$, TABI, AFRSI $\gamma = 1, \varepsilon = 0.85$
- ACC $\gamma = 0$, TABI, AFRSI $\gamma = 1$, Material Dependent $\varepsilon$

Windward  $\leftarrow|\rightarrow$  Leeward

$z (\text{m})$  

$T (^{\circ} \text{F})$
Shuttle Temperature Contours (STS-2)
M=24, 72.4 km, radiative equilibrium wall
RCG kinetics, $\varepsilon=0.85$

1500 °F
500 °F
750 °F
1000 °F
1750 °F
2000 °F
2500 °F

Reacting Flow Environments Branch
NASA Ames Research Center
Heat Transfer Profile Along the Windward Centerline
STS-2, Mach 24.3, $\varepsilon=.85$, RCG kinetics
Radiative Equilibrium Wall

$q (W/cm^2)$

$\sigma$  

Experiment
Computation

$x (m)$

Reacting Flow Environments Branch  NASA Ames Research Center
COMPUTATIONAL ISSUES ASSOCIATED WITH TEMPORALLY DEFORMING GEOMETRIES SUCH AS THRUST VECTORING NOZZLES

Kishore Boyalakuntla, Bharat K. Soni, Hugh J. Thornburg and Robert Yu

NSF Engineering Research Center for Computational Field Simulation
Mississippi State University
Mississippi State, MS 39759
ABSTRACT

During the past decade Computational Simulation of fluid flow around complex configurations has progressed significantly and many notable successes have been reported, however unsteady time–dependent solutions are not easily obtainable. The present effort involves unsteady time dependent simulation of temporally deforming geometries. Grid generation for a complex configuration can be a time consuming process and temporally varying geometries necessitate the regeneration of such a grid for every time step. Traditional grid generation techniques have been tried and demonstrated to be inadequate to such simulations. NURBS based techniques provide a compact and accurate representation of the geometry. This definition can be coupled with a distribution mesh for a user defined spacing. The present method greatly reduces cpu requirements for time dependent remeshing, facilitating the simulation of more complex unsteady problems. A Thrust Vectoring Nozzle has been chosen to demonstrate the capability as it is of current interest in aerospace industry for better maneuverability of fighter aircraft in close combat and in post stall regimes. This current effort is the first step towards multidisciplinary design optimization which involves coupling the aerodynamic heat transfer and structural analysis techniques. Applications include simulation of temporally deforming bodies and aeroelastic problems.

A NURBS based volume grid generation technique is used for remeshing at each timestep. Remeshing is easily accomplished by varying the control points and time dependent motion is contained in the motion of the control points. Timestep controls the movement of control points. Great flexibility in geometric definition is achieved. The grid generation code is successfully coupled with UBIFLOW and INS3d which are compressible and incompressible flow solvers respectively.

Various geometries such as converging diverging nozzle, duct and thrust vectoring nozzle have been simulated and will be presented.
OBJECTIVE:

The objective of this work is to develop a capability for CFD simulation of temporally deforming geometries such as thrust vectoring nozzles.

1. Efficient grid generation system.

2. Flow solver that can handle moving geometries.
MOTIVATION:

1. Biomedical flows (heart valve, blood flow in arteries or veins).

2. Flapping of wing, helicopter rotor.

3. Thrust vectoring.

4. Wear and deformation on bodies, molding.

5. Coupling with finite element for structural analysis for design optimisation.
DESIRED FEATURES:

1. Good control over mesh point spacing.
2. Grid quality.
3. Grid generation must be user independent.
4. Geometric fidelity.
5. Consume moderate CPU time.
6. Handle severe deflections.
7. Flowsolver that can handle moving boundaries.
OVERVIEW OF STUDY:

1. Approach.
2. Grid generation.
3. Why NURBS?
5. Why thrust vectoring?
6. Ongoing research
7. Results.

PRETTY PICTURES IN BETWEEN

and ?????????
APPROACH:

1. Grid Generation is discretised representation of volume in interest.

2. Temporally deforming bodies require remeshing every time-step.

3. TFI and Elliptic Grid Generation techniques have been found to be inadequate.

4. NURBS based volume grid generation techniques are well suited for temporally varying geometries.
NURBS based volume grid generation (YU Method):

1. SIGNIFICANT FEATURE IS THAT IT ONLY REQUIRES THE CONTROL POINTS, WEIGHTS AND DISTRIBUTION MESH.

2. REDUCES CPU REQUIREMENT.

3. REMESHING IS EASILY DONE.

4. CONTROL POINTS AND THE ORDER DEFINE THE GEOMETRY.

5. USER DEFINED SPACING IS ACCOMPLISHED THROUGH USE OF DISTRIBUTION MESH.
CONTROL POINTS FOR AIRCRAFT WING (4 x 2 x 9)
6. TIME DEPENDENT MOTION IS ACCOMPLISHED BY MOVING THE CONTROL POINTS OR BY INTERPOLATION.

7. THE AMOUNT OF MOVEMENT IS CONTROLLED BY THE TIME STEP.

8. FLEXIBILITY IN GEOMETRIC DEFINITION

9. ADAPTATION CAN BE EASILY ACHIEVED BY ADAPTING THE DISTRIBUTION MESH.
FLOW SOLVER:

- Grid generation code is coupled with
  
  UBIFLOW
  INS3D

- Time metrics are calculated as

  \[ \frac{dx}{dt}(i,j,k) = \frac{(x(i,j,k) - x_0(i,j,k))}{dt} \]
  \[ \frac{dy}{dt}(i,j,k) = \frac{(y(i,j,k) - y_0(i,j,k))}{dt} \]
  \[ \frac{dz}{dt}(i,j,k) = \frac{(z(i,j,k) - z_0(i,j,k))}{dt} \]

UBIFLOW calculates the time metrics

Time metrics have to be calculated in grid code in INS3D
UBIFLOW (Whitfield and Arabshahi):

- Multiblock compressible Navier–Stokes solver
- Cell centered finite volume scheme
- Flux difference splitting on the RHS and upwind difference using flux splitting on LHS
- Accounts for the block boundary movement
- INS3D is an incompressible Navier–Stokes solver and both can solve unsteady and time varying flow.
THRUST VECTORING:

1. Reasonably complex problem.

2. Increase performance.

3. Shrink high drag, radar reflecting horizontal and vertical tail.


5. Reduce landing and take-off distances.

6. Greater payload capability for air — to ground mission aircraft.

7. Reduce over the deck wind speed requirements for aircraft operating on aircraft carriers.
Pressure Distribution for timesteps of 0, 273, 363, 423
Circular to Rectangular Transition Duct
CIRCULAR TO RECTANGULAR TRANSITION DUCT (0 timestep)
grid size (100 x 50 x 50) k = 25 plane

Thrust Vectoring (70 timestep)
grid size (100 x 50 x 50) k = 25 plane

Thrust Vectoring (30 timestep)
grid size (100 x 50 x 50) k = 25 plane

Thrust Vectoring (150 timestep)
grid size (100 x 50 x 50) k = 25 plane
Circular to rectangular transition duct
0 timestep

THRUST VECTORING (150 timesteps)
ONGOING RESEARCH:

Adaption on the distribution mesh and link it with the moving grid code
RESULTS:

1. CPU of 2.1 secs / iteration for a grid size of 60*40*35 on a onyx 150 MHz processor

2. Memory of the moving grid code is 1.8 Megs

3. Linked with flowsolver it does not consume any memory.

4. Flow solver consumes 75 secs of CPU per / iteration.

5. Adaption is easy to perform as it is on the distribution mesh.

6. Linked succesfully to UBIFLOW and INS3D.

7. Converging —diverging , 2—D and axisymmetric thrust vectoring nozzle have been succesfully simulated.
HYBRID GRID TECHNIQUES FOR PROPULSION APPLICATIONS

Roy P. Koomullil, Bharat K. Soni, and Hugh J. Thornburg
NSF Engineering Research Center for Computational Field Simulation
Mississippi State University
Mississippi State, MS 39762

ABSTRACT

During the past decade computational simulation of fluid flow for propulsion applications has progressed significantly, and many notable successes have been reported in the literature. However, the generation of a high quality mesh for such problems has often been reported as a pacing item. Hence, much effort has been expended to speed this portion of the simulation process. Several approaches have evolved for grid generation. Two of the most common are structured multi-block, and unstructured based procedures. Structured grids tend to be computationally efficient, and high aspect ratio cells necessary for efficiently resolving viscous layers. Structured multi-block grids may or may not exhibit grid line continuity across the block interface. This relaxation of the continuity constraint at the interface is intended to ease the grid generation process, which is still time consuming. Flow solvers supporting non-contiguous interfaces require specialized interpolation procedures which may not ensure conservation at the interface. Unstructured or generalized indexing data structures offer greater flexibility, but require explicit connectivity information and are not easy to generate for three-dimensional configurations. In addition unstructured mesh based schemes tend to be less efficient and it is difficult to resolve viscous layers.

Recently, hybrid or generalized element solution and grid generation techniques have been developed with the objective of combining the attractive features of both structured and unstructured techniques. In the present work recently developed procedures for hybrid grid generation and flow simulation are critically evaluated, and compared to existing structured and unstructured procedures in terms of accuracy and computational requirements.

In the present grid generation procedure multi-body configurations are decomposed into a number of simple geometric entities. A structured grid generator is first employed to construct a high quality grid around the body with appropriate packing. One grid must be designated as a main grid and enclose the solid surfaces of all other component grids. Upon completion these structured grids are converted to the hybrid grid data structure format. Based upon an input normal distance from the surface, holes are cut in the main grid for each component grid. Overlapping and hole cells are deleted from the hybrid grid data structure. Delaunay triangulation is then used to construct cells to fill the void between the cut main grid and the truncated component grid. Upon completion of this procedure the hybrid grid is written in a format usable by the flow solver.

The non-dimensionalized Euler equations in integral form provide the mathematical formulation for this scheme. The discretized flow domain is represented by a set of non-overlapping polygons and the cell averaged variables are stored at each cell center. Each individual cell is treated as its own control volume. The numerical flux at the cell edge is calculated using Roe's approximate Riemann solver. An assumed linear distribution in each cell is employed to reconstruct the edge values, which results in a second order discretization. The flux limiting procedure of Barth is used to suppress spurious oscillations near discontinuities. An implicit pseudo-time integration procedure using the Generalized Minimum RESidual (GMRES) method for solving the sparse matrix system is employed. The results has been verified with the standard benchmark results.
Hybrid Grid Techniques For Propulsion Applications

Roy P Koomullil, Dr. Bharat K. Soni and Dr. Hugh J. Thornburg

Sponsors: Teledyne Brown Engineering and AFOSR

National Science Foundation Engineering Research Center For Computational Field Simulation
Mississippi State University.
OUTLINE

- MOTIVATION
- GRID GENERATION APPROACH
- DATA STRUCTURE
- FLOW SOLVER
- RESULTS
- CONCLUSIONS
MOTIVATION

UNSTRUCTURED GRIDS

- GREATER FLEXIBILITY IN HANDLING COMPLEX CONFIGURATIONS
- EASE OF GRID ADAPTION
- DIFFICULT TO MAKE HIGHLY STRETCHED VISCOUS GRIDS
- DIFFICULT TO RESOLVE CONVECTIVE AND VISCOUS FLUXES FOR HIGH REYNOLDS NUMBERS
- TURBULENCE MODELLING IS DIFFICULT

HYBRID GRIDS

- COMBINING ADVANTAGES OF STRUCTURED AND UNSTRUCTURED GRIDS
- GRID GENERATION TIME CAN BE REDUCED
GRID GENERATION

1. DECOMPOSE COMPLEX BODIES INTO SIMPLE ENTITIES
2. GENERATE STRUCTURED GRIDS FOR THESE GEOMETRIC ENTITIES USING STANDARD PACKAGES
3. CUT HOLES IN THE MAIN GRID WHERE THE COMPONENTS OVERLAP
4. CONNECT COMPONENT GRIDS USING UNSTRUCTURED GRIDS BY DELAUNAY TRIANGULATION OR OTHER METHODS
5. AVOID INTERPOLATIONS OF CONSERVED VARIABLES BETWEEN THE COMPONENT AND MAIN GRIDS AS IN CAMERA GRIDS
DATA STRUCTURE

EDGE BASED DATA STRUCTURE

- $\text{EDGE}(K,1) = \text{FIRST NODE (N1)}$
- $\text{EDGE}(K,2) = \text{SECOND NODE (N2)}$
- $\text{EDGE}(K,3) = \text{CELL ON LEFT (C1)}$
- $\text{EDGE}(K,4) = \text{CELL ON RIGHT (C2) or BOUNDARY CONDITION}$

ADVANTAGE: ANY ARBITRARY POLYGONS CAN BE HANDLED
CELL AREA

\[ Area = \oint_{\partial\Omega} x \, dy = \sum_{edges} x_e \, dy \]

Loop over the edge

\begin{align*}
N1 & = \text{EDGE}(I, 1) \\
N2 & = \text{EDGE}(I, 2) \\
C1 & = \text{EDGE}(I, 3) \\
C2 & = \text{EDGE}(I, 4) \\
dy & = Y(N2) - Y(N1) \\
XE & = (X(N2) + X(N1)) \times 0.5 \\
\text{AREA(C1)} & = \text{AREA(C1)} + XE \times dy \\
\text{AREA(C2)} & = \text{AREA(C2)} - XE \times dy
\end{align*}

dloop
GOVERNING EQUATIONS

\[ \frac{\partial}{\partial t} \int_{\Omega} Q \, dA + \int_{\partial\Omega} F(Q) \cdot n \, ds = 0 \]

Where

\[ F = f \, i + g \, j \quad n = n_x \, i + n_y \, j \]

\[ Q = \begin{bmatrix} Q \\ Qu \\ Qv \\ E \end{bmatrix} \quad f = \begin{bmatrix} Qu \\ Qu^2 + p \\ Quv \\ u(E + p) \end{bmatrix} \quad g = \begin{bmatrix} Qv \\ Quv \\ Qv^2 \\ v(E + p) \end{bmatrix} \]

\[ p = (\gamma - 1) \left[ E - Q \left( \frac{u^2 + v^2}{2} \right) \right] \]

Non Dimensionalization w.r.t freestream conditions
FINITE VOLUME DISCRETIZATION

\[ A_i \frac{\partial Q_i}{\partial t} = - \int_{\partial \Omega} F(Q_i) \cdot n \, ds = - \sum_{j=1}^{k} F_{ij} \cdot n_j \, ds_j \]

Where
- \( k \) Number of sides of the polygon
- \( i \) Cell number
- \( j \) Edge number

SUMMATION OF FLUXES

Loop Over the edges

CALCULATE \( F_{ij} \)

\[
\text{FLUX (C1)} = \text{FLUX(C1)} + F_{ij} \\
\text{FLUX (C2)} = \text{FLUX(C2)} - F_{ij}
\]

Endloop
APPROXIMATE RIEMANN SOLVER (ROE)

\[ F_{ij} = \frac{1}{2} \left[ F(Q_R) + F(Q_L) - |A| (Q_R - Q_L) \right] \]

where

\[ |A| (Q_R - Q_L) = |\Delta F_{1,2}| + |\Delta F_3| + |\Delta F_4| \]

Corresponding to 3 different eigen values

\[ \lambda_{1,2} = u \quad \text{and} \quad \lambda_{3,4} = u \pm c \]
EXPLICIT SCHEME
( RUNGE–KUTTA TIME INTEGRATION )

\[ Q^{(0)} = Q^{(N)} \]
\[ Q^{(1)} = Q^{(0)} + \alpha_1 \frac{\Delta t}{A} R(Q^{(0)}) \]
\[ Q^{(2)} = Q^{(0)} + \alpha_2 \frac{\Delta t}{A} R(Q^{(1)}) \]
\[ Q^{(3)} = Q^{(0)} + \alpha_3 \frac{\Delta t}{A} R(Q^{(2)}) \]
\[ Q^{(3)} = Q^{(0)} + \alpha_4 \frac{\Delta t}{A} R(Q^{(3)}) \]
\[ Q^{(N+1)} = Q^{(4)} \]

\[ \alpha_1 = 0.0833 \quad \alpha_2 = 0.2069 \quad \alpha_3 = 0.4265 \quad \alpha_4 = 1.0 \]

Where \[ R = - \sum_{edges} F_{ij} \cdot n \, ds \]
IMPLICIT SCHEME

\[ F_{ij}^{n+1} = \frac{1}{2} \left( F(Q_R^{n+1}) + F(Q_L^{n+1}) - |A| \cdot (Q_R^{n+1} - Q_L^{n+1}) \right) \]

LINEARIZATION

\[ F_{ij}^{n+1} = F_{ij}^n + \left[ \frac{\partial F_{ij}}{\partial Q_R} \right]^n \Delta Q_R + \left[ \frac{\partial F_{ij}}{\partial Q_L} \right]^n \Delta Q_L \]

APPROXIMATE ANALYTIC JACOBIANS

NUMERICAL JACOBIANS
APPROXIMATE ANALYTIC JACOBIANS

\[ F_{ij}^{n+1} = \frac{1}{2} \left( F(Q_R^{n+1}) + F(Q_L^{n+1}) - |A| \cdot (Q_R^{n+1} - Q_L^{n+1}) \right) \]

\[ D_i \Delta Q_i + \sum_{j=1}^k \left( U_{n(j)} \Delta Q_{n(j)} \right) = R^n \]

Where \( D_i = \frac{V_i}{\Delta t} I + \frac{1}{2} \sum_{j=1}^k \left( H_{i} + |\bar{H}|_{\int_{n+1}^{n+1}} \right) \)

\[ U_{n(j)} = \frac{1}{2} \left( H_{n(j)} - |\bar{H}|_{\frac{n+1}{2}} \right) \]

\[ A \cdot N_x + B \cdot N_y = H, \quad |\bar{A}| \cdot N_x + |\bar{B}| \cdot N_y = |\bar{H}| \]

\[ A = \frac{\partial f}{\partial Q}, \quad & B = \frac{\partial g}{\partial Q} \]
NUMERICAL FLUX JACOBIAN

\[ A_{ij}(Q) = \frac{F_i \left( Q + h \ e_j \right) - F_i \left( Q \right)}{h} \]

Where \( e_j \), \( j^{th} \) unit vector

\[ h = (\text{machine zero})^{1/2} \]

\( j^{th} \) Column of \( F'(Q) = \frac{F \left( Q + h \ e_j \right) - F \left( Q \right)}{h} \)
SPARSE MATRIX STRUCTURE

GRID

1 2 3

4 x x
5 x x

6 x x

MATRIX STRUCTURE

1 2 3 4 5 6

1 x x x x x x
2 x x x x x x
3 x x x x x x
4 x x x x x x
5 x x x x x x
6 x x x x x x
HIGHER ORDER SCHEME

\[ Q(x, y) = Q(x_i, y_i) + \nabla Q(x_i, y_i) \cdot \Delta r \]

Where

\[ \Delta r = (x - x_i)i + (y - y_i)j \]

Using Green's Theorem

\[ \nabla (Q) = \frac{1}{V_i} \int_{\partial \Omega} Q \cdot n \, ds \]

\[ \sum_{j=1}^{nk} \frac{Q_{c_j}}{r_j} \]

\[ Q_{n_i} = \frac{1}{nk} \sum_{j=1}^{nk} \frac{1}{r_j} \]
LIMITER (Barth)

\[ Q(x, y) = Q(x_i, y_i) + \phi_i \nabla Q(x_i, y_i) \cdot \Delta r \]

\[ Q_i^{\text{min}} = \min (Q_{ci}, Q_{adj}) \]
\[ Q_i^{\text{max}} = \max (Q_{ci}, Q_{adj}) \]

then \[ Q_i^{\text{min}} \leq Q(x, y) \leq Q_i^{\text{max}} \]

\[ \phi_{ij} = \begin{cases} 
\min \left\{ 1, \frac{Q_i^{\text{max}} - Q_{ij}}{Q_i - Q_{ij}} \right\} & \text{if } Q_i - Q_{ij} > 0 \\
\min \left\{ 1, \frac{Q_i^{\text{min}} - Q_{ij}}{Q_i - Q_{ij}} \right\} & \text{if } Q_i - Q_{ij} < 0 \\
1 & \text{if } Q_i - Q_{ij} = 0 
\end{cases} \]

and \[ \phi_i = \min(\phi_{ij}') \]
VARIFICATION OF THE RESULTS

Cp Distribution Over NACA0012
(Mach Number = 0.63, Angle of Attack = 2.0 Deg)

- First Order
- Second Order
- Alfa Data

Distance Along the Chord
COMPARISON OF CONVERGENCE

Comparison of Convergence History
No of Nodes = 4000, Number of Cells = 3872, CFL 50

- - - Approximate Analytic Jacobian
- - - Numerical Jacobian

![Graph showing comparison of convergence history with grid and line plot.](image)
SCRAMJET INLET GEOMETRY
CONCLUSION

○ DEVELOPED AND VERIFIED 2-D HYBRID FLOW SIMULATION SYSTEM.

○ NUMRICAL FLUX JACOBIANS GIVES BETTER CONVERGENCE

○ TREATED STRUCTURED AND UNSTRUCTURED GRIDS IN HYBRID SYSTEM AS A SINGLE BLOCK

FUTURE WORK

○ DEVELOP A 3-D NAVIER–STOKES FLOW SIMULATION SYSTEM USING HYBRID GRIDS

○ TEST THE SYSTEM FOR INTERNAL AND EXTERNAL FLOW PROBLEMS
A Structured Grid Based Solution-adaptive Technique for Complex Separated Flows

by

Hugh Thornburg, Bharat K. Soni, Boyalakuntla Kishore, and Robert Yu
NSF Engineering Research Center for Computational Field Simulation
Mississippi State University
Mississippi State, MS 39762

ABSTRACT

The objective of this work has been to enhance the predictive capability of widely used CFD codes through the use of solution adaptive gridding. Most problems of engineering interest involve multi-block grids and widely disparate length scales. Hence, it is desirable that the adaptive grid feature detection algorithm be developed to recognize flow structures of different type as well as differing intensity, and adequately address scaling and normalization across blocks. In order to study the accuracy and efficiency improvements due to the grid adaptation, it is necessary to quantify grid size and distribution requirements as well as computational times of non-adapted solutions. Flowfields about launch vehicles of practical interest often involve supersonic freestream conditions at angle of attack exhibiting large scale separated vortical flow, vortex-vortex and vortex-surface interactions, separated shear layers and multiple shocks of different intensity. In this work a weight function and an associated mesh redistribution procedure is presented which detects and resolves these features without user intervention. Particular emphasis has been placed upon accurate resolution of expansion regions and boundary layers.

Flow past a wedge at Mach = 2.0 is used to illustrate the enhanced detection capabilities of this newly developed weight function. Figure 1 presents weight functions evaluated using the previous procedure, lower half plane, as well as the current procedure, upper half plane.

Figure 1. Comparison of Weight Functions. Figure 2. Comparison of Adapted Grids.

It can be observed that both weight functions clearly detected the primary shock. It can also be seen
that the expansion fan, boundary layer, and the reflected shocks are much more clearly represented in the current weight function. Adapted grids using both weight function formulations are presented in Fig. 2. The high gradient regions of the expansion region are only reflected in the adapted grid using the new weight function. The reflected shock is also much sharper. Figure 3 compares the solution obtained using the current adaption procedure with that obtained using the original grid. The enhanced resolution is clearly evident.

Figure 3. Comparison of Solutions Using Adapted Grid.

Supersonic flow at Mach=1.45 and 14 degree angle of attack has been simulated around a tangent-ogive cylinder. The grid and associated flow solution constructed after two adaption cycles using hybrid differencing of the grid equations and the current weight functions is presented in Figure 4.

Figure 4. Adapted grid after two cycles. Figure 5. Adapted grid after two cycles.

Figure 5 presents the grid constructed using the previous weight function and the same flow conditions and number of adaptation cycles. Figures 3 and 6 present streamwise cuts of the two grids shown in Figs 4 and 5 at X/D = 5.5 and 7.5 respectively.
Figure 8 present the flow solution obtained using the NPARC [NASA 1993] flow solver, the KE turbulence model option and two adaptation cycles. Figure 9 presents the associated weight function.

Examples will presented to demonstrate the capability for solution–adaptive regridding of multi–block launch vehicle simulations.

References


A Structured Grid Based Solution-Adaptive Technique for Complex Separated Flows

Hugh J. Thornburg, Bharat K. Soni, Kishore Boyalakuntla, and Robert Yu

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
April 25–27, 1995
Huntsville, Alabama

1This research was supported, in part, by the Army Research Laboratory.
OBJECTIVES

Improved resolution of complex flows through the use of solution adaptive gridding

1. Develop a weight function suitable for use with a solution adaptive grid redistribution procedure for complex flows, including viscous dominated separation.

2. Minimum user inputs.

3. Appropriate feature detection for a wide range of flow features (Vortices, Shear layers, Shocks).

4. Robust redistribution procedure for use with weight function.
GOVERNING EQUATIONS FOR GRID MOVEMENT

1. Inverted form:

\[
\sum_{i=1}^{3} \sum_{j=1}^{3} g^{ij} \bar{r}_{\xi^i \xi^j} + \sum_{k=1}^{3} g^{kk} P_k \bar{r}_{\xi^k} = 0
\]

Where:
- \( \bar{r}_{ij} \): Position vector,
- \( g^{ij} \): Contravariant metric tensor,
- \( \xi^i \): Curvilinear coordinate, and
- \( P_k \): Control Function.

2. Control of distribution and characteristics of grid system can be achieved by varying control Functions \( P_k \).
\[ P_i = (P_{\text{initial geometry}})_i + c_i (P_{\text{wt}}) \quad (i = 1, 2, 3) \]

where \((P_{\text{initial geometry}})_i\) : control function based on initial grid geometry

\(P_{\text{wt}}\) : control function based on gradient of flow parameter

\(c_i\) : constant weight factors

\[ P_i^{(n)} = P_i^{(n-1)} + c_i (P_{\text{wt}})^{(n-1)} \quad (i = 1, 2, 3) \]

where

\[ P_i^{(1)} = (P_{\text{initial geometry}})^{(0)}_i + c_i (P_{\text{wt}})^{(0)} \quad (i = 1, 2, 3) \]
EVALUATION OF FORCING FUNCTIONS

1. Smoothness.

2. Near orthogonality.

3. Equidistribution of 'error' or weight function.

4. One-dimensional equidistribution law
   \[ W X_{\xi} = \text{constant, where } W \text{ is a weight factor.} \]

5. Poisson equation form, (Anderson, Thompson), obtained by differentiating equidistribution law.
   \[ W X_{\xi\xi} + W_{\xi} X_{\xi} = 0, \]
   \[ X_{\xi\xi} + P X_{\xi} = 0, \]
   \[ \text{i.e. } P = \frac{W_{\xi}}{W} \]

6. For Multiple dimensions:
   \[ P_k = \frac{W_{\xi}^k}{W}, \quad k = 1,2,3 \]
CHARACTERISTICS OF WEIGHT FUNCTIONS

1. Weight functions approximately equidistributed over solution domain.
2. Determine grid spacing and characteristics.
3. Approximation to local truncation error.
   - Use lower order derivatives to approximate high order truncation error terms.
   - Detect structures of disparate strength.
   - Minimum variation of coefficients.
EVALUATION OF WEIGHT FUNCTIONS

1. Density or pressure is not sufficient for viscous flows.
2. Boolean sums used to eliminate 'multiplying' effect.
3. Relative derivatives are necessary to detect features of varying intensity.
5. Nearly uniform flowfields require minimum normalization value.
WEIGHT FUNCTIONS

$$W = \frac{W^1}{\max(W^1, W^2, W^3)} \oplus \frac{W^2}{\max(W^1, W^2, W^3)} \oplus \frac{W^3}{\max(W^1, W^2, W^3)}$$

Where,

$$W^k = 1 + \frac{|q_{zk}|}{|q| + \varepsilon} \oplus \frac{|(qw)_{zk}|}{|(qw)| + \varepsilon} \oplus \frac{|(qv)_{zk}|}{|(qv)| + \varepsilon} \oplus \frac{|(qw)_{zk}|}{|(qw)| + \varepsilon}$$

for $k = 1, 2, 3$, and

The symbol $\oplus$ represents the Boolean sum. Note that the directional weight functions are scaled using a common maximum in order to maintain the relative strength.
OVERALL SOLUTION PROCEDURE

1. Obtain initial flow solution.
2. Adapt grid.
3. Interpolate solution onto adapted grid.
4. Restart flow solution.
5. Repeat steps 2–4 until satisfactory result.
ADAPTIVE GRID PROCEDURE

1. Read PLOT3D grid and solution files.
2. Evaluate weight function,
   (no input parameters).
3. Evaluate and smooth $P_k$.
4. Integrate grid.
5. Interpolate $P_k$ onto current adapted grid.
6. Repeat steps 4 and 5 until convergence.
7. Output adapted grid.
SOLUTION OF GRID EQUATIONS

1. Solution difficulties transferred from flow equations to grid equations.
2. Accuracy not as important for postulated law.
3. Adaptive Central/Upwind differencing scheme, based upon forcing function gradients.
4. Integrated in time using CSIP.
5. Non-linear terms are quasi-linearized.
7. Precise geometry definition is critical.
BOUNDARY POINT MOVEMENT

1. Very important.
   • Orthogonality.
   • Skewness.


3. Boundary surface redistribution based on specified region of surface.
   • Explicit.
   • Local iteration for desired distribution.
   • Can be used to keep sharp corners, and to transfer information between blocks.
Figure 1. Comparison of Weight Functions.

Figure 2. Comparison of Adapted Grids.
Figure 3. Comparison of Solutions Using Adapted Grid.
Case 1 NPARC Sol Grid 7 KE Model
Case 1 NPARC Grid 7 a2 KE Model
Gimble Nozzle, Left UBIFLOW Density, Right Weight Function
Gimble Nozzle, Left UBIFLOW Density, Right Marked Cells
Gimble Nozzle, Left UBIFLOW Density, Right Weight Function
Gimble Nozzle, Left UBIFLOW Density, Right Marked Cells
SUMMARY

1. Developed Weight function which requires no user input.

2. Implemented adaptive upwind/central difference scheme.

3. Demonstrated enhanced grid resolution.
   - Thinner shocks.
   - Stronger circular vorticies.
   - Lower values of artificial dissipation may be used.
   - Larger time steps may be used.
   - Improved convergence behaviour.
   - More closely resembles experimental data.
ONGOING WORK

1. Multiblock problems.
   - Global scaling across blocks.
   - Block interface or block point movement.
2. Local refinement (Solver of Koomullil)
3. Coupling with flow solver.
4. Coding efficiency.
5. Reacting flow.
   - Include temperature in weight function.
6. Unsteady flow problems.
   - $P_k$, viewed as velocities in temporally parabolized grid equations.
GENIE++ – A Multi-Block Structured Grid System

by

Tonya Williams, Naren Nadenthiran, Hugh Thornburg, and Bharat K. Soni

NSF Engineering Research Center for
Computational Field Simulation
Mississippi State University
Mississippi State, MS 39762

ABSTRACT

The computer code GENIE++ (Soni et al. 1992) is a continuously evolving grid system containing a multitude of proven geometry/grid techniques. The generation process in GENIE++ is based on an earlier version. The process uses several techniques either separately or in combination to quickly and economically generate sculptured geometry descriptions and grids for arbitrary geometries. The computational mesh is formed by using an appropriate algebraic method. Grid clustering is accomplished with either exponential or hyperbolic tangent routines which allow the user to specify a desired point distribution. Grid smoothing can be accomplished by using an elliptic solver with proper forcing functions. B-spline and Non-Uniform Rational B-splines (NURBS) algorithms are used for surface definition and redistribution. The built-in sculptured geometry definition with desired distribution of points, automatic Bezier curve/surface generation for interior boundaries/surfaces, and surface re-distribution is based on NURBS. Weighted Lagrange/Hermite transfinite interpolation methods, interactive geometry/grid manipulation modules, and on-line graphical visualization of the generation process are salient features of this system, which result in a significant time savings for a given geometry/grid application.

The development of the system, as well as computational examples of practical interest will be presented to demonstrate the success of these methodologies. Complete documentation is available using Mosaic. Versions are available for PC’s, X window, and SGI systems. It is planned to place this code in the public domain by August 1995.
GENIE++: A Structured Multi-Block Grid System

Bharat K. Soni
Hugh Thornburg
Tonya Williams
Nadesan Narenthiran
Grid Strategies

- Unstructured
  - Advancing Front
  - Delaunay
  - PDES
  - Other

- Structured
  - Algebraic
  - PDES
  - Other
GRID INFLUENCE

- All Positive or All Negative Volumes
- Orthogonality (Not Too Skewed)
- Smooth
- Aspect Ratio

- Accuracy
- Truncation Errors
- Stability
- Treatment of BCS
- Economy ($)
GENERATION STEPS

1. Algebraic Grid
2. Surface Grids and Patches
3. Geometry Definition
4. Distribution of Points
5. Mapping Physical Space
6. Computational Space
7. Refined Grid
STRETCHING FUNCTIONS

** Exponential \[ f(x) = \frac{e^x - 1}{e^x - 1} \]

** Hyperbolic Tangent \[ f(x) = 1 + \frac{\tanh(\alpha(x-1))}{\tanh \alpha} \]

** Hyperbolic Sine \[ f(x) = 1 - \frac{\sinh(\alpha(1-x))}{\sinh \alpha} \]
STRETCHING OPTIONS

Exponential

Hyperbolic Tangent

Options for Distributing Points
A BOUNDARY CURVE, SURFACE, OR VOLUME

\( G(1 \rightarrow I_2, \rightarrow J_1 \rightarrow J_2, K_1 \rightarrow K_2) \)

Curve: \( G(I_1 \rightarrow I_2, J \rightarrow \bar{J}, K \rightarrow \bar{K}) \)

A Surface: \( G(I_1 \rightarrow I_2, J_1 \rightarrow J_2, K \rightarrow \bar{K}) \)

A Volume: \( G(I_1 \rightarrow I_2, J_1 \rightarrow J_2, K_1 \rightarrow K_2) \)
GENIE

Grid Generation Process / Geometry Definition

Volume

Physical Configuration

Sub-Volumes

Contiguous Blocks

Surfaces

Surfaces Patches

Boundary Segments
GEOMETRY GENERATION

• Semi-Interactive Construction

• Analytic:
  Points, Line, Circle, Ellipse, Super-Ellipse,
  Polynomial, Plane, Ruled Surface, Ellipsoid,
  Hyperboloid, Paraboloid, NASA Airfoils, . . .

• Sculptured:
  Spline-Akima, B-Spline, Rational B-Spline,
  Polynomial-Hermite, LaGrange, Bezier,
  Coon's Patch, NURBS, . . .
GEOMETRY MANIPULATION

- Body of Revolution
- Ruling, Marching, TFI, Coon's Patch
- Transformations: Translation, Rotation, Scaling, Mirror Image
- Cut, Paste, Blend, ...
- Intersections and Projections
ALGEBRAIC

* Fast
* Precise Spacing Control
* Interactive User Interface
* Possible Overlapping
* Requires High Degree of Understanding
* Generalization!
* Propagation of Slope Discontinuities

PDES

* Inherent Smoothness
* Resistant To Grid Line Overlapping
* No Propagation of Slope Discontinuities
* Competitive Enhancement of Smoothness, Orthogonality and Concentration
* Readily Adaptable for Generalization
* Distribution Loss
APPROACH

Objective: Accomplish orthogonality – smoothness without any distribution loss.

- Work hard with Algebraic
  - Precise Spacing Control (Grid Spacings, Areas, Volume)
  - Inexpensive and Fast
  - Interior Bezier Curve/Surface Specification for Sub-blocks
  - Weighted Transfinite Lagrange and Hermite Interpolation
  - Precise Spacing Control (Grid Spacings, Areas, Volume)

- Use elliptic for a quick fix
  - Smart Forcing Functions
  - 3-5 Iterations (maximum)
WEIGHTED TRANSFINITE INTERPOLATION

Physical Space

Distribution Space

Computational Space

50 x 40

(i, j)

(s_i, t_j)

(x_i, y_j)
GENERAL ELLIPTIC
GENERATION SYSTEM

\[
\sum_{i-1}^{3} \sum_{j-1}^{3} g^{ij} r_{\xi}^{i} r_{\bar{\xi}}^{j} + \sum_{k=1}^{3} \phi_{k} r_{\xi}^{k} = 0
\]

\[
g^{il} = \frac{1}{g} (g_{jm} g_{kn} - g_{jn} g_{km}) \quad i = 1, 2, 3; j = 1, 2, 3; (i, j, k) \text{ and } (l, m, n) \text{ cyclic}
\]
EVALUATION OF FORCING FUNCTIONS

\[ \sum_{i-1}^{3} \sum_{j-1}^{3} g_{ij} (g_{iq})_{\xi^i} + \sum_{k=1}^{3} \phi_k g_{kq} - \sum_{i-1}^{3} \sum_{j-1}^{3} g_{ij} \left( \frac{(g_{ij})_{\xi^k} - (g_{jq})_{\xi^i}}{2} \right) = 0 \]

\[ q = 1, 2, 3 \]

\[ g_{ij} = r_i \cdot r_j = \| r_i \| \cdot \| r_j \| \cdot \cos \Theta \]
GENIE++ Characteristics
GENIE++

Geometry Mode

• Sculptured Curves

Computational Mode

• One Block at a Time
• With One Extra Block in On-Line Memory
INITIALIZATION OPTIONS
1 TOGGLE REAL TIME PLOTTING
2 TOGGLE PROMPTING
3 GIVE TITLE TO GRID
4 CHANGE CURRENT GRID BLOCK SIZE
5 CHANGE MAXIMUM GRID SIZES
6 CHANGE CURRENT BLOCK NUMBER
7 CHANGE MAXIMUM NUMBER OF BLOCKS
8 TOGGLE GRID GENERATION MODE
9 INITIALIZE DATABASE
10 INITIALIZE ZONAL INFORMATION
11 VIEW NON-BLOCK GRID
12 VIEW ONE BLOCK
13 VIEW ALL BLOCKS
14 EXIT INITIALIZATION
15 QUIT GRID GENERATION

INPUT OPTION NUMBER
Conte++ Main Menu

This chapter is about using GENE++ to generate game descriptions of the fictitious country. All instructions, commands, and information included in this chapter are not used GENB electronics. For more information on GENB, refer to the MANUAL for creating using the author.

Grid Generation Activity Options

Choosing options in the initial Grid Generation Activity menu results in the following Activity options:

Activity Option Menu
1. Choose Random Selection
2. Select First Selection
3. Volume Selection
4. Pick Random Selection
5. Get Metabolites
6. Bladder Geometry
7. Initial Bladder
8. Aorta Selection
9. Pulmonary Arteries
10. Chart/Overview
11. Utility
BOUNDARY SEGMENT DEFINED BY

1. A CURVE PROJECTED ONTO A PARALLEL PLANE
2. OTHER CURVE PROJECTION OPTIONS
3. A STRAIGHT LINE
4. A 3D BEZIER / HERMITE CUBIC CURVE
5. SCULPTURED CURVE DEFINITION
6. CURVE MANIPULATION OPTIONS

INPUT OPTION NUMBER
Generic Launch Vehicle (41x41x5, 61x91x5) M = 0.3.
GENIE++

- Semi-Interactive - Simple Minded
- Portable, Modular
- Journal File Execution Control
- Batch-Interactive Execution
- CadType Geometry Construction
- SOA Grid Generation Algorithms
- Quality Control & Extensive Error Checking
- Online Graphical Visualization of Overall Process
- User Friendly & Researcher Friendly
- SGI, X-Window, PC Versions
- bsoni@erc.msstate.edu
Surface and Volume grid generation in parametric form

by

Tzuyi Yu, Bharat K. Soni
NSF/ERC For Computational Field Simulation
Mississippi State University, MS 39762

Ted Benjamin, Robert Williams
Marshall Space Flight Center, AL 35812

ABSTRACT

The algorithms for surface modeling and volume grid generation using parametric NURBS geometric representation are presented. The enhanced re-parameterization algorithm which can yield a desired physical distribution on the curve, surface and volume is also presented. This approach bridges the gap between CAD surface / volume definition and surface / volume grid generation.

INTRODUCTION

Surface grid generation is the most labor intensive part of the overall complex three dimensional grid generation process. Also, a significant amount of effort is required in changing the resolutions (grid sizes) and / or the distribution of the grid while maintaining geometry fidelity. In the last few years, various researchers have concentrated on utilizing the Computer Aided Geometry Design (CAGD) techniques to expedite the overall surface generation process. In this presentation, a parametric formula which has been used in CAD system is extended with re-parameterization approach to numerical grid generation for modeling the surface as well as the volume grid.

There are many parametric approaches for representing sculptured geometry, such as rational or non-rational Bezier, cubic splines, rational or non-rational B-spines, ... etc. Among these representation, the Non-Uniform Rational B-Splines (NURBS) has been widely accepted among these researchers. NURBS has been widely utilized to represent and design geometry in the CAD/CAM and the graphics community due to its powerful features, such as the local control property, variation diminishing, convex hull and affine invariance [Ref 1,2]. Also the geometry tool kits, such as curve/surface interpolation, data reduction, degree elevation, knot insertion and splitting, are well-developed [Ref 1,2,]. These properties have made NURBS representation very popular in recent developments in CAD/CAM. However, the distribution requirements in CFD application are much complicated than those in CAD system. Hence, the NURBS must be cooperated with re-parameterization algorithm so that it can be more useful in grid generation. Computational examples associated with practical configurations are shown in Figure 1 and 2. The re-parameterization approach described in many research [Ref 2,3] is implemented by iteration process, which needs a lot of computation time. The more efficient and robust approach presented here needs only one interpolation process.

The development of the software based on NURBS representation package: CAGI (Computer Aided Grid Interface) was initiated by authors under the sponsorship of NASA Marshall Space Flight Center. The purpose of this presentation is to present the progress realized in enhancing the NURBS based curve / surface grid generation techniques into a 3D volume grid generation technique. To this end, various options for generating 3D volume geometry–grid are discussed. A reparameterization scheme has been developed to achieve desired distribution in physical space. Computational examples for modeling practical configurations have been exercised using the volume options and the reparameterization scheme.
Figure 1: Propulsional example.
3D NURBS control patches model the missile (with fins) geometry.

Figure 2:
3D NURBS control patches model the single rotation propfan.

References
Title: Surface and Volume grid generation in parametric form

Authors: Tzu–Yi Yu, Bharat K. Soni
Mississippi State University /Engineering Research Center
2 Research Blvd., Starkville, MS 39759, USA

Ted Benjamin, Robert Williams
NASA/Marshall Space Flight Center

Sponsor: NASA/Marshall Space Flight Center
Geometry generated in parametric space:

**Advantages:**

1> Natural for grid generation algorithm.
2> Easy manipulation. Designer's Dream.
3> Less storage & standard data structure.
4> Way of future.

**Disadvantage:**

0> Distribution differs in parametric space and physical space.
NURBS: Non–Uniform Rational BSpline is selected.

1> Stable and robust geometric properties.
   - Degree raising, splitting/join algorithm, convex hull, shape preserving, local support, ... etc.

2> IGES (Initial Graphics Exchange Specification), NASA IGES, NINO.

3> Way of Future:
   - Solid Geometry Modeling.

STEP.

CAGl: A NURBS database package.
NURBS control volume for a nozzle with square–ellipse–circle sections
(O Type Control Volume)
NURBS control volume for a nozzle with square-elliptic-circular sections

( H Type Control Volume )
Original Geometry (from CAD/CAM System)
Original NURBS control net from CAD/CAM

NURBS surfaces are defined with $C^{-1}$

<table>
<thead>
<tr>
<th>NURBS orders</th>
<th>Size of NURBS control net</th>
</tr>
</thead>
<tbody>
<tr>
<td>9 by 12</td>
<td>17 X 47</td>
</tr>
<tr>
<td>6 by 6</td>
<td>17 X 65</td>
</tr>
<tr>
<td>6 by 6</td>
<td>17 X 65</td>
</tr>
</tbody>
</table>
Manipulating CAD geometry and generating the volume grid

(O Type Grid)
Manipulating CAD geometry and generating the volume grid
(O Type Grid)
Manipulating CAD geometry and generating the volume grid
Converge/Diverge nozzle with circular-rectangular section
Converge/Diverge nozzle with circular-rectangular section

Mach Number

0.205  2.457  4.709
On Going Activities:

- Surface/Volume grid characteristic improvement.  
  (Distribution control, Orthogonality, smoothness ... )
- General algorithm allowing trim surface for structured grid.
- Enhance geometric generation/manipulation functions.
- Reparametration algorithm for unstructured grid.
- Unstructured/Hybrid grid generation.
Overview of CFD Analyses Supporting the Reusable Solid Rocket Motor (RSRM) Program at MSFC

E. Stewart, P. McConnaughey, J. Lin, E. Reske, and D. Doran, NASA/MSFC, R. H. Whitesides, ERC, Inc., Huntsville, AL, and Y.-S. Chen ESI, Huntsville, AL

During the past year, various CFD analyses were performed at MSFC to support the RSRM program. The successful completion of these analyses was realized through the cooperation of ESI, ERC, and The Computational Fluid Dynamics Branch (ED32) at MSFC and involved application of the CFD codes FDNS and CELMINT. The topics addressed by the analyses were: 1. the design and prediction of slag accumulation within the five inch test motor, 2. prediction of slag pool behavior and its response to lateral accelerations, 3. the clogging of potential insulation debonds within the nozzle by slag accumulation, 4. the behavior of jets within small voids inside nozzle joint gaps, 5. the effect of increased inhibitor stiffness on motor acoustics, and 6. the effect of a nozzle defect on particle impingement enhanced erosion. Topics 1, 2, and 5 will be discussed in some detail by other speakers at the conference and are only mentioned here for the sake of completeness. Thus, the emphasis of this presentation will be to further discuss the work involved in topics 3, 4, and 6.

Eric Stewart
RSRM CFD Analyses at MSFC

Overview of CFD Analyses Supporting the Reusable Solid Rocket Motor (RSRM) Program at MSFC

Presented at 13th Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology, MSFC, AL, April 27-30, 1995

E. Stewart, P. McConnaughey, E. Reske, J. Lin, and D. Doran, NASA/MSFC, R. H. Whitesides, ERC, Inc., Huntsville, AL, and Y.-S. Chen ESI, Huntsville, AL
Overview

- Overview of RSRM CFD analyses at MSFC
- Insulation Debond Analysis
- Potential RTV Flaw Analysis
- Nose Inlet Assembly Wetline Investigation
- Future efforts
Overview of RSRM CFD Analyses at MSFC

- Slag ($\text{Al}_2\text{O}_3$) behavior and accumulation
  - 5 inch spin motor design and analysis
  - Accumulation within RSRM at 67 seconds
  - Response of slag pool to lateral accelerations using VOF methodology

- Increase in Nitrile Butadiene Rubber (NBR) stiffness
  - Aerodynamic torque on nozzle
  - Potential effect on internal acoustics/pressure oscillations
    - Change in inhibitor deflections
    - Vortex shedding by the inhibitors

- Code validation for the 8-percent ASRM cold flow model

- Insulation debond analysis
- Potential RTV flaw analysis
- Nose inlet assembly wetline Investigation
Insulation Debond Analysis

Issues  • Prediction of flow/clogging through potential insulation defect flow paths during motor operation

Approach  • Use two-phase flow and condensation models to predict propensity for pore clogging during motor operation
            - prescribed thermal boundary conditions

Results  • Small (0.01") pores probably clog quickly (.05 sec) under severe thermal gradients
          • Lower probability of clogging during start pressurization transient

Impact  • Joint gap clogging prediction methodology is available to support potential anomalies
• Results (Problem 1, debond vent to ambient)

<table>
<thead>
<tr>
<th>gap width</th>
<th>time to clog</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.010&quot;</td>
<td>0.05 sec.</td>
</tr>
<tr>
<td>0.005&quot;</td>
<td>0.02 sec.</td>
</tr>
<tr>
<td>0.002&quot;</td>
<td>0.006 sec.</td>
</tr>
</tbody>
</table>

• Results (Problem 2, start-up transient through debond)

- clogging of debond predicted in 0.61 sec. after initiation of particle flow
- lower mass flow rate (4X less) due to cavity fill results in fewer particles to condense on pore wall
Joint Gap Clogging Prediction
Potential RTV Flaw Analysis

Issues
- Hot gas jet impingement environments on O-rings within RSRM nozzle due to potential RTV flaws
- Predict jet spreading within irregularly shaped cavities

Approach
- Predict 3-D Jet Spreading for potential joint gap/cavity flows for input into thermal models

Results
- Hot gas jet spreading within joint cavities is smaller than that used in previous non-CFD analyses

Impact
- Jet spread width used in thermal models should be 0.7" (rather than 1.25"")
- Jet spreading predictions are available to support potential anomalies
RSRM CFD Analyses at MSFC

Geometry of Internal Nozzle, Joint #4

Computational Geometry
**Analysis matrix and results**

<table>
<thead>
<tr>
<th>case</th>
<th>flowrate (lbm/s)</th>
<th>domain width (in)</th>
<th>corners</th>
<th>inlet width (in)</th>
<th>jet spread width (in)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0001</td>
<td>8.0</td>
<td>rounded</td>
<td>0.1</td>
<td>1.5</td>
</tr>
<tr>
<td>2</td>
<td>0.0002</td>
<td>8.0</td>
<td>rounded</td>
<td>0.1</td>
<td>1.1</td>
</tr>
<tr>
<td>3</td>
<td>0.0005</td>
<td>8.0</td>
<td>rounded</td>
<td>0.1</td>
<td>0.6</td>
</tr>
<tr>
<td>4</td>
<td>0.001</td>
<td>8.0</td>
<td>rounded</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>5</td>
<td>0.00155</td>
<td>8.0</td>
<td>rounded</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>6</td>
<td>0.00155</td>
<td>16.0</td>
<td>rounded</td>
<td>0.1</td>
<td>0.5</td>
</tr>
<tr>
<td>7</td>
<td>0.00155</td>
<td>8.0</td>
<td>square</td>
<td>0.1</td>
<td>0.75</td>
</tr>
<tr>
<td>8</td>
<td>0.00155</td>
<td>8.0</td>
<td>rounded</td>
<td>0.2</td>
<td>0.75</td>
</tr>
</tbody>
</table>
Velocity Magnitudes

Average Velocity Magnitudes

Center Line Velocity Profile Through Corner A
Nose Inlet Assembly Wetline Investigation

**Issues**
- Enhancement of nozzle erosion due to presence of defect
- Effect of defect on slag particle impingement

**Approach**
- Assume wedge shaped nozzle defect
- Use two-phase flow results to assess flow environment near defect
- Use current data/experience base to assess potential flow deviations

**Results**
- Size of defect relative to local boundary layer is not sufficient to significantly alter flow external to boundary layer
- Main source of particle impingement is external to boundary layer
- Erosion enhancement due to particle impingement is not significantly altered by presence of defect

**Impact**
- Recommend nozzle in question for flight
Geometry of Nozzle Nose Region
RSRM CFD Analyses at MSFC

RSRM 67 Sec. Burn Time, 150 Micron Particle Trajectories

ED32 Results

ERC Results
Future Effort

• Continue code validation

• Continue to enhance modeling capabilities
  - two-phase flow
  - combustion
  - turbulence
  - slag accumulation
  - unsteady flow

• Improve readiness to address potential anomalies
  - Perform similar analyses at additional burn times
RSRM Chamber Pressure Oscillations:
Transit Time Models and Unsteady CFD

Tom Nesman and Eric Stewart
Fluid Dynamics Division
National Aeronautics and Space Administration, MSFC
Marshall Space Flight Center, AL 35812

Abstract

Space Shuttle solid rocket motor (SRM) low frequency internal pressure oscillations have been observed since early testing. The same type of oscillations are also present in the redesigned solid rocket motor (RSRM). The oscillations, which occur during RSRM burn, are predominantly at the first three motor cavity longitudinal acoustic mode frequencies. Broadband flow and combustion noise provide the energy to excite these modes at low levels throughout motor burn, however, at certain times during burn the fluctuating pressure amplitude increases significantly. The increased fluctuations at these times suggests an additional excitation mechanism.

The RSRM has inhibitors on the propellant forward facing surface of each motor segment. The inhibitors are in a slot at the segment field joints to prevent burning at that surface. The aft facing segment surface at a field joint slot burns and forms a cavity of time varying size. Initially the inhibitor is recessed in the field joint cavity. As propellant burns away the inhibitor begins to protrude into the bore flow. Two mechanisms (transit time models) that are considered potential pressure oscillation excitations are cavity edge-tones, and inhibitor hole-tones. Estimates of frequency variation with time of longitudinal acoustic modes, cavity edge-tones, and hole-tones compare favorably with frequencies measured during motor hot firing. It is believed that the highest oscillation amplitudes occur when vortex shedding frequencies coincide with motor longitudinal acoustic modes.

A time accurate CFD analysis was made to replicate the observations from motor firings and to observe the transit time mechanisms in detail. FDNS is the flow solver used to detail the time varying aspects of the flow. The fluid is approximated as a single-phase ideal gas. The CFD model was an axisymmetric representation of the RSRM at 80 seconds into burn. Deformation of the inhibitors by the internal flow was determined through an iterative structural and CFD analysis. The analysis domain ended just upstream of the nozzle throat. This is an acoustic boundary condition that caused the motor to behave as an closed-open organ pipe. This differs from the RSRM which behaves like a closed-closed organ pipe.

The unsteady CFD solution shows RSRM chamber pressure oscillations predominantly at the longitudinal acoustic mode frequencies of a closed-open organ pipe. Vortex shedding in the joint cavities and at the inhibitors contribute disturbances to the flow at the second longitudinal acoustic mode frequency. Further studies are planned using an analysis domain that extends downstream of the nozzle throat.
RSRM - Chamber Pressure Oscillations: Transit Time Models and Unsteady CFD

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology

Tom Nesman and Eric Stewart
Fluid Dynamics Division - NASA - MSFC
Introduction

- Space Shuttle SRM Pc oscillations issues have surfaced at various times in past
  - Pre- STS-1 loads analysis
  - Post STS-1 loads evaluation
  - STD to HPM change
  - FWC testing (ASRM)
  - HPM to RSRM change
  - Inhibitor stiffening evaluation (present study)
- SRM Pc oscillation evaluation based primarily on test and flight data
- Mechanisms evaluated empirically
- Unsteady CFD activities initiated in early 1990’s (funded thru 1993)
- Unsteady RSRM CFD activities revived for inhibitor stiffening evaluation

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Background

- Space Shuttle solid rocket motor low frequency internal pressure oscillations observed since early testing
- Same type oscillations present in redesigned solid rocket motor (RSRM)
- Predominantly at first three motor internal longitudinal acoustic mode frequencies
- Broadband flow and combustion noise provide energy to excite these modes at low levels throughout motor burn
- At certain times during burn fluctuating pressure amplitude increases significantly
- Increased fluctuations at these times suggests an additional excitation mechanism
Head-end Chamber Pressure (Pc) Measurement

Typical RSRM Pc Isoplot

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Typical RSRM Pc Timehistory
(Bandpass Filtered Data)

Head-end Chamber Pressure (Pc) Measurement

This is actually an HPM

Filter settings

Longitudinal Acoustic Mode 1

Longitudinal Acoustic Mode 2

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
General Characteristics

- RSRM inhibitors on propellant forward facing surface of each motor segment
- Inhibitors in slot at segment field joints to prevent burning at forward facing surface
- Aft facing segment surface at field joint slot burns to form cavity of time varying size
- Initially inhibitor recessed in field joint cavity
- As propellant burns away inhibitor protrudes into bore flow
- Two mechanisms considered potential pressure oscillation exciters
  - Feedback transit-time models
  - 1. Cavity edge-tones
  - 2. Inhibitor hole-tones
- Estimates of frequency variation with time of longitudinal acoustic modes, cavity edge-tones, and hole-tones compare favorably with frequencies measured during motor hot firing
- Highest oscillation amplitudes occur when vortex shedding frequencies coincide with motor longitudinal acoustic modes

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Excitation Mechanisms

Cavity Edge-Tone Mechanism

Feedback Transit Time Models

Empirical estimates - time for disturbance to travel downstream plus time for feedback to sensitive region
Analytical estimates - sum of phase of downstream traveling wave and phase of upstream traveling acoustic wave
RSRM Static Firing Head End Chamber Pressure

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Predicted Hole Tone Frequencies

Hole Tones:
- \( St = f(L, D, Re) \)
- \( St = .8, 1.6, 2.4, ... \)
- Ref. Chanaud, Powell, Flatau

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Predicted Cavity Edge Tone Frequencies

Cavity Edge Tone:
- St = f(L, D, M)
- Ref. Rossiter, East, Block, Plumlee

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Background

- Numerical simulation of "edge-tone" phenomenon (NASA CR 4581)
  - Performed by Rockwell-Huntsville in 1992 using USA flow solver
- Dipole nature of edge-tone
  - Performed by Rockwell-Huntsville in 1993 using USA flow solver
- Numerical simulation of RSRM (NAS8-38550)
  - S+80 sec and S+105 sec burn time geometry and flow
  - Solved Navier-Stokes equation for low speed flows
  - Objective: evaluate the effect of inhibitors on Pc oscillations
- Head-end p' dominated by 1L, 2L, and 3L organ pipe modes
- Inhibitors generate oscillations, however, head-end p' lower with inhibitor than without inhibitor (not tuned?)
Present Study

- Time accurate CFD analysis made to replicate observations from motor firings and observe transit time mechanism details
- CFD model is axisymmetric representation of RSRM at 80 seconds into burn
- Objective: determine effect of stiffer inhibitors on Pc oscillations
- Deformation of inhibitors by internal flow determined through iterative structural and CFD analysis
- FDNS is flow solver used to detail time varying aspects of flow
- Fluid approximated as single-phase ideal gas
- Analysis domain ends upstream of nozzle throat

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Aft segment pressure contours

Contour Levels:
- 4.24000
- 4.24250
- 4.24500
- 4.24750
- 4.25000
- 4.25250
- 4.25500
- 4.25750
- 4.26000
- 4.26250
- 4.26500
- 4.26750
- 4.27000
- 4.27250
- 4.27500
- 4.27750
- 4.28000
- 4.28250
- 4.28500
- 4.28750
- 4.29000
- 4.29250
- 4.29500
- 4.29750
- 4.30000

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Aft segment pressure contour sequence

\begin{align*}
  t &= 0 \text{ sec} \\
  t &= 0.01 \text{ sec} \\
  t &= 0.02 \text{ sec} \\
  t &= 0.03 \text{ sec} \\
  t &= 0.04 \text{ sec} \\
  t &= 0.05 \text{ sec} \\
  t &= 0.06 \text{ sec} \\
  t &= 0.07 \text{ sec}
\end{align*}
RSRM Unsteady CFD Timeplot

Fluctuating Pressure Downstream of Inhibitor Tip

Inhibitor

- FWD nominal
- MID nominal
- AFT nominal

Workshop for CFD Applications in Rocket Propulsion and Launch Vehicle Technology
Summary of Present Unsteady CFD Results

- RSRM Pc oscillations dominated by organ pipe modes
- These acoustic modes excite or are excited by shedding of vortices within motor
- Vortex shedding in joint cavities and at inhibitors
- Vortex shedding at second open-closed organ pipe frequency
- Essential questions remain unanswered
  - Do vortices gain energy from feedback mechanism?
  - What is the feedback mechanism?
- Further studies planned with this model
  - Extend analysis domain downstream of nozzle throat
  - Evaluate using pressure gradient magnitude
A Coupled CFD/FEM Structural Analysis to Determine Deformed Shapes of the RSRM Inhibitors

Richard A. Dill and R. Harold Whitesides  
ERC, Incorporated, Hunstville, AL 35816

Abstract

Recent trends towards an increase in the stiffness of the NBR insulation material used in the construction of RSRM propellant inhibitors prompted questions about possible effects on RSRM performance. The specific objectives of the CFD task included: 1) the definition of pressure loads to calculate the deformed shape of stiffer inhibitors, 2) the calculation of higher port velocities over the inhibitors to determine shifts in the vortex shedding or edge tone frequencies and 3) the quantification of higher slag impingement and collection rates on the inhibitors and in the submerged nose nozzle cavity.

A coupled CFD/Finite element structural analysis was required to calculate the deformed inhibitor geometry. Since the NBR inhibitor material erodes at a different rate than the motor propellant burns, an inhibitor stub which protrudes above the propellant into the port cavity is created during motor operation. The impinging port flow causes the inhibitor stub to bend in the downstream flow direction. Since a stiffer NBR inhibitor material would cause the inhibitor to bend less, it was necessary to know the difference in the bending of the original NBR material compared to the stiffer NBR material. The CELMINT CFD computer code was used to perform the fluid dynamic calculations of the motor flow field. The structural bending effect of the pressure loads from the CFD code was analyzed by ED28. Initially, the CELMINT code was used to determine the flow field and inhibitor pressure loads for unbent motor inhibitors. This pressure loading on the inhibitors was used by ED28 to generate the bending which would occur in the inhibitor. The computed bent inhibitor geometry was then used again by the CFD code to compute a new pressure loading on the inhibitors. This iterative computation between the CFD code and the structural analysis code was continued until convergence in the inhibitor bent geometry was achieved.

The CFD solution was then used to assess the effect of higher flow velocities and edge tone frequencies from the reduced inhibitor bending on the maximum oscillating pressure amplitudes that occur during resonance between the edge tones and the motor longitudinal modes. Also, a comparison of the difference in slag accumulation between the two NBR materials was also made to determine if the stiffer material increases slag collection in the field joints and the submerged nozzle cavity.

The coupled CFD/FEM structural analysis was successful in defining the effect of inhibitor stiffness on inhibitor geometry and the shift in edge tone frequencies. Also, the two-phase CFD analysis showed that there was a small increase in the rate of slag accumulation at the aft inhibitor; however, motor trajectory analyses of slag debris shed from the inhibitors showed that the debris would pass out the motor nozzle and therefore create no additional slag accumulation in the slag pool around the nozzle.

1189
A COUPLED CFD/FEM STRUCTURAL ANALYSIS TO
DETERMINE DEFORMED SHAPES OF THE RSRM
INHIBITORS

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

Thirteenth Workshop for CFD Applications in Rocket Propulsion
NASA Marshall Space Flight Center
Huntsville, Alabama
April 25-27, 1995
Background

- In October, 1994, Thiokol reported the use of NBR material in RSRM’s with properties significantly different from the historical database.

- A 30% to 40% increase in modulus was reported.

- This increased stiffness had the potential to affect the amplitude of chamber pressure oscillations in the SRM:
  --By changing the inhibitor structural response
  --By indirectly changing the flow/acoustic interaction

- The slag accumulation in the field joints and submerged nozzle region might also be increased thereby increasing the potential for pressure and thrust perturbations.
Objectives of Coupled CFD/FEM and Two-Phase CFD Analyses

- Determine deformed geometry of NBR inhibitors at the forward, center and aft joints for both nominal and stiff NBR materials using a coupled CFD/FEM analysis.

- Determine effect of inhibitor properties/geometry on inhibitor hole velocities to evaluate effect on hole edge tone (vortex shedding) frequencies.

- Determine effect of inhibitor properties/geometry on slag accumulation on both the inhibitor surfaces and underneath the nozzle nose.
Coupled CFD/FEM Analysis Approach

1) Perform single-phase gas CFD analysis of entire RSRM port at 80 second burn time using straight inhibitor lengths from erosion analysis.

2) Perform FEM structural analysis on inhibitors to determine deformations using surface pressure distributions from CFD analysis.

3) Perform CFD analysis using deformed inhibitor geometries from step 2).

4) Repeat steps 2) and 3) until convergence of inhibitor geometry is achieved.

5) Provide velocity profile at each inhibitor location for both nominal and stiff inhibitors as input to flow/acoustic interaction analysis.
Two-Phase Flow CFD Methodology

CELMINT Code
(Combined Eulerian Lagrangian Multi-Dimensional Implicit Nonlinear Time-Dependent)

- **Navier-Stokes Solution**
  - Fully implicit, density-based, conservative, ensemble-averaged Navier-Stokes code
  - Low and high Reynolds number and wall injection $\kappa-\varepsilon$ models
  - Equilibrium and finite-rate chemistry for multi-species flows

- **Two-phase Flow Models**
  - Coupled Eulerian-Lagrangian for solid and liquid phases
  - Hermsen aluminum burn rate model for particle combustion
  - Specification of particle properties (density, size distribution)
  - Particle break-up based on Weber number
  - Agglomeration based on collisions between discrete phase particles and continuous phase smoke particles
  - Programmable for various particle capture criteria
Propellant Thermochemical Properties and Motor Operating Conditions
RSRM 80 Second Burn Time

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propellant</td>
<td>TP-H1148</td>
</tr>
<tr>
<td>Pressure</td>
<td>625 psia</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>6093° R</td>
</tr>
<tr>
<td>Molecular Weight</td>
<td>28.04</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>6.189x10^{-5} bm/ft-sec</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.138</td>
</tr>
<tr>
<td>Flow Rate, Forward Segment</td>
<td>1555.9 lbm/sec</td>
</tr>
<tr>
<td>Flow Rate, Center Segment 1</td>
<td>2587.5 lbm/sec</td>
</tr>
<tr>
<td>Flow Rate, Center Segment 2</td>
<td>2578.6 lbm/sec</td>
</tr>
<tr>
<td>Flow Rate, Aft Segment</td>
<td>2849.0 lbm/sec</td>
</tr>
<tr>
<td>Flow Rate, Total</td>
<td>9571.0 lbm/sec</td>
</tr>
<tr>
<td>Throat Diameter</td>
<td>55.42 inches</td>
</tr>
</tbody>
</table>
### Computational Grid Resolution

<table>
<thead>
<tr>
<th>Description</th>
<th>Resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Port</td>
<td>400X50</td>
</tr>
<tr>
<td>Field Joints</td>
<td>30X20</td>
</tr>
<tr>
<td>Inhibitor Stub</td>
<td>4X20</td>
</tr>
<tr>
<td>Submerged Region</td>
<td>70X20</td>
</tr>
<tr>
<td>Overall Grid</td>
<td>488X70</td>
</tr>
</tbody>
</table>
Computational Grid, Full Motor
RSRM 80 Second Stiff NBR Inhibitor
Computational Grid, Forward Slot

RSRM 80 Second Stiff NBR Inhibitor
Computational Grid, Center Slot
RSRM 80 Second Stiff NBR Inhibitor
Computational Grid, Aft Slot

RSRM 80 Second Stiff NBR Inhibitor
Computational Grid, Submerged Nozzle

RSRM 80 Second Stiff NBR Inhibitor
Flowfield Velocity Magnitude
RSRM 80 Second Stiff NBR Inhibitor
Flowfield Static Pressure

RSRM 80 Second Stiff NBR Inhibitor
Velocity Vectors, Forward Slot
RSRM 80 Second Stiff NBR Inhibitor
Velocity Vectors, Aft Slot
RSRM 80 Second Stiff NBR Inhibitor
Velocity Vectors, Submerged Nozzle
RSRM 80 Second Stiff NBR Inhibitor
Forward Inhibitor Radial Pressure Distribution
RSRM 80 Seconds Burn Time
Stiff NBR

Pressure (psia)

Motor Radius (inches)
Center Inhibitor Radial Pressure Distribution
RSRM 80 Seconds Burn Time
Stiff NBR

Pressure (psia)

Motor Radius (inches)
Center Inhibitor Deformations
Nominal and Stiff NBR
RSRM 80 Seconds Burn Time

Motor Radius (inches)

Axial Motor Station (inches)

- Propellant
- Undeformed Inhibitor
- Nominal NBR
- Stiff NBR
Aft Inhibitor Deformations
Nominal and Stiff NBR
RSRM 80 Seconds Burn Time
Port Velocity Profile at Forward Inhibitor
Nominal and Stiff NBR
RSRM 80 Seconds Burn Time

Motor Radius (Inches)

Axial Velocity (ft/sec)

Pressure: 626 psia
Mass Flow Rate: 1503 lbm/sec
Average Velocity, Stiff: 133 ft/sec
Average Velocity, Nominal: 131 ft/sec
Port Velocity Profile at Center Inhibitor
Nominal and Stiff NBR
RSRM 80 Seconds Burn Time

Pressure: 623 psia
Mass Flow Rate: 4158 lbm/sec
Average Velocity, Stiff: 273 ft/sec
Average Velocity, Nominal: 263 ft/sec

ERC, Inc.
Port Velocity Profile at Aft Inhibitor
Nominal and Stiff NBR
RSRM 80 Seconds Burn Time

Motor Radius (Inches)

Pressure: 619 psia
Mass Flow Rate: 6784 lbm/sec
Average Velocity, Stiff: 379 ft/sec
Average Velocity, Nominal: 378 ft/sec

Axial Velocity (ft/sec)
Comparison of the Motor Port Velocity Profiles Immediately Upstream of Nozzle Nose
RSRM 80 Seconds Burn Time

Motor Radius (Inches) vs. Axial Velocity (ft/sec)
Coupled CFD/FEM Analysis Conclusions

- The coupled CFD/FEM inhibitor structural analysis was successfully iterated to convergence to determined the deformed geometry of inhibitors at the forward, center and aft joints.

- The velocity through the inhibitor hole for the stiff inhibitors is somewhat higher which would increase the hole true frequency and delay tuning with the acoustic mode until a later burn time.

- The velocity profile at the nozzle entrance just upstream of the nose is not affected by the inhibitor stiffness/geometry and thus nozzle internal aerotorque would not be impacted.
Two-Phase CFD Analysis Approach

- Perform two-phase CFD analysis of RSRM port at 80 second burn time using final deformed inhibitor geometries for both nominal and stiff inhibitors.

- Calculate slag captured on both nominal and stiff inhibitors at all three field joints.

- Perform trajectory analysis for slag debris shed from inhibitor tips for all above cases to determine whether it passes through nozzle or accumulates underneath nozzle nose.
Slag Debris Trajectories

RSRM 80 Second Stiff NBR Inhibitor

Debris Diameter: 0.2 Inches
Slag Debris Trajectories

RSRM 80 Second Stiff NBR Inhibitor

Debris Diameter: 0.2 Inches
<table>
<thead>
<tr>
<th>Release Location</th>
<th>Debris Diameter</th>
<th>Nominal NBR</th>
<th>Stiff NBR</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>0.2 inches</td>
<td>Exits Nozzle</td>
<td>Exits Nozzle</td>
</tr>
<tr>
<td></td>
<td>0.4 inches</td>
<td>Exits Nozzle</td>
<td>Exits Nozzle</td>
</tr>
<tr>
<td></td>
<td>0.8 inches</td>
<td>Exits Nozzle</td>
<td>Exits Nozzle</td>
</tr>
<tr>
<td></td>
<td>1.6 inches</td>
<td>Exits Nozzle</td>
<td>Exits Nozzle</td>
</tr>
</tbody>
</table>
Two-Phase CFD Analysis Conclusions

- The rate of slag accumulation for both the nominal and stiff inhibitors at all joints is a very small percentage of the total motor slag accumulation rate.

- The rate of slag accumulation on the center inhibitor is approximately four times greater for the stiff NBR compared to the nominal NBR.

- Slag debris shed from the nominal inhibitors at all three joints exits the nozzle throat plane.

- Slag debris shed from the stiff inhibitors at the forward and center joints exits the nozzle throat plane. Slag from the aft joint stiff inhibitor impacts the nozzle entrance ramp.

- No excess slag collected on the stiff inhibitors is transported underneath the nozzle nose to add to the normal slag pool.
CFD Flow Analysis and Code Validation for the MSFC Eight-Percent ASRM Cold Flow Model
Part II

Thirteenth Workshop for CFD Applications in Rocket Propulsion
April 25-27, 1995

Jeff Lin
Ed Reske

NASA, Marshall Space Flight Center
Computational Fluid Dynamics Branch
Overview

- Introduction
  - purpose for code benchmarks
  - experimental results from the MSFC 8% ASRM Cold Flow Model

- Codes under Consideration
  - CELMINT
  - GASP
  - FDNS

- Physical and Geometrical Parameters

- Grid Issues

- Results
  - comparison between CFD and experiment
  - flow visualization

- Summary and Future Plans
Introduction

- Why a Code Benchmark is needed
  - To assure validity of CFD models used to predict
    -- motor performance
    -- internal nozzle aerodynamics (loads and hinge moments)
    -- heat transfer to the nozzle, insulation, and joints
    -- slag accumulation (and potential pressure spikes)
    -- chamber pressure oscillations

- Potential problem areas
  -- flow field anomalies attributed to grid irregularities
    --- complex mappings
    --- skewness
    --- kinks
    --- zonal interfaces

  -- appropriate methodology?
Experimental Results from the MSFC 8% ASRM Cold Flow Model

- static wall pressures
- velocity rakes
- gimbal angles of 0, 4, and 8 degrees

Codes Under Consideration

- CELMINT (SRA)
  - benchmark complete for 0 and 8 degrees

- GASP (Aerosoft)
  - benchmark complete for 0 degrees
  - benchmark in progress for 4 and 8 degrees

- FDNS (MSFC and ESI)
  - benchmark complete for 0 degrees
  - benchmark in progress for 4 and 8 degrees
HEAT TRANSFER GAUGES
4 AXIAL STATIONS
AS SHOWN

Note: Station 31 is the Gimbal Ring Vent Hole
ASRM Aft Section/Nozzle Model

Boundary Conditions at the Probe Tip Axial Location

5-13-93

1-D Conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Throat Diameter</td>
<td>4.36 inches</td>
</tr>
<tr>
<td>T₀</td>
<td>530° R</td>
</tr>
<tr>
<td>P₀</td>
<td>572 psia</td>
</tr>
<tr>
<td>Mach Number</td>
<td>28.97</td>
</tr>
<tr>
<td>Static Pressure</td>
<td>954.28 psia</td>
</tr>
<tr>
<td>Static Temperature</td>
<td>826.25° R</td>
</tr>
<tr>
<td>Mass Flow Rate</td>
<td>197.285 lbm/sec</td>
</tr>
<tr>
<td>Local Velocity</td>
<td>236.777 lbs/s</td>
</tr>
<tr>
<td>Local Sonic Velocity</td>
<td>1123.4 lbs/s</td>
</tr>
<tr>
<td>Local Density</td>
<td>2.8487 lbm/lb^3</td>
</tr>
</tbody>
</table>

Reynolds Number based on above conditions and viscosity equal to 1.9458 x 10⁻⁵ ps–s

R = 31.8 x 10⁶
GEOMETRY

ARAN Cold Flow Model: 8/19/93
8% ASRM COLD FLOW MODEL, AXISYMMETRIC CASE

Pressure vs. Arc Length for Points on Nozzle Nose

---

[Graph showing pressure vs. arc length for points on nozzle nose, with experimental data and FDNS (Preliminary) curves marked.]
8% ASRM COLD FLOW MODEL, AXISYMMETRIC CASE

Pressure vs. Arc Length in Exit Cone from Throat to Exit Plane

- Experimental Data
- FDNS (Preliminary)

Pressure in PSIA

PS-11
PS-12
PS-13
PS-14
PS-15
PS-16
PS-17
PS-18
PS-19
PS-20
PS-21
Summary and Future Plans

- Good agreement is attained for GASP at 0 degrees.
  - The 4- and 8-degree cases are in progress.

- The case for FDNS at 0 degrees will be readdressed.
  - The 4- and 8-degree cases will be attempted once the discrepancy is resolved.

- The final results from these benchmarks will be submitted for future publication.
Application of Two-Phase CFD to the Design and Analysis of a Subscale Motor Experiment to Evaluate Propellant Slag Production

R. Harold Whitesides and Richard A. Dill
ERC, Incorporated, Huntsville, AL 35816

Abstract

The RSRM Pressure Perturbation Investigation Team concluded that the cause of recent pressure spikes during both static and flight motor burns was the expulsion of molten aluminum oxide slag from a pool which collects in the aft end of the motor around the submerged nozzle nose during the last half of motor operation. It is suspected that some motors produce more slag than others due to differences in aluminum oxide agglomerate particle sizes which may relate to subtle differences in propellant ingredient characteristics such as particle size distributions, contaminants, or processing variations.

In order to determine the effect of suspect propellant ingredient characteristics on the propensity for slag production in a real motor environment, a subscale motor experiment was designed to accomplish this objective. An existing 5 inch ballistic test motor was selected as the basic test vehicle due to low cost and quick turn around times. The standard converging/diverging nozzle was replaced with a submerged nozzle nose design to provide a positive trap for the slag which would increase both the quantity and repeatability of measured slag weights. CFD was used to assess a variety of submerged nose configurations to identify the design which possessed the best capability to reliably collect slag. CFD was also used to assure that the final selected nozzle design would result in flow field characteristics such as dividing streamline location, nose attach point, and separated flow structure which would have similitude with the RSRM submerged nozzle nose flow field. It was also decided to spin the 5 inch motor about its longitudinal axis to further enhance slag collection quantities. Again, CFD was used to select an appropriate spin rate along with other considerations, including the avoidance of burn rate enhancement from radial acceleration effects.

The CFD analyses were performed with the CELMINT code which is a two-phase Navier-Stokes coded employing an Eulerian/Lagrangian scheme, a low Reynolds number $k$-$\varepsilon$ turbulence model modified for wall injection, and both surface and distributed particle combustion models which include particle agglomeration and break-up. Aluminum oxide particle distributions were measured with RSRM propellant in a combustion bomb with particle quench capability. Predictions for slag weights and slag distribution patterns were compared with slag weight data from defined zones in the motor and nozzle. Various parameters were investigated to reconcile differences between CFD predictions and data. General comparisons were acceptable considering combustion bomb data on particle sizes was not available for each propellant sample. Confidence in using this methodology in the RSRM was enhanced by this successful subscale experiment.
Application of Two-Phase CFD to the Design and Propellant Slag Production

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

Thirteenth Workshop for CFD Applications in Rocket Propulsion
NASA Marshall Space Flight Center
Huntsville, Alabama
April 25-27, 1995

1246
Flight and static test data for the Space Shuttle Reusable Solid Rocket Motors reveals roughness and small spikes in the pressure trace for some motors during the 65-75 second time period.

An extensive investigation has determined that periodic expulsion of aluminum oxide slag is the cause of pressure perturbations.

Excessive slag production by some motors is suspected as making these motors more susceptible to slag expulsion.

Excessive slag production is related to propellant ingredient characteristics including but not limited to aluminum and ammonium perchlorate particle size distributions.

A low cost, quick turn-around experimental method was needed to evaluate effects of subtle changes in propellant ingredient characteristics on the propensity for slag production.
Experimental Program Objective and Approach

Objective:
Develop and employ a subscale rocket test motor capable of measuring relative slag production of propellants with subtle changes in ingredient characteristics.

Approach:
- Use an existing Thiokol 5-inch diameter ballistic test motor and static test spin stand.
- Modify existing converging/diverging nozzle entrance geometry by incorporating submerged nose to enhance slag capture and retention.
- Select motor operating pressure to match full scale motor pressure. Select spin rate to enhance slag capture but avoid propellant burn rate augmentation.
- Use CFD to determine overall viability of experiment, to aid in design of motor components, to support selection of test conditions, and to analyze test results.
Specific CFD Analysis Tasks

- Evaluate candidate nozzle entrance designs for slag capturing characteristics.

- Select submerged nose nozzle geometry that qualitatively simulates the primary flow pattern and features relative to nozzle nose attachment and recirculation pattern in the RSRM.

- Determine viability of experiment design before hardware manufacture by evaluating sensitivity of slag capture weights to small changes in aluminum oxide particle size distribution.

- Determine effect of spin rate on slag capture weights to support final selection of test spin rate.

- Perform post-test analysis of data including parametric studies as required to validate and calibrate two-phase CFD model.

- Use analysis results to upgrade two-phase CFD model for RSRM slag predictions.
FIVE INCH SPIN MOTOR NOZZLE
SLAG ZONES

ERC, Inc.
Velocity Vectors, Submerged Nozzle
RSRM 80 Second Stiff NBR Inhibitor
Velocity Vectors, Submerged Nozzle
Spin Motor at 50% Web Time
Two-Phase Flow CFD Methodology
CELMINT Code
(Combined Eulerian Lagrangian Multi-Dimensional Implicit Nonlinear Time-Dependent)

- Navier-Stokes Solution
  - Fully implicit, density-based, conservative, ensemble-averaged Navier-Stokes code
  - Low and high Reynolds number and wall injection $\kappa-\varepsilon$ models
  - Equilibrium and finite-rate chemistry for multi-species flows

- Two-phase Flow Models
  - Coupled Eulerian-Lagrangian for solid and liquid phases
  - Hermsen aluminum burn rate model for particle combustion
  - Specification of particle properties (density, size distribution)
  - Particle break-up based on Weber number
  - Agglomeration based on collisions between discrete phase particles and continuous phase smoke particles
  - Programmable for various particle capture criteria

ERC, Inc.
Effect of Spin Rate on Nozzle Slag Accumulation
WECCO AP, Surface Combustion, 400RPM

![Graph showing the effect of spin rate on normalized slag accumulation rate. The graph includes two lines: one for CFD and another for experimental data. The x-axis represents spin rate (RPM) ranging from 0 to 400, and the y-axis represents normalized slag accumulation rate ranging from 0 to 2.0. The CFD line is slightly lower than the experimental line.]
<table>
<thead>
<tr>
<th>Location</th>
<th>15% Web</th>
<th>50% Web</th>
<th>85% Web</th>
</tr>
</thead>
<tbody>
<tr>
<td>Port</td>
<td>50X50</td>
<td>50X65</td>
<td>50X65</td>
</tr>
<tr>
<td>Nozzle Closure</td>
<td>105X85</td>
<td>110X95</td>
<td>115X80</td>
</tr>
</tbody>
</table>
Spin Motor At 50 % Web Time

Computational Grid
Computational Grid
Spin Motor At 85 % Web Time
### RSRM Propellant Thermochemical and Nominal Particle Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propellant</td>
<td>TP-H1148</td>
</tr>
<tr>
<td>Pressure</td>
<td>625 psia</td>
</tr>
<tr>
<td>Total Temperature</td>
<td>6093° R</td>
</tr>
<tr>
<td>Molecular Weight</td>
<td>28.04</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>$6.189 \times 10^{-5}$ bm/ft-sec</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.138</td>
</tr>
<tr>
<td>Particle Distribution</td>
<td>Polynomial Fit to Wecco Quench Bomb Data</td>
</tr>
<tr>
<td>Particle Density</td>
<td>60lbm/ft³</td>
</tr>
<tr>
<td>Ratio of Initial Particle/Gas Velocity</td>
<td>1.0</td>
</tr>
<tr>
<td>Aluminum Oxide Caps Fraction (Discrete Phase)</td>
<td>28.33%</td>
</tr>
</tbody>
</table>
## Spin Motor Operation Conditions

<table>
<thead>
<tr>
<th>Parameter</th>
<th>15% Web</th>
<th>50% Web</th>
<th>85% Web</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chamber Pressure:</td>
<td>610.6 psia</td>
<td>628.8 psia</td>
<td>610.6 psia</td>
</tr>
<tr>
<td>Mass Flow Rate:</td>
<td>2.613 lbm/s</td>
<td>2.691 lbm/s</td>
<td>2.613 lbm/s</td>
</tr>
<tr>
<td>Propellant Burning Area:</td>
<td>115.47 in²</td>
<td>114.67 in²</td>
<td>112.35 in²</td>
</tr>
<tr>
<td>Throat Diameter:</td>
<td>.916 inches</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Burn Time:</td>
<td>2.71 seconds</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Experimental Data and Curve Fit Distribution Functions
WECCO Quench Bomb Data- 500 psi
3-inch Quench Distance

- WECCO Data
- Polynomial Distribution, 
  $D_m=113 \mu m$
- Log-Normal Distribution, 
  $D_m=96 \mu m, \sigma=0.30$
- Log-Normal Distribution, 
  $D_m=80 \mu m, \sigma=0.42$
Experimental Data and Curve Fit Density Functions
WECCO Quench Bomb Data- 500 psi
3-inch Quench Distance

![Graph showing mass fraction vs. particle diameter (microns)]
Experimental Data and Curve Fit Distribution Functions
Kerr McGee Quench Bomb Data - 500 psi
3-inch Quench Distance

Cumulative Mass Fraction

- Kerr McGee Data
- Polynomial Distribution, $D_m=140 \, \mu m$
- Log-Normal Distribution, $D_m=90 \, \mu m, \sigma =0.38$

Particle Diameter (microns)
Nozzle Slag Accumulation
WECCO and Kerr McGee AP
Surface Combustion, 400 RPM

Slag Accumulation (grams)

Percent Motor Web Time

- WECCO AP, Dm=113 \mu m
- Kerr McGee AP, Dm=140 \mu m
Velocity Magnitude
Spin Motor at 15% Web Time
Spin Motor at 50% Web Time

Velocity Magnitude

Velocity (ft/sec)
Velocity Magnitude
Spin Motor at 85% Web Time
Velocity Field In The Submerged Nozzle Region

Spin Motor At 15 % Web Time
Spin Motor At 50 % Web Time

Velocity Field In The Submerged Nozzle Region
Velocity Field In The Submerged Nozzle Region
Spin Motor At 85 % Web Time
50 Micron Diameter Particle Trajectories
Spin Motor At 85% Web Time
150 Micron Diameter Particle Trajectories

Spin Motor At 85% Web Time
Spin Motor Circumferential Velocity
WECCO, Surface Combustion, 400 RPM
15% Web Time

Circumferential Velocity (feet/second)

Radius (inches)

Axial Station Near Head End
Axial Station Near After End
Grain
Axial Station in Nozzle Entrance

ERC, Inc.
50 Micron Diameter Particle Trajectories
Spin Motor At 15% Web Time
50 Micron Diameter Particle Trajectories
Spin Motor At 50 % Web Time
50 Micron Diameter Particle Trajectories
Spin Motor At 85 % Web Time
Nozzle Slag Accumulation Rate
WECCO AP, Surface Combustion, 400 rpm

Slag Accumulation Rate (grams/second)

Percent Motor Web Time

- B, Analysis
- C, Analysis
- D, Analysis
- B+C+D, Analysis

ERC, Inc.
Nozzle Slag Accumulation
WECCO AP, Surface Combustion, 400 rpm

Slag Accumulation (Grams)

Percent Motor Web Time

B, Analysis
C, Analysis
D, Analysis
B+C+D, Analysis
B, Experimental
C, Experimental
D, Experimental
B+C+D, Experimental
FIVE INCH SPIN MOTOR SLAG DEPOSITS

CFD and Experimental

Slag Thickness – 4x

- CFD
- Experimental

ERC, Inc.
Slag Accumulation Per Unit Area Along The Nozzle
WECCO AP, Surface Combustion, 400 rpm

Slag Accumulation Per Unit Area (g/sq. in.)

Length Along Nozzle Surface From The Throat (in.)

Throat
Underneath Tangent Point
Nose Tip
End of Zone B
End of Zone C
End of Zone D

- Impinging Slag
- Captured Slag

ERC, Inc.
Effect of Initial Particle Velocity on Slag Accumulation
WECCO AP, Surface Combustion, 400 rpm

![Graph showing the effect of initial particle velocity on slag accumulation.](image-url)
Effect of Particle Density on Slag Impingement Rates
WECCO AP, Surface Combustion, 400 RPM
85% Web Time

Slag Impingement Rate (grams/second)

Density=100 lbm/cu.ft
Density=60 lbm/cu.ft

Zone B: 24.03, 19.163
Zone C: 1.023, 3.59
Zone D: 3.207, 3.164
Zones B+C+D: 28.08, 25.92

Nozzle Slag Zones
Nozzle Slag Accumulation
WECCO AP, Distributed Combustion, 400 rpm

Slag Accumulation (Grams)

Percent Motor Web Time

B, Analysis
C, Analysis
D, Analysis
B+C+D, Analysis
B, Experimental
C, Experimental
D, Experimental
B+C+D, Experimental

ERC, Inc.
Unburned Aluminum in Zone B
WECCO AP, Distributed Combustion, 400 rpm

Percent Unburned Aluminum

29.15

33.06

CFD

Experimental
Conclusions

- The use of two-phase CFD analysis was highly successful in determining the viability of an experimental motor test program being designed to measure the propensity for slag production of propellants with various ingredient variations.

- A submerged nose nozzle design was successfully developed and motor test spin rate selected to maximize slag capture and retention weights using two-phase CFD.

- The two-phase CFD model for the 5-inch spin test motor proved to be a credible analysis tool in evaluation of the slag weight distributions in motor.

- The slag capture criteria is the most important factor in the prediction model for slag capture. Uncertainties in particle properties appear to be less important.
Investigation of Slag Behavior for RSRM

P. Liaw, Y-S Chen, H. Shang, and M. Shih
Engineering Sciences, Inc., Huntsville, AL

and

D. Doran and E. Stewart
NASA MSFC, Huntsville, AL

ABSTRACT

It is known that the flowfield of the SRM (Solid Rocket Motor) is very complicated due to the complex characteristics of turbulent multi-phase flow, chemical reaction, particle combustion, evaporation, breakup and agglomeration, etc. It requires multi-phase calculations, chemical reaction simulation, and particle combustion, evaporation, and breakup models to obtain a better understanding of thermophysics for the SRM design using numerical methods. Also, the slag buildup due to the molten particles is another factor affecting the performance of the SRM. To achieve this goal, the VOF (Volume Of Fluid) method is used to capture the free surface motion so as to simulate the accumulation of the molten particles (slag) of SRM. A Finite-rate chemistry model is used to simulate the chemical reaction effects. For multi-phase calculations, Hermsen combustion model is used for the AL particle combustion analysis and Taylor Analogy Breakup (TAB) model is used for the particle breakup analysis. An interphase mass-exchange model introduced by Spalding is used for the evaporation calculation. The particle trajectories are calculated using a one-step implicit method for several groups of particle sizes by which the drag forces and heat fluxes are then coupled with the gas phase equations.

The preliminary results predicted a reasonable physical simulation of the particle effects using a simple 2-D solid rocket motor configuration. It shows that the AL/AL2O3 particle sizes are reduced due to the combustion, evaporation, and breakup. The flowfield is disturbed by the particles. Mach number distributions in the nozzle are deformed due to the effect of particle concentrations away from the center line.

The RSRM (Redesigned Solid Rocket Motor) geometry at 67 seconds is employed to investigate the slag behavior in the aft-end cavity with the combustion, evaporation, and breakup models. The particulate phase was assumed to be aluminum oxide (AL2O3) for the preliminary study. It is assumed that the propellant grain of the aft-end cavity has burned out completely at 67 seconds. The geometry and mass flow rate information were provided by the NASA Marshall Space Flight Center. The slag may flow out of the cavity and enter the nozzle due to the accelerations. The molten particles entering the aft-end cavity merge with the slag. The volume of the slag will grow and affect the performance of the RSRM. This shows that the effects of particles and slag on the flowfield are very significant. From the calculation, a flow vortex exists in the aft-end cavity of the RSRM. A stagnation point on the wall is captured. This flow impingement may cause the erosion of the wall. The shape of the vortex is changed due to the slag. The particles entering the cavity may become slag and either flow into or out of the cavity depending on the temperature and the surface tension of the molten particles. An axial gravity force of 2.4g is assumed to simulate the RSRM flowfield at 67 seconds.

The flowfield analysis using the FDNS code in the present research using the proposed models should provide a design guide for the solid rocket motors. The obtained results can give the designer a basic guide line for the use of materials and the nozzle geometry to improve the performance of SRMs.
Numerical Investigation of Slag Behavior for RSRM

P. Liaw, Y. S. Chen, H. M. Shang, and M. Shih
Engineering Sciences, Inc.

and

D. Doran and E. Stewart
ED32, NASA Marshall Space Flight Center

13th Workshop for CFD Applications in Rocket Propulsion

April 25-27, 1995
OBJECTIVE

1. BACKGROUND AND GENERAL APPROACH

2. NUMERICAL METHOD

3. APPLICATION

4. CONCLUSIONS

5. FOLLOWING WORK
BACKGROUND & GENERAL APPROACH

- ACCUMULATED SLAG WAS FOUND IN THE AFT-END CAVITY AND NOZZLE.

  ==> WILL THIS AFFECT THE MOTOR PERFORMANCE DUE TO ITS EFFECT ON THE PRESSURE?

- WILL VOF METHOD WORK FOR THE ANALYSIS OF SLAG BEHAVIOR?


**VOF Model**

The VOF transport equation is given below:

\[
\frac{\partial \alpha}{\partial t} + (u - u_g)_i \frac{\partial \alpha}{\partial x_i} = S_\alpha
\]

where \( \alpha = 1 \) stands for liquid and \( \alpha = 0 \) is for gas. The interface is located at \( 1 > \alpha > 0 \). For a given solution of \( \alpha \) field, equation (6) can be recast as:

**for compressible gas**:

\[
\frac{\partial \rho_m \phi}{\partial t} + \frac{\partial \rho_m (u - u_g)_i \phi}{\partial x_i} = S_\phi , \alpha < 0.01
\]

**for incompressible gas**:

\[
\rho_m \frac{\partial \phi}{\partial t} + \rho_m (u - u_g)_i \frac{\partial \phi}{\partial x_i} = S_\phi , \alpha \geq 0.0
\]

and

\[
\rho_m = \text{Max}\{\rho_g, \alpha \rho_\ell\}
\]

The interface \( \alpha \) solution compression procedure is expressed as:

\[
\alpha_{\text{new}} = \text{Max}\{0, \text{Min}[1, 0.5 + f(\alpha_{\text{old}} - 0.5)]\}
\]
\[ f = \frac{(\text{Interface volume})_{\text{new}}}{(\text{Interface volume})_{\text{initial}}} \]

The surface tension forces in the continuum surface force model is formulated as continuous body forces across the interface. These forces can be written as:

\[ F_x = -\sigma(\nabla \hat{n}) \alpha_x \]
\[ F_y = -\sigma(\nabla \hat{n}) \alpha_y + \left( \frac{\alpha_y}{y} \right), \text{for 2D, axisymmetric case only} \]
\[ F_z = -\sigma(\nabla \hat{n}) \alpha_z, \text{ for 3D case only} \]

where

\[ \hat{n} = \alpha_{xx} + \alpha_{yy} + \alpha_{zz} \]

\( \alpha \) is 0.5 for the free surface. The VOF method is used to represent the tracking of the free surface between the liquid and gas phase.
APPLICATIIONS

SLAG BEHAVIOR ANALYSIS FOR RSRM CONFIGURATION AT 67 SECONDS

(1) NO PRE-ACCUMULATED SLAG IN THE AFT-END CAVITY
    --- 2D AND 3D

(2) ASSUMED PRE-ACCUMULATED SLAG IN THE AFT-END CAVITY
    --- 2D
2D Grid system of RSRM configuration.

Mach number contours of RSRM, no particles.
Velocity vectors near the aft-end cavity, no particles.

Particle trajectories.
Slag accumulation in the aft-end cavity

\[ T = 0.01125 \text{ sec.} \]

\[ T = 0.28238 \text{ sec.} \]

\[ T = 0.91531 \text{ sec.} \]

\[ T = 2.31444 \text{ sec.} \]
Slag accumulation at $T = 2.3144$ sec.
Slag buildup history in the aft-end cavity.
Slag Accumulation for 3D RSRM at $T = 0.031687$
Slag Accumulation for 3D RSRM at $T = 0.031687$
Slag Accumulation for 3D RSRM at $T = 0.031687$
Slag Accumulation for 3D RSRM at $T = 0.031687$
Slag Accumulation for 3D RSRM at $T = 0.031687$
CONCLUSIONS AND RECOMMENDATIONS

- THE OBTAINED PRELIMINARY NUMERICAL RESULTS USING FDNS CODE SHOW THAT THE SLAG BEHAVIOR CAN BE PREDICTED NUMERICALLY. THE PREDICTED FLOW FIELD IS REASONABLE BASED ON THE PHYSICAL POINT OF VIEW.

- A DIRECT SIMULATION USING CHEMICAL REACTION, COMBUSTION/EVAPORATION/BREAKUP/AGGLOMERATION MODELS IS NECESSARY FOR A MORE ACCURATE NUMERICAL ANALYSIS OF SLAG BEHAVIOR.
In recent years, there has been a resurgence of interest in the development of hybrid rocket engines for advanced launch vehicle applications. Hybrid propulsion systems use a solid fuel such as hydroxyl-terminated polybutadiene (HTPB) along with a gaseous/liquid oxidizer. The performance of hybrid combustors depend on the convective and radiative heat fluxes to the fuel surface, the rate of pyrolysis in the solid-phase and the turbulent combustion processes in the gaseous-phase. These processes in combination specify the regression rates of the fuel surface and thereby the utilization efficiency of the fuel. In this paper, we employ computational fluid dynamic techniques in order to gain a quantitative understanding of the physical trends in hybrid rocket combustors.

The computational modeling is tailored to ongoing experiments at Penn State that employ a 2D slab-burner configuration. The co-ordinated computational/experimental effort enables model validation while providing an understanding of the experimental observations. Computations to date have included the full-length geometry with and without the aft-nozzle section as well as shorter-length domains for extensive parametric characterization. HTPB is used as the fuel with 1,3 butadiene being taken as the gaseous product of the pyrolysis. Pure gaseous oxygen is taken as the oxidizer. The fuel regression rate is specified using an Arrhenius rate reaction, while the fuel surface temperature is given by an energy balance involving gas-phase convection and radiation as well as thermal conduction in the solid-phase. For the gas-phase combustion, a two-step global reaction set is used. The standard \( k - \varepsilon \) model is used for turbulence closure. Radiation is presently treated using a simple diffusion approximation which is valid for large optical path lengths, representative of radiation from soot particles.

Computational results are obtained to determine the trends in the fuel burning or regression rates as a function of the head-end oxidizer mass flux, \( G = \rho_e U_e \), and the chamber pressure. Furthermore, computations of the full slab-burner configuration have also been obtained for various stages of the burn. Comparisons with available experimental data from small-scale tests conducted by General Dynamics-Thiokol-Rocketdyne suggest reasonable agreement in the predicted regression rates. Future work will include: (1) a model for soot generation in the flame for more quantitative radiative transfer modeling, (2) parametric study of combustion efficiency and (3) transient calculations to help determine the possible mechanisms responsible for combustion instability in hybrid rocket motors.
Combustion Processes in Hybrid Rocket Engines

S. Venkateswaran and C. L. Merkle
Propulsion Engineering Research Center
The Pennsylvania State University
University Park, PA 16802

Presented at
The 13th Workshop for CFD Applications in Rocket Propulsion, MSFC, April 25-27, 1995
Presentation Outline

- Introduction
  - Research Issues
  - Penn State Slab Burner Configuration

- Physical Modeling
  - Gas/Surface Coupling
  - Radiation

- Computational Results
  - Representative Solutions
  - Characterization of Regression Rates

- Conclusions
Introduction

- Advantages of Hybrid Propulsion
  - Reduced Cost
  - Safety
  - Improved Reliability
  - Thrust Tailoring
  - Environmentally Friendly

- Hybrids Development
  - Intermittent Testing Since 60's
  - JIRAD
  - AMROC
  - France & Japan

- Small-Scale Testing
  - JPL/Strand *et al.*
  - ONERA
  - UAH
  - Penn State
Research Issues

- Characterization of Fuel Surface Regression
  - Fuel Pyrolysis and Surface Chemistry
  - Heat Fluxes - Convection and Radiation

- Combustion Efficiency

- Combustion Instability

- Modeling Issue:
  - Boundary Layer vs. Navier-Stokes
Schematic of Hybrid Rocket Motor
Experimental Configuration

Top View

Profile View
Experimental Configuration

- Test Conditions
  - Fuel - HTPB
  - Oxidizer - GOX
  - Pressures - 300 to 900 psi
  - GOX Flow Rates - 0.2 to 0.8 $\text{lbm/s}$
  - GOX Mass Flux ($G = \rho U$) - $0.1$ to $0.5$ $\text{lbm/in}^2 \cdot s$
Physical Modeling

- Gas-Phase Navier-Stokes Equations
  — Standard $k - \epsilon$ Model

- Gas-Phase/Combustion Model:
  — Butadiene—Product of Pyrolysis
  — Two-Step Global Kinetics Model

$$C_4H_6 + 3.5O_2 \rightarrow 4CO + 3H_2O$$

$$CO + 0.5O_2 \leftrightarrow CO_2$$

- Solid-Phase/Pyrolysis:
  — Arrhenius Pyrolysis Rate

$$\rho_s r_b = A_s exp\left(\frac{-E_s}{R_u T_s}\right)$$
Solid/Gas Coupling

• Surface Mass Balance

\[ \rho v = - \rho_s r_b \]

• Surface Energy Balance

\[-\lambda \frac{\partial T}{\partial y} + Q_{rad} + \rho v h - \sum_{i=1}^{N} \rho D_{im} \frac{\partial Y_i}{\partial y} h_i = -\lambda_s \left( \frac{\partial T}{\partial y} \right)_s - \rho_s r_b h_s \]
Radiation Modeling

• Gaseous Molecular Radiation
  — Optically Thin Approximation

\[ Q_{rad,k} = \sum_{i,j} \frac{4\pi k_{i,j} T_{i,j}^4}{J_{i,j} F_{i,j} - k} \]

• Particulate (Soot) Radiation
  — Optically Thick Approximation

\[ Q_{rad,k} = -\lambda_R \frac{\partial T}{\partial y} \]

\[ \text{where } \lambda_R = \frac{4}{3} \pi \frac{C}{K} T^3 \]
Representative Solution

Grid Geometry

Temperature Contours

$T_g \times 1000K$

500K

3500K
Representative Solution

Axial Velocity

Mach Number Contours
Representative Solutions

Carbon Dioxide Mass Fraction

GOX Mass Fraction
Representative Results

Centerline Variation of Mass Flux (G)

**Legend**
- Start of Burn
- Midway Through Burn
- End of Burn

**Mass Flux G**

- 1.000 kg/m²-s
- 0.500 kg/m²-s
- 0.350 kg/m²-s
- 0.250 kg/m²-s

**Axial Distance, m**

0.000 to 0.700
Representative Results

Surface Regression Rate

\[ \tau_b, \text{in/s} \]

\[ \tau_s, K \]

Fuel Surface Temperature

Wall Heat Fluxes

\[ Q_{\text{rad}}, \text{W/m}^2 \]

\[ Q_{\text{conv}}, \text{W/m}^2 \]

CONVECTIVE HEAT FLUX

RADIATIVE HEAT FLUX

Axial Distance, m
Parametric Studies
Different Stages in Burn

Start of Burn

Midway Through Burn

End of Burn
Parametric Studies
Different Stages in Burn

W/O Radiation

Surface Regression Rate

Convective Wall Heat Fluxes

Axial Distance, m

1330
Parametric Studies
Different Stages in Burn

With Radiation/Optically Thick

Surface Regression Rate

Radiative Wall Heat Fluxes

Axial Distance, m
Parametric Studies
Effect of GOX Flow Rate

Temperature Contours

Temperature Profiles
Parametric Studies
Effect of GOX Flow Rate

W/O Radiation

Surface Regression Rate

Convective Wall Heat Fluxes
Parametric Studies
Effect of GOX Flow Rate

With Radiation/Optically Thick

Surface Regression Rate

Radiative Wall Heat Fluxes
Parametric Characterization of Fuel Surface Regression

![Graph showing regression rate vs. specific mass flow rate](image)

- **Current results with radiation**
- **Current results w/o radiation**
Conclusions

- Navier-Stokes Analysis of Hybrid Motor
  - Planar Slab Burner Configuration
  - Arrhenius-Rate for Pyrolysis
  - Global Chemistry
  - Turbulence Model
  - 'Thick/Thin' Radiation Model

- Computational Results
  - Parametric Characterization
  - Fuel Surface Temperatures 900 to 1100 K
  - Regression Rates of 0.01 to 0.07 in/s
  - Radiative Fluxes - Significant Contribution

- Ongoing/Future Work:
  - Radiation Properties - Soot Concentration
  - Combustion Efficiency - Downstream Mixing
  - Combustion Instability - Transient Calculations
CFD ANALYSIS OF THE 24-INCH JIRAD HYBRID ROCKET MOTOR

Pak-Yan Liang, Ronald Ungewitter, Scott Claflin
CFD Technology Center
Rocketdyne Division, Rockwell International
Mail Code IB39, 6633 Canoga Avenue
Canoga Park, CA 91303

ABSTRACT

A series of multispecies, multiphase CFD analyses of the 24-inch diameter joint government/industry IR&D (JIRAD) hybrid rocket motor is described. The 24-inch JIRAD hybrid motor operates by injection of liquid oxygen (LOX) into a vaporization plenum chamber upstream of ports in the hydroxyl-terminated polybutadiene (HTPB) solid fuel. Injector spray pattern had a strong influence on combustion stability of the JIRAD motor so a CFD study was initiated to define the injector-end flow field under different oxidizer spray patterns and operating conditions. By using CFD to gain a clear picture of the flow field and temperature distribution within the JIRAD motor, it is hoped that the fundamental mechanisms of hybrid combustion instability may be identified and then suppressed by simple alterations to the oxidizer injection parameters such as injection angle and velocity.

The simulations in this study were carried out using the GALACSY (General ALgorithm for Analysis of Combustion SYstems) multiphase combustion codes. GALACSY consists of a comprehensive set of droplet dynamic submodels (atomization, evaporation, etc.) and a computationally efficient hydrocarbon chemistry package built around a robust Navier-Stokes solver optimized for low Mach number flows. Lagrangian tracking of dispersed particles describes a closely coupled spray phase.

The CFD cases described in this paper represent various levels of simplification of the problem. They include: (A) gaseous oxygen with noncombusting fuel vapor blowing off the walls at various oxidizer injection angles and velocities, (B) gaseous oxygen with combusting fuel vapor blowing off the walls, and (C) liquid oxygen with combusting fuel vapor blowing off the walls. The study used an axisymmetric model and the results indicate that the injector design significantly effects the flow field in the injector-end of the motor. Markedly different recirculation patterns are observed in the vaporization chamber as oxygen velocity and/or spray pattern is varied. The ability of these recirculation patterns to stabilize the diffusion flame above the surface of the solid fuel gives a plausible explanation for the experimentally determined combustion stability characteristics of the JIRAD motor, and suggests how combustion stability can be assured by modifications to the injector design. Planned future activities to the submodels which allow for additional degree of realism will be discussed.
CFD ANALYSIS OF 24 INCH JIRD HYBRID ROCKET MOTOR

Dr. Pak Liang, Ronald Ungewitter, Scott Claflin

Rocketdyne Div./Rockwell International
OVERVIEW

- ISSUE: SIGNIFICANT INCREASES IN PRESSURE OSCILLATION RECORDED DURING TESTING OF ANGLED AND STRAIGHT LOX INJECTORS

STRAIGHT

ANGLED

- GOAL: TO GAIN AN UNDERSTANDING OF THE PHYSICAL MECHANISMS OF HYBRID ROCKET MOTOR COMBUSTION AND THE CONDITIONS LEADING TO COMBUSTION OSCILLATIONS
LIQUID / GAS OXIDIZER INJECTED INTO VAPORIZATION CHAMBER

SOLID FUEL SUBLIMATES WHICH THEN REACTS WITH OXIDIZER

INJECTOR-END RECIRCULATION PATTERN DEEMED CRITICAL TO COMBUSTION STABILITY AND FLAME HOLDING MECHANISMS
PLAN

APPROACH:

- USE GALACSY CFD COMBUSTION CODE FOR CFD ANALYSIS
- CONDUCT SERIES OF ANALYSES AT SEVERAL LEVELS OF SOPHISTICATION TO IDENTIFY BASIC FLUID DYNAMIC MECHANISMS

- MODEL

- TWO ZONE, AXISYMMETRIC (STEPS 1, 2, & 3)
- WALL BLOWING FUEL REPRESENTING SUBLIMATION PROCESS
- O2/RP-1 CHEMISTRY

- PLAN:

- STEP 1 - THREE NON REACTING GOX FLOW CALCULATIONS WITH DIFFERENT INLET CONDITIONS (COMPLETED)
- STEP 2 - ANGLED AND STRAIGHT REACTING GOX CASES (COMPLETED)
- STEP 3 - LOX STREAM REPRESENTED AS FULLY ATOMIZED DROPS; STRAIGHT AND ANGLED FLOW (IN PROGRESS)
- STEP 4 - MULTI-PORT COMBUSTOR, STRAIGHT AND ANGLED FLOW
GALACSY CODE

- PRESSURE-BASED, EXTENDED SIMPLE-S SEQUENTIAL SOLVER METHODOLOGY (REACT PLATFORM)

- LAGRANGIAN DROPLET TRACKING, "ONION SKIN" EVAPORATION MODEL

- EXPLICIT INTER-ZONAL COUPLING FOR MULTIZONE PROBLEMS

- GLOBAL FINITE RATE REACTION FOR HYDROCARBON FUEL PLUS H/O EQUILIBRIUM CHEMISTRY (VALIDATED FOR CH4)
  - RP1 (REPRESENTED AS C_{12.449}H_{24.47}) CHEMISTRY REQUIRES 10 SPECIES
STEP 1 – NON REACTING RESULTS

CASE 1 RESULTS*

CONDITIONS: TWO ZONE: 90x40, 100x16
GOX, Tin=293°C K, Uin=47 m/s

* IMAGE TRUNCATES COMPUTATIONAL DOMAIN AND PARTICLE TRACES DO NOT MOVE BETWEEN ZONES
STEP 1 – NON REACTING RESULTS

CASE 2 RESULTS*

CONDITIONS:  TWO ZONE: 90x40, 100x16
              GOX, Tin=811° K, Uin=132 m/s

* IMAGE TRUNCATES COMPUTATIONAL DOMAIN AND PARTICLE TRACES DO NOT MOVE BETWEEN
STEP 1 - NON REACTING RESULTS

CASE 3 RESULTS*

CONDITIONS:  TWO ZONE: 90x40, 100x16
GOX, Tin=1810° K, Uin=296 m/s

* IMAGE TRUNCATES COMPUTATIONAL DOMAIN AND PARTICLE TRACES DO NOT MOVE BETWEEN ZONES
STEP 2 - REACTING RESULTS

CASE 1 STRAIGHT FLOW*

CONDITIONS: TWO ZONE: 90x40, 100x16
GOX, Tin=811° K, Uin=132 m/s

* IMAGE TRUNCATES COMPUTATIONAL DOMAIN AND PARTICLE TRACES DO NOT MOVE BETWEEN
CASE 2 ANGLED FLOW

CONDITIONS: TWO ZONE: 90x40, 100x16
GOX, Tin=811° K, Uin=132 m/s

* IMAGE TRUNCATES COMPUTATIONAL DOMAIN AND PARTICLE TRACES DO NOT MOVE BETWEEN ZONES
CONCLUSIONS

- COMPLETED CASES CONSISTENT WITH SINGLE POINT TEST DATA

- RECIRCULATION PATTERN CAN CHANGE COMPLETELY DUE TO DIFFERENCES IN INJECTOR PATTERN

- CHEMISTRY PACKAGE NEED ADDITIONAL VALIDATION

- NEW CALCULATIONS (STEP 3) INDICATE PERFORMANCE DECREASE OF NUMERICAL SCHEME DUE TO DROPLET COUPLING AND INTERZONAL BOUNDARIES
  
  - NEED ALGORITHMIC REFINEMENT TO INCREASE PERFORMANCE
  - POSSIBLE OPTION: REASSEMBLY OF MULTIZONE MATRICES BACK INTO SINGLE ZONE MATRIX BEFORE CALLING MATRIX SOLVER
## Thirteenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology

### 6. AUTHOR(S)
R.W. Williams, Compiler

### 7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
George C. Marshall Space Flight Center
Marshall Space Flight Center, Alabama 35812

### 9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546-0001

### 11. SUPPLEMENTARY NOTES
Prepared by Structures and Dynamics Laboratory, Science and Engineering Directorate.

### 12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified-Unlimited
Subject Category 34

### 13. ABSTRACT (Maximum 200 words)
This conference publication includes various abstracts and presentations given at the 13th Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion and Launch Vehicle Technology held at the George C. Marshall Space Flight Center April 25-27 1995. The purpose of the workshop was to discuss experimental and computational fluid dynamic activities in rocket propulsion and launch vehicles. The workshop was an open meeting for government, industry, and academia. A broad number of topics were discussed including computational fluid dynamic methodology, liquid and solid rocket propulsion, turbomachinery, combustion, heat transfer, and grid generation.

### 14. SUBJECT TERMS
- computational fluid dynamics
- rocket propulsion
- launch vehicle technology
- liquid rocket
- solid rocket
- turbopump
- turbomachinery
- combustion
- methodology
- impeller
- inducer
- heat transfer
- grid generation
- nozzle
- plume spray
- injector

### 15. NUMBER OF PAGES
668

### 16. PRICE CODE
A99

### 17. SECURITY CLASSIFICATION
Unclassified

### 18. SECURITY CLASSIFICATION OF THIS PAGE
Unclassified

### 19. SECURITY CLASSIFICATION OF ABSTRACT
Unclassified

### 20. LIMITATION OF ABSTRACT
Unlimited

NSN 7540 - 01 - 280 - 5500