Tenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion

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

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

<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>TECHNOLOGY TEST BED REVIEW (H.V. McConnaughey)</td>
<td>1</td>
</tr>
<tr>
<td>ADVANCED SOLID ROCKET MOTOR PROJECT STATUS (K.D. Coates)</td>
<td>27</td>
</tr>
<tr>
<td>SPACE TRANSPORTATION MAIN ENGINE (J.C. Monk)</td>
<td>45</td>
</tr>
<tr>
<td>THE IMPACT OF TIME STEP DEFINITION ON CODE CONVERGENCE AND ROBUSTNESS (S. Venkateswaran, J.M. Weiss, and C.L. Merkle)</td>
<td>83</td>
</tr>
<tr>
<td>COMPARISON BETWEEN THE PISO ALGORITHM AND PRECONDITIONING METHODS FOR COMPRESSIBLE FLOW (C.L. Merkle, P.E.O. Buelow, and S. Venkateswaran)</td>
<td>123</td>
</tr>
<tr>
<td>A COMPARISON OF ARTIFICIAL COMPRESSIBILITY AND FRACTIONAL STEP METHODS FOR INCOMPRESSIBLE FLOW COMPUTATIONS (D.C. Chan, A.D. Darian, and M.M. Sindir)</td>
<td>147</td>
</tr>
<tr>
<td>CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER (G.C. Cheng, Y.S. Chen, and R.W. Williams)</td>
<td>201</td>
</tr>
<tr>
<td>CFD APPLICATIONS IN PUMP FLOWS (C. Kiris, L. Chang, and D. Kwak)</td>
<td>219</td>
</tr>
<tr>
<td>IMPELLER TANDEM BLADE STUDY WITH GRID EMBEDDING FOR LOCAL GRID REFINEMENT (G. Bache')</td>
<td>259</td>
</tr>
<tr>
<td>THREE-DIMENSIONAL FLOW FIELDS INSIDE A SHROUDED INDUCER AT DESIGN AND OFF-DESIGN CONDITIONS (CFD STUDY) (C. Hah, O. Kwon, D.A. Greenwald, and R. Garcia)</td>
<td>289</td>
</tr>
<tr>
<td>EFFECTS OF CURVATURE AND ROTATION ON TURBULENCE IN THE NASA LOW-SPEED CENTRIFUGAL COMPRESSOR IMPELLER (J.G. Moore and J. Moore)</td>
<td>315</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMIC DESIGN OF ROCKET ENGINE PUMP COMPONENTS (W.C. Chen, G.H. Prueger, D.C. Chan, and A.H. Eastland)</td>
<td>339</td>
</tr>
<tr>
<td>Title</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>SSME HPOTP IMPELLER BACKCAVITY CFD ANALYSIS (W.W. Hsu and S.J. Lin)</td>
<td>361</td>
</tr>
<tr>
<td>NLS CLUTCHING BEARING CAVITY FLOW ANALYSIS (K. Tran, D.C. Chan, and A. Darian)</td>
<td>389</td>
</tr>
<tr>
<td>CFD ANALYSIS TO OPTIMIZE A DESIGN MODIFICATION OF BSMT (M. Ratcliff, R. Avva, and R. Williams)</td>
<td>419</td>
</tr>
<tr>
<td>COMBUSTION INSTABILITY ANALYSIS FOR LIQUID PROPELLANT ROCKET ENGINES (Y.M. Kim, C.P. Chen, and J.P. Ziebarth)</td>
<td>441</td>
</tr>
<tr>
<td>INVERSE DESIGN OF A PROPER NUMBER, SHAPES, SIZES, AND LOCATIONS OF COOLANT FLOW PASSAGES (G.S. Dulikravich)</td>
<td>467</td>
</tr>
<tr>
<td>NUMERICAL ANALYSIS OF THE HOT-GAS-SIDE AND COOLANT-SIDE HEAT TRANSFER IN LIQUID ROCKET ENGINE COMBUSTORS (T.S. Wang and V. Luong)</td>
<td>487</td>
</tr>
<tr>
<td>AN EFFICIENT AND ROBUST GRID OPTIMIZATION ALGORITHM (B.K. Soni and S. Yang)</td>
<td>503</td>
</tr>
<tr>
<td>ENHANCEMENTS TO THE GRIDGEN STRUCTURED GRID GENERATION SYSTEM FOR INTERNAL AND EXTERNAL FLOW APPLICATIONS (J.P. Steinbrenner and J.R. Chawner)</td>
<td>543</td>
</tr>
<tr>
<td>CAGI: COMPUTER AIDED GRID INTERFACE—A WORK IN PROGRESS (B.K. Soni, T.-Y. Yu, and D. Vaughn)</td>
<td>577</td>
</tr>
<tr>
<td>USING ADAPTIVE GRID IN MODELING ROCKET NOZZLE FLOW (A.S. Chow and K.-R. Jin)</td>
<td>615</td>
</tr>
<tr>
<td>COMPLEX THREE-DIMENSIONAL INTERNAL FLOWS IN THE ASRM AND RSRM AFT END SEGMENTS (E.J. Reske, D.F. Billings, and J.W. Cornelison)</td>
<td>647</td>
</tr>
<tr>
<td>AN ANALYSIS OF THE FLOW FIELD IN THE REGION OF THE ASRM FIELD JOINTS (R.A. Dill and H.R. Whitesides)</td>
<td>663</td>
</tr>
<tr>
<td>EFFECT OF INCLUDING VARIABLE GAS PROPERTIES AND ENTRAINED PARTICLES IN THE FLOW ANALYSIS OF THE ASRM NOZZLE (C.D. Clayton)</td>
<td>689</td>
</tr>
<tr>
<td>A TWO-PHASE RESTRICTED EQUILIBRIUM MODEL FOR COMBUSTION OF METALIZED SOLID PROPELLANTS (J.S. Sabnis, F.J. de Jong, and H.J. Gibeling)</td>
<td>713</td>
</tr>
<tr>
<td>Title</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>CURRENT STATUS OF THE DEVELOPMENT OF AN IGNITION TRANSIENT MODEL FOR SOLID ROCKET MOTORS (G.D. Luke and H.A. Dwyer)</td>
<td>725</td>
</tr>
<tr>
<td>SRMAFTE FACILITY CHECKOUT MODEL FLOW FIELD ANALYSIS (R.A. Dill and H.R. Whitesides)</td>
<td>763</td>
</tr>
<tr>
<td>A COMPARATIVE STUDY OF THE EFFECTS OF INHIBITOR STUB LENGTH ON SOLID ROCKET MOTOR COMBUSTION CHAMBER PRESSURE OSCILLATIONS: RSRM AT T=80 SECONDS, PRELIMINARY RESULTS (D. Chasman, D. Burnette, J. Holt, and R. Farr)</td>
<td>787</td>
</tr>
<tr>
<td>OVERVIEW OF THE RELEVANT CFD WORK AT THIOKOL CORPORATION (P. Chwalowski and H.-T. Loh)</td>
<td>791</td>
</tr>
<tr>
<td>A STATUS OF THE ACTIVITIES OF THE NASA/MSFC COMBUSTION DEVICES TECHNOLOGY TEAM (P.K. Tucker)</td>
<td>807</td>
</tr>
<tr>
<td>CFD ANALYSIS OF THE STME NOZZLE FLOWFIELD (A. Krishnan, P.K. Tucker)</td>
<td>831</td>
</tr>
<tr>
<td>NLS NOZZLE BASE FLOW CHARACTERISTICS (J.J. Erhart)</td>
<td>849</td>
</tr>
<tr>
<td>HEAT TRANSFER IN ROCKET ENGINE COMBUSTION CHAMBERS AND NOZZLES (P.G. Anderson, Y.S. Chen, and R.C. Farmer)</td>
<td>863</td>
</tr>
<tr>
<td>APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO THE DESIGN OF THE FILM COOLED STME SUBSCALE NOZZLE FOR THE NATIONAL LAUNCH SYSTEM (J.L. Garrett)</td>
<td>897</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF SPACE SHUTTLE MAIN ENGINE MULTIPLE PLUME FLOWS AT HIGH-ALTITUDE FLIGHT CONDITIONS (N.S. Dougherty, J.B. Holt, B.L. Liu, and S.L. Johnson)</td>
<td>923</td>
</tr>
<tr>
<td>DIRECT NUMERICAL SIMULATION OF A COMBUSTING DROPLET WITH CONVECTION (P.Y. Liang)</td>
<td>945</td>
</tr>
<tr>
<td>NUMERICAL MODELING FOR DILUTE AND DENSE SPRAYS (C.P. Chen, Y.M. Kim, H.M. Shang, J.P. Ziebarth, and T.S. Wang)</td>
<td>987</td>
</tr>
<tr>
<td>MODELING OF SSME FUEL PREBURNER ASI (P.Y. Liang)</td>
<td>1013</td>
</tr>
</tbody>
</table>

v
<table>
<thead>
<tr>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD MODELING OF TURBULENT FLOWS AROUND THE SSME MAIN INJECTOR ASSEMBLY USING POROSITY FORMULATION (G.C. Cheng, Y.S. Chen, and J.H. Ruf)</td>
<td>1033</td>
</tr>
<tr>
<td>COMPUTATIONAL FLUID DYNAMICS ANALYSIS OF SSME PHASE II AND PHASE II+ PREBURNER INJECTOR ELEMENT HYDROGEN FLOW PATHS (J. H. Ruf)</td>
<td>1071</td>
</tr>
<tr>
<td>STME HYDROGEN MIXER STUDY (R. Blumenthal, D. Kim, and G. Bache')</td>
<td>1093</td>
</tr>
<tr>
<td>AN EXPERIMENTAL STUDY OF THE FLUID MECHANICS ASSOCIATED WITH POROUS WALLS. (N. Ramachandran, J. Heaman, and A. Smith)</td>
<td>1117</td>
</tr>
<tr>
<td>TURBINE DISK CAVITY AERODYNAMICS AND HEAT TRANSFER (B.V. Johnson and W.A. Daniels)</td>
<td>1163</td>
</tr>
<tr>
<td>A NUMERICAL STUDY OF TWO-DIMENSIONAL VORTEX SHEDDING FROM RECTANGULAR CYLINDERS (A.H. Hadid, M.M. Sindir, and R.I. Issa)</td>
<td>1181</td>
</tr>
<tr>
<td>A STATUS OF THE TURBINE TECHNOLOGY TEAM ACTIVITIES (L.W. Griffin)</td>
<td>1205</td>
</tr>
<tr>
<td>A CRITICAL EVALUATION OF A THREE-DIMENSIONAL NAVIER-STOKES CFD AS A TOOL TO DESIGN SUPERSONIC TURBINE STAGES (C. Hah, O. Kwon, and M. Shoemaker)</td>
<td>1227</td>
</tr>
<tr>
<td>NAVIER-STOKES ANALYSIS OF AN OXIDIZER TURBINE BLADE WITH TIP CLEARANCE (H.J. Gibeling and J.S. Sabnis)</td>
<td>1243</td>
</tr>
<tr>
<td>NUMERICAL SIMULATION OF TURBOMACHINERY FLOWS WITH ADVANCED TURBULENCE MODELS (B. Lakshminarayana, R. Kunz, J. Luo, and S. Fan)</td>
<td>1275</td>
</tr>
<tr>
<td>DEVELOPMENT OF A CFD CODE FOR INTERNAL FLOWS IN LIQUID FUELED ENGINES (Y. Dakhoul)</td>
<td>1307</td>
</tr>
<tr>
<td>DEVELOPMENT OF THE KIVA-II CFD CODE FOR ROCKET PROPULSION APPLICATIONS (R.V. Shannon, Jr. and A.L. Murray)</td>
<td>1349</td>
</tr>
<tr>
<td>A COMPUTATIONAL DESIGN SYSTEM FOR RAPID CFD ANALYSIS (E.P. Ascoli, S.L. Barson, M.E. DeCroix, and M.M. Sindir)</td>
<td>1379</td>
</tr>
<tr>
<td>OPTIMUM DESIGN OF NINETY DEGREE BENDS (V. Modi)</td>
<td>1397</td>
</tr>
<tr>
<td>Title</td>
<td>Page</td>
</tr>
<tr>
<td>----------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>A MULTIDOMAIN METHOD FOR SUBSONIC VISCOUS FLOWS (D.C. Chan and M.M.</td>
<td>1427</td>
</tr>
<tr>
<td>Sindir)</td>
<td></td>
</tr>
<tr>
<td>LARGE EDDY SIMULATION OF COMPRESSIBLE TURBULENT CHANNEL FLOWS</td>
<td>1453</td>
</tr>
<tr>
<td>(R.A. Beddini and J.P. Ridder)</td>
<td></td>
</tr>
<tr>
<td>TREATING CONVECTION IN SEQUENTIAL SOLVERS (W. Shyy and S. Thakur)</td>
<td>1469</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>
TECHNOLOGY TEST BED REVIEW

H.V. McConnaughey
April 28, 1992
AGENDA

- WHAT IS TTB?
- TTB OBJECTIVES
- TTB MAJOR ACCOMPLISHMENTS
- SOME CFD CHALLENGES
- FUTURE PLANS
SSME TECHNOLOGY TEST BED HISTORY

PLANNING INITIATED IN 1982

MOTIVATION: SSME PROGRAM BENEFITS (CHARACTERIZATION OF ENGINE INTERNAL OPERATING ENVIRONMENTS AND ASSESSMENT OF PROTOTYPE HARDWARE)

OAST PROPULSION TECHNOLOGY SYSTEM-LEVEL VALIDATION

SITE: SATURN S1-C STAGE TEST STAND AT MSFC

FACILITY MODIFICATIONS: 1984 - 1988

TESTING INITIATED: FALL, 1988
TTB OBJECTIVES

- Assess propulsion technology advances in an engine systems environment
- Enhance the process for implementation of technology into emerging and operational programs
- Provide system test capability for evaluation of prototype hardware
- Support NASA programs with anomaly resolution on an as-needed basis
- Develop and maintain in-house, hands-on rocket propulsion hardware and test experience/capability at MSFC
TTB MAJOR ACCOMPLISHMENTS

- 31 SSME TESTS CONDUCTED/ ~3000 SECONDS CUMULATIVE TEST TIME
- EVALUATION OF A MODIFIED SSME
- ENGINE 3001 (HIGHLY INSTRUMENTED SSME) ENVIRONMENT CHARACTERIZATION
- DEMONSTRATION OF SSME ADVANCED DEVELOPMENT CONCEPTS
- SUPPORT OF SHUTTLE FLIGHT AND DEVELOPMENT INVESTIGATIONS
- ASSESSMENT OF ADVANCED PROPULSION TECHNOLOGY CONCEPTS
- NUMEROUS IMPROVEMENTS DERIVED FROM IN-HOUSE, HANDS-ON INVOLVEMENT
# TTB Activities To Date

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Advanced Engine (0208)</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Downtime in support of 17&quot; disconnect</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 w/ Std RDNN Pumps</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Downtime in support of phase II HGM</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 w/ Instr RDNN Pumps</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 w/ Hydrostatic Brg HPOTP</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Technology Items:</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exit plane spectroscopy of SSME</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Plume temperature measurements</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Optical plume anomaly detector</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Sodium resonant line absorption meas.</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSME exit plane holography</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Laser induced fluorescence</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SAFD</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>TTB/HOSC Expert Systems</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nozzle optic assembly</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HPOTP Preburner pump end hydrostatic bearing retrofit</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Low cost controller</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
TTB CONTRIBUTIONS TO SSME

ACCOMPLISHMENT: LARGE THROAT SSME CHARACTERIZATION
SYSTEM PERFORMANCE DEFINED
MORE BENIGN ENVIRONMENT DEMONSTRATED
STABLE COMBUSTION DEMONSTRATED
TURBOMACHINERY REDESIGN REQUIREMENTS IDENTIFIED

ENGINE: 0208

BENEFIT: EARLY ASSESSMENT OF PROPOSED SSME IMPROVEMENTS
PERFORMANCE OF ASSESSMENT WAS FREE FROM USUAL DEVELOPMENT
PRESSURES AND WAS ACCOMPLISHED WITHOUT INTERFERENCE TO SSC

IMPACT: TTB RESULTS FORMED THE BASIS FOR ADVANCED FAB LTMCC BASELINE
STABLE COMBUSTION WITHOUT BAFFLES OR ACOUSTIC CAVITIES IS
APPLICABLE TO FUTURE DESIGNS
ACCOMPLISHMENT: PHASE II SSME INTERNAL ENVIRONMENT CHARACTERIZATION

~ 630 TTB-UNIQUE ENGINE MEASUREMENTS

DEFINES INTEGRATION PARAMETERS FOR ALTERNATE COMPONENTS (e.g., ATDs)

ENGINE: 3001

BENEFIT: INCREASED UNDERSTANDING OF OPERATING ENVIRONMENT

BASELINE DATA FOR COMPARISON WITH ATDs AND PHASE II+ POWERHEAD

CALIBRATION AND IMPROVEMENT OF POWER BALANCE MODEL, DIGITAL TRANSIENT MODEL, AND ADVANCED ANALYTICAL MODELS

IMPACT: TO BE FULLY REALIZED. EXAMPLES: SIGNIFICANT DEFICIENCIES IN SSME POWER BALANCE MODEL HAVE BEEN DISCOVERED. HAVE MADE NEW DISCOVERIES ABOUT TURBOPUMP OPERATION.
MAJOR IMPACTS OF 3001 TESTING

1. ACQUISITION OF HERETOFORE UNMEASURED SSME HOT-FIRE DATA - COMPLETE MAPPING OF ENGINE OPERATION FOR RANGE OF MIXTURE RATIO, POWER LEVEL, PUMP INLET (NPSP), F7 ORIFICE AND REPRESS CONDITIONS

   MAJOR FLOWRATES
   INSTRUMENTED TURBOPUMPS
   OTHER Ps, Ts, STRAINS, etc.

   ~ 630 MEASUREMENTS IN TOTAL

2. CALIBRATION AND IMPROVEMENT OF MODELS

   POWER BALANCE MODEL
   1-D TURBOPUMP MODELS
   CFD MODELS
   THERMAL MODELS
   STRUCTURAL MODELS
   STRESS MODELS

3. NUMEROUS LESSONS LEARNED

   INSTRUMENTATION DESIGN AND IMPLEMENTATION
   TEST OPERATION EFFICIENCY ENHANCEMENTS
MAJOR IMPACTS OF 3001 TESTING

NEW DATA

- IMPROVED CHARACTERIZATION OF SSME OPERATION
  - ENHANCED DATABASE
  - INCREASED UNDERSTANDING OF HOT-FIRE ENVIRONMENT

  - IMPROVED ANOMALY OR ISSUE RESOLUTION
  - IMPROVED MODELS
  - IMPROVED DESIGNS
  - CALIBRATION OF SUBSCALE TESTING

  - MORE DEFINITIVE CONCLUSIONS
  - EXPEDITIOUS RETURN TO FLIGHT/TEST
  - BETTER PREDICTIONS
  - IMPROVED HDWR RELIABILITY/DURABILITY
  - REDUCTION OF LARGE-SCALE TEST REQTS
SOME CHALLENGES TO CFD

0 COMPARE HOT-FIRE PREDICTIONS WITH 3001 DATA

- UTILIZE DATA TO CALIBRATE/IMPROVE MODELS
- UTILIZE MODELS (+ KNOWLEDGE OF FLUID MECHANICS, etc.)
  TO EXPLAIN DATA

0 TAKE AN ACTIVE ROLE AND ENCOURAGE CONCURRENT ENGINEERING IN THE DESIGN PROCESS

- PLACEMENT OF SENSORS IN INSTRUMENTED TEST ARTICLES
- HARDWARE DESIGN (EARLY INVOLVEMENT, TIMELY INPUT)

0 EXAMPLES ...
HPFTP 2ND DISC AFT CAVITY AVERAGE TEMPERATURE 109% VS. 100% P.L.

TEMPERATURE (deg R)

- 109%
- 100%
- UKHD 100

AXIAL LOCATION
Three-dimensional basecase results: temperature.
HPFTP 2ND DISC AFT CAVITY AVERAGE
PRESSURE 109% P.L. VS. 100% P.L.

AXIAL LOCATION

PRESSURE (psig)
Three-dimensional basecase results: static pressure.
HIFTF 1st NOZZLE TEMPERATURES

PREBURNER BAFFLES

100% POWER LEVEL

t = 5 sec
TEST 801-023
(801-022)

LEGEND

- U/S TEMP
- U/S PRES
- D/S TEMP (Δ)
- D/S PRES
Nozzle Wall Static Pressure

Nozzle Wall Static Pressure

Throat centerline

Atmospheric pressure influences

Pressure, psia

Distance from thrust chamber throat, inches

Per Klems:

Increase associated with sea level, not expected at vacuum conditions.

To run CFD code to compare:

- TDK 100% Ab
- 100% - 021 Ab
- 100% - 023 Ab
- 100% - 025 Ab
- 100% - 026 Ab
- 100% - 027 Ab
- 100% - 028 Ab

The location of pressure increase is a surprise. It is farther upstream than expected.
3001 LESSONS LEARNED (CONT.)

DESIGN AND OPERATIONS

• ENGINE DESIGN/TESTING

INSTRUMENTATION:

• GLASS BRAIDED TYPE THERMOCOUPLE WIRE SHOULD BE REPLACED BY TEFILON COATED T/C WIRE TO AVOID UNRAVELING OF COATING DUE TO NORMAL FIELD OPERATION. (i.e. RUBBING WIRES DURING PUMP INSTALLATION)

• ADDITIONAL EFFORT SHOULD BE MADE ON INSTRUMENTATION PLACEMENT TO AVOID LOCATIONAL EFFECTS WHICH MISREPRESENT THE DESIRED DATA: (i.e. ELBOWS, STAGNATION AND RECIRCULATION REGIONS)

• EFFORT SHOULD BE MADE TO EQUALIZE SENSE LINE VOLUMES ON BOTH HIGH AND LOW SIDE OF DELTA PRESSURE TRANSDUCERS WHERE ACCURATE TRANSIENT DATA IS REQUIRED

• PRESSURE SENSE LINES REQUIRED TO MEASURE HOT GAS ENVIRONMENTS AND THAT ARE ROUTED "NEAR" COOLANT CIRCUITS SHOULD BE CONSIDERED FOR PURGING DURING TESTING TO PREVENT ICING.
HPOP PREBURNER PUMP BEARING COOLANT
FLOW TEMPERATURE 109% VS. 100% P.L.

TEMPERATURE (deg R)

160 180 200 220 240 260

DMP SL EX  BRG IN  BRG EX  PMP IN

AXIAL LOCATION

109%
100%
UXHDT

isolator value
probably more representative of brg ext.

may mean
so close to pip work
⇒ recirculating flow.
FUTURE PLANS

CONTINUATION OF PHASE II ENVIRONMENT CHARACTERIZATION
  • INSTRUMENTED RKDN TURBOPUMPS
  • P&W ALTERNATE TURBOPUMPS

CONTINUATION OF TECHNOLOGY ITEM INTEGRATION AND EVALUATION
  • HEALTH MONITORING SYSTEMS
  • TURBOMACHINERY
  • COMBUSTION DEVICES
  • INSTRUMENTATION
  • CONTROLLER

CHARACTERIZATION OF FUTURE PROPULSION SYSTEM DESIGNS
  • SSME PROTOTYPE BLOCK II (PHASE II+ POWERHEAD, ATDs, LTMCC)
  • SSME PRODUCIBILITY IMPROVEMENTS
  • HLLV STME PROTOTYPE
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ADVANCED ENGINE (0208)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DOWNTIME IN SUPPORT OF 17&quot; DISCONNECT</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ STD RKDN PUMPS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DOWNTIME IN SUPPORT OF PHASE II + HGM</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ INSTR RKDN PUMPS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ HYDROSTATIC BRG HPOP TP</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ INSTR P&amp;W FUEL PUMP AND INSTR RKDN LOX PUMP</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ 5ALOX CAGE/SPEED SENSOR HPOP T</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ INSTR P&amp;W PUMPS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3001 W/ SN BALLS/BEARING DEFLECTOMETER HPOP T</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E3003-PHASE III CHARACTERIZATION</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

TECHNOLOGY ITEMS:

- EXIT PLANE SPECTROSCOPY OF SSME
- PLUME TEMPERATURE MEASUREMENTS
- OPTICAL PLUME ANOMALY DETECTOR
- SODIUM RESONANT LINE ABSORPTION MEAS.
- SSME EXIT PLANE HOLOGRAPHY
- LASER INDUCED FLUORESCENCE
- SAFD
- TTB/HOSC EXPERT SYSTEMS
- NOZZLE OPTIC ASSEMBLY
- HPOTP PREBURNER PUMP END HYDROSTATIC BEARING RETROFIT
- LOW COST CONTROLLER
- OPTICAL PROPELLANT SENSING
- SOLID STATE H2/O2 SENSORS
- VORTEX SHEDDING FLOWMETER
- IMPROVED BEARING CAGE MATERIAL
- FABRY-PEROT SPECTROMETER
- NON-INTRUSIVE SPEED SENSOR
- ASYM. AND MAIN LEAK DETEC PRIOR TO TEST
- NON-INTRUSIVE (I.R.) GAS TEMP SENSOR
- SILICON NITRIDE BALLS
- BEARING DEFLECTOMETER
- HEAT FLUX SENSORS
- VPS BLADES
- ULTRASONIC FLOWMETER
- BRUSHLESS TORQUEMETER
- ELECTROMECHANICAL ACTUATORS
- PIEZO-ELECTRIC SENSOR AUTO-CALIBRATION
- ADVANCED MAIN COMBUSTION CHAMBER
SUMMARY

TECHNOLOGY TEST BED HAS PROVEN TO BE EFFECTIVE IN CHARACTERIZING PROPULSION SYSTEM DESIGNS

TECHNOLOGY TEST BED HAS PROVIDED VALUABLE HOT-FIRE DATA FOR INCREASED UNDERSTANDING OF THE INTERNAL OPERATING ENVIRONMENT OF THE SSME AND FOR CALIBRATION OF SSME MODELS

TECHNOLOGY TEST BED HAS ENHANCED SHUTTLE DEVELOPMENT TESTING AND ANOMALY RESOLUTION

TECHNOLOGY TEST BED HAS PROVIDED A VALUABLE PLATFORM FOR ASSESSMENT OF ADVANCED PROPULSION TECHNOLOGIES

TECHNOLOGY TEST BED FUTURE PLANS INCLUDE CHARACTERIZATION OF FORTHCOMING PROPULSION SYSTEM DESIGNS AND EVALUATION OF EMERGING PROPULSION TECHNOLOGIES
ADVANCED SOLID ROCKET MOTOR
PROJECT STATUS

Tenth Workshop for Computational Fluid Dynamics (CFD)
Applications in Rocket Propulsion

28 April 1992

Keith Coates
EE 71
MSFC
OUTLINE

- Project Objectives
- Team
- Locations
- Motor Design
- Schedule
- Technical Issues
PROJECT OBJECTIVES

- Improve System Safety and Reliability
  - Design Features
  - Enhanced Quality
  - Reproducibility

- Improve Shuttle Payload Performance: 12,000 lb

- Optimize Program Cost

- Promote Competitive SRM Industry
  - Construct and Operate Government Owned Manufacturing and Test Installations
ASRM PROJECT TEAM LOCATIONS

AEROGLEET
- PROPULSION DIVISION
  - IGN DESIGN & MFG
  - SYS ANAL
  - PILOT PLANT
  SACRAMENTO, CA

LMSC - CO MGMT
- SYSTEMS ANAL
  SUNNYVALE, CA

LMSC SANTA CRUZ
- IGNITION COMPONENTS
  SANTA CRUZ, CA

D3 TECH INC
- GSE DESIGN
  SAN DIEGO, CA

LOCKHEED AUSTIN DIV.
- GSE MFG
  - GSE ENGR MGMT
  - DATA ACQUISITION CONTROL SYSTEM
  AUSTIN, TX

MAF
- FACILITY CONSTRUCTION
  - NOZZLE MFG
  MICHOUD, LA

SSC - TEST PROJ MGMT
- FACILITY CONSTRUCTION
  - SYSTEMS TEST
  BAY ST. LOUIS, MS

IUKA/YELLOW CREEK
- PROJECT MGMT
  - FACILITIES CONSTR
  - MOTOR MFG
  IUKA, MS

MSFC - PROJ MGMT
- SYSTEMS ENGR
  - TESTING - HSTA
  STA
  TPTA
  48" MTR
  HUNTSVILLE, AL

RUST INTERNATIONAL
- FACILITIES A&E
  - CONSTRUCTION MGMT
  BIRMINGHAM, AL

KSC - GSE PROJ MGMT
- SYS OPS & LOGISTICS
  - LAUNCH OPERATIONS
  - RECOVERY OPERATIONS
  KENNEDY SPACE CENTER, FL

KAMAG/HULS AMERICA, INC
- KDT (TRANSPORTER)
  - VERTICAL BORING MILL
  PDPI-PROPELLANT
  CURATIVE
  ULM, GERMANY
ASRM MOTOR DESIGN HIGHLIGHTS

**INSULATION**
- Uses RSRM J-Seal Design at Field Joint
- Castable Inhibitor-to-Stress Relief Flap Joint Location Minimizes Potential for Jetting Into Sidewall Insulation
- Asbestos Free Formulation

**NOZZLE**
- Minimum Joints and Inlet/throat rings
- Improved process ablative materials
- Eliminates flexseal cowl and boot assembly

**CASE**
- Two Field Joints
- High Fracture Toughness
- High Stress Corrosion Resistance
- Welded Factory Joints
- Integral Stiffeners and ET Attach Ring
  - In Aft Segment Eliminates Failure Points
  - Experienced With Bolt-On Stiffeners and Ring

**PROPELLANT**
- Industry Proven HTPB Propellant with over 60M lbs Successfully Produced
- Formulated and Proven for Continuous Mix Process
- Positive Margins

**GRAIN/IGNITION**
- High Safety Margin in Forward Fin Trailing Edge
- Minimum Igniter Chamber Bolt Leak Paths
- Expendable Carbon Filament Chamber
- TBI Initiator Eliminates S&A Device Leak Paths
ASRM DESIGN PARAMETERS

- Diameter/Length, in: 150/1,513.43
- Average Thrust Vacuum, lbf; Web time: 2,654,776
- Delivered Isp Vacuum, sec: 268.1
- Area Ratio, (Ae/At): 7.48
- Motor Weight, lb: 1,351,092
- Propellant Weight, lb: 1,209,589
- Motor Propellant Mass Fraction, (Wp/Wt): 0.895
- Inert Weight, lb: 141,503
  - Metal Case Weight/Number of Segments, lb: 98,553/3
  - Single Nozzle Weight, lb: 18,800
- Solid Propellant Type: HTPB
- Average Chamber Pressure, psia; Action Time: 612
- Burn Rate at 625 psia, in/sec: 0.350
- Action Time, sec: 130.9
- Thrust Vector Control: Flexible Bearing
- Recovery/Reuse: Yes
CASE PREP BUILDING
CASE PREP AUTOCLAVE
ON-SITE FABRICATION AT YELLOW CREEK
MICHOUD PILING AND PERIMETER WALL
VERTICAL BORING MILL (VBM)
HEAT TREAT/CHILLER BUILDING
SSC ASRM TEST SITE
ASRM TECHNICAL TOP 5
March 27, 1992

1. Mix/Cast Construction, Outfitting & Process Verification
2. Soluble Casting Mandrels
3. Integration Effects; P_c Dot; Loads; Recovery; Overpressure; Moldlines
4. Low-Density Nozzle Ablatives Performance
5. Forward-Facing Cast Inhibitor
SOLUBLE CORE DESIGN
APRIL 28, 1992

Jan C. Monk
Chief Engineer
Space Transportation Main Engine
Marshall Space Flight Center
### STME DEFINITION

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Phase</td>
<td>STME Phase A</td>
<td>STME Advanced Development</td>
<td>STME Phase B</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>STAS</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ALS</td>
<td>ALS Phase I</td>
<td>ALS Phase II</td>
<td>ALDP</td>
<td>NLS</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- **Over 7500 STME Trades and Studies**
  - Engine System Level Trades
  - Engine Component Level Trades
  - Manufacturing Studies
  - Assembly Facilities Studies
  - Test Facilities Studies
  - Overhaul Facilities Studies

- **STME Design Concept Formed in Total System Context**
  - Interactive Engine/Vehicle/Operability Trades
  - Broad range of Vehicles and Missions
  - QFD established Strategic Quality Characteristics, Pugh Concept Selection
  - National Consensus Cycle Decision

- **Design Effort Supported by Extensive Data Base**
  - ADP Hardware
  - Lessons Learned: SSME, J-2, LR87, F-100, F-119, etc.
DESIGN PHILOSOPHY

- ROBUST DESIGN
- DESIGN FOR OPERABILITY
- DESIGN FOR RELIABILITY
- DESIGN FOR LOW COST
ROBUST DESIGN

- ACHIEVABLE REQUIREMENTS
  - Mission Life - 10
  - Chamber Pressure - 2250 psia
  - Weight - 9100 pounds
- SERIES TURBINES
  - Enhances benign system response to fuel turbopump failures
- DESIGN MARGINS
  - Design based on internal operating environment worst case plus development margin
- REDUCED INTERNAL ENVIRONMENTS TO ASSURE ROBUST PROCESSES
  - No sheet metal liners, no overlays, no platings
- NEAR NET SHAPE PROCESSES
  - Minimum number of welds, capable processes, reduced process steps
MDC = NOMINAL PARAMETER VALUE
+ RSS OF PREDICTED COMPONENT VARIATIONS
+ THRUST/MR CALIBRATION ERRORS
+ FLIGHT EFFECTS
+ DEVELOPMENT MARGIN

MAXIMUM DESIGN CONDITION

FUEL PUMP DISCHARGE PRESSURE

<table>
<thead>
<tr>
<th>Pressure (psia)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3400</td>
</tr>
<tr>
<td>3600</td>
</tr>
<tr>
<td>3800</td>
</tr>
<tr>
<td>4000</td>
</tr>
<tr>
<td>4200</td>
</tr>
<tr>
<td>4400</td>
</tr>
<tr>
<td>4600</td>
</tr>
<tr>
<td>4800</td>
</tr>
</tbody>
</table>

- 10% Development Margin
- Flight Effects
- F & MR Tolerance
- 2 Sigma Component
- Nominal
CASTINGS VS MACHINED AND WELDED FORGINGS

- Material costs $15,182
- Machine and welding 440,824

Total cost $456,006

SSME Turbopump Volute

I-718 cast Machining $14,527*

Total cost $20,700

IR&D Cast Volute

* Based on vendor quote for 6 parts
OPERABILITY CONSIDERATIONS

- TANK-HEAD START
  - No start valve or ducting
- ELECTROMECHANICAL ACTUATORS
  - Simplify pre-launch checkout
  - No pneumatics or hydraulics
- COMMONALITY FACILITATES INVENTORY AND ON-PAD REPLACEMENT
  - Electromechanical actuators/seals/fasteners
- REDUCTION OF POTENTIAL LEAK PATHS
  - Proven joint and seal designs
  - Reduced number of joints
- SIMPLE COMPONENT INSTALLATION
  - No stretch bolt joints
- AUTOMATED PRE-LAUNCH CONTROLS CHECKOUT
- MINIMUM FLIGHT-CRITICAL INSTRUMENTATION
HIGH RELIABILITY DESIGN PHILOSOPHY

• DESIGN FOR RELIABILITY IS AN INTEGRAL PART OF THE STME DESIGN PROCESS
  - Concurrent engineering
  - Design to reliability goals
  - Reliability lessons learned
  - Bottoms-up failure modes and effects analysis
  - Tops-down fault tree analysis
  - Reliability tracking
  - New manufacturing techniques
  - Reliability demonstration program
  - Development margin
ENGINE RELIABILITY ENHANCEMENT

- NON-INTRUSIVE OXIDIZER HEAT EXCHANGER (FOR TANK PRESSURIZATION)
- SIMPLE SAFETY MONITORING APPROACH. SAFE ENGINE SHUTDOWN TRIGGERED BY ABNORMAL VALUES OF:
  - Chamber pressure
  - Interpropellant seal purge pressure
  - Gas generator temperature
- MECHANICALLY LINKED GAS GENERATOR VALVES (TO PREVENT GG MIXTURE RATIO EXCURSIONS)
- PRUDENT USE OF REDUNDANCY IN CONTROL SYSTEM
  - Dual EMA motors and resolvers
  - Duplex / triplex controller electronics
  - Dual power source
LOW COST DESIGN PHILOSOPHY

- DESIGN-TO-COST IS AN INTEGRAL PART OF THE STME DESIGN PROCESS
  - New manufacturing techniques
  - Advanced Development Programs used to investigate low cost ideas
  - Suppliers integral part of design effort
  - Customer (Government) integral part of design effort
  - Costs continually estimated and tracked
  - Cost drivers identified and worked
  - Trade studies used to select lowest cost concepts
  - Zero RID'S
  - Zero MR'S
ENGINE SYSTEM REQUIREMENTS

Dual Thrust (Step)
  Normal Thrust (100%): 650,000 lbf
  Minimum Thrust (70%): 455,000 lbf
Mixture Ratio: 6.0
Specific Impulse (100%): 428.5 sec.
Specific Impulse (70%): 427.3 sec.
Chamber Pressure: 2250 psia
Dry Weight: 9100 lbm
Area Ratio: 45:1
Length: 160 inches
Nozzle Diameter: 96 inches
Design Life: 10 missions
Verified Reliability: 0.995
500th Unit Cost Goal: $5.3M
STME COMPARISONS

J-2
Gas Generator

STME
Gas Generator

SSME
Staged Combustion

<table>
<thead>
<tr>
<th></th>
<th>J-2</th>
<th>STME</th>
<th>SSME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thrust (klbf-vac)</td>
<td>225</td>
<td>650</td>
<td>489</td>
</tr>
<tr>
<td>Chamber Pressure (psia)</td>
<td>700</td>
<td>2250</td>
<td>3130</td>
</tr>
<tr>
<td>Fuel Pump Power Density</td>
<td>30</td>
<td>35</td>
<td>100</td>
</tr>
<tr>
<td>Thrust-to-weight Ratio</td>
<td>68</td>
<td>67</td>
<td>70</td>
</tr>
<tr>
<td>Specific Impulse (vac)</td>
<td>425</td>
<td>428</td>
<td>452</td>
</tr>
<tr>
<td>Mixture Ratio</td>
<td>5.5</td>
<td>6.0</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>27:1</td>
<td>45:1</td>
<td>77:1</td>
</tr>
</tbody>
</table>
STME SCHEMATIC

Rated Conditions

LH₂
- P = 45
- T = 37.5
- m = 217
- N = 24404

P = 3969
- T = 73

P = 2232
- T = 1600
- m = 74

ΔP = 1109
- P = 295
- T = 1190
- m = 74

Pc = 2250

Fuel Repress

GOX Repress

Internal Mixer

HEX

OTBP

MOV

LOX
- P = 130
- T = 166
- m = 1299
- N = 7913

F (vac) = 650Klb
Isp (vac) = 428.5
MR = 6.0
EPS = 45

Legend:
- Pump Stage
- Turbine Stage
- Shutoff Valve
- Dual Position Valve
- Orifice

LH₂
- Hot Gas

LOX
MAJOR ENGINE COMPONENTS

- Fuel Turbopump
- Oxidizer Turbopump
- Main Injector
- Main Combustion Chamber
- Gas Generator
- Nozzle
- Igniters
- Controller, Actuators, Valves
FUEL TURBOPUMP

BASELINE DESIGN FEATURES
- CAST HOUSINGS
- INTEGRALLY BLADED TURBINE
- CAST PUMP IMPELLERS
- HYDROSTATIC BEARINGS

OPTIONAL FEATURES
- LOW COST MACHINED IMPELLERS
- BACKUP DESIGNS FOR TURBINE AND BEARINGS

DEVELOPMENT STATUS
- FABRICATION PROCESSES IN TRIAL
- HYDROSTATIC BEARING RIG TESTING UNDERWAY
- SUPPLIERS ON CONCURRENT ENGINEERING TEAM

92 PLANS
- CONTINUED FAB DEVELOPMENT
- HYDROSTATIC BEARING TESTING

LEAD: STPT-ROCKETDYNE/CANOOGA PARK CA,
AEROJET/SACRAMENTO CA
## FUEL TURBOPUMP COMPARISON

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>198</td>
<td>60</td>
</tr>
<tr>
<td>Number of weld counts</td>
<td>801</td>
<td>1</td>
</tr>
<tr>
<td>Weld overlays</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Sheet Metal Liners</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Cost (TFU)</td>
<td>$4.1M</td>
<td>$1.02M</td>
</tr>
<tr>
<td>Discharge Pressure</td>
<td>6320 psia</td>
<td>3969 psia</td>
</tr>
<tr>
<td>Turbine Inlet Temperature</td>
<td>1860 deg R</td>
<td>1600 deg R</td>
</tr>
<tr>
<td>Speed</td>
<td>35180 rpm</td>
<td>23000 rpm</td>
</tr>
<tr>
<td>Weight</td>
<td>771 lbs</td>
<td>1877 lbs</td>
</tr>
</tbody>
</table>
LOX TURBOPUMP

- Vaneless diffuser mixes and equalizes inlet flow
- Volutes reduce turbine side loads and minimize flow disturbances
- Materials
  - Low Cost
- Turbine Airfoils
  - Hollow
  - Damped
  - Unshrouded
- Rotodynamics
  - Subcritical
  - Damper Seals
- Integral Disk & Shaft
  - Single forging
  - H2 resistant mat'l
- Cast Impeller
  - Low Tip Speed
- Axial Pump Inlet
- Splitter to reduce side loads
- Cast Housings
  - Turbine & Pump
- Bearings
  - Same at Both Locations
LOX TURBOPUMP

BASELINE DESIGN FEATURES
- CAST HOUSINGS
- SUB-CRITICAL SPEED DESIGN
- CAST PUMP IMPELLORS
- SINGLE PIECE FORGED DISK/SHAFT

OPTIONAL FEATURES
- LOW COST MACHINED IMPELLORS
- BACKUP DESIGNS FOR SHAFT AND BEARINGS

DEVELOPMENT STATUS
- FABRICATION PROCESSES IN TRIAL
- SPIN RIG INSTRUMENTATION UNDERWAY
- SUPPLIERS ON CONCURRENT ENGINEERING TEAM

92 PLANS
- CONTINUED FAB DEVELOPMENT
- SPIN TESTING OF IMPELLER

LEAD: STPT-PRATT & WHITNEY/WEST PALM BEACH FL
## LOX TURBOPUMP COMPARISON

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>153</td>
<td>87</td>
</tr>
<tr>
<td>Number of weld counts</td>
<td>128</td>
<td>0</td>
</tr>
<tr>
<td>Weld overlays</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Disk gold plating</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Cost (TFU)</td>
<td>5.16M</td>
<td>1.38M</td>
</tr>
<tr>
<td>Discharge Pressure</td>
<td>4360/7340 psia</td>
<td>3313 psia</td>
</tr>
<tr>
<td>Turbine Inlet Temperature</td>
<td>1510 deg R</td>
<td>1190 deg R</td>
</tr>
<tr>
<td>Speed</td>
<td>28200 rpm</td>
<td>7913 rpm</td>
</tr>
<tr>
<td>Weight</td>
<td>570 lbs</td>
<td>1712 lbs</td>
</tr>
</tbody>
</table>
MAIN INJECTOR

- Distribution/Filtration Plates
- Swirl Elements, Uniform Length
- Injector Body, Sand/AQD Casting
- Fuel Manifold Torus
- Torch Igniter
- Oxidizer Posts
- Regimesh Faceplate
- No Stability Aids
- Integral H2 Mixer
MAIN INJECTOR

BASELINE DESIGN FEATURES
- TWO MAJOR COMPONENTS (DOME AND BODY)
- BRAZED IN ELEMENTS
- 100 % INSPECTIBLE

OPTIONAL FEATURES
- ELEMENTS INTEGRAL TO BODY (SINGLE PIECE)

DEVELOPMENT STATUS
- FABRICATION PROCESSES IN TRIAL DEVELOPMENT
- 40K SUBSCALE AND UNIELEMENT TESTING INITIATED
- SUPPLIERS ON CONCURRENT ENGINEERING TEAM
- FIRST FULL SCALE TEST INJECTOR DELIVERED

92 PLANS
- PATTERN OPTIMIZATION
- 40K TEST FOR PERFORMANCE VERIFICATION
- 1ST FULL SCALE TEST (STABILITY DATA)

LEAD: STPT-AEROJET/SACRAMENTO CA
# Injector Comparison

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>&gt;3200</td>
<td>2213</td>
</tr>
<tr>
<td>Number of processes</td>
<td>170</td>
<td>151</td>
</tr>
<tr>
<td>Number of welds</td>
<td>&gt;360</td>
<td>13</td>
</tr>
<tr>
<td>Number of inspections</td>
<td>90</td>
<td>58</td>
</tr>
<tr>
<td>Cost (TFU)</td>
<td>2.71M</td>
<td>0.88M</td>
</tr>
<tr>
<td>Chamber Pressure</td>
<td>3126 psia</td>
<td>2250 psia</td>
</tr>
<tr>
<td>Weight</td>
<td>394 lbs</td>
<td>1339 lbs</td>
</tr>
</tbody>
</table>
COMBUSTION CHAMBER

Fuel In

VPS
NARloy-Z
Layer 1

VPS
NARloy-Z
Layer 2

Vacuum Plasma
Sprayed NARloy-Z
Coolant Liner

Cast JKK-75
Jacket, Manifold,
Forward Flange

Fuel Out to Injector
COMBUSTION CHAMBER

BASELINE DESIGN FEATURES
- CAST STRUCTURAL JACKET AND MANIFOLD
- VACUUM PLASMA SPRAY NARLOY Z LINER

OPTIONAL PROCESSES
- LIQUID INTERFACE DIFFUSION BOND ASSY
- PLATELET LINER

DEVELOPMENT STATUS
- FABRICATION PROCESSES IN TRIAL
- 1ST FAB DEVELOPMENT JACKET CAST @ PCC -PORTLAND, OR
- 1ST LIDB CHAMBER FAB COMPLETE
- VPS SPRAY TESTS UNDERWAY

92 PLANS
- LIDB CHAMBER HOT FIRE TEST WITH LSI
- CONTINUE CASTING AND VPS DEVELOPMENT

LEAD: STPT-ROCKETDYNE/CANOOGA
## COMBUSTION CHAMBER COMPARISON

### SSME

![Combustion Chamber Image]

### STME

![Combustion Chamber Image]

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>60</td>
<td>6</td>
</tr>
<tr>
<td>Number of welds</td>
<td>96 (incl. overlays)</td>
<td>4</td>
</tr>
<tr>
<td>Coating operations</td>
<td>7 (plating)</td>
<td>2 (vps)</td>
</tr>
<tr>
<td>Cost</td>
<td>4300K</td>
<td>800K</td>
</tr>
<tr>
<td>Chamber Pressure</td>
<td>3126 psia (104%)</td>
<td>2250 psia</td>
</tr>
<tr>
<td>Weight</td>
<td>466 lbs</td>
<td>1834 lbs</td>
</tr>
</tbody>
</table>
CAST NOZZLE COOLANT TURBINE MANIFOLD

TURBINE DRIVE EXHAUST GAS

STIFFENERS

ATTACHMENT CONE

HEAT SHIELD

INFLATION FORMED / DIFFUSION BONDED TUBES, JACKET

SECONDARY PRIMARY COOLANT FILM INJECTION

CONVECTIVE COOLANT GAS OUT TO AMBIENT

TUBULAR SKIRT / JACKET

SECTION A-A

- ALL MATERIALS ARE INCO 625
NOZZLE

DESIGN FEATURES
- ALL COMPONENTS MADE OF SAME ALLOY
- TUBULAR SKIRT/JACKET - NO WELDS
- BOLT TO CHAMBER
- CAST MANIFOLDS

DEVELOPMENT STATUS
- FABRICATION PROCESSES IN TRIAL
- BASELINE TO BE SELECTED EARLY 92
- SUBSCALE NOZZLE IN FABRICATION

92 PLANS
- SUBSCALE NOZZLE HOT FIRE TESTING
- DESIGN UPDATE AND CONTINUED FAB DEVELOPMENT

LEAD: STPT-PRATT & WHITNEY/W. PALM BEACH FL
# NOZZLE COMPARISON

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>1600</td>
<td>611</td>
</tr>
<tr>
<td>Number of weld counts</td>
<td>113</td>
<td>33</td>
</tr>
<tr>
<td>Number of processes</td>
<td>520</td>
<td>207</td>
</tr>
<tr>
<td>Number of inspections</td>
<td>633</td>
<td>34</td>
</tr>
<tr>
<td>Cost (TFU)</td>
<td>5.6M</td>
<td>1.56M</td>
</tr>
<tr>
<td>Weight</td>
<td>1328 lbs</td>
<td>1945 lbs</td>
</tr>
</tbody>
</table>
GAS GENERATOR

DESIGN FEATURES
- UNCOOLED CHAMBER
- OPERATING PRESSURE SAME AS MAIN CHAMBER

DEVELOPMENT STATUS
- WORKHORSE GAS GENERATOR TESTED
- DESIGN OPTIONS IDENTIFIED

92 PLANS
- IGNITER DESIGN • CONCEPTUAL GAS GENERATOR DESIGN

LEAD: STPT-AEROJET/SACRAMENTO CA
### GAS GENERATOR/PREBURNER COMPARISON

#### STME

![STME diagram]

#### SSME

![SSME diagram]

<table>
<thead>
<tr>
<th>Discriminators</th>
<th>SSME*</th>
<th>STME</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of parts</td>
<td>591</td>
<td>70</td>
</tr>
<tr>
<td>Number of weld counts</td>
<td>20</td>
<td>5</td>
</tr>
<tr>
<td>Weld overlays</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>Sheet Metal Liners</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>Cost (TFU)</td>
<td>1.17</td>
<td>243 K</td>
</tr>
<tr>
<td>Discharge Pressure</td>
<td>5460 PSIA</td>
<td>2205 PSIA</td>
</tr>
<tr>
<td>Turbine Inlet Temperature</td>
<td>1900 R</td>
<td>1600 R</td>
</tr>
<tr>
<td>Weight</td>
<td>143 LBS</td>
<td>111 LBS</td>
</tr>
</tbody>
</table>

* Fuel Preburner Only
SPACE TRANSPORTATION PROPULSION TEAM (STPT)

Senior Executive Council
- "Board of Directors"
- Delegates Authority to team
- Establishment of Policy
- Provision of Overall Guidance
- Final Resolution for Team Conflicts
- One Senior Executive from each Company

Team Program Office (TPO)
- Project Director is Final Authority
- Overall Program Management Responsibility
- Principal Director from each Company

Aerojet Propulsion Division
ALS/STME Program
- Thrust Chamber Assembly (L)
- Control System (L)
- Injector (L)
- Gas Generator
- Igniter
- Engine Systems D&D
- Fuel Turbopump (Turbine)
- System Components

Pratt & Whitney Space Propulsion & Systems
ALS/STME Program
- Engine Systems D&D (L)
- Oxidizer Turbopump (L)
- Nozzle (L)
- Engine Systems Assy & Test
- Control Systems
- System Components

Rocketdyne Division
ALS/STME Program
- Fuel Turbopump Assy (L)
- Main Combustion Chamber (L)
- Engine System D&D
- Engine Systems Test
- Control Systems
- System Components

Team Project Director is Final Authority
Senior Executive Overall Program Management Council Principal Director from each Company

One Senior Executive from each Company

Aerojet Propulsion Division
ALS/STME Program

Pratt & Whitney Space Propulsion & Systems
ALS/STME Program

Rocketdyne Division
ALS/STME Program
SPACE TRANSPORTATION MAIN ENGINE PARTICIPANTS

- AEROJET
- PHILLIPS LABORATORY
- ROCKETDYNE
- LEWIS RESEARCH CENTER
- SPACE TRANSPORTATION PROPULSION TEAM HEADQUARTERS
- MARSHALL SPACE FLIGHT CENTER
- STENNIS SPACE CENTER
- PRATT & WHITNEY
PROGRAM STATUS

THRUST LEVEL INCREASED TO 650K
- REQUIRED TO SUPPORT 20K VEHICLE
- DESIGNS BEING REVISED TO REFLECT INCREASE

FULL SCALE DEVELOPMENT
- REQUEST FOR PROPOSALS IN PREPARATION
- SCHEDULE FOR RELEASE - JUNE 1992
- FIRST ENGINE SYSTEM TEST - MAY 1996
SUMMARY

- STME IS PROCESS FOCUSED
- PRODUCT OF INTEGRATED SYSTEMS PROCESS
- INNOVATIVE APPROACH TO PROJECT
- DEFINITION MATURE
- BENEFITS
  - Robust Propulsion
  - Legacy of New Process
A large fraction of the Navier-Stokes codes in use today are based on the so-called 'time-marching' procedures, wherein the unsteady form of the governing equations are solved in time. For compressible flows, these methods perform very well in the transonic and supersonic flow regimes and have been applied to solving a wide variety of problems. There are, however, several disadvantages associated with these methods. It is well known that at low Mach numbers, the convergence of these schemes deteriorate dramatically. The reason for the behaviour is the wide disparity in the eigenvalues of the system at low speeds. Furthermore, highly viscous regions of the flow and the presence of strong source terms also introduce convergence difficulties. These again are due to the very disparate time scales involved in these processes.

Preconditioning offers a means of controlling the time-step size for a wide variety of flow situations. Originally, preconditioning methods were developed as a means of circumventing the disparity in the eigenvalues at low Mach numbers. Essentially, this involves altering the time derivative of the equations of motion such that the acoustic speed is scaled down to the level of the fluid velocity. This 'inviscid' preconditioning enables Mach number-independent convergence to be obtained. We have also extended the preconditioning approach to handling very viscous flows. Here, the acoustic speed is altered such that the local CFL number is approximately the same order as the viscous time step. This enables excellent convergence rates to be maintained over a wide range of Reynolds numbers. We are currently investigating extending this approach to various source terms of interest—particularly related to reacting flowfields.

We have implemented preconditioning for multi-species reacting flows in two independent codes—an implicit (ADI) code developed in-house and the RPLUS code (developed at NASA-Lewis Research Center). The RPLUS code has been modified to work on a 4-stage Runge-Kutta scheme. The performance of both the codes have been tested and show that preconditioning can improve convergence by a factor of two to a hundred depending on the problem. Our efforts are currently focussed on evaluating the effect of chemical sources and on assessing how preconditioning may be applied to improve convergence and robustness in the calculation of reacting flows.
The Impact of Time-Step Definition on Code Convergence and Robustness

S. Venkateswaran
Jonathan M. Weiss
Charles L. Merkle

Propulsion Engineering Research Center
The Pennsylvania State University
University Park, PA 16802
Problems Associated with Conventional Codes

- Convergence is poor at low Mach Numbers.
  - e.g., the combustion zone in rocket engines.

- Convergence is poor in viscous regions.
  - e.g., boundary layers, recirculation zones, etc.

- Large source terms induce instabilities.
  - e.g., combustion, turbulence, axisymmetry, etc.

- High aspect ratio grids cause poor convergence.
  - in regions where strong local grid stretching is used.
Time-Step Definition in Conventional Codes

- Eigenvalues of Jacobian A define inviscid time-step.
  — Eigenvalues are $u+c$, $u-c$, $u$, $u$, $u$, etc.

\[ CFL_\lambda = \frac{\lambda \Delta t}{\Delta x} \]

- In viscous regions, a viscous time-step is defined.

\[ VNN = \frac{\nu \Delta t}{\Delta s^2} \]

- Time step is fixed based on $CFL$ and $VNN$ conditions.

- Variable time stepping is normally used.
Problems with Conventional Definition

- At low Mach numbers, $CFL_{u+c} \gg CFL_u$.
  
  — Acoustic wave speeds dominate over particle convection speed.

- In viscous regions, $VNN \gg CFL$.
  
  — Diffusion time scales are much larger than wave speeds for acoustic and particle convection.

- For high aspect ratio cells, $CFL_{u+c} \gg CFL_{u+c}$.
  
  — Wave speeds are much higher in the direction normal to the flow direction.

- Source terms introduce additional time scales.
Preconditioning the Equations of Motion

- Alter the time derivative by multiplying with a preconditioning matrix $\Gamma$.

\[ \Gamma \frac{\partial Q_v}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = H + \frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} \]

— $Q_v = (P, u, v, T, Y_1, Y_2, \ldots )$

- Steady state solution remains unaltered.

- Eigenvalues of $\Gamma^{-1}A$ now define CFL number.
Choosing the Preconditioning Matrix

- Define \( \Gamma \) so that the acoustic speeds are altered.

- At low Mach numbers, keep acoustic speeds of the same order as fluid velocities.

- In viscous regions, alter acoustic speed so that the inviscid time-step is of the same order as the viscous time-step.

- Scale acoustic speed in a similar manner to account for large source terms.

- Extend philosophy to high aspect ratio grid cells.
Example of Preconditioning

Large Area Ratio Nozzle

AREA RATIO = 150:1
Comparison of Convergence

PCOMAXIVN (150:1) INV NOZZLE
CFL=4 71X61

RESIDUAL

No Preconditioning

With Preconditioning

ITERATION STEP
Comparison of Solutions
Implementation of Preconditioning

- Incorporate preconditioning in implicit, reacting flow code.
  - Euler Implicit/ADI Algorithm
  - Two D/Axisymmetric Code
  - Multi-Component Species Transport
  - Multi-Step Finite Rate Chemistry

- Incorporate preconditioning into RPLUS code.
  - Developed at NASA-Lewis Research Center.
  - Multi-Stage Explicit Runge-Kutta (RPLUS/RK)
Preconditioning in the RPLUS Code

• Modified Runge-Kutta Stage:

\[ \Delta Q_v = -\alpha_k \Delta t \Gamma^{-1} \left( \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} - H - \frac{\partial E_v}{\partial x} - \frac{\partial F_v}{\partial y} \right) \]

• Time-Step definition altered using the new eigenvalues of the system.
Convergence of RPLUS/RK

Flat Plate Boundary Layer, $M = 0.30$

Density Residual vs. Iteration

- No Preconditioning
- Preconditioning

$Re = 1,000$
$Re = 10,000$

26 Apr 92 20:27:28
Convergence of RPLUS/RK

Flat Plate Boundary Layer, $M = 0.01$

No Preconditioning

-7.0

-9.0

-11.0

-13.0

-15.0

Iteration

Density Residual

Preconditioning

$Re = 1,000$

$Re = 10,000$

$Re = 1,000$

$Re = 10,000$

26 Apr 92 20:19:34
Computation of Reacting Shear Layer

Grid Geometry

$H_2O$ Mass Fraction
Convergence of RPLUS/RK

Multi-Species Shear Layer

No Preconditioning

Preconditioning

M = 0.10

M = 0.05

Density Residual

Iteration

0.  1000.  2000.  3000.

26 Apr 92 20:38:56
Convergence of RPLUS/RK

Reacting Shear Layer

Preconditioning

$M = 0.10$

Density Residual

Iteration

0.  1000.  2000.  3000.  4000.  5000.
Convergence of Implicit-ADI

\[ H_2-O_2, \quad M = 0.1 \]

\[ \text{Re} = 10^2, \quad \text{Re} = 10^3, \quad \text{Re} = 10^4, \quad \text{Re} = 10^5 \]
Convergence of Implicit ADI

Comparison of convergence between single and multiple species

$\text{H}_2-\text{O}_2, \text{Re}=10$

$\text{O}_2, \text{Re}=10$

$\text{H}_2-\text{O}_2, \text{Re}=1.0E5$

$\text{O}_2, \text{Re}=1.0E5$

$y$

$\text{Residual}$

$x$

Iteration step
Definition of Time-Step

\[ \Delta t = \frac{CFL \Delta x}{\lambda} \]

- Based on the maximum eigenvalue:

\[ \lambda = Max \left( u + c, \ v + c \right) \]

- Based on an average eigenvalue:

\[ \lambda = \sqrt{(u + c^2) + (v + c^2)} \]

\[ \lambda = 1/2( u + c + v + c) \]

- Based on the eigenvalue in the direction of flow:

\[ \lambda = u + c \]
2D Convergence

Maximum Eigenvalue

CFL=4 (Max ev)

71X121

71X61

71X31

Iteration Step

0 200 400 600 800 1000 1200 1400

-17.0 -15.0 -13.0 -11.0 -9.0 -7.0 -5.0 -3.0 -1.0

Residual, ΔQ/Q

X-Eigenvalue

CFL=4

71X31

71X61

71X121

Iteration Step

0 200 400 600 800 1000 1200 1400

-17.0 -15.0 -13.0 -11.0 -9.0 -7.0 -5.0 -3.0 -1.0
Axisymmetric Convergence

Maximum Eigenvalue

X-Eigenvalue

CONVERGENCE

CONVERGENCE WITH CFL BASED ON X_Eigenvalue

RESIDUAL, $\Delta Q/Q$

RESIDUAL, $\Delta Q/Q$

CFL=4

CFL=5

71 x 121

71 x 61

71 x 31

AXISYMMETRIC

AXISYMMETRIC

0. 200. 400. 600. 800. 1000. 1200. 1400.

0. 200. 400. 600. 800. 1000. 1200. 1400.

ITERATION STEP

ITERATION STEP
Convergence Based on X-Eigenvalue

Axisymmetric

CFL (Based on X_Eigen_Value) = 2

RESIDUAL ΔQ/Q

71X601 CFLY=30
71X241 CFLY=12
71X61 & 71X121 CFLY=8

ITERATION STEP
**Time Step Definition**

- Important to use proper eigenvalue in CFL.
  - $u+c$
  - $\text{Max}(u+c,v+c)$
  - $\sqrt{(u + c)^2 + (v + c)^2}$

- Preliminary results for H-grids show:
  - best choice is $u+c$
  - control convergence with grid refinement
  - control convergence in near wall region

- Additional work needed to generalize
Conclusions

- The definition of time-step has a profound impact on the performance of time-marching codes.

- Preconditioning is a powerful method of controlling the time-step.
  - Low Mach number preconditioning or characteristic time-stepping has been used widely.
  - Preconditioning has been successfully extended to viscous dominated flows.
  - Similar extensions are currently being investigated for combustion and other sources of interest.

- Time-step should be defined based on the eigenvalue in the direction of flow.
  - Important when the grid aspect ratio is very high.
Development of CFD Code Evaluation Criteria and a Procedure for Assessing Predictive Capability and Performance

S.J. Lin, D.C. Chan, M.M. Sindir, and S.L. Barson
Rockwell International, Rocketdyne Division
Canoga Park, California

Careful validation of Computational Fluid Dynamic codes is essential if they are to be used as engineering design tools. Validation must be carried out in a systematic manner to ensure that all code aspects as they apply to the application of interest are understood and, to the greatest extent possible, quantified.

A study is being conducted in which a general code validation procedure is defined and demonstrated. A four phase validation procedure is defined in which a series of validation test cases are computed and compared with available analytical solutions and test data. The procedure is demonstrated using the REACT CFD code to compute validation cases for each of the four phases. For phase 4, the application of interest, the SSME high pressure fuel turbopump impeller flowfield is computed.
DEVELOPMENT OF CFD CODE EVALUATION CRITERIA AND PROCEDURE FOR ASSESSING PREDICTIVE CAPABILITY AND PERFORMANCE

S.J. Lin, D.C. Chan, M.M. Sindir, and S.L. Barson
Rockwell International, Rocketdyne Division

Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion

April 28-30, 1992
NASA Marshall Space Flight Center
DEVELOPMENT OF CODE EVALUATION CRITERIA AND A PROCEDURE

- TASK OBJECTIVES
  - PROVIDE CODE EVALUATION CRITERIA, CLASSIFICATION SCHEME, NUMERICAL ERROR ASSESSMENT TECHNIQUES, AND A PROCEDURE FOR COMPREHENSIVE CODE EVALUATION AND CERTIFICATION
  - ENSURE INTEGRITY, ACCURACY, AND APPLICABILITY OF CFD CODES
  - PROVIDE PROCEDURES AND GUIDELINES FOR CFD SOFTWARE QUALITY CONTROL
  - DEMONSTRATE CODE EVALUATION PROCEDURE USING 2-D AND 3-D BENCHMARK EXPERIMENTS.

- PRESENTATION FOCUS
  - CODE VALIDATION PROCEDURE
  - DEMONSTRATION OF PROCEDURE
THOUGHTS ON VALIDATION

• GENERAL VALIDATION PROCEDURE FOR ALL APPLICATIONS IS POSSIBLE

• NO GENERAL AND ABSOLUTE VALIDATION POSSIBLE FOR ALL CASES

• QUANTITATIVE VALIDATION ONLY MEANINGFUL WITHIN A LIMITED CLASS OF APPLICATIONS

• LEVEL OF VALIDATION DEPENDS ON FINAL APPLICATION

• VALIDATION PROCESS MUST BE REALISTICALLY ACHIEVABLE
## FOUR PHASE CODE VALIDATION PROCEDURE DEFINED

<table>
<thead>
<tr>
<th>PHASE 1</th>
<th>PHASE 2</th>
<th>PHASE 3</th>
<th>PHASE 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT PROBLEM</td>
<td>BENCHMARK CASES</td>
<td>SIMPLIFIED PARTIAL FLOWPATH</td>
<td>ACTUAL HARDWARE</td>
</tr>
<tr>
<td>• SINGLE FLOW FEATURE</td>
<td>• MORE THAN ONE FLOW FEATURE</td>
<td>• MULTIPLE RELEVANT FLOW FEATURES</td>
<td>• COMPLETE FLOW PHYSICS</td>
</tr>
<tr>
<td>• ANALYTIC SOLUTION OR HIGH FIDELITY COMPUTATIONAL SOLUTION (DNS) AVAILABLE</td>
<td>• SIMPLE FLOW PHYSICS</td>
<td>• ACTUAL FLOW PHYSICS</td>
<td>• HARDWARE TEST DATA</td>
</tr>
<tr>
<td></td>
<td>• BENCHMARK EXPERIMENT DATA</td>
<td>• HIGH QUALITY TEST DATA</td>
<td></td>
</tr>
</tbody>
</table>

**INCREASING GEOMETRIC AND FLOW COMPLEXITY**

<table>
<thead>
<tr>
<th>PHASE 1</th>
<th>PHASE 2</th>
<th>PHASE 3</th>
<th>PHASE 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>RUN UNIT PROBLEMS</td>
<td>RUN BENCHMARK CASES</td>
<td>RUN SIMPLIFIED PARTIAL FLOWPATH</td>
<td>RUN ACTUAL CONFIGURATION</td>
</tr>
<tr>
<td>• VERIFY INTEGRITY</td>
<td>• ASSESS PHYSICAL MODELS</td>
<td>• ASSESS AGREEMENT WITH DATA</td>
<td>• COMPARE WITH TEST DATA</td>
</tr>
<tr>
<td>• ASSESS ACCURACY, CONVERGENCE, AND FUNCTIONALITY</td>
<td>• ESTABLISH GRID DISTRIBUTION REQUIREMENTS</td>
<td>• ESTABLISH GRID DISTRIBUTION REQUIREMENTS</td>
<td></td>
</tr>
</tbody>
</table>

**DECREASING DATA AVAILABILITY AND ACCURACY**

---

Rockwell International
Rockwell Division
DECREASING NUMBER OF CASES REQUIRED FOR LATTER VALIDATION PHASES

UNIT PROBLEMS

BENCHMARK CASES

SIMPLIFIED PARTIAL FLOWPATHS

ACTUAL HARDWARE

FUTURE APPLICATIONS
PROBLEM OF INTEREST SUCCESSIVELY DECOMPOSED INTO LESS COMPLEX CASES

EXAMPLE: SSME HPFTP IMPELLER

<table>
<thead>
<tr>
<th>PHASE 1</th>
<th>PHASE 2</th>
<th>PHASE 3</th>
<th>PHASE 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>UNIT PROBLEMS</td>
<td>BENCHMARK CASES</td>
<td>SIMPLIFIED FLOWPATHS</td>
<td>ACTUAL HARDWARE</td>
</tr>
<tr>
<td>flat plate</td>
<td>square duct with 90° bend</td>
<td>3-D turbine blade cascade</td>
<td>SSME HPFTP impeller (2 sets</td>
</tr>
<tr>
<td>straight duct</td>
<td>S-shaped duct</td>
<td>backward facing step (turb.)</td>
<td>partial blades)</td>
</tr>
<tr>
<td>diffuser</td>
<td></td>
<td>orifice flow (turb.)</td>
<td></td>
</tr>
<tr>
<td>sudden contraction (lam.)</td>
<td></td>
<td>flow around confined bluff</td>
<td></td>
</tr>
<tr>
<td>backward facing step (lam.)</td>
<td></td>
<td>bodies</td>
<td></td>
</tr>
<tr>
<td>driven cavity</td>
<td></td>
<td>2-D turbine cascade</td>
<td></td>
</tr>
<tr>
<td>rotating concentric cylinders (Taylor-Couette flow)</td>
<td></td>
<td>rotating disk</td>
<td></td>
</tr>
</tbody>
</table>

Rockwell International Rocketdyne Division
REACT* CODE DESCRIPTION

- CO-DEVELOPED BY ROCKETDYNE/UNIVERSITY OF LONDON
- 2-D/3-D, STEADY STATE OR TRANSIENT, FULL NAVIER-STOKES
- MULTI-ZONE FINITE-VOLUME IN GENERALIZED COORDINATES
- PRESSURE-VELOCITY COUPLING THROUGH "SIMPLE" AND "PISO"
- STONE'S STRONGLY IMPLICIT AND CONJUGATE GRADIENT SOLVERS
- VARIOUS 2-EQUATION TURBULENCE MODELS
- CONJUGATE FLUID-SOLID HEAT TRANSFER CAPABILITY
- MULTI-SPECIES CAPABILITY
- PRIMARY USE FOR TURBOMACHINERY APPLICATIONS

* Rocketdyne Elliptic Analysis Code for Turbomachinery
UNIT PROBLEMS COMPUTED FOR PHASE 1
DUCT FLOW

Verify Single Zone and Multizone Convergence

Establish Multizone Grid Matching Requirements

Non-smooth Interface

Smooth Interface

Refined Grid Computations Approach Exact Solution
BENCHMARK CASES COMPUTED FOR PHASE 2
CURVED DUCT FLOW

90° Bend With Square Cross Section

Refined Grid Yields Improved Agreement With Experimental Data

Secondary Flow Features Captured

Pressure Contours

Velocity Profiles At 77.5° Location

PARTIAL FLOWPATH COMPUTED FOR PHASE 3
HPFTP FIRST STAGE TURBINE CASCADE

- STUDY EFFECTS OF SINGLE ZONE H GRID AND MULTIZONE O-H GRID
- PERFORM COARSE AND FINE GRID COMPUTATIONS ON EACH AND COMPARE WITH TEST DATA

SSME HPFTP First Stage Stator
Multizone O-H Grid

Static Pressure At Stator Mid-section
Fine Grid Solution

Rockwell International
Rocketdyne Division

CFD 82-030-018/02/SL.0
COMPUTATIONS ON ACTUAL HARDWARE CONFIGURATION IN PROGRESS FOR PHASE 4

- PROBLEM FEATURES
  - HIGHLY THREE-DIMENSIONAL
  - TWO SETS OF PARTIAL BLADES
  - STRONG CURVATURE
  - HIGH ROTATIONAL SPEEDS
  - TURBULENT FLOW

- MODELING APPROACH
  - 3-D MULTIZONE MODEL
  - k-ε TURBULENCE MODEL

- STATUS AND PLANS
  - COARSE GRID SOLUTION COMPLETED WITH ASSUMED INLET CONDITION
  - TWO FINE GRID COMPUTATIONS PLANNED
    - ASSUMED INLET CONDITION
    - INLET CONDITION FROM TEST DATA
  - DATA ACQUISITION IN PROGRESS
    - INLET FLOW DATA NOW AVAILABLE
    - OUTLET DATA AVAILABLE SOON
GENERAL VALIDATION PROCEDURE DEFINED FOR ALL APPLICATIONS

- FOUR PHASE PROCEDURE OUTLINED
- QUANTITATIVE VALIDATION ONLY MEANINGFUL WITHIN LIMITED CLASS OF APPLICATIONS
- CRITERIA BEING DEFINED

REALISTIC VALIDATION PROCEDURE DEMONSTRATED ON ACTUAL HARDWARE

- SSME HPFTP IMPELLER
- DATA ACQUISITION IN PROGRESS
- FINAL COMPUTATIONS AND DATA COMPARISONS TO FOLLOW
Two widely used family of algorithms, pressure-based and density-based methods, have been developed for CFD problems over the years. Pressure-based methods (such as SIMPLE and PISO) use a Poisson-like equation for updating pressure instead of the continuity equation, while density-based methods use the continuity equation to update density (an equation of state is used to provide density in pressure based schemes and pressure in density based schemes). Pressure-based methods were developed originally for incompressible flows at low Reynolds numbers and were then extended to high Reynolds numbers and compressible applications. On the other hand, density based methods were originally developed for transonic flows and have been extended down to low Mach numbers through the use of preconditioning techniques. Both methods have enjoyed considerable success in solving complex flowfields, though the relative effectiveness of the schemes has long been argued. Generally, pressure-based methods are more robust while density-based schemes are more temperamental but provide more accurate solutions.

In the present paper, we compare these two very different approaches to solving the Navier-Stokes equations in order to gain an understanding of their similarities and differences. Specifically, we consider the PISO scheme as a representative pressure-based method and contrast it with a recently developed preconditioning scheme. To facilitate the comparison, we write both schemes in a vector formulation. Our findings indicate that the PISO scheme is very closely related to the philosophy of the preconditioning scheme. In particular, preconditioning causes the density-based scheme to appear pressure-based at low speeds but to remain density-based at high speeds. Furthermore, both schemes alter the sonic speed so that the equations stay well conditioned in the limit of low Mach numbers.

We also compare the relative performance of the PISO algorithm with an Euler implicit algorithm that is employed to solve the preconditioned equations by means of a vector stability analysis. The results of the stability analysis indicate that the PISO algorithm, which is a multi-step (one predictor step followed by several corrector steps), uncoupled (i.e., sequential) solution procedure, is conditionally stable. Good convergence is promised at low CFL numbers, while at high CFL numbers, both low wave number and high wave number instabilities are present. The high wave number instability appears to be ‘compressible’ in origin, arising from the treatment of the equation of state. The low wave number instability is ‘incompressible’ in origin since it is present when the incompressible limit of the equations are examined. An important finding, in this regard, is that the overall scheme may be unstable even when the individual predictor and corrector stages are themselves stable. In contrast, the Euler implicit algorithm shows unconditional stability. It should be noted, however, that multi-dimensional solution of the equations demands the use of approximate factorization which limits CFL numbers to about 10. Thus, the two algorithms still remain quite competitive in solving practical flow problems.
The Relationship Between Pressure- and Density-Based Algorithms

Charles L. Merkle, Sankaran Venkateswaran
and Philip E. O. Buelow
The Pennsylvania State University
Propulsion Engineering Research Center
Department of Mechanical Engineering

Presented at

Computational Fluid Dynamics Workshop
Marshall Space Flight Center
April 28-30, 1992
Introduction

Compare Pressure-Based and Density-Based Methods

Pressure-Based
- SIMPLE, MAC, PISO etc.
- Replace Continuity by Poisson Equation
- Solve by Sequential Procedure

Density-Based
- ADI, LU, Lax-Wendroff
- Solve Continuity Directly
- Solve by Simultaneous, Coupled Procedure

Express in Common Vector Form for Comparison
Development of Pressure Poisson Relation

Use Continuity with Source:

\[ \nabla \cdot \mathbf{V} = D \]

Discretize Momentum:

\[ \frac{u^{n+1} - u^n}{\Delta t} + \left( \frac{\partial u^2}{\partial x} + \frac{\partial u v}{\partial y} + \frac{\partial p}{\partial x} \right)^* = 0 \]

\[ \frac{v^{n+1} - v^n}{\Delta t} + \left( \frac{\partial u v}{\partial x} + \frac{\partial v^2}{\partial y} + \frac{\partial p}{\partial y} \right)^* = 0 \]

Take Divergence of Momentum and Combine with Continuity:

\[ \nabla^2 p^* + \sigma^* + \frac{1}{\Delta t} (\nabla \cdot \mathbf{V})^{n+1} - \frac{1}{\Delta t} (\nabla \cdot \mathbf{V})^n = 0 \]

where:

\[ \sigma = \frac{\partial^2 u^2}{\partial x^2} + 2 \frac{\partial^2 u v}{\partial x \partial y} + \frac{\partial^2 v^2}{\partial y^2} \]
Solution of Poisson Equation

Solve by Point Jacobi with OverRelaxation:

- Express as Equivalent Time Marching

\[
\frac{4\Delta t}{\omega \Delta x^2} \frac{\partial p^{n+1}}{\partial t} = \nabla^2 p^* + \sigma^* - \frac{1}{\Delta t} (\nabla \cdot \mathbf{V})^n
\]

where: \((\nabla \cdot \mathbf{V})^{n+1} = 0\)

By Comparison of Equations:

\[
\frac{4\Delta t}{\omega \Delta x^2} \frac{\partial p^{n+1}}{\partial t} = \frac{1}{\Delta t} (\nabla \cdot \mathbf{V})^{n+1} = \frac{1}{\Delta t} D^{n-1}
\]

Hence the Equivalent Equation Can Be Written

\[
\frac{4\Delta t^2}{\omega \Delta x^2} \frac{1}{\Delta t} \left[ \frac{\partial p^{n+1}}{\partial t} - \frac{\partial p^n}{\partial t} \right] = \nabla^2 p^* + \sigma^*
\]

Poisson Method is Hyperbolic if \(\frac{1}{\Delta t} D^n\) Is Retained

Poisson Method is Parabolic if \(\frac{1}{\Delta t} D^n\) Is Dropped
Poisson Equation in Compressible PISO Method

Continuity Equation:

\[
\frac{\rho^{**} - \rho^n}{\Delta t} + \left( \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} \right)^{**} = 0
\]

Or, Using Perfect Gas Relation

\[
\frac{\rho^{**} - \rho^n}{RT^n \Delta t} + \left( \frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} \right)^{**} = 0
\]

Momentum Equations:

\[
\frac{\rho^{***} u^{***} - \rho^n u^n}{\Delta t} + \left( \frac{\partial u^2}{\partial x} + \frac{\partial uv}{\partial y} \right)^* + \frac{\partial p^{**}}{\partial x} = 0
\]

\[
\frac{\rho^{***} v^{***} - \rho^n v^n}{\Delta t} + \left( \frac{\partial uv}{\partial x} + \frac{\partial v^2}{\partial y} \right)^* + \frac{\partial p^{**}}{\partial y} = 0
\]

Combine from Divergence of Momentum
Characteristic Speed in PISO Poisson Equation

- Combined Continuity and Momentum Equations

\[
\frac{p^{***} - p^n}{RT^n \Delta t^2} + \frac{1}{\Delta t} \left( \frac{\partial p u}{\partial x} + \frac{\partial p v}{\partial y} \right)^n - \sigma^* - \nabla^2 \rho^{***} = 0
\]

- Replace Divergence with Density Derivative

\[
\frac{p^{***} - 2p^n + p^{n-1}}{RT^n \Delta t^2} = \nabla^2 \rho^{***} + \sigma^*
\]

- PISO Poisson Equation is Hyperbolic
  - Characteristic Speeds Are the Acoustic Speeds

- LowMach Number Convergence Requires:
  - Multiple Sweeps of Continuity Equation
  - Re-scaling of Time Derivative---Preconditioning
Equations of Motion

\[ \frac{\partial Q}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} = L_v(Q_v) \]

where

\[ Q = (p, pu, pv, e)^T \]

\[ E = (pu, pu^2 + p, puv, eu + pu)^T \]  (2)

\[ F = (pv, puv, pv^2 + p, ev + pv)^T \]

\[ L_v(Q_v) = \frac{\partial}{\partial x} R_{xx} \frac{\partial}{\partial x} Q_v + \frac{\partial}{\partial x} R_{xy} \frac{\partial}{\partial y} Q_v + \frac{\partial}{\partial y} R_{yx} \frac{\partial}{\partial x} Q_v + \frac{\partial}{\partial y} R_{yy} \frac{\partial}{\partial y} Q_v \]
Formulation of PISO Algorithm

Split Flux Vectors:
\[ E = E_L + E_N \]
\[ E_L = (\rho u, p, 0, 0)_T \]
\[ E_N = (0, \rho u^2, \rho uv, (e + p)u)_T \]

Use Predictor-Corrector Procedure:

\[
\begin{align*}
\left\{ l_c + l_m \left[ \Gamma + \Delta t \left( \frac{\partial}{\partial x} A_N^n + \frac{\partial}{\partial y} B_N^n - L_v \right) \right] \right\} \Delta Q^* &= -\Delta t l_m \left[ \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} - L_v(Q_v) \right]_n \\
\left\{ \Gamma + \left[ \frac{\partial}{\partial x} A_L^* + \frac{\partial}{\partial y} B_L^* \right] \right\} \Delta Q^{**} &= +l_e \left( \frac{\partial}{\partial x} A_N^* + \frac{\partial}{\partial y} B_N^* - L_v \right) \Delta Q^{**} \\
&= -\Delta t \left\{ \frac{\partial}{\partial x} A_L^* Q_v^* + \frac{\partial}{\partial y} B_L^* Q_v^* \right\} + l_e \left( \frac{\partial}{\partial x} A_N^* Q_v^* + \frac{\partial}{\partial y} B_N^* Q_v^* - L_v(Q_v^*) \right)
\end{align*}
\]
Stability Analysis

Represent Disturbance Growth by Amplification Matrix

\[ Q^{n+1} = G Q^n \]

Result Provides Four Amplification Factors
Plot Maximum of These Four

For PISO Scheme,

\[ Q^* = G^* Q^n, \quad Q^{**} = G^{**} Q^*, \quad Q^{n+1} = G^{***} Q^{**} \]

\[ G = G^{***} G^{**} G^* \]
DENSITY-BASED STABILITY RESULTS

EULER IMPLICIT ALGORITHM

CFL=1, M=0.3

CFL=1, M=0.03
MACH NUMBER EFFECT FOR PISO SCHEME

CFL=1, M=0.3

CFL=1, M=0.03

x-wavenumber

y-wavenumber

π

π
TIME DERIVATIVE EFFECT ON PISO SCHEME

CFL = 5, M = 0.3

COUPLED SYSTEM

UNCOUPL ED SYSTEM

x-wavenumber

y-wavenumber

0.0 0.4 0.8

0.0 0.4 0.6

0.0 0.4 1.0

0.0 0.4 1.2

0.0 0.4 1.4

x-wavenumber

y-wavenumber
Comparison of Convergence

PISO: 1-D Incompressible

![Graphs showing comparison of convergence]
Comparison of Convergence

inflo v/outflow B.C.'s

Euler Implicit

PISO

PISO/UP (P) 1+2 AV=0.5

steps

steps
PISO: Fully Implicit

CFL = 1

$M = 0.3$

$M = 0.03$
PISO: ADI

CFL = 1

M = 0.3

M = 0.03
PISO: ADI with Poisson Time Scaling

CFL = 1

M = 0.3

M = 0.03
PISO: Gauss Seidel (1 sweep)

CFL = 1

\[
M = 0.3 \quad M = 0.03
\]
PISO: Gauss Seidel (10 sweeps)

CFL = 1

$M = 0.3$

$M = 0.03$
PISO: Gauss Seidel (1 sweep)

No time deriv. in Poisson

CFL = 1

\[ M = 0.3 \quad \text{and} \quad M = 0.03 \]
PISO: Gauss Seidel (10 sweeps)

No time deriv. in Poisson

CFL = 1

\[ M = 0.3 \]

\[ M = 0.03 \]
Summary

- Differences in Pressure- and Density-Based Methods.
  - Upwind Direction
  - Choice of Solution Variables
  - Coupled vs. Uncoupled Equations

- Pressure-Based Methods for Incompressible Flows Are Hyperbolic
  - Not Parabolic

- Pressure-Based Methods for Compressible Flow are Hyperbolic
  - Characteristics of Poisson Equation are Stiff
  - Time-step Control is Needed for Convergence
  - Can be Offset by Multiple Sweeps of the Poisson Equation
Summary (Contd.)

- Vector Form of Pressure-Based Method Facilitates Comparison

- PISO Vector Stability Analysis Indicates:
  - Conditionally Stable
  - Low Wave Number Instability (Incompressible)
  - High Wave Number Instability (Compressible)

- Code Convergence Verifies Stability Predictions

- Approximate Factorization of Poisson Equation
  - Low Mach Number Stiffness
  - Mitigate by Scaling Time Step
  - Circumvented by Gauss-Seidel
A Comparison of Artificial Compressibility and Fractional Step Methods for Incompressible Flow Computations
Daniel C. Chan
Department of Aerospace Engineering, University of Southern California, Los Angeles, California
and
Rocketdyne Division, Rockwell International Corporation
Armen Darian and Munir Sindir
Rocketdyne Division, Rockwell International Corporation
Canoga Park, California

We have applied and compared the efficiency and accuracy of two commonly used numerical methods for the solution of Navier-Stokes equations. The artificial compressibility method, postulated by Chorin, augments the continuity equation with a transient pressure term and allows one to solve the modified equations as a coupled system. Due to its implicit nature, one can have the luxury of taking a large temporal integration step in the expenses of higher memory requirement and larger operation counts per step. Meanwhile, the fractional step method, developed independently by Chorin and Temam, splits the Navier-Stokes equations into a sequence of differential operators and integrates them in multiple steps. The memory requirement and operation count per time step are low, however, the restriction on the size of time marching step is more severe.

To explore the strength and weakness of these two methods, we used them for the computation of a two-dimensional driven cavity flow with Reynolds number of 100 and 1000, respectively. Three grid sizes, 41x41, 81x81 and 161x161 were used. The computations were considered converged after the $L_2$-norm of the change of the dependent variables in two consecutive time steps has fallen below $10^{-5}$. Same programming style is applied to the development of these codes. All computations were performed on the NASA-Marshall Convex C240 computer with double precision arithmetic.

In summary, we find that the artificial compressibility method requires twice as much memory per grid points and is less efficient for grid resolution below 81x81. Fractional step method, on the other hand, is more efficient in both memory requirement and computational speed for coarse grid computations, however, due to its explicit nature, its convergence rate deteriorates dramatically for fine grid computations.
A COMPARISON OF FRACTIONAL STEP AND ARTIFICIAL COMPRESSIBILITY METHODS

BY:

DANIEL C. CHAN
ARMEN DARIAN
MUNIR M. SINDIR

CFD TECHNOLOGY CENTER
ROCKETDYNE DIVISION
ROCKWELL INTERNATIONAL

PRESENTED AT NASA MARSHALL SPACE FLIGHT CENTER
TENTH WORKSHOP FOR COMPUTATIONAL FLUID DYNAMIC APPLICATIONS IN ROCKET PROPULSION

APRIL 28-30, 1992
AGENDA

- MOTIVATION
- APPROACH
- TEST CASE DESCRIPTION
- RESULTS
MOTIVATION

- EXPLORE AN ALTERNATIVE NAVIER-STOKES SOLVER FOR ENGINEERING APPLICATIONS
  - INTERNAL, INCOMPRESSIBLE FLOWS

- IDENTIFY THE STRENGTH AND WEAKNESS
  - LEVEL OF EXPERTISE REQUIRED
  - ACCURACY
  - SPEED
APPROACH

• COUPLED METHOD
  • EXTENSION OF COMPRESSIBLE FLOW FORMULATION
  • HIGHLY IMPLICIT
  • ELEGANT MATHEMATICAL FORMULATION

• ITERATIVE METHOD
  • LESS IMPLICIT
  • LONGER TRACK RECORD
  • LESS COMPLICATED TO FORMULATE
GOVERNING EQUATIONS FOR ARTIFICIAL COMPRESSIBILITY METHOD

\[
\frac{\partial \tilde{q}}{\partial t} + \frac{\partial \tilde{F}}{\partial x} + \frac{\partial \tilde{G}}{\partial y} - \frac{1}{Re[D]} \nabla^2 \tilde{q} = 0
\]

\[
\tilde{q} = \begin{bmatrix}
\frac{p}{a^2} \\
u \\
v \\
u v
\end{bmatrix}, \quad \tilde{F} = \begin{bmatrix}
u^2 \\
u^2 + p
\end{bmatrix}
\]

\[
\tilde{G} = \begin{bmatrix}
v \\
u v \\
v^2 + p
\end{bmatrix}, \quad [D] = \begin{bmatrix}0 & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1
\end{bmatrix}
\]
NUMERICS

- SECOND ORDER CENTRAL DIFFERENCING FOR ALL SPATIAL DERIVATIVES

- IMPLICIT TEMPORAL INTEGRATION IS NEEDED TO OVERCOME STIFFNESS

- APPROXIMATE FACTORIZATION IN DELTA FORM PLUS SECOND ORDER IMPLICIT AND FOURTH ORDER EXPLICIT DAMPING
BOUNDARY CONDITIONS

- VELOCITY
  - NO-SLIP ALONG SOLID WALLS

- PRESSURE
  - NEUMANN CONDITION DERIVED FROM MOMENTUM EQUATIONS

- EXPLICIT IMPLEMENTATION
FRACTIONAL STEP METHOD

- INTEGRATE DIFFERENTIAL OPERATORS IN A SEQUENCE OF STEPS

\[
\frac{\hat{u}_i - u_i^n}{\Delta t} = L(u_i) - N(u_i)
\]

\[
\frac{u_i^{n+1} - \hat{u}_i^n}{\Delta t} = -\frac{\delta \phi}{\delta x_i} \quad \text{where} \quad \frac{\delta \phi}{\delta x_i} \sim \frac{\delta P}{\delta x_i} - L(u_i)
\]

CONTINUITY REQUIRES

\[
\frac{1}{\Delta t} \frac{\delta \hat{u}_i^n}{\delta x_i} = -\frac{\delta^2 \phi}{\delta x_i \delta x_i}
\]
FRACTIONAL STEP METHOD (CONT'D)

- VARIOUS MULTI-STEP METHODS CAN BE INCORPORATED TO ACHIEVE DESIRED TEMPORAL ACCURACY AND STABILITY
  - FULLY IMPLICIT, LEAP-FROG, CRANK-NICOLSON, RUNGE-KUTTA

- BOUNDARY CONDITION
  - NO-SLIP ALONG WALL
  - GREEN'S THEOREM REQUIRES:

\[ \int \frac{\partial \phi}{\partial n} ds = \int \ddot{u} \cdot \ddot{n} ds \Rightarrow \frac{\partial \phi}{\partial n} = 0 \]
A COMPARISON OF THE TWO METHODS

<table>
<thead>
<tr>
<th>TIME INTEGRATION</th>
<th>ARTIFICIAL COMPRESSIBILITY</th>
<th>FRACTIONAL STEP</th>
</tr>
</thead>
<tbody>
<tr>
<td>(NEWTON'S LINEARIZATION)</td>
<td>FULLY IMPLICIT</td>
<td>CONVECTION: EXPLICIT</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DIFFUSION: IMPLICIT</td>
</tr>
<tr>
<td>SPATIAL DISCRETIZATION</td>
<td>CENTRAL DIFFERENCING</td>
<td>PRESSURE: IMPLICIT</td>
</tr>
<tr>
<td>GRID ARRANGEMENT</td>
<td>COLLOCATION POINTS</td>
<td>CONVECTION: LINEAR</td>
</tr>
<tr>
<td></td>
<td></td>
<td>UPWIND</td>
</tr>
<tr>
<td>ARTIFICIAL DAMPING</td>
<td>4TH ORDER EXPLICIT;</td>
<td>NON-LINEAR, EXPLICIT 4TH</td>
</tr>
<tr>
<td></td>
<td>2ND ORDER IMPLICIT FOR ALL EQUATIONS</td>
<td>AND 2ND ORDER FOR</td>
</tr>
<tr>
<td>BOUNDARY CONDITION</td>
<td>EXPLICIT</td>
<td>PRESSURE STEP</td>
</tr>
<tr>
<td>ACCURACY</td>
<td>2ND ORDER</td>
<td>EXPPLICIT ON V</td>
</tr>
<tr>
<td>MEMORY</td>
<td>30 WORDS/POINT</td>
<td>IMPLICIT ON ( \phi )</td>
</tr>
<tr>
<td>ADJUSTABLE CONSTANTS</td>
<td>( a_i, \Delta t, \varepsilon_i, \varepsilon_e )</td>
<td>( \Delta t )</td>
</tr>
</tbody>
</table>
COMPUTATIONAL MODEL FOR THE FRACTIONAL STEP METHOD

\[ \begin{align*}
    u &= 1 \\
    v &= 0 \\
    \frac{\partial \phi}{\partial y} &= 0 \\
    \frac{\partial \phi}{\partial x} &= 0 \\
    u &= 0 \\
    v &= 0 \\
    u &= 0 \\
    v &= 0
\end{align*} \]
COMPUTATIONAL MODEL FOR THE ARTIFICIAL COMPRESSIBILITY METHOD

\[
\begin{align*}
    u &= 1 \\
    v &= 0 \\
    \frac{\partial P}{\partial y} &= \frac{1}{Re} \frac{\partial^2 v}{\partial y^2} \\
    \frac{\partial P}{\partial x} &= \frac{1}{Re} \frac{\partial^2 u}{\partial x^2}
\end{align*}
\]
METHOD OF COMPARISON

- CONSISTENT PROGRAMMING STYLE
  - ONE DEVELOPER
  - NO EXPLICIT VECTORIZATION

- USE ONE COMPUTER WITH SAME COMPILATION OPTION
  - CONVEX C-240
  - fc -02 -pd8

- MAINTAIN CONSTANT $\Delta T$ AT EACH GRID POINT

- 'OPTIMIZE' INPUT PARAMETERS WITH 41x41 GRID RESOLUTION

- $\varepsilon_l$ AND $\varepsilon_e$ FIXED AT 0.2 AND 0.1, RESPECTIVELY

- $a=1$
PARAMETERS TO COMPARE

- RANGE OF OPERATION
- CPU TIME REQUIREMENT AS A FUNCTION OF GRID POINTS USED
- REYNOLDS NUMBER DEPENDENCY
- ACCURACY
CONVERGENCE HISTORY FOR
41 x 41 GRID, Re=100

$\beta=4.0, \Delta t=0.01$

$\beta=2.0, \Delta t=0.01$

Rockwell International
Rocketdyne Division
CONVERGENCE HISTORY FOR
41 x 41 GRID, Re=100

$\beta=2.0$, $\Delta t=0.025$

$\beta=2.0$, $\Delta t=0.05$
CONVERGENCE HISTORY FOR
41 x 41 GRID, Re=100

$\beta=1.0$, $\Delta t=0.01$

$\beta=1.0$, $\Delta t=0.025$

L2 NORM OF $\Delta Q$

CONTINUITY DEFECT

NUMBER OF TIME STEPS

10^{-1}

10^{-2}

10^{-3}

10^{-4}

10^{-5}

10^{-6}

10^{-7}

0

200

400

600

800

1000

1200

0.03

0.02

0.01

0.00

0.01

0.02

0.03
CONVERGENCE HISTORY FOR
41 x 41 GRID, Re=100

$\beta=1.0, \Delta t=0.05$

$\beta=1.0, \Delta t=0.1$
CONVERGENCE HISTORY FOR
81 x 81 GRID, Re=1000

AC METHOD
\( \Delta T=0.05 \)

FS METHOD
\( \Delta T=0.004 \)
EFFECT OF TIME STEP ON CONVERGENCE RATE

Re=100, 41X41 GRID

CONVEX C240 CPU TIME(SECOND)

MAXIMUM CFL NUMBER

TIME STEP

FS
β=0.5, AC
β=1.0, AC
β=2.0, AC
VELOCITY PROFILES ALONG THE GEOMETRIC CENTERLINES OF A DRIVEN CAVITY FLOW
Re=100
VELOCITY PROFILES ALONG THE GEOMETRIC CENTERLINES OF A DRIVEN CAVITY FLOW
Re=1000
SUMMARY OF RESULTS

- APPROXIMATE FACTORIZATION/ARTIFICIAL COMPRESSIBILITY METHOD
  - MEMORY INTENSIVE
  - SENSITIVE TO REYNOLDS NUMBER
  - REQUIRES MORE USER INTERACTIONS

- FRACTIONAL STEP METHOD
  - MEMORY EFFICIENCY
  - SENSITIVE TO TIME STEP USED
  - COMPUTING INTENSIVE FOR FINE GRID
A Status of the Activities of the NASA/MSFC Pump Stage Technology Team

R. Garcia, R. Williams, and Y. Dakhoul

The Consortium for Computational Fluid Dynamics (CFD) Application in Propulsion Technology was established to aid the transfer of CFD related advancements among academia, government agencies, and industry. The specific goals of the Consortium are to develop CFD methodologies necessary to solve propulsion problems, to validate these methodologies, and to apply these methodologies in the design process. To accomplish these goals, a team of experts in various related fields has been formed, a schedule of activities necessary to meet the goals has been generated, and funding for the activities has been obtained from NASA. During the past year (3/91-3/92) the team's activities have focused on preliminary code validation and on the design of an advanced impeller. Six codes were used to calculate the flow in a Rocketdyne 0.3 flow coefficient inducer and the results were compared to L2F data available for the inducer. This activity identified shortcomings in the experimental data sets and in the analytical solutions which must be surmounted in any future team activity. The design of the advanced impeller relied heavily on CFD results to obtain an optimized geometry. The optimized geometry has been analyzed using four different codes and at design and off-design conditions. Activities for the next year include the optimization of a tandem blade impeller design, benchmark of CFD codes for diffuser and volute flows, the collection of L2F data for "state-of-the-art" impeller and inducer, and the verification of the advanced pump team impeller design in a water rig.
A Summary of the Activities of the NASA/MSFC Pump Stage Technology Team

R. Garcia
NASA, Marshall Space Flight Center

R. Williams
NASA, Marshall Space Flight Center

Y. Dakhoul
Sverdrup Technologies

Presented:
CFD Applications Workshop
MSFC, April 28–30, 1992
A Summary of the Activities of the NASA/MSFC Pump Stage Technology Team

Overview

- Structure/objectives
- Approach
- Validation data
- CFD Analysis:
  - Benchmark activity
  - Advanced hardware
- Summary/conclusions
A Summary of the Activities of the NASA/MSFC
Pump Stage Technology Team

Structure/Objectives

The consortium for CFD application in propulsion technology

Objectives:
- Validation of state-of-the-art CFD codes
- Application of CFD in design of advanced hardware concepts
- Verification testing of advanced hardware concepts

Pump Stage Team

Objectives:
- Coordinate/focus MSFC pump technology activities
- Provide a forum for interaction/technology transfer
- Provide peer review for pump technology activities

Turbine Stage Team

Combustion Devices Team
A Summary of the Activities of the NASA/MSFC Pump Stage Technology Team

Approach

- Assemble a team of experts
  - Team members from academia, industry, and government agencies

- Implement a plan to coordinate pump team activities
  - Set milestone dates consistent with rocket engine development requirements

- Hold quarterly meetings to:
  - Critique activities
  - Raise unexpected/new issues and requirements
  - Maintain focus on the deliverable product and on the schedule
A Summary of the Activities of the NASA/MSFC
Pump Stage Technology Team

Pump Team Members

Consortium for CFD Application In Propulsion Technology Pump Stage Technology Team

- NASA Marshall Space Flight Center (MSFC)
- NASA Ames Research Center (ARC)
- NASA Lewis Research Center (LeRC)
- David Taylor Research Center
- Rocketdyne (RDYN)
- Pratt & Whitney (P&W)
- Aerojet
- Ingersoll-Rand
- Computational Fluid Dynamics (CFD) Research Corporation
- SECA
- Scientific Research Associates (SRA)
- The University of Alabama in Huntsville (UAH)
- Pennsylvania State University (PSU)
- University of Cincinnati
- Virginia Polytechnic Institute
- California Institute of Technology
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>4th</td>
<td>1st</td>
<td>2nd</td>
<td>3rd</td>
<td>4th</td>
<td>1st</td>
</tr>
<tr>
<td>STME Program</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>LLDR / PDR</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>CDR</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>TurboPump Test</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Advanced Hardware</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline Impeller</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Conventional Design</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>CFD Analysis</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Manufacture</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Water Test</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Phase 1 Impeller</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>CFD Parametrics</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Design</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>CFD Analysis</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Manufacture</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Water Test</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
<td>▼</td>
</tr>
<tr>
<td>Validation Data</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inducer (Φ = 0.3)</td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSME Impeller</td>
<td></td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Baseline Impeller</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inducer (Φ = 0.08)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Diffuser Data 1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
## PUMP DESIGN TECHNOLOGY

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1st</td>
<td>2nd</td>
<td>3rd</td>
<td>4th</td>
<td>1st</td>
<td>2nd</td>
</tr>
<tr>
<td>Code Development</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>AEROVISCE</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FDNS3D</td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HAH3D</td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>INS3D-UP</td>
<td></td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MINT</td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>REACT3D</td>
<td></td>
<td></td>
<td>▼</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>MEFP</td>
<td></td>
<td></td>
<td></td>
<td>▼</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Team Meetings</td>
<td>● ● ● ●</td>
<td>● ● ● ●</td>
<td>● ● ● ●</td>
<td>● ● ● ●</td>
<td>● ● ● ●</td>
<td>● ● ● ●</td>
</tr>
</tbody>
</table>
A Summary of the Activities of the NASA/MSFC Pump Stage Technology Team

CFD Code Verification Inducer
Data Planes and Geometry Definition
A Summary of the Activities of the NASA/MSFC Pump Stage Technology Team

Pump CFD Code Validation Tests SMSME HPFTP Impeller
A Summary of the Activities of the NASA/MSFC
Pump Stage Technology Team

CFD Analysis

• Advanced hardware development
  • Conventionally designed advanced impeller optimized using CFD
    • Impeller design to satisfy STME fuel pump requirements with two stages
    • CFD study of 15 parameters: $b_2$, $\beta_2$, axial length, total wrap angle, and discharge wrap angle difference (hub-to-tip)
    • Viability of CFD parametrics demonstrated
    • Baseline and optimized geometry analyzed by five team members
    • All solutions show higher efficiency and reduced impeller discharge flow distortion
    • Off-design analysis under way
    • Impeller being manufactured; performance to be verified in water rig in the fall of 1992
  • Tandem blade impeller concept
    • Concept has potential for increased head coefficient and efficiency
    • CFD parametric study to begin in May 1992
    • Study to include position of blade split, blade clocking, and chordwise spacing
    • Final configuration to rely entirely on results of parametric study
    • Impeller will be sized to satisfy STME fuel pump requirements
### Cases Postprocessed:

<table>
<thead>
<tr>
<th>Organization</th>
<th>Inlet Shroud</th>
<th>Exit Walls</th>
<th>Flows</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ames</td>
<td>Fixed</td>
<td>Slip</td>
<td>80%, 100%, 120%</td>
</tr>
<tr>
<td>(204 X 33 X 52)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Lewis</td>
<td>Fixed</td>
<td>Rotating</td>
<td>100%</td>
</tr>
<tr>
<td>(73 X 23 X 30)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rocketdyne</td>
<td>Rotating</td>
<td>Rotating</td>
<td>100%</td>
</tr>
<tr>
<td>(122 X 24 X 30)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SECA: Case 1</td>
<td>Fixed</td>
<td>Fixed</td>
<td>100%</td>
</tr>
<tr>
<td>Case 2</td>
<td>Fixed</td>
<td>Slip</td>
<td>100%</td>
</tr>
<tr>
<td>Case 3</td>
<td>Rotating</td>
<td>Fixed</td>
<td>100%</td>
</tr>
<tr>
<td>Case 4</td>
<td>Rotating</td>
<td>Rotating</td>
<td>100%</td>
</tr>
<tr>
<td>Case 5</td>
<td>Rotating</td>
<td>Slip</td>
<td>100%</td>
</tr>
<tr>
<td>(103 X 23 X 30)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SRA</td>
<td>Fixed</td>
<td>Slip</td>
<td>100%</td>
</tr>
<tr>
<td>(121 X 26 X 51)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
OPTIMIZED IMPELLER: EXIT CM VS. X

- SRA
- SECA
- RKDN
- LRC
- ARC

Relative x (shroud = 0.0)
OPTIMIZED IMPELLER: EXIT CM BLADE-TO-BLADE NEAR THE SHROUD

\[ \frac{c_m}{u_{tip}} \]

relative angle (full blade suction = 0.0)
Optimized Impeller: Flow Split
fraction between full, suction and partial

Q between full, suction and partial

arc 100 arc 80 arc 120 lrc rkd 1 sec 2 sec 3 sec 4 sec 5 sra
OPTIMIZED IMPELLER: CM VS. X
FOR $R_{rel} = 0.95$ (NEAR SHROUD)

OPTIMIZED IMPELLER: BETA VS. X
FOR $R_{rel} = 0.95$
OPTIMIZED IMPELLER: CM VS. X
FOR $R_{rel} = 0.50$ (MID HEIGHT)

OPTIMIZED IMPELLER: BETA VS. X
FOR $R_{rel} = 0.50$
OPTIMIZED IMPELLER: CM VS. X
FOR $R_{rel} = 0.05$ (NEAR HUB)

![Graph showing averaged CM/U_{tip} vs. normalized axial position for different cases.]

OPTIMIZED IMPELLER: BETA VS. X
FOR $R_{rel} = 0.05$

![Graph showing beta angle vs. normalized axial position for different cases.]

195
AMES RESULTS: EXIT CM VS. X
FOR VARYING FLOW RATE

Graph showing the relationship between cm/\mu t_{tip} and relative x for different flow rates: 100% flow, 80% flow, and 120% flow. The graph compares the flow rates at various points labeled as shroud and hub.
AMES RESULTS: HEAD COEFF. VS. X
FOR VARYING FLOW RATE

head coefficient

- 100% flow
- 80% flow
- 120% flow

relative x
AMES RESULTS: CM UPSTREAM OF IMPELLER
FOR 80% FLOW

relative $r$

$cm/u_{tip}$

- $x = 0.000$
- $x = -0.0449$
- $x = -0.1023$
- $x = -0.2518$

198
AMES RESULTS: CU UPSTREAM OF IMPELLER
FOR 80% FLOW

relative $r$

$cu/\left<u_{lip}\right>$
Summary/Conclusions

- Technology team in place and functioning efficiently
  - Participation by industry, universities, and government

- Detailed experimental data sets suitable for benchmarking have been or are being generated

- Preliminary evaluation of six different codes complete

- CFD codes being used to reduce the design development time and improve performance of advanced impellers

- Verification of advanced impeller predictions planned for the fall of 1992

- Future work to include impeller-diffuser interaction and inducer non-cavitating analysis
CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER

Gary C. Cheng*, Y.S. Chen†, and R.W. Williams‡

Abstract

Current design of high performance turbopumps for rocket engines requires effective and robust analytical tools to provide design impact in a productive manner. The main goal of this study is to develop a robust and effective computational fluid dynamics (CFD) pump model for general turbopump design and analysis applications. A Navier-Stokes flow solver, FDNS, embedded with the extended k-ε turbulence model and with appropriate moving interface boundary conditions, is developed to analyze turbulent flows in the turbomachinery devices. The FDNS code has been benchmarked with its numerical predictions of the pump consortium inducer, and provides satisfactory results. In the present study, a CFD analysis of the pump consortium impeller will be conducted with the application of the FDNS code. The pump consortium impeller, with partial blades, is the new design concept of the advanced rocket engine. A 3-D flow calculation with 81 x 41 x 41 grid system was conducted for the team base-line impeller. The result shows a massive flow separation occurs between the full-blade pressure surface and the partial-blade suction surface. Similar result was predicted by the other consortium members. A pump consortium optimized impeller, a revision based on the base-line impeller, was then designed by Rocketdyne to remove the flow separation. A 3-D flow analysis, with 103 x 23 x 30 mesh system and with the inlet flow conditions provided by Rocketdyne, was performed for the optimized impeller. The numerical result indicates no flow separation occurs inside the flow passage, which is also consistent with the other consortium members’ predictions. However, the flow field inside the optimized impeller as calculated by the team members showed great variations, especially near the exit shroud region. The discrepancy is suspected to be due to different exit boundary conditions used by the consortium members. Therefore, three different exit wall boundary conditions will be further examined by the FDNS code, those are fixed-wall, wall-slip (symmetry), and rotating wall boundary conditions. The computed results will be compared in order to address the effect of exit boundary conditions on the impeller flow field. Meanwhile, two off-design cases of the optimized impeller, 80% and 120% of the design flow, will also be analyzed with a particular exit boundary condition. All CFD analysis of the pump consortium base-line impeller, and the optimized impeller with various exit boundary conditions will be presented in the coming CFD workshop meeting.

* SECA, Inc., 3313 Bob Wallace Ave., Suite 202, Huntsville, AL
† Engineering Sciences, Inc., 4920 Corporate Dr., Suite K, Huntsville, AL
‡ ED 32, NASA/ Marshall Space Flight Center, Huntsville, AL
CFD ANALYSIS OF PUMP CONSORTIUM IMPELLER

By

Gary C. Cheng, SECA, Inc.

Y.S. Chen, ESI

AND

R.W. Williams
NASA/Marshall Space Flight Center

NASA Contract No. NAS8-38868

TENTH ANNUAL CFD WORKSHOP MEETING, APRIL, 1992
### INLET/EXIT WALL B.C. TESTED

<table>
<thead>
<tr>
<th>Inlet B.C.</th>
<th>Exit B.C.</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Fixed-Wall</td>
</tr>
<tr>
<td>Fixed-Wall</td>
<td>Case 1</td>
</tr>
<tr>
<td>Rotating-Wall</td>
<td>Case 3</td>
</tr>
</tbody>
</table>

### CALCULATED MASS FLOW RATE SPLIT

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
<th>Case 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>S.F. - P.P. /S.P. - P.F.</td>
<td>50.4/49.6</td>
<td>49/51</td>
<td>43.2/56.8</td>
<td>42.4/57.6</td>
<td>40.6/59.4</td>
</tr>
</tbody>
</table>
DEFINITION OF PERFORMANCE PARAMETERS

- $C_u = \frac{c_u}{U_{tip}}$; $C_M = \frac{c_M}{U_{tip}}$ where $c_u =$ Absolute Tangential Velocity,
  $c_M =$ Meridional Velocity, $U_{tip} =$ Wheel Tip Velocity

- $\beta =$ Relative Flow Angle Relative to Tangential Direction

- Relative Radius $= \frac{(R_i - R_{hub})}{(R_{shroud} - R_{hub})}$

- Relative $X = \frac{(X_i - X_{shroud})}{(X_{hub} - X_{shroud})}$

- Relative Angle $= \frac{(\text{Angle}_{i} - \text{Angle}_{suction})}{(\text{Angle}_{pressure} - \text{Angle}_{suction})}$

- $\Psi$ (Head Coefficient) $= \frac{\Delta H_{tg}}{U_{tip}^2}$

- $\eta$ (Efficiency) $= \text{Head Rise} / \text{Euler Head Rise}$
Case 1

Case 3

VELOCITY VECTORS NEAR SUCTION SIDE OF BLADE
VELOCITY VECTORS NEAR PRESSURE SIDE OF SPLITTER
Case 1

Case 3

VELOCITY VECTORS NEAR SUCTION SIDE OF SPLITTER
VELOCITY VECTORS NEAR PRESSURE SIDE OF BLADE

Case 1

Case 3
Case 2

Case 5

VELOCITY VECTORS NEAR SUCTION SIDE OF BLADE
VELOCITY VECTORS NEAR PRESSURE SIDE OF SPLITTER

Case 2

Case 5
Case 2

Case 5

VELOCITY VECTORS NEAR SUCTION SIDE OF SPLITTER
Case 2

Case 5

VELOCITY VECTORS NEAR PRESSURE SIDE OF BLADE
OPTIMIZED IMPELLER: CM VS. X
FOR $R_{rel} = 0.05$ (NEAR HUB)

OPTIMIZED IMPELLER: BETA VS. X
FOR $R_{rel} = 0.05$
OPTIMIZED IMPELLER: CM VS. X
FOR \( R_{rel} = 0.95 \) (NEAR SHROUD)

DISTANCE FROM IMPELLER INLET: \( (X-X_{in})/D_{tip} \)

OPTIMIZED IMPELLER: BETA VS. X
FOR \( R_{rel} = 0.95 \)
OPTIMIZED IMPELLER:
EFFICIENCY VS. X

OPTIMIZED IMPELLER:
HEAD COEFFICIENT VS. X
CONCLUSIONS

- THE PRESENT CFD RESULTS HAVE SHOWN SENSITIVITY OF INLET AND EXIT WALL BOUNDARY CONDITIONS ON THE FLOW STRUCTURE INSIDE THE OPTIMIZED CONSORTIUM IMPELLER DESIGN

- INLET SHROUD WALL BOUNDARY TREATMENTS HAVE SIGNIFICANT EFFECT ON THE FLOW SPLIT AROUND THE PARTIAL BLADE (MORE FLOW THROUGH THE PARTIAL/FULL-PRESSURE PASSAGE WHEN THE INLET SHROUD WALL IS ASSUMED ROTATING)

- ONLY MINOR IMPACT ON THE OVERALL IMPELLER PERFORMANCE DATA WAS REVEALED FOR DIFFERENT BOUNDARY CONDITIONS IMPOSED
Abstract of a proposed paper for the presentation at workshop for CFD Applications in Rocket Propulsion to be held at NASA Marshall Space Flight Center, AL, April 28-30, 1992

CFD APPLICATIONS IN PUMP FLOWS

Cetin Kiris, Liang Chang
MCAT Institute, Moffett Field, CA

and

Dochan Kwak
NASA-Ames Research Center, Moffett Field, CA

The objective of the proposed paper is to develop a computational procedure that solves incompressible Navier-Stokes equations for pump flows. The solution method is based on the pseudocompressibility approach and uses an implicit-upwind differencing scheme together with the Gauss-Seidel line relaxation method. The equations are solved in steadily rotating reference frames and the centrifugal force and the Coriolis force are added to the equation of motion. As a benchmark problem, the flow through the Rocketdyne inducer is numerically simulated. A coarse grid solution is obtained with a single zone by using an algebraic turbulence model. In multi-zone fine grid computation, one-equation Baldwin-Barth turbulence model is utilized. Numerical results are compared with experimental measurements and a good agreement is found between the two. The resulting computer code is then applied to the flow analysis inside two-stage fuel pump impeller operating at 80 %, 100 %, and 120 % of design flow.
CFD APPLICATIONS IN PUMP FLOWS

Cetin Kiris, Dochan Kwak, and Leon Chang
NASA-Ames Research Center

Workshop for CFD Applications in Rocket Propulsion
NASA-MSFC, April 28-30, 1992
Introduction

- Motivation
  - Increasing efficiency and reliability of the liquid rocket engine components is an important task.
  - Understand fluid dynamics of fuel and oxidizer flows from fuel tank to plume.
  - Role of CFD toward a better design.

- Goal
  - To implement CFD technology to simulate the flow through the pump components.
  First Step: Benchmark problems and component analysis.
  Second Step: Unsteady flows through the entire pump (future work).
Method of Solution

- Available algorithms for pump applications are: INS3D-UP and IND3D-LU.

- Currently INS3D-UP is used.

- Based on method of artificial compressibility.

- Both steady-state and time-accurate formulation.

- Steady-state formulation in steadily rotating reference frame.

- Multi-Zone and Operlapped grid scheme capability.

- Computing time: $\approx 1 \times 10^{-4}$ sec/grid point/iteration

- Memory Usage: $\approx 45$ times number of grid points in words
INS3D-UP Algorithm

- Central differencing for viscous fluxes
- Upwind differencing for convective fluxes
  - 3rd and 5th order flux-difference splitting is used for the right hand side terms
- Gauss-Seidel line relaxation relaxation
- Unlimited time step usage in steady-state formulation.
- Inflow and Outflow boundaries based on Method of Characteristics
  - Inflow Boundary: Three velocity components specified
  - Outflow Boundary: Static pressure specified
- Quasi-implicit boundary conditions at zonal interfaces.
Steady-State Formulation

- Introduce artificial compressibility term to the continuity equation

\[
\frac{\partial p}{\partial \tau} = -\beta \left( \frac{\partial \hat{U}}{\partial \xi} + \frac{\partial \hat{V}}{\partial \eta} + \frac{\partial \hat{W}}{\partial \zeta} \right)
\]

\[
\frac{\partial \hat{q}}{\partial \tau} = -\frac{\partial}{\partial \xi} (\hat{e} - \hat{e}_v) - \frac{\partial}{\partial \eta} (\hat{f} - \hat{f}_v) - \frac{\partial}{\partial \zeta} (\hat{g} - \hat{g}_v) = -\hat{r} + S
\]

\(\beta\) is an artificial compressibility constant
\(\tau\) is a pseudo-time step
\(S\) is a source term as centrifugal and coriolis forces.

- Euler Implicit time discretization

- Solve system of equations iteratively in pseudo-time until solution converges to a steady state
Time-accurate Formulation

• Discretize the time term in momentum equations using second-order three-point backward-difference formula

\[
\left( \frac{\partial U}{\partial \xi} + \frac{\partial V}{\partial \eta} + \frac{\partial W}{\partial \zeta} \right)^{n+1} = 0
\]

\[
\frac{3\hat{q}^{n+1} - 4\hat{q}^n + \hat{q}^{n-1}}{2\Delta t} = -\hat{r}^{n+1}
\]

• Introduce a pseudo-time level and artificial compressibility

\[
\frac{1}{\Delta \tau}(\hat{p}^{n+1,m+1} - \hat{p}^{n+1,m}) = -\beta \nabla \cdot q^{n+1,m+1}
\]

\[
\frac{1.5}{\Delta t}(\hat{q}^{n+1,m+1} - \hat{q}^{n+1,m}) = -\hat{r}^{n+1,m+1} - \frac{3\hat{q}^{n+1,m} - 4\hat{q}^n + \hat{q}^{n-1}}{2\Delta t}
\]

• Iterate the equations in pseudo-time for each time step until incompressibility condition is satisfied.
Inducer Computations

- Grid size: 187 x 27 x 35
- Baldwin-Lomax algebraic turbulence model.
- Tip clearance region is included.
- Computer time: 5 - 6 Cray-YMP hours
- Multi-zone computation (currently underway)
- Grid 1: 63 x 37 x 74 Upstream of inducer
- Grid 2: 115 x 37 x 48 Inducer blades
- Grid 3: 51 x 37 x 20 Bull-nose cavity
- Grid 4: 51 x 37 x 49 Downstream of blades
- One-equation Baldwin-Barth turbulence model.
VALIDATION OF INS3D-UP
Pump Design Code Development for STME

Rocketdyne inducer geometry

Schematic of the experimental measurement planes

Comparison of relative total velocities

Computed surface pressure for Rocketdyne inducer
<table>
<thead>
<tr>
<th>Impeller Water Test Conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Number of Blades</strong></td>
</tr>
<tr>
<td><strong>Design Speed, RPM</strong></td>
</tr>
<tr>
<td><strong>Design Flow, GPM</strong></td>
</tr>
<tr>
<td><strong>Reynolds Number, per inch</strong></td>
</tr>
<tr>
<td><strong>Inlet Tip Diameter, inch</strong></td>
</tr>
<tr>
<td><strong>Inlet Hub Diameter, inch</strong></td>
</tr>
<tr>
<td><strong>Outflow Diameter, inch</strong></td>
</tr>
<tr>
<td><strong>Discharge Tip Speed, fps</strong></td>
</tr>
</tbody>
</table>
NLS Pump Impeller

Relative Velocity Vectors Colored by Pressure

Old Design

New Design
NLS Pump Impeller

120%  100%  80%

Particles are released near shroud
Velocity Vectors Colored by Pressure

downstream of full blade L.E.

Pressure

-1.12000
-1.11000
-1.10000
-1.09000
-1.08000
-1.07000
-1.06000
-1.05000
-1.04000
-1.03000
-1.02000
-1.01000
-1.00000
-0.99000
-0.98000
-0.97000
-0.96000
-0.95000
-0.94000
-0.93000
-0.92000
-0.91000

100% of Design Flow

120% of Design Flow
Velocity Vectors Colored by Pressure
downstream of full blade L.E.

Pressure

-1.12000
-1.11000
-1.10000
-1.09000
-1.08000
-1.07000
-1.06000
-1.05000
-1.04000
-1.03000
-1.02000
-1.01000
-1.00000
-0.99000
-0.98000
-0.97000
-0.96000
-0.95000
-0.94000
-0.93000
-0.92000
-0.91000

80% of Design Flow

60% of Design Flow
Velocity Vectors Colored by Pressure upstream of discharge plane

Pressure

-0.42250
-0.42000
-0.41750
-0.41500
-0.41250
-0.41000
-0.40750
-0.40500
-0.40250
-0.40000
-0.39750
-0.39500
-0.39250
-0.39000
-0.38750
-0.38500
-0.38250
-0.38000
-0.37750
-0.37500
-0.37250
-0.37000
-0.36750
-0.36500
-0.36250
-0.36000
-0.35750
-0.35500
-0.35250
-0.35000
-0.34750
-0.34500
-0.34250
-0.34000
-0.33750
-0.33500
-0.33250
-0.33000
-0.32750
-0.32500

100% of Design Flow
Velocity Vectors Colored by Pressure
upstream of discharge plane

Pressure
-0.42750
-0.42500
-0.42250
-0.42000
-0.41750
-0.41500
-0.41250
-0.41000
-0.40750
-0.40500
-0.40250
-0.40000
-0.39750
-0.39500
-0.39250
-0.39000
-0.38750
-0.38500
-0.38250
-0.38000
-0.37750
-0.37500
-0.37250
-0.37000
-0.36750
-0.36500
-0.36250
-0.36000
-0.35750
-0.35500
-0.35250
-0.35000
-0.34750
-0.34500
-0.34250

80% of Design Flow
Velocity Vectors Colored by Pressure
upstream of discharge plane

Pressure

-0.49500
-0.49000
-0.39500
-0.39000
-0.39500
-0.38000
-0.37500
-0.37000
-0.36500
-0.36000
-0.35500
-0.35000
-0.34500
-0.34000
-0.33500
-0.33000
-0.32500
-0.32000
-0.31500
-0.31000
-0.30500
-0.30000
-0.29500
-0.29000

120% of Design Flow
IMPELLER EXIT CM DISTRIBUTION

CM

Relative X (from shroud to hub)

- 100% of Design Flow
- 120% of Design Flow
- 80% of Design Flow
- 60% of Design Flow
IMPELLER EXIT BETA DISTRIBUTION

BETA

Relative X (from shroud to hub)

- 100% of Design Flow
- 120% of Design Flow
- 80% of Design Flow
- 60% of Design Flow
CM @ R/Rtip = 1.0  120% of Design Flow

Relative Angle (from suction side to pressure side)
## IMPELLER OVERALL PARAMETERS

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Downstream Boundary</td>
<td>no-slip wall</td>
<td>no-slip wall</td>
<td>slip b.c.</td>
<td>slip b.c.</td>
<td>slip b.c.</td>
<td>slip b.c.</td>
</tr>
<tr>
<td>Design Flow</td>
<td>100 %</td>
<td>100 %</td>
<td>100 %</td>
<td>120 %</td>
<td>80 %</td>
<td>60 %</td>
</tr>
<tr>
<td>Efficiency</td>
<td>.946</td>
<td>.983</td>
<td>.943</td>
<td>.931</td>
<td>.955</td>
<td>.963</td>
</tr>
<tr>
<td>Head</td>
<td>.663</td>
<td>.636</td>
<td>.659</td>
<td>.631</td>
<td>.667</td>
<td>.691</td>
</tr>
<tr>
<td>Coefficient</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Relative Flow Angle, Deg</td>
<td>25.5</td>
<td>18.3</td>
<td>19.8</td>
<td>21.1</td>
<td>19.2</td>
<td>17.6</td>
</tr>
<tr>
<td>Absolute Flow Angle, Deg</td>
<td>7.01</td>
<td>6.44</td>
<td>6.63</td>
<td>7.31</td>
<td>4.33</td>
<td>3.10</td>
</tr>
<tr>
<td>Meridional Velocity</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Summary

- The present capability to compute a 3-D flow through a complex internal geometry is demonstrated. Advanced impeller analysis showed that overall parameters do not have a significant change between 80%, 100%, and 120% of design flow.

- Solution algorithm was tested and validated in model problems. Inducer computations indicate less than 10% error in velocities. Tip and wake regions show biggest discrepancies. Future work will be focused on turbulence modeling.

- Solution procedure obtained here can be used in the design process of pump components.
COMPUTATION OF THE FLOW FIELD IN A CENTRIFUGAL IMPELLER WITH SPLITTER BLADES†

Frederik J. de Jong, Sang-Keun Choi, T.R. Govindan and Jayant S. Sabnis*
Scientific Research Associates, Inc.
Glastonbury, CT

ABSTRACT

To support the design effort of the STME Fuel Pump Stage, viscous flow calculations have been performed in a centrifugal impeller with splitter blades. These calculations were carried out with SRA's Navier-Stokes solver (MINT), which employs a linearized block-implicit ADI procedure to iteratively solve a finite difference form of the system of conservation equations of mass, momentum, and energy in body-fitted coordinates. A computational grid was generated algebraically for the "channel" between two main blades of the impeller and extended both upstream of the impeller inlet and downstream of the impeller exit so that the appropriate boundary conditions could be applied (viz. specified velocity profiles at the inflow boundary, and specified pressure at the outflow boundary).

The results of the calculations show that although the overall level of flow distortion near the impeller exit is not very large, there is a noticeable difference between the flow patterns in the two "passages" (one passage between the pressure side of the full blade and the suction side of the splitter blade, and the other one between the pressure side of the splitter blade and the suction side of the next full blade). For example, the pressure distribution shows that the splitter blade is loaded less heavily than the main blade. At the same time, the flow distortion near the suction side of the main blade is larger than that near the suction side of the splitter blade. A better understanding of these results can be obtained by studying "particle traces" (streamlines in a frame of reference fixed to the rotating impeller). These traces show that a significant amount of low momentum fluid (originating from the hub and shroud boundary layers) moves from the pressure side to the suction side in the impeller "channel" ahead of the splitter blade, and ends up in the passage between the pressure side of the splitter blade and the suction side of the full blade. This explains why the region of flow distortion in this passage is larger than that in the other passage, and why the mass flow through this channel is less. The understanding of the physics of impeller flow fields that results from analyzing viscous flow calculations such as the one described above is very valuable in pump stage design.

† This work was supported by NASA Marshall Space Flight Center under Contract NAS8-38866.
* Currently at United Technologies Research Center, East Hartford, CT
OBJECTIVES

- DEVELOPMENT OF A PREDICTIVE TOOL FOR THE VISCOS FLOW IN CENTRIFUGAL IMPELLERS
- SUPPORT OF DESIGN EFFORT OF STME FUEL PUMP STAGE
- ENHANCEMENT OF UNDERSTANDING OF PHYSICAL FLOW PHENOMENA

APPROACH

- EQUATIONS SOLVED:
  REYNOLDS-AVERAGED NAVIER-STOKES EQUATIONS IN ROTATING BODY-FITTED COORDINATES
- METHOD OF SOLUTION:
  LINEARIZED BLOCK IMPLICIT (ADI) SCHEME
- TURBULENCE MODEL:
  MIXING LENGTH OR TWO-EQUATION (K-ε)
GEOMETRY

- SHROUDED IMPELLER
- SIX FULL ("MAIN") BLADES, SIX PARTIAL ("SPLITTER") BLADES
- ROTATING VANELESS DIFFUSER SECTION DOWNSTREAM
- ROTATING HUB AND NON-ROTATING END WALL UPSTREAM
- COMPUTATIONAL DOMAIN = "DUCT" BETWEEN TWO MAIN BLADES, WITH UPSTREAM AND DOWNSTREAM EXTENSIONS
BOUNDARY CONDITIONS

- INFLOW:
  VELOCITY PROFILES, STAGNATION TEMPERATURE
- OUTFLOW:
  CONSTANT PRESSURE
- WALLS:
  NO-SLIP, ADIABATIC
- UPSTREAM AND DOWNSTREAM MAIN BLADE EXTENSIONS:
  PERIODICITY

WATER TEST CONDITIONS

- DIMENSIONS

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
<th>Unit</th>
<th>Conversion</th>
</tr>
</thead>
<tbody>
<tr>
<td>INLET TIP DIAMETER</td>
<td>6.0</td>
<td>INCH</td>
<td>0.1524 M</td>
</tr>
<tr>
<td>INLET HUB DIAMETER</td>
<td>3.9</td>
<td>INCH</td>
<td>0.0991 M</td>
</tr>
<tr>
<td>EXIT TIP DIAMETER</td>
<td>9.045</td>
<td>INCH</td>
<td>0.2297 M</td>
</tr>
</tbody>
</table>

- FLOW CONDITIONS

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
<th>Conversion</th>
</tr>
</thead>
<tbody>
<tr>
<td>DESIGN SPEED</td>
<td>6322</td>
<td>RPM</td>
<td>662 RAD/S</td>
</tr>
<tr>
<td>TIP SPEED</td>
<td>249.5</td>
<td>FT/S</td>
<td>76.05 M/S</td>
</tr>
<tr>
<td>DESIGN FLOW</td>
<td>1205</td>
<td>GPM</td>
<td>0.0760 M³/S</td>
</tr>
<tr>
<td>AVERAGE INFLOW VELOCITY</td>
<td>23.7</td>
<td>FT/S</td>
<td>7.22 M/S</td>
</tr>
<tr>
<td>FLOW COEFFICIENT</td>
<td>0.095</td>
<td></td>
<td></td>
</tr>
<tr>
<td>REYNOLDS NUMBER</td>
<td>5.5 x 10⁴</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>2.9 x 10⁶</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
COMPUTATIONAL GRID

• ALGEBRAICALLY GENERATED GRID

• NUMBER OF GRID POINTS = 160,446
  - 121 POINTS IN "STREAMWISE" DIRECTION
    (WITH 61 POINTS ON THE MAIN BLADE AND 41 POINTS
    ON THE PARTIAL BLADE)
  - 26 POINTS FROM HUB TO SHROUD
  - 51 POINTS IN CIRCUMFERENTIAL DIRECTION
    (FROM BLADE TO BLADE)

• GRID SPACING (BASED ON TIP DIAMETER)
  - $10^5$ NEAR HUB AND SHROUD
  - $5 \times 10^5$ - $10^4$ NEAR BLADE SURFACES
IMPELLER GEOMETRY
MERIDIONAL SURFACE

PRESSURE IN MERIDIONAL SURFACE
BETWEEN SPLITTER PRESSURE SIDE AND BLADE SUCTION SIDE
VELOCITY MAGNITUDE IN MERIDIONAL SURFACE
BETWEEN BLADE PRESSURE SIDE AND SPLITTER SUCTION SIDE

VELOCITY MAGNITUDE IN MERIDIONAL SURFACE
BETWEEN SPLITTER PRESSURE SIDE AND BLADE SUCTION SIDE
STREAMWISE VELOCITY IN MERIDIONAL SURFACE
BETWEEN BLADE PRESSURE SIDE AND SPLITTER SUCTION SIDE

STREAMWISE VELOCITY IN MERIDIONAL SURFACE
BETWEEN SPLITTER PRESSURE SIDE AND BLADE SUCTION SIDE
IMPELLER GEOMETRY
MID-SPAN BLADE-TO-BLADE SURFACE

PRESSURE
VELOCITY MAGNITUDE

STREAMWISE VELOCITY
IMPELLER GEOMETRY
CROSS-SECTION NEAR TRAILING EDGE

PRESSURE
VELOCITY MAGNITUDE

STREAMWISE VELOCITY
SUMMARY

- NAVIER-STOKES PROCEDURE HAS BEEN USED TO SIMULATE THE FLOW FIELD IN A CENTRIFUGAL IMPELLER WITH SPLITTER BLADES
- ANALYSIS OF THE RESULTS PROVIDES UNDERSTANDING OF THE PHYSICAL FLOW PHENOMENA
  - VALUABLE IN PUMP STAGE DESIGN
  - USEFUL TO GUIDE CFD CODE DEVELOPMENT
Flow non-uniformity at the discharge of high power density impellers can result in significant unsteady interactions between impeller blades and downstream diffuser vanes. These interactions result in degradation of both performance and pump reliability. The MSFC Pump Technology Team has recognized the importance of resolving this problem and has thus initiated the development and testing of a high head coefficient impeller. One of the primary goals of this program is to improve impeller performance and discharge flow uniformity. The objective of the present work is complimentary. Flow uniformity and performance gains were sought through the application of a tandem blade arrangement. The approach adopted was to numerically establish flow characteristics at the impeller discharge for the baseline MSFC impeller and then parametrically evaluate tandem blade configurations. A tandem design was sought that improves both impeller performance and discharge uniformity. The Navier-Stokes solver AEROVISC was used to conduct the study. Grid embedding is used to resolve local gradients while attempting to minimize model size. Initial results indicate that significant gains in flow uniformity can be achieved through the tandem blade concept and that blade clocking rather than slot location is the primary driver for flow uniformity.
IMPELLER TANDEM BLADE STUDY WITH GRID EMBEDDING FOR LOCAL GRID REFINEMENT

GEORGE BACHE'
AEROSPACE ENGINEERING DIVISION

10th WORKSHOP FOR COMPUTATIONAL FLUID DYNAMIC APPLICATION IN ROCKET PROPULSION
APRIL 28-30, 1992
Flow non-uniformity at the discharge of high power density impellers can result in significant unsteady interactions between impeller blades and downstream diffuser vanes. These interactions result in degradation of both performance and pump reliability. The MSFC Pump Technology Team has recognized the importance of resolving this problem and has thus initiated the development and testing of a high head coefficient impeller. One of the primary goals of this program is to improve impeller performance and discharge flow uniformity. The objective of the present work is complimentary. Flow uniformity and performance gains were sought through the application of a tandem blade arrangement. The approach adopted was to numerically establish flow characteristics at the impeller discharge for the baseline MSFC impeller and then parametrically evaluate tandem blade configurations. A tandem design was sought that improves both impeller performance and discharge uniformity. The Navier-Stokes solver AEROVISC was used to conduct the study. Grid embedding is used to resolve local gradients while attempting to minimize model size. Initial results indicate that significant gains in flow uniformity can be achieved through the tandem blade concept and that blade clocking rather than slot location is the primary driver for flow uniformity.
MOTIVATION FOR WORK

- HIGH POWER DENSITY IMPELLERS USED IN ROCKET PROPELLION TURBOPUMPS PRODUCE SIGNIFICANT UNSTEADY INTERACTIONS BETWEEN IMPELLER BLADES AND DOWNSTREAM DIFFUSER VANES.

- PERFORMANCE DEGRADATION

- REDUCED COMPONENT LIFE (RELIABILITY)

- GOAL: USE TANDEM BLADE CONCEPT TO IMPROVE IMPELLER PERFORMANCE AND/OR DISCHARGE FLOW UNIFORMITY
- Predict flow in MSFC pump technology team baseline impeller

- Numerically study several tandem blade configurations

- Parametrically vary slot location and clocking

- Perform numerical parametrics with AeroVisc

- Judge improvements based on performance and discharge flow uniformity
Aerovisc Numerics

- **Formulation**
  - Reynolds Stress Averaged Navier-Stokes Equations
  - Cartesian Primitive Variable Approach
  - Strongly Conservative, Colocated Form
  - k-ε and ARS Turbulence Models With Wall Functions
  - 2 Layer Turbulence Model

- **Discretization**
  - Flux Element Based Finite Volume Method
  - General Non-Orthogonal Boundary-Fitted Structured Grid
  - Advection Schemes Have Two Components
    - Upwind Skew Scheme (UDS, MWS, LPS) ==> Transverse Gradients
    - Physically Based Correction Term (PAC) ==> Streamwise Gradients
  - Rhie Type 4th Order Pressure Redistribution
    - Pressure /Velocity Coupling For I.C. Flows

- **Solver**
  - Vectorized Gauss-Siedel or Incomplete Cholesky Base Solver
  - Multigrid (Large Grids) and Block Correction (High Aspect Ratio Grids)
GenCorp Demonstrated Code Capabilities

- Applicable Flow Ranges
  - Incompressible
  - Subsonic, Transonic and Supersonic
  - Laminar, Turbulent or Inviscid
  - Multi-Component
  - 2-Phase (Solid Particle / Gas)
- Coriolis and Centrifugal Terms For Turbomachinery Applications
- Conjugate Heat Transfer or Specified Wall Temperature / Flux
- Flexible Geometric Modeling Features
  - Arbitrary Periodicity
  - Multiple Blocked Regions
  - Grid Embedding or Attaching
- Fixed, Moving or Rotating Walls
- Variable Fluid and / or Solid Properties
The Krain Impeller – Introduction

- 24 Blades, 22,363 rpm, 4.7 Design Pressure Ratio, 4.0 kg/s Total Mass Flow
- Computational Domain Includes Inlet, Tip Region, and Diffuser
- Absolute Frame Total Pressure and Total Temperature at Inlet, Mass Flow Specified at Outlet, Log-Law Used at Walls
- 80,000 Nodes, Second-Order-Accurate Skew Upwind Scheme, Coupled Multigrid
- Note: Inlet Geometry Was Estimated
The Krain Impeller – Comparisons to Data

- Total Run Time About 150 Hours on a Personal Iris 4D/25
- Measured Total Pressure Ratio Was About 4.1, the Code Predicted 4.26

Meridional Velocity Components at Outlet Data (Left) Calculation (Right)

Circumferentially Averaged Shroud Static Pressure
INDUCER CODE VALIDATION

PLANE B, X=2.725 IN., ANGLE=-133.755 DEG.
POSITION 5, RADIUS= 2.365 IN.

LEGEND
- DATA
- SECA
- RKDN
- LERC
- AERO
- SRA
- ARC
INDUCER CODE VALIDATION

PLANE B, X=2.725 IN., ANGLE=-133.755 DEG.
POSITION 5, RADIUS= 2.365 IN.
- Fluid: Hydrogen
- 6 Full + 6 Partial
- N = 30108 RPM
- Tip Speed = 1857 fps
- Outlet Blade Angle = 49.5 deg.
- Specific Speed = 1141
- Head Coeff. = 0.572
BASELINE IMPELLER, VISCOSOUS SOLUTION

Values mass avg'd from hub to shroud

Utip = 1858 fps, 60 Deg. mass flow split 45% - 55%
PARTIAL

HUB

FULL

SHROUD

\[ r / r_{tip} = 0.99 \]
\[ \text{avrg} = 8 \text{ deg.} \]

\[ r / r_{tip} = 1.05 \]
\[ \text{avrg} = 9 \text{ deg} \]

\[ r / r_{tip} = 1.10 \]
\[ \text{avrg} = 9 \text{ deg} \]

ROTATION

\[ \Delta \text{DEG} \]

13 DEG

6.5 DEG

0.0 DEG

6.5 DEG

13 DEG.
IMPELLER DOWNSTREAM PROFILE
BASELINE CONFIGURATION 34K NODES

\[ r/rtip = 0.99 \]
\[ \text{avrg} = 1221 \text{ psi} \]

\[ r/rtip = 1.05 \]
\[ \text{avrg} = 1331 \text{ psi} \]

\[ r/rtip = 1.10 \]
\[ \text{avrg} = 1389 \text{ psi} \]
Definitions

\[ c = r_3 - r_2 \]
\[ a = r_2 - r_1 \]
\[ s = r_1 \times (\text{delta theta})_{\text{full - partial}} \]
\[ h = r_1 \times (\text{delta theta})_{\text{shift}} \]
FULL BLADE

PARTIAL BLADE

DISCHARGE INVESTIGATION LOCATIONS

GENCORP
AEROJET
Propulsion Division

R/Rtip = 1.10
R/Rtip = 1.05
R/Rtip = 1.02
R/Rtip = 1.0

0.14
0.09
0.07
0.05

a/c Locations
CONFIGURATIONS AND STATUS

Propulsion Division

- a/c = 0.0
  - h/s = 0.0
  - Complete

- a/c = 0.15
  - h/s = 0.0
  - Complete

- a/c = 0.14
  - h/s = 0.0
  - Complete
  - h/s = 0.033
  - Complete
  - h/s = 0.067
  - Complete

- a/c = 0.09
  - h/s = 0.0
  - Complete

- a/c = 0.07
  - h/s = 0.0
  - Complete

- a/c = 0.05
  - h/s = 0.0
  - Complete

100x8x43 Main Grid: 36x8x28 and 36x8x40 Embedded Grids
GenCorp Aerojet
Propulsion Division

Impeller Discharge Circ. Cm Distribution
Mass Averaged Hub to Tip

$H/H_{tip} = 1.05$
IMPPELLER DISCHARGE CIRC. Cm DISTRIBUTION
Mass Averaged Hub to Tip

\[ R/R_{tip} = 1.10 \]
BASELINE IMPELLER, VISCOS SOLUTION

Values mass avg'd from hub to shroud.

U_{tip} = 1858 \text{ fps}, \quad \text{Wrd Deg, mass flow split} = 45\% - 55\%.
GenCorp
Aerojet
Propulsion Division

\( \alpha/c = 0.14, \; h/s = 0.067, \text{ VISCOUS SOLUTION} \)

Values mass avg'd from hub to shroud

\( U_{tip} = 1858 \text{ fps}, \; \text{Miss. Deg. mass flow split: 42\% - 58\%} \)
BASELINE vs. a/c = 0.14; 1 Deg. CLOCKED

[Graph and chart showing data comparison]
Cm DISTRIBUTION JUST DOWNSTREAM OF SLOT LOCATION
R/RTIP = 0.94

BASELINE VISCOS

Partial Blade Location

Full Blade Location

a/c=0.14; h/s=0.067
CONCLUSIONS

- FLOW NON-UNIFORMITY FOR THE BASELINE CASE WAS FOUND TO BE SIGNIFICANT

- SIGNIFICANT GAINS IN UNIFORMITY CAN POTENTIALLY BE ACHIEVED WITH TANDEM BLADE CONCEPT

- CLOCKING WAS SHOWN TO BE THE PRIMARY DRIVER FOR IMPELLER DISCHARGE FLOW UNIFORMITY

- CARE MUST BE TAKEN IN THE DESIGN PROCESS TO ACHIEVE POSITIVE PERFORMANCE GAINS RATHER THAN LOSSES

- GRID EMBEDDING CAN SIGNIFICANTLY REDUCE MODEL SIZE WHILE NOT SACRIFICING FLOW GRADIENT RESOLUTION

FURTHER OPTIMIZATION REQUIRED FOLLOWED BY GRID REFINEMENT
Three-dimensional flow phenomena in a shrouded inducer have been studied with a three-dimensional Navier-Stokes method.

The details of the three-dimensional flow structure inside the inducer at design and off-design conditions are analyzed and the results are compared with some flow visualization results obtained at the California Institute of Technology.
THREE-DIMENSIONAL FLOW FIELDS INSIDE A SHROUDED INDUCER AT DESIGN AND OFF-DESIGN CONDITIONS (CFD STUDY)

C. HAH, O. KWON, AND D. A. GREENWALD

NASA LEWIS RESEARCH CENTER

R. GARCIA

NASA MARSHALL SPACE FLIGHT CENTER
OBJECTIVES

3-D FLOW STRUCTURE AT LOW FLOW COEFFICIENT

FORMATION OF BACKFLOWS

LATERAL FORCES IN AXIAL FLOW INDUCERS
OVERALL APPROACHES

EXPERIMENTAL INVESTIGATION

CALIFORNIA INSTITUTE OF TECHNOLOGY

PH.D THESIS BY BHATTACHARYYA (PROF. ACOSTA)

ASME PAPER BY BHATTACHARYYA, ACOSTA, BRENNEN
AND CAUGHEY ON "BACKFLOW IN INDUCER"

CFD INVESTIGATION

CURRENT SUBJECT
INDUCER 10

SHROUDED INDUCER (3 BLADES)

BLADE ANGLE = 12 DEGREES

TIP RADIUS = 1.9115 INCHES

TESTED FLOW COEFFICIENTS (0.074 & 0.041)
LATERAL FORCE AT VARIOUS FLOW COEFFICIENTS
(FROM CALTECH STUDY)
CALTECH SHROUDED INDUCER

296
FLOW COEFF. = 0.074

FLOW ON SHROUD (FROM CALTECH STUDY)
FLOW COEFF. = 0.074

FLOW ON HUB (FROM CALTECH STUDY)
VELOCITY VECTORS NEAR SHROUD (FLOW COEFF.=0.074)
VELOCITY VECTORS NEAR HUB (FLOW COEFF.=0.074)
FLOW COEFF. = 0.041

FLOW ON SHROUD (FROM CALTECH STUDY)
FLOW COEFF. = 0.041

FLOW ON HUB (FROM CALTECH STUDY)
ELOCITY VECTORS NEAR SHROUD (FLOW COEFF.=0.041)
VELOCITY VECTORS NEAR HUB (FLOW COEFF.=0.041)
OIL FLOW ON HUB (FLOW COEFF.=0.041)
OBSERVATIONS FROM CURRENT EXERCISE

MAJOR PHENOMENA WELL CALCULATED

FURTHER QUANTITATIVE COMPARISON NEEDED

CAVITATION MODELING NECESSARY
EFFECTS OF CURVATURE AND ROTATION ON TURBULENCE
IN THE
NASA LOW-SPEED CENTRIFUGAL COMPRESSOR IMPELLER

Joan G. Moore and John Moore
Mechanical Engineering Department
Virginia Polytechnic Institute and State University
Blacksburg, Virginia

The flow in the NASA Low-Speed Impeller is affected by both
curvature and rotation. The flow curves due to

a) geometric curvature, e.g. the curvature of the hub and shroud
profiles in the meridional plane and the curvature of the
backswept impeller blades, and

b) secondary flow vortices, e.g. the tip leakage vortex.

Is the turbulence and effective turbulent viscosity in the
impeller significantly affected by the curvature and rotation?
Do these changes significantly affect the overall
three-dimensional flow development?
And do they also impact on the overall performance of the
impeller?

An answer to these questions is obtained by comparing two
predictions of the flow in the impeller - one with, and one
without modification to the turbulent viscosity due to rotation
and curvature.

Some experimental and theoretical background for the modified
mixing length model of turbulent viscosity will also be presented.
Prediction of Flow in the Impeller using a Mixing Length Model for Turbulent Viscosity

\[ \nu_t = \rho L^2 \frac{du}{dy} \]

\[ L = \text{smaller of } 0.41y, 0.08 \delta \]

Van Driest correction used in 0.41y region

Calculated secondary flow velocity vectors on cross-sectional plane three quarters of the way through the impeller. Note contour of zero primary velocity near the shroud wall.

Throughflow velocity vectors projected onto the unrolled blade-to-blade plane at 80% of blade height.

Meridional view of the velocity vectors at mid-passage.
The effects of tip leakage dominate the calculated flow. The calculation was made with a tip gap which varied from 2.6% of blade height at the impeller inlet to 4% at the impeller exit. The secondary flow velocity vectors in the cross-sectional plane are dominated by the flow over the blade tip and the resulting vortex in the passage. The velocity vectors in the meridional view at mid passage show the extent of the backflow region near the shroud due to the tip leakage. The vectors in the blade-to-blade view show the penetration of the high loss/low velocity tip leakage fluid at 80% of the blade height.

The entropy on four cross-sectional planes show the build up of the losses to be dominated by the tip leakage flow with the high loss fluid covering the pressure-side/shroud quarter of the passage at the impeller exit.
Background of the Modification of the Mixing Length due to Curvature and Rotation

MIXING LENGTH MODEL

\[ \nu_t = \rho L^2 \frac{d\mu}{dy} \]

= Equilibrium form of one equation k-L model

Turbulence Kinetic Energy Equation

\[ \rho \mu \cdot \nabla \mu - \nabla \cdot \nu_{\text{eff}} \mu = P_k - D_k \]

Convection Diffusion Production Dissipation

Equilibrium

\[ P_k = D_k \]

\[ \nu_t = \rho L^2 \left[ \frac{u_{ij}}{\partial x_j} \right]^{1/2} \]

The mixing length turbulence model may be viewed as the equilibrium form of a one-equation turbulence model. It can be derived by setting the production of turbulence kinetic energy equal to its dissipation.

The modification to the turbulent viscosity due to curvature and rotation may be derived by considering the Reynolds stress equations. So (1978), derived the reduction by considering the situation when rotation and curvature act in a plane. He used the equilibrium form (production = dissipation) of the three normal stress equations and one shear stress equation.
MIXING LENGTH MODEL WITH CURVATURE AND ROTATION

\[ \nu_t = \rho (L_o F)^2 \frac{\partial u}{\partial y} \]

= Equilibrium form of 6 equation \( u'_i u'_j - L \) model

Reynolds Stress Equations

\[ \rho \partial_t \overline{u_i' u_j'} - \nabla \cdot (\rho \overline{u_i u_j}) \varepsilon_{\text{eff}} \nabla \overline{u_i' u_j'} = P_{ij} - D_{ij} \]

Convection Diffusion Production Dissipation

Equilibrium

\[ P_{ij} = D_{ij} \]

implies

\[ \nu_t = \rho (L_o F)^2 \frac{\partial u}{\partial y} \]

\[ F = F(\text{Gradient Richardson Number}) \]

MODIFICATION FACTOR FROM 2-D FLOW LITERATURE

\[ F = 1 - 8 \text{ Ri} \quad \text{for } \text{Ri} < 0 \]

\[ F = \frac{1}{1 + 8 \text{ Ri}} \quad \text{for } \text{Ri} > 0 \]

Curvature and Rotation Acting in a Plane (So)

\[ \text{Ri} = \frac{(2u/R - 2\Omega) (\partial u/\partial y + u/R - 2\Omega)}{(\partial u/\partial y - u/R)^2} \]

\( (R = \text{radius of curvature}) \)
Gillis et al. measured the six Reynolds stresses and the velocity profile on the suction side of a curved channel. The local turbulent viscosity and the local mixing length for either a zero equation (Prandtl mixing length) or one-equation (k-L) turbulence model may then be determined from their measurements.
Modified L Model using local Factor:

\[ L = L_0 F \]

\[ F = 1/(1 + 3Ri) \]

When \( L \), determined from the measurements, is compared to \( L \) from the model, to obtain \( \beta \), it is found that using a local modification factor results in the wrong shape for the \( L \) versus \( y \) profile through the boundary layer. A mean factor applied to \( L \) in the outer part of the boundary layer gives better results.
PREDICTED FLOW IN THE IMPELLER

using a CURVATURE/ROTATION
MODIFIED MIXING LENGTH MODEL FOR 3-D FLOW

\[ \mu_{e} = \rho L^{2} \frac{du}{dy} \]

\[ L = \text{smaller of } 0.41"y" \]

\[ 0.086 \bar{F} \]

Van Driest correction used in 0.41y region

\[ F = 1 - B \text{ Ri} \quad \text{for } \text{Ri} < 0 \]
\[ F = \frac{1}{1 + B \text{ Ri}} \quad \text{for } \text{Ri} > 0 \]

\[ B = 4 \]

\[ \text{Ri} = \left[ 2\varepsilon_{nkj}u_{k}u_{i}\frac{\partial u_{j}}{\partial x_{i}} - 2\varepsilon_{nkj}u_{k}u_{i}\Omega_{j} \right] \]
\[ \cdot \left[ \varepsilon_{nal} \varepsilon_{akn} \frac{\partial u_{i}}{\partial x_{n}} \left( \varepsilon_{kji} \frac{\partial u_{j}}{\partial x_{i}} - 2\Omega_{k} \right) \right] \]
\[ / \left[ \varepsilon_{nkj}u_{k}u_{i} \left( \frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}} \right) \right]^{2} \]

The mixing length model, with L modified by a mean factor for curvature and rotation, was then used to obtain another prediction of the flow in the NASA low speed centrifugal impeller. \( B=4 \) was used with a generalized 3-d form of the Richardson number which reduces to the correct 2-d form in 2-d situations.
Modification factor for the turbulent viscosity, $F^2$, 3/4ths of the way through the impeller.

The classical 2-d boundary layer modifications can be seen with enhancement due to rotation in the pressure side boundary layer, reduction due to rotation in the suction side boundary layer, and enhancement due to curvature on the concave hub wall. In the tip leakage vortex there is an enhancement due to rotation near mid-passage where the entropy gradient is in the same sense as in a pressure side boundary layer. The tip vortex also sees a reduction due to curvature near mid-height where the entropy gradient is in the same sense as for a shroud wall (convex wall) boundary layer.
Secondary flow velocity vectors and entropy contours on cross-sectional planes 3/4ths through, and near the exit of the impeller. Contour interval $\delta s = 2 \text{ J/kgK}$.

The character of the flow in the impeller and the mean (mass-averaged) levels of entropy were essentially the same as for the first prediction. The shape and location of the tip leakage vortex were slightly modified by the increased turbulence at mid-shroud and the reduced turbulence in the tip vortex near the pressure side near mid-height.
Comparison of Rotor Performance Predictions at 1920 RPM and 30 kg/s (66 lbm/s) with Preliminary Measurements

<table>
<thead>
<tr>
<th></th>
<th>NASA Design Prediction (Streamline Curvature)</th>
<th>Prediction Mixing Length Turb. Model</th>
<th>Prediction Curv/Rot Modified L</th>
<th>NASA Preliminary Measurements</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{t2}/P_0$</td>
<td>1.173</td>
<td>1.128</td>
<td>1.127</td>
<td>1.13</td>
</tr>
<tr>
<td>$n_{t-ti2}$</td>
<td>0.934</td>
<td>0.922</td>
<td>0.922</td>
<td>0.93</td>
</tr>
<tr>
<td>$\Delta H/U^2$</td>
<td>0.607</td>
<td>0.469</td>
<td>0.467</td>
<td>0.50</td>
</tr>
<tr>
<td>$M_{rel, tip}$</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
<td></td>
</tr>
<tr>
<td>$M_{rel2}$</td>
<td>0.20</td>
<td>0.272</td>
<td>0.272</td>
<td></td>
</tr>
<tr>
<td>$M_{abs2}$</td>
<td>0.287</td>
<td>0.244</td>
<td>0.244</td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td>0.763</td>
<td>0.843</td>
<td>0.846</td>
<td></td>
</tr>
</tbody>
</table>

Plane 1 is at the impeller inlet; Plane 2 is at the impeller exit.

TOTAL PRESSURE RISE IN THE IMPELLER

- NASA Preliminary Measurements

Predictions

- MEFP, Mixing Length Turbulence Model
- MEFP, Rot/Curv Modified L Model
- NASA Design (Streamline Curvature)

Both MEFP predictions compare well with the preliminary measurements made by NASA. Comparisons with preliminary measurements are included here because NASA has decided to reduce the tip clearance before making detailed measurements of flow in the impeller.

325
CONCLUSIONS

- A local modification factor due to curvature and rotation is inappropriate in a mixing length model.
  
  Data suggests that an average factor for the layer is appropriate.

- Typical modification factors for the turbulent viscosity in the NASA Low Speed Centrifugal Impeller were 2-4.

- Changes to the predicted flow were small:
  
  Slight increase in secondary flow velocities on the pressure side.
  Slight change in shape and relocation of the tip leakage vortex.

- No change in overall performance.

- The NASA Low Speed Centrifugal Impeller is a good test case for verification of 3-d N-S solvers which include mixing length, one-equation, or two-equation turbulence models.

- The Upwind Control Volume* discretization used in MEFP makes this code a good vehicle for the evaluation of turbulence models in 3-d flow. Calculated vortex structure is sensitive to small changes in the turbulence model.

* The Upwind Control Volume approach introduces no numerical mixing either directly through second or fourth order smoothing or indirectly through inconsistencies in the discretization of the convection term such as upwind differencing.
EFFECTS OF CURVATURE AND ROTATION ON TURBULENCE IN THE NASA LOW-SPEED CENTRIFUGAL COMPRESSOR IMPELLER

Joan G. Moore and John Moore

Mechanical Engineering Department
Virginia Polytechnic Institute and State University
Blacksburg, Virginia

The flow in the NASA Low-Speed Impeller is affected by both curvature and rotation. The flow curves due to

a) geometric curvature, e.g. the curvature of the hub and shroud profiles in the meridional plane and the curvature of the backswept impeller blades, and

b) secondary flow vortices, e.g. the tip leakage vortex.

Is the turbulence and effective turbulent viscosity in the impeller significantly affected by the curvature and rotation? Do these changes significantly affect the overall three-dimensional flow development? And do they also impact on the overall performance of the impeller?

An answer to these questions is obtained by comparing two predictions of the flow in the impeller - one with, and one without modification to the turbulent viscosity due to rotation and curvature.

Some experimental and theoretical background for the modified mixing length model of turbulent viscosity will also be presented.
Prediction of Flow in the Impeller
using a Mixing Length Model for Turbulent Viscosity

\[ \nu_t = \rho L^2 \frac{du}{dy} \]

\[ L = \text{smaller of } 0.41 y, 0.08 s \]

Van Driest correction used in 0.41y region

Calculated secondary flow velocity vectors on cross-sectional plane three quarters of the way through the impeller. Note contour of zero primary velocity near the shroud wall.

Throughflow velocity vectors projected onto the unrolled blade-to-blade plane at 80% of blade height.

Meridional view of the velocity vectors at mid-passage.
The effects of tip leakage dominate the calculated flow. The calculation was made with a tip gap which varied from 2.6% of blade height at the impeller inlet to 4% at the impeller exit. The secondary flow velocity vectors in the cross-sectional plane are dominated by the flow over the blade tip and the resulting vortex in the passage. The velocity vectors in the meridional view at mid passage show the extent of the backflow region near the shroud due to the tip leakage. The vectors in the blade-to-blade view show the penetration of the high loss/low velocity tip leakage fluid at 80% of the blade height.

The entropy on four cross-sectional planes show the build up of the losses to be dominated by the tip leakage flow with the high loss fluid covering the pressure-side/shroud quarter of the passage at the impeller exit.
Background of the Modification
of the Mixing Length
due to Curvature and Rotation

**MIXING LENGTH MODEL**

$$\nu_t = \rho L^2 \frac{du}{dy}$$

= Equilibrium form of one equation k-L model

Turbulence Kinetic Energy Equation

$$\rho u \cdot \nabla k - \nabla \cdot \nu_{eff} \nabla k = P_k - D_k$$

Convection Diffusion Production Dissipation

Equilibrium

$$P_k = D_k$$

$$\nu_t = \rho L^2 \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \left( \frac{\partial u_i}{\partial x_j} \right)^{1/2}$$

$$\nu_t = \rho L^2 \frac{du}{dy}$$

The mixing length turbulence model may be viewed as the equilibrium form of a one-equation turbulence model. It can be derived by setting the production of turbulence kinetic energy equal to its dissipation.

The modification to the turbulent viscosity due to curvature and rotation may be derived by considering the Reynolds stress equations. So (1978), derived the reduction by considering the situation when rotation and curvature act in a plane. He used the equilibrium form (production = dissipation) of the three normal stress equations and one shear stress equation.
MIXING LENGTH MODEL WITH CURVATURE AND ROTATION

\[ \nu_t = \rho(L_0 F)^2 \frac{d\bar{u}}{dy} \]

= Equilibrium form of 6 equation \( u_i^2 u_j^j - L \) model

Reynolds Stress Equations

\[ \rho \nabla \cdot (u_i' u_j') - \nabla \cdot (\nu / \sigma)_{\text{eff}} \nabla (u_i' u_j') = P_{ij} - D_{ij} \]

Convection Diffusion Production Dissipation

Equilibrium

\[ P_{ij} = D_{ij} \]

\[ \Rightarrow \]

\[ \nu_t = \rho (L_0 F)^2 \frac{d\bar{u}}{dy} \]

F = F(Gradient Richardson Number)

MODIFICATION FACTOR FROM 2-D FLOW LITERATURE

\[ F = 1 - \beta \text{ Ri} \quad \text{for } \text{Ri} < 0 \]

\[ F = 1/(1 + \beta \text{ Ri}) \quad \text{for } \text{Ri} > 0 \]

Curvature and Rotation Acting in a Plane (So)

\[ \text{Ri} = \frac{(2u/R - 2\Omega) (\partial u/\partial y + u/R - 2\Omega)}{(\partial u/\partial y - u/R)^2} \]

(R = radius of curvature)
Modified Mixing Length from Experimental Data
2-D Flow in a Curved Channel (Gillis et al.)

\[-\rho u'v' = \mu_t \frac{du}{dy},\]

\[\circ \quad \mu_t = \rho L^2 \frac{du}{dy}\]

\[\times \quad \mu_t = C_{\mu} \rho L \kappa^{1/2}\]

Gillis et al. measured the six Reynolds stresses and the velocity profile on the suction side of a curved channel. The local turbulent viscosity and the local mixing length for either a zero equation (Prandtl mixing length) or one-equation (k-L) turbulence model may then be determined from their measurements.
Modified L Model using local Factor:

\[ L = L_0 F \]

\[ F = \frac{1}{1 + BR_i} \]

Modified L Model using mean Factor:

\[ L = \text{smaller of } 0.41"y" \]

\[ 0.086 \int_0^5 F \, dy \]

When \( L \), determined from the measurements, is compared to \( L \) from the model, to obtain \( B \), it is found that using a local modification factor results in the wrong shape for the \( L \) versus \( y \) profile through the boundary layer. A mean factor applied to \( L \) in the outer part of the boundary layer gives better results.
PREDICTED FLOW IN THE IMPELLER

using a CURVATURE/ROTATION

MODIFIED MIXING LENGTH MODEL FOR 3-D FLOW

\[ \nu_t = \rho L^2 \frac{\partial y}{\partial y} \]

\[ L = \text{smaller of } 0.41"y" \]

\[ 0.088 \frac{F}{\bar{F}} \]

Van Driest correction used in 0.41y region

\[ F = 1 - B \text{ Ri} \quad \text{for } \text{Ri} < 0 \]

\[ F = \frac{1}{1 + B \text{ Ri}} \quad \text{for } \text{Ri} > 0 \]

\[ B = 4 \]

\[ \text{Ri} = \left[ 2\varepsilon_{nkj}u_k u_i \partial u_j / \partial x_i - 2\varepsilon_{nlk} \varepsilon_{kji} u_l u_i n_j \right] \]

\[ \times \left[ \varepsilon_{mla} \varepsilon_{akn} u_m (\varepsilon_{kji} \partial u_j / \partial x_i - 2n_k) \right] \]

\[ / \left[ \varepsilon_{nkj} u_k u_i (\partial u_i / \partial x_j + \partial u_j / \partial x_i) \right]^2 \]

The mixing length model, with L modified by a mean factor for curvature and rotation, was then used to obtain another prediction of the flow in the NASA low speed centrifugal impeller. B=4 was used with a generalized 3-d form of the Richardson number which reduces to the correct 2-d form in 2-d situations.
Modification factor for the turbulent viscosity, $F^2$, 3/4ths of the way through the impeller.

The classical 2-d boundary layer modifications can be seen with enhancement due to rotation in the pressure side boundary layer, reduction due to rotation in the suction side boundary layer, and enhancement due to curvature on the concave hub wall. In the tip leakage vortex there is an enhancement due to rotation near mid-passage where the entropy gradient is in the same sense as in a pressure side boundary layer. The tip vortex also sees a reduction due to curvature near mid-height where the entropy gradient is in the same sense as for a shroud wall (convex wall) boundary layer.
Secondary flow velocity vectors and entropy contours on cross-sectional planes 3/4ths through, and near the exit of the impeller. Contour interval $\delta s = 2 \text{ J/kgK}$.

The character of the flow in the impeller and the mean (mass-averaged) levels of entropy were essentially the same as for the first prediction. The shape and location of the tip leakage vortex were slightly modified by the increased turbulence at mid-shroud and the reduced turbulence in the tip vortex near the pressure side near mid-height.
Comparison of Rotor Performance Predictions at 1920 RPM and 30 kg/s (66 lbm/s) with Preliminary Measurements

<table>
<thead>
<tr>
<th></th>
<th>NASA Design (Streamline Curvature)</th>
<th>Prediction Mixing Length Turb. Model</th>
<th>Prediction Curv/Rot Modified L</th>
<th>NASA Preliminary Measurements</th>
</tr>
</thead>
<tbody>
<tr>
<td>$p_{t2}/p_o$</td>
<td>1.173</td>
<td>1.128</td>
<td>1.127</td>
<td>1.13</td>
</tr>
<tr>
<td>$n_{t-ti2}$</td>
<td>0.934</td>
<td>0.922</td>
<td>0.922</td>
<td>0.93</td>
</tr>
<tr>
<td>$\Delta H/U^2$</td>
<td>0.607</td>
<td>0.469</td>
<td>0.467</td>
<td>0.50</td>
</tr>
<tr>
<td>$M_{reli,tip}$</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
<td></td>
</tr>
<tr>
<td>$M_{rel2}$</td>
<td>0.20</td>
<td>0.272</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$M_{abs2}$</td>
<td>0.287</td>
<td>0.244</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction</td>
<td>0.763</td>
<td>0.843</td>
<td>0.846</td>
<td></td>
</tr>
</tbody>
</table>

Plane i is at the impeller inlet; Plane 2 is at the impeller exit.

TOTAL PRESSURE RISE IN THE IMPELLER

- - - NASA Preliminary Measurements

Predictions

- MEFP, Mixing Length Turbulence Model
- - MEFP, Rot/Curv Modified L Model
- - NASA Design (Streamline Curvature)

Both MEFP predictions compare well with the preliminary measurements made by NASA. Comparisons with preliminary measurements are included here because NASA has decided to reduce the tip clearance before making detailed measurements of flow in the impeller.
CONCLUSIONS

- A local modification factor due to curvature and rotation is inappropriate in a mixing length model.
  Data suggests that an average factor for the layer is appropriate.

- Typical modification factors for the turbulent viscosity in the NASA Low Speed Centrifugal Impeller were 2-4.

- Changes to the predicted flow were small:
  Slight increase in secondary flow velocities on the pressure side.
  Slight change in shape and relocation of the tip leakage vortex.

- No change in overall performance.

- The NASA Low Speed Centrifugal Impeller is a good test case for verification of 3-d N-S solvers which include mixing length, one-equation, or two-equation turbulence models.

- The Upwind Control Volume* discretization used in MEFP makes this code a good vehicle for the evaluation of turbulence models in 3-d flow. Calculated vortex structure is sensitive to small changes in the turbulence model.

* The Upwind Control Volume approach introduces no numerical mixing either directly through second or fourth order smoothing or indirectly through inconsistencies in the discretization of the convection term such as upwind differencing.
Computational Fluid Dynamic Design of Rocket Engine Pump Components
by
Wei-Chung Chen, George H. Prueger, Daniel C. Chan and Anthony H. Eastland
Rocketdyne Division, Rockwell International Corp.

Integration of CFD for design and analysis of turbomachinery components is needed as the requirements of pump performance and reliability become more stringent for the new generation of rocket engine. A fast grid generator, designed specially for centrifugal pump impeller, which allows a turbomachinery designer to use CFD to optimize the component design will be presented. The CFD grid is directly generated from the impeller blade G-H blade coordinates. The grid points are first generated on the meridional plane with the desired clustering near the end walls. This is followed by the marching of grid points from the pressure side of one blade to the suction side of a neighboring blade. This fast grid generator has been used to optimize the consortium pump impeller design. A grid dependency study has been conducted for the consortium pump impeller. Two different grid sizes, one with 10,000 grid points and one with 80,000 grid points were used for the grid dependency study. The effects of grid resolution on the turnaround time, including the grid generation and completion of the CFD analysis, is discussed. The impeller overall mass average performance is compared for different designs. Optimum design is achieved through systematic change of the design parameters. In conclusion, it is demonstrated that CFD can be effectively used not only for flow analysis but also for design and optimization of turbomachinery components.
CFD DESIGN OF ROCKET ENGINE COMPONENT

by

Wei-Chung Chen, George H. Prueger
Daniel C. Chan, Anthony H. Eastland

Rocketdyne Division
Rockwell International Corporation

Presented at NASA Marshall Space Flight Center
Tenth Workshop for Computational Fluid Dynamic Applications
in Rocket Propulsion
April 28-30, 1992
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

TYPICAL TURBOPUMP LAYOUT

- KEY COMPONENTS
  - INDUCER, STATOR, IMPELLER, VOLUTE, TURBINE, BEARING
  - IMPELLER SERVES AS THE HEART OF PUMP

- ADVANCED IMPELLER DESIGNS REQUIRE HIGH HEAD COEFFICIENTS
  - RELIABILITY AND COST REQUIREMENTS LIMIT MAXIMUM ALLOWABLE TIP SPEED
  - LOW COST REQUIRES MINIMUM NUMBER OF STAGES

- HIGH PUMP HEAD COEFFICIENTS INCREASE FLOW TURNING AND DIFFUSION
  - INCREASE EXIT FLOW NON-UNIFORMITY

- CFD INCORPORATED FOR PUMP IMPELLER DESIGN
  - IDENTIFY FLOW PROBLEMS INSIDE IMPELLER PASSAGE AND DISCHARGE
  - OPTIMIZE IMPELLER CONFIGURATION DURING DESIGN PROCESS
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT
CONSTRUCTION OF IMPELLER BLADE

- GENERATE BLADE MEANLINE G-H CURVE ALONG EACH STREAMLINE
  - REDUCE 3-D PROBLEM TO 2-D COORDINATE SYSTEM
  - EACH STREAMLINE INDEPENDENTLY GENERATED TO MATCH FLOW FIELD
  - CHANGE BLADE WRAP ANGLE TO CONTROL SOLIDITY
  - STACK BLADE TO ACHIEVE OPTIMUM IMPELLER PERFORMANCE

- CONSTRUCT BLADE SURFACE FROM MEANLINE COORDINATES
  - SURFACE COORDINATE GENERATED ACCORDING TO BLADE THICKNESS DISTRIBUTION
  - FLEXIBILITY OF USING MEANLINE, PRESSURE SIDE, SUCTION SIDE OR HYBRID FAIRING

- SURFACE INFORMATION DIRECTLY USED FOR
  - BLADE LAYOUT IN CATIA MODEL
  - CFD GRID GENERATION
IMPELLER MEANLINE BLADE GENERATION

CONSORTIUM PUMP IMPELLER STREAMLINE DEFINITION (ITERATION 22)

CONSORTIUM PUMP IMPELLER BLADE G-H CURVE (ITERATION 22)
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

FAST GRID GENERATOR

- USE BLADE SURFACE PRESSURE AND SUCTION SURFACE G-H INFORMATION
  - BOTH FULL AND PARTIAL BLADES

- GENERATE GRID IN MERIDIONAL PLANE
  - DETERMINE L.E. TO T.E. STATION NUMBER
  - SELECT HUB TO TIP STREAMLINE NUMBER

- INTERPOLATE BLADE SURFACE COORDINATES TO GRID MESH POINTS
  - REQUIRE SURFACE INTERPOLATION

- CREATE 3-D GRID POINTS BETWEEN TWO BLADE SURFACES

- EXTEND GRID POINTS OUTWARDS
  - ACCORDING TO INLET AND OUTLET BLADE ANGLE DISTRIBUTION

- H-GRID ALGEBRAICALLY GENERATED
  - POISSON GRID Smoother Incorporated

Rockwell International
Rocketdyne Division
DEVELOPMENT OF 10K IMPELLER GRID

COMPUTATIONAL GRID

COMPUTATIONAL GRID

Rockwell International
Rocketdyne Division
DEVELOPMENT OF 80K IMPELLER GRID

COMPUTATIONAL GRID

AXIAL DISTANCE

RADIAL DISTANCE

COMPUTATIONAL GRID

MERIDIONAL DISTANCE

CIRCUMFERENTIAL ANGLE

Rockwell International
Rocketdyne Division
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

BOUNDARY AND INITIAL CONDITIONS

- **BOUNDARY CONDITIONS AT INLET**
  - MERIDIONAL AND TANGENTIAL VELOCITY PRESCRIBED FROM MEASUREMENT OR PREDICTION

- **BOUNDARY CONDITION AT OUTLET**
  - VELOCITY EXTRAPOLATED FROM INTERIOR POINTS
  - MASS AND ANGULAR MOMENTUM CONSERVED

- **ALONG BLADE SURFACES AND END WALLS**
  - NO-SLIP BOUNDARY CONDITION IMPOSED
  - SLIP END WALL EFFECTS STUDIED

- **PERIODICITY APPLIED AT INLET AND OUTLET IN BLADE-TO-BLADE DIRECTION**

- **INITIAL CONDITIONS**
  - UNIFORM VELOCITY ASSUMED AT EACH STREAMWISE STATION
  - VELOCITY BASED ON 1-D PREDICTION
  - FLOW DIRECTION IS ALIGNED WITH LOCAL GRID ORIENTATION
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT
ACCURACY OF CFD RESULTS

- FLOW SOLVER REACT3D
  - ROCKETDYNE ELLIPTIC ANALYSIS CODE FOR TURBOMACHINERY USED
  - VALIDATED FOR INDUCTER AND TURBINE PERFORMANCE

-COMPARE TO ONE-DIMENSIONAL PROGRAM FOR IMPELLER PERFORMANCE
  - IMPELLER HYDRO EFFICIENCY AGREE WITH 1-D PROGRAM (94.5%)
  - EULER HEAD 8% HIGHER THAN THAT OF 1-D PREDICTION

- EULER HEAD DISCREPENCY ATTRIBUTED TO DIFFERENCES IN DEVIATION ANGLE AND BLOCKAGE
  - 1-D MODEL EXTRAPOLATED TO HIGH HEAD COEFFICIENT
  - UNCERTAINTY IN SOLIDITY CALCULATION WITH PARTIAL BLADE
  - SIMPLIFIED BLADE TRAILING EDGE MODELING IN REACT3D

- WATER TEST PLANNED TO RESOLVE THESE ISSUES
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

DISCUSSION OF GRID DEPENDENCY

- 2 GRID SIZE USED FOR IMPELLER ANALYSIS
  - 10K (59X14X11, STREAMWISE X BLADE-TO-BLADE X HUB-TO-TIP)
  - 80K (119X30X23, STREAMWISE X BLADE-TO-BLADE X HUB-TO-TIP)
- GRID GENERATED ON APOLLO WORKSTATION
  - 2 CPU MINUTES FOR 10K CASE
  - 10 CPU MINUTES FOR 80K CASE
- CFD ANALYSIS ALSO CARRIED OUT ON APOLLO WORKSTATION
  - 4 CPU HOURS FOR 10K CASE
  - 15 CPU HOURS FOR 80K CASE
- COMPARISON OF CFD RESULTS BETWEEN 10K AND 80K
  - CONSISTENT RESULTS FOR BOTH BASELINE AND OPTIMUM DESIGN
  - EFFICIENCY AND PUMP HEAD WITHIN 2%
  - TRENDS IN EVALUATION CRITERIA SIMILAR
  - USE 10K FOR IMPELLER DESIGN OPTIMIZATION
  - USE 80K FOR FINAL OPTIMUM DESIGN FLOW ANALYSIS
## COMPARISON OF CFD SOLUTIONS

<table>
<thead>
<tr>
<th>CASE</th>
<th>HEAD (FT)</th>
<th>EFFICIENCY</th>
<th>FLOW SPLIT</th>
<th>IMPELLER DISCHARGE</th>
<th>CROSSOVER INLET</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>EC 1</td>
<td>EC 2</td>
</tr>
<tr>
<td>10K BASELINE</td>
<td>1256.6</td>
<td>94.2%</td>
<td>+/- 6%</td>
<td>3.26 DEG</td>
<td>4.35 DEG</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.142</td>
<td>0.0809</td>
</tr>
<tr>
<td>80K BASELINE</td>
<td>1256.5</td>
<td>94.1%</td>
<td>+/- 6.6%</td>
<td>5.48 DEG</td>
<td>7.67 DEG</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.148</td>
<td>0.0657</td>
</tr>
<tr>
<td>10K OPTIMUM</td>
<td>1255.6</td>
<td>94.2%</td>
<td>+/- 2.5%</td>
<td>2.00 DEG</td>
<td>2.996 DEG</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.121</td>
<td>0.0603</td>
</tr>
<tr>
<td>80K OPTIMUM</td>
<td>1272.9</td>
<td>94.8%</td>
<td>+/- 5%</td>
<td>2.36 DEG</td>
<td>5.516 DEG</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.142</td>
<td>0.0536</td>
</tr>
</tbody>
</table>

*HIGH ON FULL BLADE PRESSURE SURFACE TO PARTIAL BLADE SUCTION SURFACE*
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT
DISCUSSION OF END WALL EFFECTS

- 3 CASES USED FOR PARAMETRIC STUDY
  - CASE 1: ROTATING WALL WITH NO SLIP AT IMPELLER UPSTREAM AND DOWNSTREAM
  - CASE 2: SAME AS CASE 1 EXCEPT SLIP CONDITONS AT DOWNSTREAM
  - CASE 3: SAME AS CASE 1 EXCEPT STATIONARY SHROUD AT UPSTREAM
- SMALL CHANGE FOR ALL PARAMETERS EVALUATED
  - DIFFERENCE OF HEAD AND EFFICIENCY WITHIN 1%
  - SMALL CHANGE OF IMPELLER DISCHARGE CM AND CU DISTRIBUTION
- FOURTH CASE WITH STATIONARY DOWNSTREAM WALL DID NOT FULLY CONVERGE
  - UNCONVERGED RESULTS (RESIDUALS 10-2) SHOWED EXTENSIVE RECIRCULATION AT DISCHARGE HUB AND SHROUD

Rockwell International
Rocketdyne Division
IMPELLER EXIT CM DISTRIBUTION

OPTIMIZED GEOMETRY WITH VARIOUS
BOUNDARY CONDITIONS

LEGEND

○ ROTATING WALLS W/NO SLIP CONDITION AT DISCHARGE
● ROTATING WALLS W/SLIP CONDITION AT DISCHARGE
△ ROTATING WALLS W/NO SLIP CONDITION AT DISCHARGE
STATIONARY SHROUD INLET WALL

X, INCH

0.00

0.02

0.04

0.06

0.08

0.10

0.12

NON-DIMENSIONAL CM (CM/UTIP)
IMPELLER EXIT CU DISTRIBUTION
OPTIMIZED GEOMETRY WITH VARIOUS
BOUNDARY CONDITIONS

LEGEND

○ ROTATING WALLS W/NO SLIP CONDITION AT DISCHARGE
● ROTATING WALLS W/SLIP CONDITION AT DISCHARGE
▲ ROTATING WALLS W/NO SLIP CONDITION AT DISCHARGE
STATIONARY SHROUD INLET WALL

NON-DIMENSIONAL CU (CU/UTIP)

SHROUD

HUB

X, INCH

2.5
3.0
3.5

0.6
0.7
0.8
0.9
1.0
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

DESIGN PARAMETRIC CHANGES

- IMPPELLER DISCHARGE HUB-TO-TIP WIDTH
  - RANGE FROM 0.88 TO 1.20 INCHES

- IMPPELLER DISCHARGE BLADE ANGLE FROM TANGENTIAL
  - RANGE FROM 38 TO 54 DEGREES

- IMPPELLER AXIAL LENGTH FROM 2.38 TO 3.0 INCHES

- BLADE TOTAL WRAP ANGLES
  - 65 TO 90 DEGREES FOR TIP
  - 65 TO 115 DEGREES FOR HUB

- WRAP DIFFERENCE BETWEEN HUB AND TIP
  - RANGE FROM 0 TO 5.0 DEGREES

- ALL CHANGES ACHIEVE REQUIRED HEAD WITH CONSTANT RPM AND DIAMETER

Rockwell International
Rocketdyne Division
## Consortium Impeller Configuration Studied

<table>
<thead>
<tr>
<th>Key Parameter</th>
<th>Discharge B2 Width</th>
<th>Discharge Blade Angle</th>
<th>Axial Length</th>
<th>Total Wrap Tip (Hub)</th>
<th>Discharge Wrap Difference</th>
<th>CFD Grid Point</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>1.00</td>
<td>47.2</td>
<td>2.50</td>
<td>65 (85)</td>
<td>5</td>
<td>10000</td>
</tr>
<tr>
<td>Baseline</td>
<td>1.00</td>
<td>47.2</td>
<td>2.50</td>
<td>65 (85)</td>
<td>5</td>
<td>80000</td>
</tr>
<tr>
<td>Case 1</td>
<td>1.00</td>
<td>47.2</td>
<td>2.50</td>
<td>65 (85)</td>
<td>0</td>
<td>10000</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.88</td>
<td>54.0</td>
<td>2.38</td>
<td>55 (76)</td>
<td>3</td>
<td>10000</td>
</tr>
<tr>
<td>Case 3</td>
<td>1.12</td>
<td>39.0</td>
<td>2.62</td>
<td>77 (100)</td>
<td>2.7</td>
<td>10000</td>
</tr>
<tr>
<td>Case 4</td>
<td>1.20</td>
<td>34.4</td>
<td>2.70</td>
<td>90 (115)</td>
<td>5.0</td>
<td>10000</td>
</tr>
<tr>
<td>Case 5</td>
<td>1.00</td>
<td>47.2</td>
<td>3.00</td>
<td>72 (92)</td>
<td>5.0</td>
<td>10000</td>
</tr>
<tr>
<td>Optimum</td>
<td>1.12</td>
<td>38.0</td>
<td>2.82</td>
<td>83 (105)</td>
<td>2.6</td>
<td>10000</td>
</tr>
<tr>
<td>Optimum</td>
<td>1.12</td>
<td>38.0</td>
<td>2.82</td>
<td>83 (105)</td>
<td>2.6</td>
<td>80000</td>
</tr>
</tbody>
</table>
REACT USED TO OPTIMIZE HIGH PERFORMANCE IMPELLER

FLOW RECIRCULATION ELIMINATED AT IMPELLER DISCHARGE

BASELINE GEOMETRY 80K GRID IMPELLER DISCHARGE RADIAL VELOCITY (FPS)

OPTIMUM GEOMETRY 80K GRID IMPELLER DISCHARGE RADIAL VELOCITY (FPS)

PRESSURE (psf)

55100
42700
30300
17900
5500

Rockwell International
Rocketdyne Division
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

CONCLUSIONS

- A FAST GRID GENERATOR HAS BEEN DEVELOPED
  - USED FOR DESIGN OPTIMIZATION OF CONSORTIUM IMPELLER
  - USED FOR SSME HPFTP IMPELLER FOR CFD CODE VALIDATION
  - SUCCESSFULLY APPLIED FOR INDUCER

- CFD INCORPORATED INTO PUMP DESIGN PROCESS
  - IMPELLER PERFORMANCE EVALUATION CRITERIA DEVELOPED
  - OPTIMUM DESIGN ACHIEVED THROUGH CFD ANALYSIS
  - TURNAROUND TIME ACCEPTABLE FOR DESIGN PROCESS
CFD DESIGN OF ROCKET ENGINE PUMP COMPONENT

CONCLUSIONS (continued)

- CFD GRID DEPENDENCY
  - SMALL GRID SIZE ACCEPTABLE FOR PARAMETRIC STUDIES
  - FINE GRID REQUIRED (300K) TO FINALIZE GRID DEPENDENCY STUDY

- END WALL BOUNDARY EFFECTS
  - CHANGE OF IMPELLER HEAD AND EFFICIENCY WITHIN 1%
  - SMALL CHANGE OF IMPELLER DISCHARGE CM AND CU DISTRIBUTION

- TEST DATA REQUIRED
  - CONFIRM HIGH HEAD COEFFICIENT IMPELLER PERFORMANCE
  - VALIDATE CFD RESULTS FOR CENTRIFUGAL PUMP APPLICATION
SSME HPOTP IMPELLER BACKCAVITY CFD ANALYSIS

W.W. HSU and S.J. LIN

Rockwell International Corp., Rocketdyne Division
6633 Canoga Avenue, MS IA34
Canoga Park, California 91303

The ball bearings behind the SSME HPOTP preburner pump have a history of premature wear requiring their replacement. Extensive tests have been conducted in an attempt to identify the operating factors that contribute to the wear. It has been conjectured that the coolant inflow velocity swirl pattern can aid bearing operation by matching ball orbit speed and thus affect bearing life. However, control of the velocity distribution up to now could only be achieved by trial and error following hardware testing. Observation of hardware from recent flight and development operation led to the hypothesis that certain assemblies with more extensive grinding patterns on the backwall of the impeller for rotor balancing correlated with improved bearing wear.

To analytically evaluate the effect of cavity configuration on the flowfield, 3-D CFD analyses of various geometries was successfully executed using REACT3D. Height of the anti-vortex ribs on the stationary wall was varied, as was the configuration of the rotating wall, from smooth to simulations of various grindout patterns. The results obtained indicate the effects of the various geometries and provide valuable guidelines for cavity modification to optimize bearing cooling.
SSME HPOTP PREBURNER IMPELLER BACKCAVITY
CFD ANALYSIS

W.W. HSU, S.J. LIN
ROCKWELL INTERNATIONAl, ROCKETFYNE DIVISION
APRIL 1992
BACKGROUND

SSME HPOTP BALL BEARINGS #1 AND #2 BEHIND PREBURNER IMPELLER HAVE HISTORY OF WEAR AND PREMATURE REPLACEMENT.

EXTENSIVE ENGINE AND SUBCOMPONENT TESTS HAVE IDENTIFIED VARIOUS OPERATING FACTORS THAT AFFECT WEAR:

- **LUBRICATION**: CAGE COATINGS SUCH AS FEP, BRAYCOTE
- **AXIAL PRE-LOAD**: CORRECT SPRING STIFFNESS TO MAINTAIN PRE-LOAD
- **MATERIALS**: SILICON NITRIDE BALLS, PLATINGS
- **COOLANT FLOW**: MAINTAIN ADEQUATE VAPOR MARGIN

RECENT TESTS CORRELATED MORE EXTENSIVE ROTOR BALANCING GRINDOUTS ON IMPELLER REAR FACE WITH REDUCED BEARING WEAR.
PHASE II
HIGH PRESSURE OXYGEN TURBOPUMP
INCENTIVE FOR CFD ANALYSIS

COOLANT INFLOW SWIRL DISTRIBUTION CONJECTURED TO AFFECT BEARING WEAR

MATCHING BALL ORBIT SPEED REDUCES INFLOW RESISTANCE AND DRAG TORQUE ON BALLS AND CAGE

SUGGESTS DESIGN CHANGES TO UPSTREAM CAVITY COULD REDUCE BEARING WEAR

OPTIMIZE MAGNITUDE AND RADIAL DISTRIBUTION OF INLET SWIRL VELOCITY

ADJUST HEIGHT OF ANTI-VORTEX RIBS AND/OR SIZE OF IMPELLER GRINDOUTS TO ACHIEVE DESIRED DISTRIBUTION
CFD ANALYSIS OBJECTIVES

DEFINE VARIATION OF INLET SWIRL VELOCITY WITH GEOMETRY

EFFECT OF ANTI-VORTEX RIBS AND IMPELLER GRINDOUTS INDIVIDUALLY AND TOGETHER

UNDERSTAND FLOW WELL ENOUGH TO SUGGEST DESIGN CHANGES

NARROW CAVITY WITH HIGH WALL TANGENTIAL VELOCITY

RIBS ON STATIONARY WALL

HIGH VELOCITY JET AT INLET
CFD MODELING

REACT3D STEADY NAVIER-STOKES ANALYSIS, SINGLE ZONE

GRINDOUTS ANALYZED IN ROTATING FRAME OF REFERENCE
RIBS ANALYZED IN STATIONARY FRAME OF REFERENCE
QUASI-STEADY APPROACH PROPOSED FOR GRINDOUT-RIB COMBINATIONS

FLOWFIELD SIMPLIFICATIONS

SMALL LEAKAGE PATH PARALLEL TO BEARINGS IGNORED
EFFECTS OF ROTATING CAGE AND ROLLING BALLS NOT SIMULATED

GRID EXTENDED DOWNSTREAM BEYOND BEARING INLET
AIDS CONVERGENCE WITH POTENTIAL BACKFLOW
GEOMETRIES INVESTIGATED

FOUR BASIC GEOMETRIES TO BE INVESTIGATED (3 COMPLETE)

RIBBED STATIONARY WALL / SMOOTH ROTATING WALL
SMOOTH STATIONARY WALL / SMOOTH ROTATING WALL
SMOOTH STATIONARY WALL / ROTATING WALL WITH GRINDOUTS
RIBBED STATIONARY WALL / ROTATING WALL WITH GRINDOUTS

GRINDOUTS SIMULATED WITH SMOOTH INDENTATIONS ON ROTATING WALL

RADIAL HEIGHT MATCHED TO AVERAGE OBSERVED
TOTAL CIRCUMFERENTIAL EXTENT 180 DEG.

GEOMETRY VARIATIONS

RIB HEIGHTS 100%, 50%, 25% OF CURRENT DESIGN
GRINDOUTS WITH 4 LOBES 0.05" DEEP, 6 LOBES 0.1" DEEP
**CFD ANALYSIS CONDITIONS**

<table>
<thead>
<tr>
<th>GRID SIZE</th>
<th>MERIDIONAL DIRECTION I = 75 TO 85</th>
</tr>
</thead>
<tbody>
<tr>
<td>CIRCUMFERENTIAL DIRECTION J = 15 TO 30</td>
<td></td>
</tr>
<tr>
<td>NORMAL DIRECTION K = 15 TO 20</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>GEOMETRY</th>
<th>IMPELLER ROTATION 29141 RPM</th>
</tr>
</thead>
<tbody>
<tr>
<td>HUB SEAL RADIUS</td>
<td>1.67 INCH</td>
</tr>
<tr>
<td>SEAL GAP</td>
<td>0.005 INCH</td>
</tr>
<tr>
<td>HUB INNER RADIUS</td>
<td>1.06 INCH</td>
</tr>
<tr>
<td>CAVITY</td>
<td>AXIAL WIDTH 0.113 INCH</td>
</tr>
<tr>
<td>RIB HEIGHT</td>
<td>0.05 INCH</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>FLUID</th>
<th>LOX, DENSITY 57.4 LB/CUB. FT.</th>
</tr>
</thead>
<tbody>
<tr>
<td>FLOW RATE</td>
<td>10.8 LB/SEC</td>
</tr>
<tr>
<td>JET VELOCITY</td>
<td>AXIAL 245 FT/SEC (SEAL EXIT CHAMFERED)</td>
</tr>
<tr>
<td></td>
<td>TANGENTIAL 50% WHEEL SPEED (PHASE 1)</td>
</tr>
<tr>
<td></td>
<td>30% WHEEL SPEED (PHASE 2)</td>
</tr>
</tbody>
</table>

Rockwell International
Rocketdyne Division
RESULTS

REACT3D RESULTS QUALITATIVELY AS EXPECTED

GRINDOUTS SIGNIFICANTLY INCREASE INLET TANGENTIAL VELOCITY

RIBS SIGNIFICANTLY DECREASE INLET TANGENTIAL VELOCITY

STRONG VORTEX MOTIONS DRIVEN BY JET FROM DAMPING SEAL AND ROTATING WALLS

NO SIGNIFICANT IMPACT ON AXIAL VELOCITY

FLOW SEPARATIONS OFF INNER RACE MAY SIGNIFICANTLY REDUCE THROUGHFLOW

LESS SIGNIFICANT SEPARATION AT BOTTOM OF RIBS

RIB DEPTH CAN BE USED TO CONTROL INLET VELOCITY

JET INLET TANGENTIAL VELOCITY HAS LITTLE EFFECT ON BEARING INLET VELOCITY PROFILE
Smooth Rotating Wall with Rib

Tangential Velocity (ft/s)
Smooth Rotating Wall without Rib

Tangential Velocity (ft/s)
Ground Rotating Wall without Rib
Tangential Velocity (ft/s)
Smooth Rotating Wall with Rib

Axial Velocity (ft/s)
Smooth Rotating Wall without Rib
Axial Velocity (ft/s)
Ground Rotating Wall without Rib
Axial Velocity (ft/s)
Smooth Rotating Wall with Rib
Smooth Rotating Wall with Rib
Smooth Rotating Wall without Rib
Smooth Rotating Wall without Rib
Ground Rotating Wall without Rib
Ground Rotating Wall without Rib
HPOTP PREBURNER BACKCAVITY (AT INNER-RACE INLET)

$U_T = 50\%$ ROTATIONAL SPEED

$\triangle$ W/ RIB, FULL RIB HEIGHT

$U_T = 30\%$ ROTATIONAL SPEED

$\square$ W/ RIB, FULL RIB HEIGHT

$\odot$ W/ RIB, HALF RIB HEIGHT

$\triangledown$ W/ RIB, QUARTER RIB HEIGHT

$\bigcirc$ W/O RIB.

$\bullet$ W/O RIB, HUB GRIND 4 LOBES/0.05 DEEP

$\blacksquare$ W/O RIB, HUB GRIND 6 LOBES/0.10 DEEP

RADIUS - INCH

TANGENTIAL VELOCITY, DIFFERENTIAL (C/0.43u) FPS

04/22/92
CONCLUSIONS / FUTURE WORK

REACT3D CFD RESULTS PROVIDE INSIGHT INTO COMPLEX FLOW PHYSICS

ADVANCED POST-PROCESSING ESSENTIAL TO UNDERSTANDING FLOW

RESULTS CAN BE USED TO DIRECT REDESIGN EFFORTS

CASE COMBINING RIBS AND GRINDOUTS IN WORK
In turbopumps with hydrostatic bearings, clutching bearings are one technique that can be used to control the transient axial thrust. At steady state operation, the clutching bearing inner race is decoupled from the rotating shaft and spins at a speed which is determined by the fluid driving forces in the bearing cavity and the ball bearing resistance. The life of the clutching bearing depends on the speed of rotation of the inner race; therefore, it is important to predict the latter with accuracy.

Cavity flow analysis is difficult due the complicated nature of the geometry, which often results in a totally skewed mesh. A quick study of a simple cavity flow was performed to gain insight into important parameters. It was concluded that the multi-domain (or multi-zone) approach, the double precision code, the initial condition and a good combination of relaxation factors are the 4 essential features in the search for a quick converged solution. The multi-domain approach enables the user to divide the model into small blocks which are gridded separately; therefore insuring the creation of a reasonable mesh. The double precision code solves the problem of various scales in different regions of the flow and the good initial guess in conjunction with a good selection of relaxation factors helps reduce the computational time.

A flow model of the NLS clutching bearing cavity was built for 2-D axisymmetric viscous analyses. From the CFD output, the tangential force exerted on the surfaces of the inner race was integrated to calculate the driving torque which, in conjunction with the resistance torque, was used to predict the operating speed of the inner race.

In order to further reduce the inner race rotation, the swirling flow at the cavity inlet was partially re-directed to generate an opposing torque. Thirty six slanted slots was incorporated into the anti-vortex rib to achieve this goal. A 3-D flow analysis performed on this configuration indicated a drastic reduction of the driving torque and inner race RPM.
NLS CLUTCHING BEARING FLOW ANALYSIS

By Ken Tran, Daniel C. Chan and Armen Darian

Rocketdyne Division, Rockwell International
CONTENT

- Clutching bearing description
- Objective and Methodology
- Two-Dimensional analysis
- Three-Dimensional analysis
- Concluding remarks
CLUTCHING BEARING FUNCTION

- During transient operation:
  - Bearing element is used to control transient axial thrust

- At steady state operation:
  - Bearing is decoupled from the rotating shaft
  - Balance piston takes over the control of the axial thrust
  - Inner race is induced to rotate by fluid friction
OBJECTIVE

- Determine life of the clutching bearing
- Predict the inner race RPM
- Calculate the torque acting on the bearing inner race faces
- Investigate design features to minimize inner race speed
APPROACH

- Estimate resistance torque as a function of inner race RPM from bearing mechanical characteristics
- Determine driving torque in function of inner race RPM
- Intersection of 2 torque curves determines inner race speed
- Simplify the flow geometry
  - Labys are not modeled
  - Flow through the balls not included
CFD METHODOLOGY

- Reynolds averaged Navier-Stokes solver (REACT):
  - Control volume, pressure correction method
  - Two-equation k-ε turbulence model

- Validation:
  - Daily and Nece cavity
COMPUTATIONAL MODEL

- Following elements are important to achieve converged solution:

  - Multi-domain grid: better mesh and control of Y+
  - Double precision code
  - Good initial condition
  - Relaxation factors: Taguchi parametrics for simple case performed on u,v,w (0.35, 0.5, 0.8) and p (0.1, 0.15, 0.2) relaxation factor
COMPUTATIONAL MODEL (cont.)

- Grid resolution:
  - $Y^+ \sim 50-500$
  - 4205 grid points for 2-D model
  - 95366 grid points for 3-D model: meridional grid identical to 2-D mesh

- Boundary conditions:
  - Swirling jet imposed at one circumferential node line

  The jet represents the exit condition of the hydrostatic bearing
COMPUTATIONAL MESH

SINGLE ZONE

MULTI-ZONE
TAGUCHI ANALYSIS OF RELAXATION FACTORS

RES. $10^{**3}$

LEVEL

1st 2nd 3rd
COMPUTATIONAL GRID
2-D CFD RESULTS

- Streamlines show the jet diffusion is slow
- Swirl flow is still present near the inner race front face
- Radial pressure distribution on the inner race front face is relatively uniform except at stagnation point:
  - 1-D model can be used to estimate axial load
- Predicted axial load is independent of inner race speed
2-D CFD RESULTS (cont.)

- Integrated torque acting on the inner race indicates that:
  - At zero RPM, the inner race front face contributes over 50% to the total torque
  - At 10000 RPM, this contribution is only 25%
- Resistance and driving torque characteristics determine inner race RPM:
  - Predicted RPM is about 9000
MERIDIONAL STREAMLINES
TORQUE VERSUS INNER RACE RPM

- □ 2-D CFD PREDICTION
- ○ BEARING RESISTANCE
PRESSURE DISTRIBUTION ON INNER RACE FRONT FACE

\[ \square = \text{UPSTREAM} \]
\[ \bigcirc = \text{DOWNSTREAM} \]

PRESSURE (PSI)

RADIUS (IN.)
METHODS TO REDUCE INNER RACE SPEED

- Increase axial load to raise resistance torque:
  - Tighter laby clearance: unacceptable due to high assembly cost

- Reduce the effect of the swirl:
  - Anti-vortex ribs: limited result because of small contribution of the front face torque
  - Redirecting the jet against direction of rotation to offset driving torque
3-D CFD ANALYSIS

- Feature: 36 Slots on rib used to redirect jet (20 deg. from axial)

- Results:
  - Driving torque reduced significantly
  - Inner race speed lowered to 7000 RPM
  - 12 Hrs of YMP Cray CPU time per case

- Slots can be modified to increase effectiveness
TORQUE VERSUS INNER RACE SPEED

- □ 2-D CFD PREDICTION
- ○ BEARING RESISTANCE
- △ 3-D PRED. WITH 36 SLOTS
NLS CLUTCHING BEARING

COMPUTATIONAL GRID
CONCLUDING REMARKS

- Multi-domain methodology is needed for complicated geometry

- REACT is a mature code:
  - Enables the exploration of new design concepts
  - Helps determine optimum configurations

- Graphics post-processing is an important tool:
  - Assists users in the understanding of the flow field
  - Highlights flow features
The Bearings, Seals and Material Tester (BSMT) is a test article being used at MSFC to evaluate the performance of conventional rolling contact bearings. Pressure differentials between the BSMT inlet and exit cavities are found to cause large parasitic axial loads on the bearing-carrier walls. These parasitic loads, besides being detrimental to the life of bearings, make the testing and evaluation of bearing performance very difficult, and need to be eliminated if at all possible.

CFDRC is currently under contract to MSFC to perform a detailed analysis of the flow fields inside the BSMT cavities and manifolds. The objectives of this study are to estimate the hydrodynamic loads on the bearings and to recommend feasible design modifications for BSMT to eliminate the parasitic loads.

Three-dimensional computational analyses of inlet and exit cavities in their baseline configuration were performed with REFLEQS which is an advanced finite-volume Navier-Stokes code. Computations were performed with and without a 1/4 inch diameter temperature probe included in each of the cavities. The results of the analyses indicate that the temperature probes substantially alter the flow field and reduce the pressure drop/rise in the cavities. The overall pressure drop across the tester compares quite well with the measurements.

One of the potential design modifications to reduce the parasitic loads on the bearings is to place baffles in the inlet cavities to isolate the coolant flow from the slinger wall. Three-dimensional analyses of the inlet cavities with the baffle were performed to assess the effect of baffles on the axial load. The baffle length was varied as a parameter. Results suggest that axial loading should be reduced considerably with the baffle extended inward to the radius of the outer race.

Thermal analyses of the inlet cavities were performed to determine the temperature rise due to viscous dissipation. The deflection of the baffle due to the hydrodynamic pressure load was also determined by performing structural analysis. The analyses suggest that the temperature rise and the baffle deflection are not of much concern. Therefore the considered design modification seems feasible and should be investigated further from structural, manufacturing, and test assembly considerations.
CFD ANALYSIS TO OPTIMIZE A DESIGN MODIFICATION OF BSMT

by

Mark Ratcliff and Ram Avva
CFD Research Corporation
Huntsville, Alabama 35805

and

Robert Williams
NASA Marshall Space Flight Center
Marshall Space Flight Center, Alabama

April 28, 1992
- Overview

- Modeling of Inlet and Exit Cavities

- Results of 3D Analyses

- Summary and Concluding Remarks
Bearings, Seals & Material Tester

Exit Ports

Exit Cavity

Inlet Cavity No. 1

Inlet Cavity No. 2

Slinger

Bearings

Bearing Carriers

Axial Loader

Inlet Ports for LOX

Radial Loader
STATEMENT OF PROBLEM

- Problem
  - Tester Designed with Improper Cavity Pressure Distribution
  - Pressure Differentials Between BSMT Inlet and Exit Cavities Cause Large Parasitic Axial Loads

- Objective of Design Modification
  - Reduce Axial Loads in BSMT
- Density Averaged N-S Equations
- Finite Volume
- Pressure-Based Algorithm (SIMPLEC)
- Incompressible and Compressible Flows
- Cartesian, Axisymmetric and BFC Options
- Turbulence and Combustion Models
MODELING OF BSMT CAVITIES

LOX at -279°F (100°K) and 600 PSI (4×10^6 Pa)

Assumptions:

- Incompressible
- Isothermal
- Constant Properties
- Single Phase
MODELING OF INLET CAVITIES
Mass Flux Matched

ACTUAL GEOMETRY

INLET CAVITY NO. 1

INLET CAVITY NO. 2

MODELED GEOMETRY
MODELING OF EXIT CAVITY

Actual Geometry

Modeled Geometry

Flow from bearings

Flow from bearings

Stationary

Rotating
BOUNDARY CONDITIONS

Specified Mass Flow

INLET CAVITY NO. 1
- Extrapolation
- Fixed Pressure

INLET CAVITY NO. 2
- Blocked Region
- Cage
- Inner Race
CAVITY NO. 1
COMPUTATIONAL GRID
30 x 30 x 34

rθ - PLANE
Xr - PLANE
CAVITY NO. 1 (BASELINE)

STATIC PRESSURE ON CARRIER WALL
INLET & EXIT LOCATIONS

Tester Exit

Bearing Exit

Bearing Inlet

Tester Inlet
## SUMMARY OF 3D ANALYSES

Baseline with Probes

<table>
<thead>
<tr>
<th>Case</th>
<th>Static Pressure at (psi)</th>
<th>Net Δ P across Tester (psi)</th>
<th>Bearing Inlet Swirl % cage speed</th>
<th>Net Load lbf</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cavity No. 1 with probe</td>
<td>Tester Inlet 600</td>
<td>Bearing Inlet 570</td>
<td>Bearing Exit 555</td>
<td>Tester Exit 578</td>
</tr>
<tr>
<td>Cavity No. 2 with probe</td>
<td>Tester Inlet 600</td>
<td>Bearing Inlet 556</td>
<td>Bearing Exit 541</td>
<td>Tester Exit 564</td>
</tr>
</tbody>
</table>
Problem: Large axial (parasitic) loads

Cause: Strong vortex generated due to slinger rotation

Solution: Isolate fluid from slinger
CAVITY NO. 1 (66% BAFFLE)

STATIC PRESSURE ON CARRIER WALL
CAVITY NO. 1 (100% Baffle)

STATIC PRESSURE ON CARRIER WALL
## SUMMARY OF 3D ANALYSES

<table>
<thead>
<tr>
<th>Case</th>
<th>Static Pressure at (psi)</th>
<th>Net ΔP across Tester (psi)</th>
<th>Bearing Inlet Swirl % cage speed</th>
<th>Net Axial Load on Bearings lbf</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Tester Inlet</td>
<td>Bearing Inlet</td>
<td>Bearing Exit</td>
<td>Tester Exit</td>
</tr>
<tr>
<td><strong>Drive-side</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline</td>
<td>600</td>
<td>570</td>
<td>555</td>
<td>578</td>
</tr>
<tr>
<td>66% Baffle</td>
<td>600</td>
<td>585</td>
<td>570</td>
<td>593</td>
</tr>
<tr>
<td>100% Baffle</td>
<td>600</td>
<td>591</td>
<td>576</td>
<td>599</td>
</tr>
<tr>
<td><strong>Load-side</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline</td>
<td>600</td>
<td>556</td>
<td>541</td>
<td>564</td>
</tr>
<tr>
<td>66% Baffle</td>
<td>600</td>
<td>586</td>
<td>571</td>
<td>594</td>
</tr>
<tr>
<td>100% Baffle</td>
<td>600</td>
<td>591</td>
<td>576</td>
<td>599</td>
</tr>
</tbody>
</table>
BSMT THERMAL ANALYSIS

1st Law of Thermodynamics

Assuming adiabatic walls

\[ \dot{w} = m[(h + KE)_{out} - (h + KE)_{in}] \]

Bulk \( \Delta T = \frac{h_{out} - h_{in}}{c_p} \)

Solving Enthalpy Equation \( \Rightarrow \) Peak \( \Delta T \)

Inlet Cavity No. 1 (Drive-Side)

<table>
<thead>
<tr>
<th>Case</th>
<th>Power HP</th>
<th>Bulk Temp Rise °F</th>
<th>Peak Temp Rise °F</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>24.9</td>
<td>6.72</td>
<td>17.5</td>
</tr>
<tr>
<td>33% baffle</td>
<td>27.5</td>
<td>7.4</td>
<td></td>
</tr>
<tr>
<td>66% baffle</td>
<td>25.5</td>
<td>7.0</td>
<td>24.3</td>
</tr>
<tr>
<td>100% baffle</td>
<td>22.9</td>
<td>6.5</td>
<td></td>
</tr>
</tbody>
</table>
\[ \nabla^4 \omega = \frac{q}{D} \]

where

- \( E \) = Elasticity Modulus
- \( \nu \) = Poisson's ratio
- \( t \) = thickness
- \( \omega \) = deflection

**66% Baffle**

<table>
<thead>
<tr>
<th>( \omega ) (mil)</th>
<th>( t ) (inches)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>1/16</td>
</tr>
<tr>
<td>3.3</td>
<td>1/32</td>
</tr>
</tbody>
</table>
CONCLUDING REMARKS

- Predicted axial (Parasitic) loads are comparable to earlier estimates

- Axial loads can be reduced with baffles

- Baffle deflection and temperature rise are of little concern
Combustion Instability Analysis For Liquid Propellant Rocket Engines

Y.M. Kim, C.P. Chen, and J.P. Ziebarth

University of Alabama in Huntsville
Huntsville, Alabama 35899

Abstract

The multi-dimensional numerical model has been developed to analyze the nonlinear combustion instabilities in liquid-fueled engines. The present pressure-based approach can handle the implicit pressure-velocity coupling in a non-iterative way. The additional scalar conservation equations for the chemical species, the energy, and the turbulent transport quantities can be handled by the same predictor-corrector sequences. This method is time-accurate and it can be applicable to the all-speed, transient, multi-phase, and reacting flows.

Special emphasis is given to the acoustic/vaporization interaction which may act as the crucial rate-controlling mechanism in the liquid-fueled rocket engines. The subcritical vaporization is modeled to account for the effects of variable thermophysical properties, non-unitary Lewis number in the gas-film, the Stefan flow effect, and the effect of transient liquid heating. The test cases include the one-dimensional fast transient non-reaction and reacting flows, and the multi-dimensional combustion instabilities encountered in the liquid-fueled rocket thrust chamber. The present numerical model successfully demonstrated the capability to simulate the fast transient spray-combusting flows in terms of the limiting-cycle amplitude phenomena, correspondence between combustion and acoustics, and the steep-fronted wave & flame propagation. The investigated parameters include the spray initial conditions, air-fuel mixture ratios, and the engine geometry. Stable and unstable operating conditions are found for the liquid-fueled combustors. Under certain conditions, the limiting cycle behavior of the combusting flowfields is obtained. The numerical results indicate that the spray vaporization processes play an important role in releasing thermal energy and driving the combustion instability.
COMBUSTION INSTABILITY ANALYSIS FOR LIQUID PROPELLANT ROCKET ENGINES

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

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

- To predict the nonlinear instability phenomena in liquid-fueled rocket engines.
- To gain deeper understanding of the effects of the design parameters.
- To get the detailed information about driving mechanism of combustion instabilities influenced by the physical processes such as atomization, vaporization, and drop breakup & collision.
- To develop an efficient, accurate, and stable numerical model (pressure-based) for fast transient spray-combusting flows.
APPROACH

- Eulerian-Lagrangian Formulation
  - Pressure-based method
  - Applicable to all speed flows
  - Non-iterative for transient calculations

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

- Equilibrium, Non-equilibrium, PDF Combustion Models.

- Infinitive & Effective Conductivity Vaporization Model

- Second-Order Upwind Scheme
ISSUES

- Physical processes involved in the driving mechanism of combustion instability.
- Correlation between the vaporization response characteristics and the oscillating flowfield.
- Prediction capabilities for the limiting cycle and the triggered instability.
- Effects of operating conditions, combustor geometry, and stabilization devices.
- Validation of numerical model for nonlinear chamber wave phenomena.
SHOCK TUBE PROBLEM

\[
\begin{array}{c}
u = 0.0 \\
\rho = 1.0 \\
\rho_p = 1.0
\end{array}
\quad
\begin{array}{c}
u = 0.0 \\
\rho = 0.125 \\
\rho_p = 0.1
\end{array}
\]
Shock tube problem ($CFL = 0.5, N = 100, t = 0.143s$)
Shock tube problem ($CFL = 0.5, N = 100, t = 0.143s$)
FLAME PROPAGATION IN A CLOSED TUBE

\[ T_{ig} \quad T_i \quad \Delta x_s \]

\[ 0 \quad x_{ig} \quad 5 \text{ cm} \]

- \( T_{ig} : 1500 \text{ K} \)
- \( x_{ig} : 0.25 \text{ cm} \)
- \( \Delta x_s : 0.19 \text{ cm} \)
- \( \Delta x : 0.05 \text{ cm} \)
- \( \phi : 1.0 \)
Premixed flame propagation($T_i = 600K, D = 1.8 \times 10^{-4} m^2/s$)
Premixed flame propagation \( T_i = 800K, D = 1.8 \times 10^{-4} \text{m}^2/\text{s} \)
Premixed flame propagation \(T_i = 800K, D = 9.0 \times 10^{-4} m^2/s\)
Spray flame propagation ($T_i = 800K$, $D = 9.0 \times 10^{-4} m^2/s$, $r_{k,o} = 15 \mu m$)
Spray flame propagation ($T_i = 800K$, $D = 9.0 \times 10^{-4} \, m^2/s$, $r_{k,o} = 30 \mu m$)
Spray flame propagation \( L = 500 \text{K}, D = 9.0 \times 10^{-4} \text{m} \) \( \phi = 154 \text{m} \) 

OXYGEN MASS FRACTION

0.002637

0.17403

0.232770

FUEL MASS FRACTION

0.013489

0.043053

0.072617
Table 1. Dimensions of Three Liquid-Fuel Rocket Engines.

<table>
<thead>
<tr>
<th>Engine</th>
<th>$L_1$ (m)</th>
<th>$L_2$ (m)</th>
<th>$R$ (m)</th>
<th>$R_t$ (m)</th>
<th>$\Theta$ (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>E1</td>
<td>0.3534</td>
<td>0.2094</td>
<td>0.2266</td>
<td>0.1309</td>
<td>65.44</td>
</tr>
<tr>
<td>E2</td>
<td>0.1767</td>
<td>0.2094</td>
<td>0.2266</td>
<td>0.1309</td>
<td>65.44</td>
</tr>
<tr>
<td>E3</td>
<td>0.1767</td>
<td>0.1047</td>
<td>0.2266</td>
<td>0.1309</td>
<td>47.57</td>
</tr>
</tbody>
</table>
(Atomization + Vaporization + Turbulent Mixing + Chemical Reaction + etc)

\[ \tau_c \quad (\text{Overall Combustion Time Scale}) \]

IF \( \tau_c \approx \tau_a \) (characteristic acoustic time scale) \( \rightarrow \) Combustion Instability
Figure 2  Pressure oscillations for three perturbation levels;
\((r_k, o = 100 \mu m, \phi = 1.3, X / L_1 = 1.0, Y / R = 0.5, E_1)\)
Figure 3  Pressure oscillations for three overall equivalence ratios;
\((r_{k,o}=100\mu m, C_{pert}=0.02, X/L_1=0.0, Y/R=0.5, E_1)\)
Figure 4  Pressure oscillations for three initial spray conditions;
($\phi=1.3$, $C_{pert}=0.02$, $X/L_1=0.0$, $Y/R=0.5$, $E_1$)

Figure 5  Spray parcel distribution for three initial spray conditions;
($\phi=1.3$, $C_{pert}=0.02$, $E_1$)

Figure 6  Contours of temperature for three initial spray conditions;
($\phi=1.3$, $C_{pert}=0.02$, $X/L_1=0.0$, $E_1$)
Figure 7  Flow oscillations of pressure, velocity, and temperature;
($r_{k,o}=120\mu m$, $\phi=1.3$, $C_{pert}=0.02$, $X/L_1=0.0$, $Y/R=0.5$, $E_1$)

Figure 8  Limit-cycle flow oscillations of pressure, velocity, and temperature;
($r_{k,o}=120\mu m$, $\phi=1.3$, $X/L_1=0.0$, $Y/R=0.5$, $E_1$)
Figure 9  Spray parcel distribution and temperature contours; 
($r_{k,0}=120\mu m$, $\phi=1.3$, $C_{pert}=0.02$, $E_1$)
Figure 10  Spray parcel distribution, temperature contours, and velocity vectors for three engines; ($r_{k,o} = 70\mu m$, $\phi=0.65$, $C_{pert}=0.02$, $Y/R=0.5$)
Figure 11  Stability characteristics for three engines;
SUMMARIES

• Successful predictions for the unsteady non-reacting, flame-propagating, and spray-combusting flows.
• Variations in the droplet size, the combustor length, and the nozzle length of converging section have a significant effect on the combustion instability.
• Extension to transverse mode instability analysis and incorporation of physical submodels dominantly involved in the driving mechanism of combustion instability.
• Validation of numerical model for linear and nonlinear chamber wave phenomena.
Inverse Design of a Proper Number, Shapes, Sizes, and Locations of Coolant Flow Passages

George S. Dulikravich
Department of Aerospace Engineering, 233 Hammond Building
The Pennsylvania State University, University Park, PA 16802, USA

During the past several years we have developed an inverse method that allows a thermal cooling system designer to determine proper sizes, shapes, and locations of coolant passages (holes) in, say, an internally cooled turbine blade, a scram jet strut, a rocket chamber wall, etc. Using this method the designer can enforce a desired heat flux distribution on the hot outer surface of the object, while simultaneously enforcing desired temperature distributions on the same hot outer surface as well as on the cooled interior surfaces of each of the coolant passages. This constitutes an over-specified problem which is solved by allowing the number, sizes, locations and shapes of the holes to adjust iteratively until the final internally cooled configuration satisfies the over-specified surface thermal conditions and the governing equation for the steady temperature field.

The problem is solved by minimizing an error function expressing the difference between the specified and the computed hot surface heat fluxes. The computed outer surface heat flux \( q_{\text{out}}^{\text{comp}} \) will not be the same as the specified outer surface heat flux, \( q_{\text{out}}^{\text{spec}} \). A properly scaled L-2 norm of the difference between the specified outer surface heat flux, \( q_{\text{out}}^{\text{spec}} \), and the computed outer surface heat flux, \( q_{\text{out}}^{\text{comp}} \), is then minimized by iteratively changing the sizes, shapes, and locations of coolant passages. Starting with a large number of guessed holes, all unnecessary coolant passages are efficiently eliminated when their sizes reduce below a prespecified minimal allowable value. The minimization has been performed automatically using a standard optimization algorithm of Davidon-Fletcher-Powell. Local minimas in the optimization process were successfully avoided by changing the formulation for the objective function whenever the local minimas were detected. The temperature field analysis was performed using our highly accurate boundary integral element code with linearly varying temperature along straight surface panels. Examples of the inverse design applied to internally cooled turbine blades and scram jet struts (coated and non-coated) having circular and non-circular coolant flow passages will be shown.
1. Mathematical model

Steady heat conduction in internally cooled objects is modeled as a boundary value problem for Laplace's equation over a multiply-connected domain.

Assumptions are:
- temperature field is steady
- solid material of the blade is thermally isotropic.
- thermal expansion is neglected

Governing equation is Laplace's equation:

\[ \nabla^2 T = 0 \]  

(1)

2. Objectives

Determine:
- the exact number of the holes,
- radii of the holes,
- locations of the holes,

such that relative error between specified and computed heat fluxes at the outer boundary is minimized.

3. Boundary Conditions - Ill Posed Boundary Value Problem

Both, Dirichlet and Neumann boundary conditions are specified on the outer boundary. Such an overspecified problem can be solved by inverse (design) approach. The problem is solvable since the domain is multi-connected: positions, shapes and dimensions of the holes will provide additional degrees of freedom.

4. Constraints

Besides minimizing the heat flux error, optimized shape has to satisfy these constraints:
- minimum distance between holes,
- minimum distance between holes and the outer boundary
5. Objective Functions

Two different definitions of objective function were used. The difference between the specified and the heat flux and heat flux obtained by the current design can be computed as a global error:

$$F_1(x) = \frac{\sum_{j=1}^{N} (q_j^c - q_j^r)^2}{\sum_{j=1}^{N} (q_j^r)^2}$$  \hspace{1cm} (2)

or as a local error in heat flux at each node:

$$F_2(x) = \sum_{j=1}^{N} \frac{(q_j^c - q_j^r)^2}{(q_j^r)^2}$$  \hspace{1cm} (3)

Two constraints were incorporated into the objective function using a barrier function

$$B(g(x),w_b) = \frac{1}{w_b} \sum_{i=1}^{M} \left[ \sum_{j=1}^{N_2} \frac{d^s}{(D_j^s-d_j^s-r_i)} + \sum_{k=1}^{M} \frac{d^h}{(D_k^h-d_k^h-r_i-r_k)} \right]$$  \hspace{1cm} (4)

The composite objective function can have two forms:

$$F_i(g(x),w_b) = F_i(x) + B(g(x),w_b), \hspace{0.5cm} i = 1, 2$$  \hspace{1cm} (5)

depending whether global or local objective function is used for its evaluation.
2. The Optimization Procedure

The optimization procedure consists of the following steps:

(1) Specify shape of the outer surface and coating of the turbine blade.
(2) Specify desired temperature $T_{jr}$ values on the outer and inner surfaces.
(3) Specify desired heat flux $q_{jr}$ values on the outer surface.
(4) Specify manufacturing constraints:

   (i) minimum distance $d^S$ between holes and the outer surface,
   (ii) minimum distance $d^h$ between any two neighboring holes.

(5) Specify initial guess for the number of holes, $M$, their dimensions, $r_i$, and locations of the centers of the holes, $x_i$ and $y_i$. Thus, there will be $3\times M$ design variables if we limit ourselves to circular holes only.

(6) Using the Boundary Element Method, the Laplace's equation for a given domain and temperature boundary conditions is solved and heat fluxes at the outer boundary are computed. The Laplace's equation is solved $3\times M$ times, ones for each perturbed design variable to compute the gradient.

(7) Determine relative error between specified and computed heat fluxes and evaluate the objective function. At the same time the barrier function has to be evaluated to determine the composite objective function $F_i$.

(8) Davidon-Powel-Fletcher technique is used to find the new values of design variables repeating the optimization procedure from the step (6) until the corresponding composite objective function $F$ is less than a prespecified value. If the dimension of a hole becomes less than a prespecified value, the hole is eliminated from further optimization. If the optimization procedure stalls in a local minimum the objective function formulation is changed from Eq. 2 to Eq. 3 while continuing with optimization from the step (6).
References


---

Fig. 1 Geometry and manufacturing constraints

Fig. 2b Linear element
Fig. 1.1 Initial configuration (three holes) and final configuration (one large centrally located hole and two dots marked with arrows) corresponding to a solution with 0.1% integrated flux error.

Fig. 1.2 Integrated heat flux error ($L_2$ norm) convergence history during the optimization.
Fig. 1.3 Initially symmetrically located holes of identical size maintain a symmetric configuration throughout the iterative process.

Fig. 1.4 Convergence history of the three-hole symmetrical configuration.
Fig. 1.5 Coated disk problem with initially ten holes. Convergence history shows five holes are reduced to zero. Hole elimination method was not used.

Fig. 1.6 Convergence history for a circular domain with initially ten holes. Hole elimination method was not used. Minimization process terminates in a local minimum.
Fig. 1.7  Coated disk problem with initially ten holes. Hole elimination method was used together with objective function switching.

Fig. 1.8  Convergence history for a circular domain with initially ten holes when hole elimination method is applied together with objective function switching. Discontinuities represent changing of the objective function.
Fig. 1.9 Initial (-----), intermediate (---) and final (--.--.) heat flux distribution through the outer boundary for a cylinder with ten holes initially. Hole elimination method was not used.

Fig. 1.10 Initial (-----), after 5 cycles (---), after 10 cycles (----) and final (-----) heat flux distribution through the outer boundary for a cylinder with ten holes initially. Hole elimination method and objective function switching was used.
Fig. 1.11  Total CPU time (IBM 3090) vs. number of iterations for a circular cylinder with ten holes initially. Hole elimination method was not applied. Total number of analysis code calls (BEM code) was 1428.

Fig. 1.12  Total CPU time (IBM 3090) vs. number of iterations for a circular cylinder with ten holes initially when hole elimination method was applied together with objective function switching.
Fig. 1.13 A five-hole coated turbine blade from which thermal boundary conditions were used represents an actual solution for the case of the turbine blade with ten holes initially.

Fig. 1.14 Initial guess for a coated turbine blade configuration with ten holes using thermal boundary conditions from the five-hole configuration.
Fig. 1.15 Optimized solution for initial configuration with ten holes. Number of holes is minimized to six, where the sixth hole continues to shrink. Hole elimination method was used together with objective function switching.

Fig. 1.16 Convergence history for a coated turbine blade with ten holes initially when hole elimination method was applied together with objective function switching. Discontinuities represent changing of the objective function.
Fig. 1.17  Initial (........), after 5 cycles (-----) , after 10 cycles (-----) and final (------) heat flux distribution through the outer boundary for a turbine blade with initially ten holes. Hole elimination method was used together with objective function switching.

Fig. 1.18  Total CPU time (IBM 3090) vs. number of iterations for a turbine blade with ten holes initially. Hole elimination method was applied together with objective function switching.
Fig. 1 Discretized Ceramically Coated Scram Jet Combustor strut
With prescribed Temperatures and Outer Surface Heat Flux
chord length of the strut: 19.
maximum thickness of the strut: 5.

Fig. 2 Iteration sequence of case 1 (norm error = 0.554 %)
Fig. 2.1 Initial configuration (an off-center inclined almost rectangular hole) and optimized configuration (one large centrally located hole) for one-hole coated disk with intermediate hole shapes.

Fig. 2.2 Convergence history of the coated disk with one-hole configuration.
Fig. 2.3  Initial configuration consisting of three different holes (solid line) and their intermediate shapes during the first 64 optimization cycles for a coated disk.

Fig. 2.4  Intermediate shapes of the three different holes during the optimization cycles 65-121 for a coated disk.
Fig. 2.5 Initial configuration (three circular holes) and optimized configuration (ellipse, rectangle, and a square) for a coated turbine blade with intermediate hole shapes (dotted).

Fig. 2.6 Initial configuration (three unequal almost rectangular holes) and optimized configuration (three differently sized, positioned almost rectangular partially constrained holes) for the coated turbine blade airfoil with intermediate hole shapes.
Fig. 1. Geometry and boundary conditions [9].

Fig. 2. Inner and outer contours discretized with panels (O denotes fixed end points).

Fig. 4. Turbine design for case 1.

Fig. 5. Turbine design for case 2.
INVERSE DESIGN OF MULTHOLED INTERNALLY COOLED TURBINE BLADES

Iteration sequence for turbine design case 1: (a) initial configuration; (b) solution after 6 iterations; (c) solution after 14 iterations; (d) solution after 18 iterations.
Abstract

The objectives of this paper are to develop computational methods to predict the hot-gas-side and coolant-side heat transfer, and to use these methods in parametric studies to recommend optimized design of the coolant channels for regeneratively cooled liquid rocket engine combustors. An integrated numerical model which incorporates computational fluid dynamics (CFD) for the hot-gas thermal environment, and thermal analysis for the coolant channels, was developed. The model was validated by comparing predicted heat fluxes with those of hot-firing test and industrial design methods. Parametric studies were performed to find a strategy for optimized combustion chamber coolant channel design.
NUMERICAL ANALYSIS OF THE HOT-GAS-SIDE AND COOLANT-SIDE HEAT TRANSFER IN LIQUID ROCKET ENGINE COMBUSTORS

BY

TEN-SEE WANG
ED32, CFD BRANCH
NASA/MSFC

AND
VAN LUONG
ED64, THERMAL ANALYSIS BRANCH
NASA/MSFC

FOR
WORKSHOP FOR CFD APPLICATIONS IN ROCKET PROPULSION
APRIL 28-30
HUNTSVILLE, ALABAMA
OBJECTIVES

★ TO DEVELOP COMPUTATIONAL METHODS FOR THE PREDICTION OF THE COUPLED HOT-GAS-SIDE AND COOLANT-SIDE HEAT TRANSFER IN A LIQUID ROCKET ENGINE COMBUSTOR

★ TO PERFORM PARAMETRIC STUDIES TO RECOMMEND OPTIMIZED DESIGN OF THE COOLANT CHANNELS FOR REGENERATIVELY COOLED LIQUID ROCKET ENGINE COMBUSTORS
THE AERO-THERMAL MODEL

★ CFD MODEL FOR HOT-GAS-SIDE ENVIRONMENT
  - AXISYMMETRIC MCC FLOWFIELD
  - FULLY VISCOUS FLOW
  - SHOCK CAPTURING
  - SEVEN SPECIES EQUILIBRIUM CHEMISTRY

★ SINDA THERMAL MODEL FOR LINER, RIB, AND JACKET
  - THREE-DIMENSIONAL
  - VARIABLE WALL THICKNESS, CHANNEL DIMENSIONS
    AND NUMBER OF CHANNELS
  - RADIATION CORRECTED
  - WALL TEMPERATURE AND THERMAL GRADIENT

★ SINDA HYDRAULIC MODEL FOR COOLANT FLOW
  - COOLANT TEMPERATURE AND PRESSURE DROP
AERO-THERMAL MODEL

FLOW CONDITIONS
ASSUME TW

CFD ANALYSIS

Q, h

THERMAL ANALYSIS

GEOMETRY

TOLERANCE CHECK

Tw

FINAL CFD/ THERMAL RESULTS
TEST CASES

★ 40K CALORIMETER THRUST CHAMBER TEST VALIDATION

★ BASELINE STANDARD THROAT SSME MCC COMPARISON

★ LARGE THROAT AMCC DESIGN PARAMETRIC STUDIES
SCHEMATIC OF 40K TEST CONFIGURATION

Fuel

Transition

8" Axial

2" Adaptor

Nozzle

5.86"

3.58"

LOX

Coolant
40K Coolant Discharge Temperature

Temperature, Deg. F

Axial Station, Inches

- Predict
- Test Data
40K Wall Heat Flux

Q/A, Btu/Sec-In²

Axial Station, Inches
SSME MCC WALL HEAT FLUX

![Graph showing heat flux vs. axial distance from throat for different models: CFD/Thermal, RKON ReGEN, and P-W. The graph indicates the current design, FPL.](image-url)
SSME MAIN COMBUSTION CHAMBER
HOT AND COLD WALL TEMPERATURE PROFILE

LEGEND

HOT WALL
COLD WALL

AXIAL DISTANCE FROM THROAT (IN)

TEMPERATURE (DEG. F)
SSME MAIN COMBUSTION CHAMBER
HOT AND COLD GAS PRESSURE DISTRIBUTION
SSME COMBUSTION CHAMBER COOLANT LINER
COOLANT BULK TEMPERATURE PROFILE

AXIAL STATION (INCHES)

TEMPERATURE (DEG. R)

109% FPL
SSME VPS MAIN COMBUSTION CHAMBER
EFFECT OF HOT GAS WALL THICKNESS
HOT GAS WALL SURFACE TEMPERATURE

TEMPERATURE (DEG. F)

AXIAL STATION (IN)

LT/550C/WIDTH=.021"
LT/550C/WIDTH=.021"/+20%
LT/550C/WIDTH=.021"/-20%
SUMMARY

★ AN INTEGRATED CFD/THERMAL MODEL HAS BEEN DEVELOPED TO PREDICT THE HOT-GAS-SIDE AND COOLANT SIDE HEAT TRANSFER FOR LIQUID ROCKET COMBUSTION CHAMBER

★ MODEL VALIDATED FOR 40K CALORIMETER THRUST CHAMBER TEST

★ MODEL COMPARED FOR BASELINE STANDARD THROAT SSME MCC HEAT TRANSFER

★ PERFORMED LARGE THROAT AMCC DESIGN PARAMETRIC STUDIES
  - INCREASED ASPECT RATIO AND NUMBER OF CHANNELS REDUCE THE WALL TEMPERATURE AND THERMAL GRADIENT
  - REDUCED WALL THICKNESS REDUCES THE SURFACE WALL TEMPERATURE
An Efficient and Robust Grid Optimization Algorithm

By

Bharat K. Soni
Associate Professor
NSF Engineering Research Center
for Computational Field Simulation

and

Shaochen Yang
Assistant Professor
Mississippi University for Women
ABSTRACT

The development of an efficient and robust grid optimization algorithm is presented. This algorithm is developed by combining the best characteristics of algebraic, elliptic and hyperbolic grid generation techniques (Ref. 1-3). This development is based on the following observations and evaluations:

Algebraic systems are fast and economical.
Precise spacing control (well distributed grid) is always achieved with algebraic systems.
Grid generation by elliptic system is always smooth.
Algebraic system may cause grids to overlap, however, elliptic system resist grid line overlapping.
Weighted transfinite interpolation method blended with Bezier, B-spline curves/surfaces can produce well-distributed, orthogonal(at Boundaries) and smooth grids (not in all cases, but most all).
The control functions can be formulated to achieve boundary orthogonality and spacing control (near solid boundary surface) by elliptic generation system.
The control functions can be formulated to accomplish field orthogonality in a given computational direction (h, x, or z ) and spacing control by elliptic generation system by iteratively updating various terms in the generation system. This is very time consuming especially in three dimensional problems.
Algebraic systems require a high degree of understanding and visual user interaction. However, elliptic systems can be readily adaptable for generalization. This is extremely useful in grid adaptation.
The hyperbolic system preserves the orthogonality at the solid boundary and the point distribution in the field. However, its applicability is restricted to external flows where the accurate geometrical shape of the outer boundaries/surfaces are not important as long as their location is a certain distance away from the body. Also in three dimensional applications of hyperbolic system the grid quality is directly influenced by the characteristics of the surfaces associated with the computational domain.

Computational examples representing practical internal flow configurations are presented to demonstrate the success of this algorithm.
References:


Grid Methods

Direct (Algebraic)
- Fast and Economical
- Precise Spacing Control
- Propogation of Slope Discontinuities
- Interactive User Interface
- Possible Overlapping - can be avoided!
- High Degree of Understanding and Visual User Interaction
- Orthogonality and Smoothness
- Transfinite: Lagrange, Hermite, Bezier, B-Splines, NURBS

Indirect (PDES)
- Time Consuming
- Distribution Loss!
- Inherent Smoothness
- Iterative Background Crunching
- Resistant to Grid Line Overlapping
- Readily Adaptable for Generalization
- Competitive Enhancement of Smoothness, Orthogonality, and Concentration
- Elliptic Hyperbolic
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)
Conflicting Features

- Smoothness
- Resolution
- Orthogonality
Transfinite Interpolation

\[ P_\zeta = \sum_k \sum_n \phi(\zeta)r^{(k)}(\xi_n, \eta) \]

\[ P_\eta = \sum_l \sum_m \Psi(\eta)r^{(l)}(\zeta, \eta_m) \]

\[ P_\zeta P_\eta = \sum_k \sum_n \sum_{\eta_m} \phi(\zeta)\Psi(\eta)r^{(kl)}(\xi_n, \eta_m) \]

\[ P_\zeta \oplus P_\eta = P_\zeta + P_\eta - P_\zeta P_\eta \]
HERMITE TRANSFINITE INTERPOLATION

Slope Evaluation:

Going in $\xi$ direction →

\[ r_\xi \cdot r_\eta = 0 \]

\[ || r_\xi \times r_\eta || = A \]

OR

\[ r_\eta \cdot r_\eta = g_{22} \]

Going in $\eta$ direction →

\[ r_\xi \cdot r_\eta = 0 , \quad r_\xi \cdot r_\xi = 0 \]

\[ || r_\xi \times r_\eta || = A , \quad r_\xi \cdot r_\xi = g_{11} \]
A Two Dimensional Elliptic Grid System

\[ g_{22}(r_{\xi\xi} - \phi r_{\xi}) + g_{11}(r_{\eta\eta} - \Psi r_{\eta}) - 2g_{12}r_{\xi\eta} = 0 \]

\[ r = (x, y) \quad \text{physical space} \]
\[ (\xi, \eta) \quad \text{computational space} \]

\[ g_{11} = r_{\xi} \quad r_{\xi} = x_{\xi}^2 + y_{\xi}^2 \]
\[ g_{11} = r_{\eta} \quad r_{\eta} = x_{\xi}x_{\eta} + y_{\xi}y_{\eta} \]
\[ g_{22} = r_{\eta} \quad r_{\eta} = x_{\eta}^2 + y_{\eta}^2 \]

\[ \phi, \Psi \quad \text{control functions} \]
Control Functions

\[ \phi = \frac{r_{\xi \xi} \cdot r_\xi}{r_\xi \cdot r_\xi} + \frac{r_{\eta \eta} \cdot r_\xi}{r_\eta \cdot r_\eta} \]

\[ \psi = \frac{r_{\eta \eta} \cdot r_\eta}{r_\eta \cdot r_\eta} + \frac{r_{\xi \xi} \cdot r_\eta}{r_\xi \cdot r_\xi} \]
Cell Area Approach (I)

- Evaluation of $m$

\[ \begin{align*}
I_\xi \cdot I_\eta &= x_\xi x_\eta + y_\xi y_\eta = 0 \\
|I_\xi \times I_\eta| &= x_\xi y_\eta - x_\eta y_\xi = v
\end{align*} \]

\[
\begin{bmatrix}
  x_\xi & y_\xi \\
  -y_\xi & x_\xi
\end{bmatrix}
\begin{bmatrix}
  x_\eta \\
  y_\eta
\end{bmatrix} =
\begin{bmatrix}
  0 \\
  v
\end{bmatrix}
\]

\[
\begin{bmatrix}
  x_\xi & y_\xi \\
  -y_\xi & x_\xi
\end{bmatrix}
\]

\[ g_{11} = x_\xi^2 + y_\xi^2 \neq 0 \]
Cell Area Approach (II)

- Evaluation of $r_{\xi \eta}$

$$(r_{\xi} \cdot r_{\eta})_{\xi} = (x_{\xi} x_{\eta} + y_{\xi} y_{\eta})_{\xi} = 0$$

$$|r_{\xi} \times r_{\eta}|_{\xi} = (x_{\xi} y_{\eta} - x_{\eta} y_{\xi})_{\xi} = v_{\xi}$$

$$\begin{bmatrix} x_{\xi} & y_{\xi} & x_{\xi \eta} \\ -y_{\xi} & x_{\xi} & y_{\xi \eta} \end{bmatrix} = \begin{bmatrix} -x_{\xi \xi} x_{\eta} - y_{\xi \xi} y_{\eta} \\ v_{\xi} - x_{\xi \xi} y_{\eta} + x_{\eta} y_{\xi \xi} \end{bmatrix}$$
Cell Area Approach (III)

- Evaluation of $r_{\eta\eta}$

\[
(r_\xi \cdot r_\eta)\eta = (x_\xi x_\eta + y_\xi y_\eta)\eta = 0
\]

\[
| r_\xi \times r_\eta |\eta = (x_\xi y_\eta - x_\eta y_\xi)\eta = v_\eta
\]

\[
\begin{bmatrix}
  x_\xi & y_\xi \\
  -y_\xi & x_\xi
\end{bmatrix}
\begin{bmatrix}
  x_{\eta\eta} \\
  y_{\eta\eta}
\end{bmatrix}
= 
\begin{bmatrix}
  -x_\xi x_\eta - y_\xi y_\eta \\
  v_\eta - x_\xi y_\eta + x_\eta y_\xi
\end{bmatrix}
\]
Grid Spacing Approach

- Evaluation of $\gamma_\eta$
  \[(r_\xi = g_{22})\]
  \[
  (r_\xi \cdot r_\eta) \eta = (x_\xi x_\eta + y_\xi y_\eta) \eta = 0 \\
  x_\eta^2 + y_\eta^2 = g_{22}
  \]

- Evaluation of $\gamma_\xi \eta$
  \[
  (r_\xi \cdot r_\eta) \xi = (x_\xi x_\eta + y_\xi y_\eta) \xi = 0 \\
  (r_\xi \cdot r_\eta) \xi = (x_\eta^2 + y_\eta^2) \xi = (g_{22}) \xi
  \]

- Evaluation of $\gamma_\eta \eta$
  \[
  (r_\xi \cdot r_\eta) \eta = (x_\xi x_\eta + y_\xi y_\eta) \eta = 0 \\
  (r_\eta \cdot r_\eta) \eta = (x_\eta^2 + y_\eta^2) \eta = (g_{22}) \eta
  \]
Control Functions Using Metric Terms

\[ r = (x, y) \]

\[ g_{11} = x_\xi^2 + y_\xi^2 \quad (g_{11})_\eta = 2r_\xi \cdot r_\xi \eta, \quad (g_{11})_\xi = 2r_\xi \cdot r_\xi \eta \]

\[ g_{22} = x_\eta^2 + y_\eta^2 \quad (g_{22})_\eta = 2r_\eta \cdot r_\eta \eta, \quad (g_{22})_\xi = 2r_\eta \cdot r_\xi \eta \]

\[ g_{12} = x_\xi x_\eta + y_\xi y_\eta \quad (g_{12})_\xi = r_\xi \cdot r_\xi \eta + r_\eta \cdot r_\xi \eta, \quad (g_{12})_\eta = r_\eta \cdot r_\xi \eta + r_\xi \cdot r_\eta \eta \]
Similarly in 3D

Working on $\xi = \text{Constant Surface}$

\[
\begin{align*}
    r_\xi \cdot r_\eta &= 0 \\
    r_\xi \cdot r_\delta &= 0 \\
    r_\xi : (r_\eta \times r_\delta) &= \nu \\
    r_\xi \cdot r_\xi &= g_{11}
\end{align*}
\]
Control Functions Using Metric Term cont'd

Assuming \( g_{12} = (g_{12})_\xi (g_{12})_\eta - 0 \)

\[
\phi = \frac{(g_{11})_\xi}{g_{11}} - \frac{(g_{22})_\xi}{g_{22}} \\
\psi = \frac{(g_{22})_\eta}{g_{22}} - \frac{(g_{11})_\eta}{g_{11}}
\]

Assuming \( g_{12} = (g_{12})_\xi (g_{12})_\eta - 0 \)

\[
\phi = \frac{1}{2} \left[ \frac{(g_{11})_\xi}{g_{11}} - \frac{(g_{22})_\xi}{g_{22}} \right] = \frac{1}{2} \frac{d}{d\xi} \left[ \ln \left( \frac{g_{11}}{g_{22}} \right) \right] \\
\chi = \frac{1}{2} \left[ \frac{(g_{22})_\eta}{g_{22}} - \frac{(g_{11})_\eta}{g_{11}} \right] = \frac{1}{2} \frac{d}{d\eta} \left[ \ln \left( \frac{g_{22}}{g_{11}} \right) \right]
\]
Similarly In 3D

\[ \phi = \frac{1}{2} \frac{d}{d \xi} \left( \ln \left( \frac{g_{11}}{g_{22} g_{33}} \right) \right) \]

\[ \chi = \frac{1}{2} \frac{d}{d \eta} \left( \ln \left( \frac{g_{22}}{g_{11} g_{33}} \right) \right) \]

\[ \theta = \frac{1}{2} \frac{d}{d \delta} \left( \ln \left( \frac{g_{33}}{g_{11} g_{22}} \right) \right) \]
FUTURE

\[ r_\xi \cdot r_\eta = 0 \]
\[ r_\xi \cdot r_\delta = 0 \]
\[ r_\xi \cdot (r_\eta \times r_\delta) = v \]
\[ (r_\xi \cdot r_\eta) \eta = 0 \]
\[ (r_\xi \cdot r_\delta) \eta = 0 \]
\[ (r_\xi \cdot (r_\eta \times r_\delta)) \eta = v_\eta \]
\[ (r_\xi \cdot r_\eta) \delta = 0 \]
\[ (r_\xi \cdot r_\delta) \delta = 0 \]
\[ (r_\xi \cdot (r_\eta \times r_\delta)) \delta = v_\delta \]
FUTURE

- Surface Grid Optimization Using NURBS Evaluation and Elliptic System Applications to the Parametric Space
- Full 3D Applications in a Multiblock Environment
Enhancements to the GRIDGEN Structured Grid Generation System for Internal and External Flow Applications

John P. Steinbrenner and John R. Chawner
MDA Engineering, Inc.
Arlington, Texas

GRIDGEN is a government domain software package for interactive generation of multiple block grids around general configurations. Though it has freely available since 1989, it has not been widely embraced by the internal flow community due to a misconception that it was designed for external flow use only. In reality GRIDGEN has always worked for internal flow applications, and GRIDGEN ongoing enhancements are increasing the quality of and efficiency with which grids for external and internal flow problems may be constructed.

The software consists of four codes used to perform the four steps of the grid generation process. GRIDBLOCK is first used to decompose the flow domain into a collection of component blocks and then to establish interblock connections and flow solver boundary conditions. GRIDGEN2D is then used to generate surface grids on the outer shell of each component block. GRIDGEN3D generates grid points on the interior of each block, and finally GRIDVUE3D is used to inspect the resulting multiple block grid. Three of these codes (GRIDBLOCK, GRIDGEN2D, and GRIDVUE3D) are highly interactive and graphical in nature, and currently run on Silicon Graphics, Inc. and IBM RS/6000 workstations. The lone batch code (GRIDGEN3D) may be run on any of several Unix based platforms.

The ease of flow domain decomposition using GRIDBLOCK has been improved through incorporation of edge point distribution commands and a new intermediate construction entity know as a domain. Grid point dimensions and distributions are now assigned to block boundary curves (connectors) before block construction. From here, block subsurfaces are defined by domains, which are simply a loop of connectors that represent the perimeter of the surface. The bounding connectors of the domain and the grid point distributions on the connectors provide sufficient data for the automatic initialization of surface grid points, which may be later refined as necessary in the GRIDGEN2D code. Blocks are then constructed by grouping domains into faces, and then by grouping 6 faces into a block. Grouping takes place in a point-and-click environment, and the reorientations of domains and faces needed to fit these components into the developing block is maintained automatically within the code, so that block construction may proceed in an intuitive manner. Further, block to block interfaces are determined automatically on the domain level, and domains without interblock connections may be assigned flow solver boundary conditions in a graphical interface.

Surface grid generation in GRIDGEN2D is being improved with the addition of higher order surface definitions (NURBS and parametric surfaces input in IGES format and bicubic surfaces input in PATRAN Neutral File format) and double precision mathematics. In addition, two types of automation have been added to GRIDGEN2D that reduce the learning curve slope for new users and eliminate work for experienced users.

Volume grid generation using GRIDGEN3D has been improved via the addition of an advanced hybrid control function formulation that provides both orthogonality and clustering control at the block faces and clustering control on the block interior.
Enhancements to the GRIDGEN Structured Grid Generation System for Internal and External Flow Applications

NASA Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion
28-30 April 1992

by
John P. Steinbrenner (presenter)
John R. Chawner

MDA Engineering, Inc.
OUTLINE

- Overview of the GRIDGEN System
- Review of current GRIDGEN capabilities (Version 6)
- Summary of GRIDGEN enhancements (Version 8)
- Continuing GRIDGEN improvement
- Conclusions
Overview of the GRIDGEN System

- GRIDGEN is a series of four codes for the generation of 3D, multiple block, structured grids.
- GRIDBLOCK (interactive): domain decomposition.
- GRIDGEN2D (interactive): 3D surface grid generation.
- GRIDGEN3D (batch): volume grid generation.
Overview of the GRIDGEN System

- The interactive codes have been written using IRIS GL and currently run only on Silicon Graphics 4D and IBM RS/6000 workstations.
- The interactive codes also require 24-bit planes and Z-buffer.
- GRIDGEN documentation consists of an official Air Force manual and several AIAA and AGARD papers.

MDA Engineering, Inc.
Arlington, TX
Overview of the GRIDGEN System

- Version 6, USAF
  - Developed for USAF at Wright-Patterson AFB, 1987-1990.
  - Technical Supervision: Dr. Donald W. Kinsey.

- Version 8, NASA Langley
  - Technical Supervision: Dr. Robert E. Smith.

- Version 9, (currently being negotiated)
OUTLINE

- Overview of the GRIDGEN System
- Review of current GRIDGEN capabilities (Version 6)
- Summary of GRIDGEN enhancements (Version 8)
- Continuing GRIDGEN improvement
- Conclusions
Databases (Version 6)

- The user provides GRIDGEN with a geometric description (a database) of the configuration.
- A database consists of a collection of patches called networks.
- Each network is a 2D array of coordinate data on the configuration.
- The networks are not the same as the surface grid.
- The database may be obtained from a CAD system, an external user program, or GRIDGEN2D.
GRIDBLOCK (Version 6)

- GRIDBLOCK is used to decompose the domain surrounding the database into blocks.

- The user interactively draws 3D curves (connectors) that define the edges of each block.
  - Straight Line
  - Circular/Elliptical Arc
  - Piecewise Cubic
  - Line on Database

- Connectors may be drawn in any order and in any direction.
GRIDBLOCK (Version 6)

- The user groups connectors into blocks.
- Blocks may contain up to 12 edges; singularities are allowed.
- The user interactively specifies computational ($\xi, \eta, \zeta$) coordinate axes and number of points in each block.
GRIDBLOCK (Version 6)

- The user specifies interblock connections and flow solver BCs.
- Interblock connections must be set.
  - GRIDGEN2D can then ensure consistency between blocks.
  - GRIDGEN3D can then provide slope continuity across interfaces.
- The user may set TEAM (USAF Euler solver) flow BC’s.
  - TEAM restrictions on connections are checked to be sure grid is compatible.
  - GRIDGEN3D writes the connection and BC data in TEAM format.
GRIDGEN2D (Version 6)

- GRIDGEN2D is used to generate the surface grids on the six faces of each block in the system.
- It may also be used to generate single block or single surface grids without running GRIDBLOCK first.
- For each face of each block...
  - Distribute points on each of the four edges.
  - Initialize surface points using algebraic methods.
  - Refine surface points using elliptic PDE methods.
GRIDGEN2D (Version 6)

- GRIDGEN2D edge point distribution.
  - GRIBLOCK connectors are used to define edge shape or a new edge shape may be drawn interactively.
  - The edge may be divided into subedges for more control of point distribution.
  - Grid points are distributed using...
    * Two-sided tanh (Vinokur) stretching.
    * One-sided sinh and tanh stretching.
    * One-sided geometric progression.
    * Equal spacing.
    * Copy spacing from elsewhere in grid.
    * Cluster to edge curvature.
GRIDGEN2D (Version 6)

- GRIDGEN2D algebraic methods.
  - Standard TFI with computational LaGrange BF
  - Standard TFI with arclength based LaGrange BF
  - Ortho TFI with computational Hermite BF
  - Polar TFI
  - Re-distribution methods.
  - Parametric methods to fit the grid to the database.
GRIDGEN2D (Version 6)

- GRIDGEN2D's elliptic PDE methods.
  - Poisson's Equation on 3D surfaces solved using pointwise SOR with Ehrlich's optimal relaxation factor.
  - Six hybrid control function formulations.
  - Five solver types.
  - Five edge BC types
GRIDGEN2D (Version 6)

- GRIDGEN2D elliptic PDE methods cont'.

- Hybrid control functions combine background and foreground control functions.
- Background control functions tend to influence interior grid points, e.g. LaPlace, Thomas and Middlecoff, or Fixed Grid.
- Foreground control functions tend to influence grid points near the edges, e.g. Sorensen.

  (b) Thomas and Middlecoff
  (c) Sorensen
  (d) Thomas and Middlecoff plus Sorensen
GRIDGEN2D (Version 6)

- GRIDGEN2D elliptic PDE methods cont'.
  - Three conventional solvers:
    * Solve for $x, y$ and leave $z$ as is.
    * Solve for $x, y, z$.
    * Solve for $x, y$ and interpolate $z$ from the database.
  - Two parametric solvers:
    * Solve for $x, y, z$ in terms of the current surface shape.
    * Solve for $x, y, z$ in terms of a database network.
GRIDGEN2D (Version 6)

- **GRIDGEN2D tools**
  - A face may be divided into subfaces.
  - This allows the shape of and distribution of points on grid lines on the face interior to be explicitly set by the user.
  - 8600 lines of help text may be accessed via a browser.
  - There is a utility to graphically move any point.
GRIDGEN3D and GRIDVUE3D (Version 6)

- GRIDGEN3D is a batch code written for a Cray X/MP running the UNICOS operating system
- It may easily be modified for other UNIX hardware.
- Algebraic methods include standard TFI with computational or arclength based LaGrange blending functions.
- Elliptic PDE methods solve the 3D Poisson equations using point-wise SOR with optimal relaxation factors.
- LaPlace, Thomas and Middlecoff, Fixed Grid, or Sorenson control functions are available.
- GRIDVUE3D is used to visualize volume grids written in either GRIDGEN or PLOT3D format.
OUTLINE

- Overview of the GRIDGEN System
- Review of current GRIDGEN capabilities (Version 6)
- Summary of GRIDGEN enhancements (Version 8)
- Continuing GRIDGEN improvement
- Conclusions
Summary of GRIDGEN enhancements (Version 8)

- Double precision
- Add edge point grid generation to GRIDBLOCK.
  - New connector shapes: Cubic on surface and read from file.
  - New distribution function: Monotonic Quadratic Rational Spline (MQRS), allows a smooth variation of grid point spacing along the connector with explicit control over grid point locations on the interior.
  - Improved editing capability: shape or number of points can be changed and point distribution is updated automatically.
Summary of GRIDGEN enhancements (Version 8)

- Add the domain entity to GRIDBLOCK.
  - Connectors grouped into surfaces called domains. Then domains are grouped into blocks.
  - A domain may be a whole face or only a subface.
  - Point to point interblock connections will be determined \textit{automatically}.
  - Flow BCs will be set by graphically picking the domain; no more typing indices.
  - Algebraic surface grid generation will be performed automatically; the GRIDGEN2D workload is drastically reduced.
  - Changes in number of points on a single connector will be propagated semi-automatically throughout the grid.
Summary of GRIDGEN enhancements (Version 8)

- Add the domain entity to GRIDBLOCK cont'.
  - A domains is an entity between the connector and block entities in the GRIDBLOCK hierarchy.
  - They may represent a region of a single flow solver BC or an interblock connection.
  - The user creates a domain by interactively picking the individual connectors in a closed loop.
  - Domains must be computationally rectangular.

![Diagram]

This domain maps into a 45x31 region.
This domain does not map into a rectangular region.
Summary of GRIDGEN enhancements (Version 8)

- Add the domain entity to GRIDBLOCK cont'.
  - Blocks are now defined by six faces, rather than twelve edges.
  - Faces are defined by at least one domain.
  - Blocks and faces are checked for a consistent number of points during construction.
  - An example of a face consisting of four non-unique domains.

[Diagram showing a face with four domains: Domain A, Domain B, Domain C, Domain D]
Summary of GRIDGEN enhancements (Version 8)

• Change Number of Points Utility
  - Low level changes to an existing grid will be propagated semi-automatically throughout the entire blocking system.
  - Rather than edit a journal file, the code will do most of the work and prompt the user for any changes.
  - After a change in the number of points on the indicated connector, the code would ask the user to apportion the new points across the affected connectors.
Summary of GRIDGEN enhancements (Version 8)

- GRIDGEN3D upgrades.
  - Hybrid control functions added for improved grid quality.
  - Background CF $\Phi_b$: control on interior of block, e.g. LaPlace, Thomas & Middlecoff
  - Foreground CF $\Phi_f$: control near faces, e.g. Sorenson
  - Hybrid = Background + Foreground
    * Compute $\Phi_b$ on block interior, $\Phi_f$ on faces.
    * Calculate $\Phi_\Delta = \Phi_f - \Phi_b$ on faces.
    * Interpolate $\Phi_\Delta$ from faces into interior using exponentially decaying blending functions.
    * Sum $\Phi = \Phi_\Delta + \Phi_b$
Summary of GRIDGEN enhancements (Version 8)

- GRIDGEN3D upgrades.
  - Grid sequencing added: faster convergence rate in the PDE solver.
  - Robustness improved: one sided differencing based on the sign of the control function.
  - Efficiency improved: I/O of temporary files changed to reduce overhead.
  - Grid quality: several quality measures are written to a file for visualization using PLOT3D or FAST.
Summary of GRIDGEN enhancements (Version 8)

- GRIDGEN2D Customization.
  - Goal: reduce the effort required to use GRIDGEN2D.
  - Method: eliminate seldom-used buttons and text prompts.
  - Benefit: fewer keystrokes, less confusion.
  - Implementation: verbosity setting and preferencing.
    * Terse verbosity hides obscure prompts from the user (meant for novices).
    * Preferencing allows the user to pre-select certain options such as control function type (meant for experts).

- Double precision GRIDGEN2D.
Summary of GRIDGEN enhancements (Version 8)

- Standardized higher order surface models (databases).
- PATRAN Neutral File.
  - Bicubics.
- DT-IGES, a simplified form of the IGES standard.
  - Parametric surface.
  - Rational B-spline surface.
  - Implementation will use the Navy David Taylor Research Center DT_NURBS Library of surface geometry routines.
OUTLINE

- Overview of the GRIDGEN System
- Review of current GRIDGEN capabilities (Version 6)
- Summary of GRIDGEN enhancements (Version 8)
- Continuing GRIDGEN improvement
- Conclusions
Continuing GRIDGEN improvement

- Merge GRIDBLOCK and GRIDGEN2D.
  - Less user confusion (What do I do in which code?)
  - Improved usability through a single GRIDBLOCK style GUI.
  - More maintainability through elimination of duplicate functionality.
- Support higher order surface definitions (databases) in standard file formats (e.g., NASA-IGES).
- Increase user base by porting to other hardware.
Continuing GRIDGEN Improvement

- Users will gain more capabilities within a familiar GUI.
- Most of the existing GRIDGEN tools can be generalized for use with unstructured techniques.
- Incorporate unstructured grid generation techniques.
- Perform grid generation locally or remotely.
- Interactivity will improve user control over volume grid generation.
- Develop an interactive GRIDGEN3D.
OUTLINE

• Overview of the GRIDGEN System
• Review of current GRIDGEN capabilities (Version 6)
• Summary of GRIDGEN enhancements (Version 8)
• Continuing GRIDGEN improvement
• Conclusions
Conclusions

- GRIDGEN currently provides *at no cost* a practical and well-tested structured grid generation capability.
- Improvements are currently being added to the government domain version of GRIDGEN.
- GRIDGEN will remain in the government domain well into the future.
- MDA Engineering is committed to supporting GRIDGEN.
CAGI: Computer Aided Grid Interface
-A Work in Progress

By

Bharat K. Soni
Associate Professor

Tzu-Yi Yu and David Vaughn
NSF Engineering Research Center
for Computational Field Simulation
Mississippi State University
ABSTRACT

Progress realized in the development of a Computer Aided Grid Interface (CAGI) software system in integrating CAD/CAM geometric system output and/or IGES files, geometry manipulations associated with grid generation and robust grid generation methodologies is presented. CAGI is being developed in a modular fashion and will offer fast, efficient and economical response to geometry/grid preparation allowing ability to upgrade basic geometry in a step-by-step fashion interactively and under permanent visual control along with minimizing the differences between the actual hardware surface descriptions and corresponding numerical analog.

The computer code GENIE (Ref. 1-3) is used as basis. The Non-Uniform Rational B-Splines (NURBS) representation of sculptured surfaces is utilized for surface grid redistribution. The computer aided analysis system, PATRAN, is adapted as a CAD/CAM system. The progress realized in NURBS surface grid generation, the development of IGES transformer, and geometry adaption using PATRAN will be presented along with their applicability to grid generation associated with rock propulsion applications.
References:

1. B. K. Soni, J. F. Thompson, M. L. Stokes, and M. H. Shih, "GENIE++,
EAGLEView and TIGER: General Purpose and Special Purpose Graphically

2. B. K. Soni, "Geometry Processing Associated with Complex Configurations
in Computational Fluid Dynamics", Second SIAM Conference on Geometric

3. B. K. Soni, "GENIE: Generation of Computational Grids for Internal-External
Flow Configurations", Proceedings of the Second International Conference
of Numerical Grid Generation in Computational Fluid Dynamics, Miami, FL,
December 1988, edited by S. Sengupta, J. Hauser, P. R. Eiseman and J. F.
Thompson, Pineridge Press, P. 915–924.

4. David Vaughn, "The Graphically Interactive PATRAN Structured Grid
Interface (GIPSI)", student paper to be presented at the Southeastern
Conference on Theoretical and Applied Mechanics, Nashville, TN,
April 12–14, 1992.
GEOMETRY DEFINITIONS

* Analytic
* Drawings
* Discretized Points
* Combo (Combination of Above Three)
* CAD/CAM Output
* IGES
* CAD Output \(\leftrightarrow\) MASAJ Software \(\rightarrow\) Surface Data
* Scale Model
SCULPTURED SURFACES

APPLICATION - CRITERIA

* Fits Given Information
* Smoothness
* Shape Fidelity
* Parametric Representation
* Local vs Global Schemes
* Interactive Design
* Interactive Viewing
A NURB curve \( c(u) \) is a piecewise rational curve of the form

\[
c(u) = \frac{\sum_{i=0}^{m} \omega_i d_i N_i,k(u)}{\sum_{i=0}^{m} \omega_i N_i,k(u)}, \quad u \in [u_{k-1}, u_{m+1}]
\]

defined by

- an order \( k \) (\( k \) equalling the degree of the polynomials - 1),
- a set of 3D control points, \( \{d_0, \ldots, d_m\} \),
- a set of real weights, \( \{\omega_0, \ldots, \omega_m\} \).
NURB Curves cont'd

- a set of real knots \( \{ u_0, \ldots, u_{m+k} \mid u_l \leq u_{l+1}, \ l = 0 \ldots (m+k-1) \} \),

- B-spline basis functions \( N_{l,k}(u), \ u \in [u_l, u_{l+k}], \ l = 0 \ldots m, \) where

\[
N_{l,k}(u) = \frac{u - u_l}{u_{l+k+1} - u_l} N_{l,k-1}(u) + \frac{u_{l+k} - u}{u_{l+k} - u_{l+1}} N_{l+1,k-1}(u)
\]

\[
N_{l,1} = \begin{cases} 1, & u_l \leq u < u_{l+1} ; \\ 0, & \text{otherwise.} \end{cases} \quad l = 0 \ldots m,
\]

- and curve segments \( c_k(u), \ u \in [u_l, u_{l+1}], \ l = (k-1) \ldots m. \)
NURB Surfaces

\[
x(u, v) = \frac{\sum_{j=0}^{n} \sum_{l=0}^{m} \omega_{k,l} N_k(u) N_l(v)}{\sum_{j=0}^{n} \sum_{l=0}^{m} \omega_{k,l} N_k(u) N_l(v)}, \quad u \in [u_{k-1}, u_{m+1}], \quad v \in [v_{l-1}, v_{n+1}],
\]

defined by

- two orders \( k \) and \( l \) (equalling the degree of the polynomials - 1)
- a set of 3D control points \( \{d_{0,0}, \ldots, d_{m,n}\} \)
- a set of real weights \( \{\omega_{0,0}, \ldots, \omega_{m,n}\} \).
NURB Surfaces cont'd

- a set of real $u$-knots $\{u_0, \ldots, u_{m+k} \mid u_l \leq u_{l+1}, l = 0 \ldots (m + k - 1)\}$,
- a set of real $v$-knots $\{v_0, \ldots, v_{n+l+1} \mid v_j \leq v_{j+1}, j = 0 \ldots (n + l - 1)\}$,
- B-spline basis functions $N_{i,k}(u), u \in \left[u_i, u_{i+k}\right], l = 0 \ldots m$,
  $N_{i,k}(u)$ defined as for the curve case,
- B-spline basis functions $N_{j,l}(v), v \in \left[v_j, v_{j+l}\right], j = 0 \ldots n$,
  $N_{j,l}(v)$ defined as for the curve case, and
- surface segments $x_{i,j}(u,v), u \in \left[u_i, u_{i+1}\right], l = (k - 1) \ldots m$,
  $v \in \left[v_j, v_{j+1}\right], j = (l - 1) \ldots n$. 

\[\text{Diagram} \]
GIPSI
Graphically Interactive PATRAN Structured Grid Interface

IGES

PATRAN Phase 1 Geometry

Structural Analysis Model

GIPEI PCL Library

Input Review/Session File

Define Surface

Modify Surfaces

Concatenate Two Surfaces

From Patch

View Surfaces

Set Of Curves

Output

Surface File

Review/Session File

598
INPUT DIRECTIVE OR "END"
SET MENU ON
"MENU" IS NOW ON (LHS ON ).
IGES Entities: (IGES V.5.0)

Circular Arc (type 100)
Composite Curve (type 102)
Conic Arc (type 104)
  • Parabola (form 1)
  • Ellipse (form 2)
  • Hyperbola (form 3)
  • General Equation (form 0)

Copious Data
  • Center line (form 20 - 21)
  • Section (form 31 - 38)
  • Witness line (form 40)

Plane (type 108)
Line (type 110)
Parametric Spline Curve (type 112)
Parametric Spline Surface (type 114)
Point (type 116)
Ruled Surface (type 118)

Equal Relative Arc Length (form 0)
Equal Relative Parametric Value (form 1)

Surface of Revolution (type 120)
Tabulated Cylinder (type 122)
Transformation Matrix (type 124)

Orthogonal Matrix (det = 1) (form 0) right handed system
Orthogonal Matrix (det = -1) (form 1) left handed system
Rational B–Spline Curve (type 126)
- General Parameters (form 0)
- Line (form 1)
- Circular Arc (form 2)
- Elliptical Arc (form 3)
- Parabolic Arc (form 4)
- Hyperbolic Arc (form 5)

Rational B–Spline Surface (type 128)
- General (form 0)
- Plane (form 1)
- Right Circular Cylinder (form 2)
- Cone (form 3)
- Sphere (form 4)
- Torus (form 5)
- Surface of Revolution (form 6)
- Tabulated Cylinder (form 7)
- Ruled Surface (form 8)
- General Quadric Surface (form 9)
IGES Entities: (IGES V.5.0) cont.

Rational B–Spline CurveOffset Curve (type 130)
Offset Surface (type 140)
Boundary Entity (type 141)
(set of curves lying on surface)

Curve on a Parametric Surface (type 142)
Boundary Surface (type 143)
Trimmed Parametric Surface (type 144)
User Defined Surface Data Form (type 5001)
1. FIRST, USE PATRAI 2.5 TO CREATE THE PC CURVES (THE LEFT UPPER WINDOW).

2. OUTPUT THE PC CURVES TO IGES FILES.

3. USE THE CAGI-IGES TRANSLATOR TO CONVERT THE IGES FILE BY RUNNING CAGI AND READ THE "PATRAI.IGS.1" AS INPUT.

4. LEFT DOWN WINDOW SHOW THERE ARE 21 POINTS AND 17 PC CURVES BEEN CONVERTED.

5. PLOT THE PC CURVES ON THE RIGHT UPPER WINDOW.
1>. FIRST, USE PATRAN 2.5 TO CREATE THE PC SURFACE (THE LEFT UPPER WINDOW).

2>. TRANSLATE THE PARAMETER SURFACES TO IGES FILES.

3>. USE THE CAGI-IGES TRANSLATOR TO CONVERT THE IGES FILE BY RUNNING CAGI AND READ THE "BODYWING.IGS" AS INPUT.

4>. LEFT DOWN WINDOW SHOW THERE ARE 19 POINTS AND 6 PC CURVES 10 SURFACE AND 6 LINES BEEN CONVERTED.

5>. PLOT THE PC SURFACE ON THE RIGHT UPPER WINDOW.
1>. USE PATRAN TO READ THE IGES FILES WHICH CONTAIN TEN BODY OF REV. ENTITY 120 AND PLOT BY PATRAN.

2>. USE THE CAGI-IGES TRANSLATOR TO CONVERT THE IGES FILE BY RUNNING CAGI AND READ THE "BODROV.IGS" AS INPUT.

3>. LEFT DOWN WINDOW SHOW THERE ARE 12 BODY OF REV. ENTITIES (SURFACES) BEEN CONVERTED.

4>. PLOT THE SURFACES ON THE RIGHT UPPER WINDOW.

** INQUIRE FUNCTION FOR THE CONVERT IGES **

```
Indy:yu 39 > cd ...
Indy:yu 40 > cagi

PLEASE KEVIN THE IGES FILE NAME ....
bodrov.igs

** OF POINTS CONVERT : 0
** OF LINES CONVERT : 1
** OF CIRARC CONVERT : 0
** OF PCURVE CONVERT : 6
** OF PC SURF CONVERT : 0
** OF BODY OF BOV. : 6
** OF CONIC ARC : 0

***********************************************
** INQUIRE FUNCTION FOR THE CONVERT IGES **
```
```
$'point', point= 2, rl=-10.00000, 0.0000000e+00, 0.0000000e+00 
$'point', point= 3, rl=0.0000000e+00, 10.00000, 0.0000000e+00 
$'point', point= 4, rl=10.00000, 10.00000, 0.0000000e+00 
$'point', point= 5, rl=20.00000, 10.00000, 0.0000000e+00 
$'point', point= 6, rl=20.00000, 0.0000000e+00, 0.0000000e+00 
$'point', point= 7, rl=10.00000, 0.0000000e+00, 0.0000000e+00 
$'point', point= 8, rl=1.00000, 1.00000, 0.0000000e+00 
$'line', points= 10, rl=-10.00000, 0.0000000e+00, 0.0000000e+00, 
  x2=0.0000000e+00, 0.0000000e+00, 0.0000000e+00, coreout= 1 $ 
$'conicur', type='circle', points=20, angle= 0.00, 90.00, radius= 10.00, 
  coreout= 2 $ 
$'trans', corein= 2, rl= 0.00, 0.00, 0.00, r2= 0.00, 0.00, 0.00, 
  cosines=1.0, 0.0, 0.0, 0.0, 1.0, 0.0, 0.0, -1.0, coreout= 2 $ 
$'line', points= 10, rl=0.0000000e+00, 10.00000, 0.0000000e+00, 
  x2=10.00000, 10.00000, 0.0000000e+00, coreout= 3 $ 
$'line', points= 10, rl=10.00000, 10.00000, 0.0000000e+00, 
  x2=20.00000, 10.00000, 0.0000000e+00, coreout= 4 $ 
$'line', points= 10, rl=20.00000, 10.00000, 0.0000000e+00, 
  x2=20.00000, 10.00000, 0.0000000e+00, coreout= 5 $ 
$'line', points= 10, rl=10.00000, 0.0000000e+00, 0.0000000e+00, 
  x2=20.00000, 0.0000000e+00, 0.0000000e+00, coreout= 6 $ 
$'conicur', type='circle', points=20, angle= 0.00, 90.00, radius= 1.00, 
  coreout= 7 $ 
$'trans', corein= 7, rl= -1.00, 0.00, 0.00, r2= 0.00, 0.00, 0.00, 
  cosines=1.0, 0.0, 0.0, 0.0, 1.0, 0.0, 0.0, -1.0, coreout= 7 $ 
$'line', points= 10, rl= 1.00000, 1.00000, 0.0000000e+00, 
  x2=10.00000, 0.0000000e+00, 0.0000000e+00, coreout= 8 $ 
$'combine', corein=1,-8, form='plot3d', fileout=4$ 
$'end'$
```
- Simple Minded
- Portable
- Modular
- Journal File Execution Control
- FORTRAN & C
- Extensive Error Checking
- Easy Customization
- ( lucr) Genie Shortcomings
  - Spacings
  - Geometry/Grid Manipulations
  - On Line Storage
  - B Spline - NURBS Applications
  - Geometry Interface
  - Interactive Visualization
USING ADAPTIVE GRID IN MODELING ROCKET NOZZLE FLOW

By Alan S. Chow * and Kang-Ren Jin**

*NASA/Performance Analysis Branch, Marshall Space Flight Center, AL 35812.

**Department of Civil Engineering, Mississippi State University, Mississippi State, MS 39762.

ABSTRACT

The mechanical behavior of a rocket motor internal flow field results in a system of nonlinear partial differential equations which cannot be solved analytically. However, this system of equations called the Navier-Stokes equations can be solved numerically. The accuracy and the convergence of the solution of the system of equations will depend largely on how precisely the sharp gradients in the domain of interest can be resolved. With the advances in computer technology, more sophisticated algorithms are available to improve the accuracy and convergence of the solutions. An adaptive grid generation is one of the schemes which can be incorporated into the algorithm to enhance the capability of numerical modeling. It is equivalent to putting intelligence into the algorithm to optimize the use of computer memory. With this scheme, the finite difference domain of the flow field called the grid does neither have to be very fine nor strategically placed at the location of sharp gradients. The grid is self adapting as the solution evolves. This scheme significantly improve the methodology of solving flow problems in rocket nozzle by taking the refinement part of grid generation out of the hands of computational fluid dynamics (CFD) specialists and place it into the computer algorithm itself.
Using Adaptive Grid in Modeling Rocket Nozzle Flow

by

Alan S. Chow
&
Kang-Ren Jin

April 29, 1992
OBJECTIVE

- To develop a user-friendly solution-adaptive grid generator that will simplify grid generation process so that a 'perfect' grid can be generated everytime without the intervention of CFD experts.
\[
\frac{\partial U}{\partial t} + \frac{\partial E}{\partial x} + \frac{\partial F}{\partial y} + \frac{\partial G}{\partial z} = 0
\]

\[
U = \begin{bmatrix}
\rho \\
\rho u \\
\rho v \\
\rho w \\
E_t
\end{bmatrix}
\]

\[
F = \begin{bmatrix}
\rho u \\
\rho u^2 + p - \tau_{xx} \\
\rho u v - \tau_{xy} \\
\rho v^2 + p - \tau_{yy} \\
\rho v w - \tau_{yx}
\end{bmatrix}
\]

\[
E = \begin{bmatrix}
\rho u \\
\rho u^2 + p - \tau_{xx} \\
\rho u v - \tau_{xy} \\
\rho v^2 + p - \tau_{yy} \\
(E + p)u - ut_{xx} - ut_{xy} - wt_{xx} + q_x
\end{bmatrix}
\]

\[
G = \begin{bmatrix}
\rho w \\
\rho w u - \tau_{wx} \\
\rho w v - \tau_{wy} \\
\rho w^2 + p - \tau_{ww} \\
(E + p)w - ut_{xx} - ut_{yx} - wt_{ww} + q_w
\end{bmatrix}
\]
\[
\frac{\partial S}{\partial t} + \frac{\partial F_j}{\partial X_j} = \frac{1}{Re} \frac{\partial G_j}{\partial X_j}
\]

where \( S \) is a vector containing the conservation variables,

\[
S = \begin{bmatrix}
\rho \\
\rho u_j \\
E
\end{bmatrix}
\]

The \( F_j \) vectors represent the inviscid flux vectors,

\[
F_j = \begin{bmatrix}
\rho u_j \\
\rho u_j u_j + P \delta_{ij} \\
(E + P)u_j
\end{bmatrix}
\]

and \( G_j \) vectors are the viscous flux vectors

\[
G_j = \begin{bmatrix}
0 \\
\tau_{ij} \\
u_k \tau_{jk} - q_j
\end{bmatrix}
\]

\[
\xi_j = \xi_j(X_j, t)
\]

\[
\frac{\partial S}{\partial t} + \frac{\partial F_j}{\partial X_j} = \frac{1}{Re} \frac{\partial G_j}{\partial X_j}
\]

\[
S = \frac{1}{J} S
\]

\[
F_j = \frac{1}{J} \left( \frac{\partial \xi_j}{\partial X_b} S + \frac{\partial \xi_j}{\partial X_b} F_b \right)
\]

\[
G_j = \frac{1}{J} \frac{\partial \xi_j}{\partial X_b} G_b
\]
GRID GENERATION METHODS

- Complex variables (Conformal Mapping)
- Algebraic
- Partial Differential Equations (PDE)
  - Elliptic
  - Hyperbolic
\[ \nabla^2 \xi' = P' \quad (i = 1, 2) \]

\[ \xi_{xx} + \xi_{yy} = P \]

\[ \eta_{xx} + \eta_{yy} = Q \]
\[ P(\xi, \eta) = - \sum_{i=1}^{N} a_i \text{sign}(\xi - \xi_i) \exp(-c_i |\xi - \xi_i|) \]

\[ - \sum_{i=1}^{M} b_i \text{sign}(\xi - \xi_i) \exp(-d_i [(\xi - \xi_i)^2 + (\eta - \eta_i)^2]^{1/2}) \]

\(\xi = \xi_i\) Attraction line

Attraction point

\(\eta\)
\[ W(x) x_\xi = \text{constant} \]

\[ W(x) \quad : \text{weight function} \]

\[ x(\xi) : \text{point distribution} \]

\[ x_{\xi\xi} W + x_\xi W_\xi = 0 \]

\[ x_{\xi\xi} + P x_\xi = 0 \]

\[ P = -\frac{x_{\xi\xi}}{x_\xi} = \frac{W_\xi}{W} \]

\[ P_i = \frac{W_\xi}{W} \quad (i = 1, 2, 3) \]

\[ W = 1 + |\nabla P| \]
GRID
Elliptic Grid Generator
Final Grid For RSRM Nozzle
The Adaptive Grid at Time Step = 500 for Inviscid Flow.
The Adaptive Grid at Time Step = 1000 for Inviscid Flow.
The Weight Function Distribution at Time Step = 300 for Viscous Flow.
UNIFORM Y GRID
PRESSURE

UNIFORM Y GRID
UNIFORM ARC GRID
UNIFORM ARC GRID
VISCOUS NOZZLE FLOW
GRID

VISCOUS NOZZLE FLOW
The Weight Function Distribution at Time Step = 5500 for Viscous Flow.
The Weight Function Distribution at Time Step = 1000 for Viscous Flow.
CONCLUSIONS & RECOMMENDATIONS

✓ 2-D Solution-Adaptive Grid Generator has been completed and demonstrated.

✓ It simplifies grid generation and makes better use of computer and human resources.

✓ Should be verified on various CFD codes to ensure robustness.

✓ Continue development in 3-D and Time-accurate Solution-Adaptive Grid Generator.
Abstract for the Tenth CFD Working Group Meeting:

Complex Three-Dimensional Internal Flows in the ASRM and RSRM Aft End Segments

Presented By: Dr. Edward J. Reske
Dr. Dana F. Billings
Ms. Joni W. Cornelison

Results from computational fluid dynamic analyses for complex three-dimensional internal flows in the Advanced Solid Rocket Motor (ASRM) and Redesigned Solid Rocket Motor (RSRM) are presented. In particular, a parametric study for the case of a gimbaled nozzle in these motors at various burn times and gimbal angles is presented. The resultant pressure fields are used to determine the location of the center of pressure and hinge moments due to the internal flow for these geometries.
COMPLEX THREE-DIMENSIONAL FLOWS IN THE ASRM AND RSRM AFT END SEGMENTS

CFD BRANCH, ED32

Ed Reske
Dana Billings
Joni Cornelison
ASRM AFT END FLOW ANALYSIS

- **Objective**
  - Characterize flow environment in aft end of ASRM

- **Purpose**
  - Hinge moments due to internal flow for a gimballed nozzle
  - Nozzle performance
  - Heat transfer for insulation sizing

- **Approach**
  - Axisymmetric analyses
    -- CMINT (48K and 24K grid points)
    -- FDNS (14K grid points)
  - Three-dimensional gimballed nozzle analysis
    -- FDNS3D (14K X 26 planes = 366K grid points)

- **Results**
  - Axisymmetric analysis complete
  - 3-D gimballed nozzle analyses nearing completion
GEOMETRY
Green Outline = ASRM at 19 seconds
Red Outline = ASRM at 19 seconds

GRID 1
220x64

GRID 2
220x64

GRID 3
220x64

GRID 4
220x64
GEOMETRY
Green Outline = ASRM at 19 seconds
Red Outline = RSRM at 19 seconds

Pivot Point
GEOMETRY

ASPM at 115 second burn line with nozzle gimballed at 0 degrees: 3/24/92
## Hinge Moments and Loads

<table>
<thead>
<tr>
<th>MOTOR</th>
<th>BURN TIME</th>
<th>GIMBAL ANGLE</th>
<th>HINGE MOMENT</th>
<th>NORMAL LOAD</th>
<th>AXIAL LOAD</th>
<th>CENTER OF PRESSURE wrt pivot</th>
<th>CENTER OF PRESSURE wrt throat</th>
</tr>
</thead>
<tbody>
<tr>
<td>ASRM</td>
<td>19</td>
<td>4</td>
<td>627 K</td>
<td>39.6 K</td>
<td>2.52 M</td>
<td>-15.9</td>
<td>+1.7</td>
</tr>
<tr>
<td>ASRM</td>
<td>19</td>
<td>8</td>
<td>1.28 M</td>
<td>80.8 K</td>
<td>2.51 M</td>
<td>-16.0</td>
<td>+1.6</td>
</tr>
<tr>
<td>RSRM</td>
<td>19</td>
<td>4</td>
<td>730 K</td>
<td>49.5 K</td>
<td>3.18 M</td>
<td>-14.7</td>
<td>+2.9</td>
</tr>
<tr>
<td>ASRM</td>
<td>115</td>
<td>4</td>
<td>150 K</td>
<td>6.9 K</td>
<td>1.25 M</td>
<td>-21.7</td>
<td>-4.1</td>
</tr>
<tr>
<td>ASRM</td>
<td>115</td>
<td>8</td>
<td>546 K</td>
<td>12.8 K</td>
<td>1.25 M</td>
<td>-42.6</td>
<td>-25.0</td>
</tr>
<tr>
<td>RSRM</td>
<td>114</td>
<td>4</td>
<td>165 K</td>
<td>14.7 K</td>
<td>0.96 M</td>
<td>-11.2</td>
<td>+6.4</td>
</tr>
</tbody>
</table>

Note: All the above hinge moments are non-restoring torques. The axial load acts along the axis of symmetry of the nozzle, and the normal load acts in the direction perpendicular to this axis, with both components acting in a direction away from the motor. The center of pressure is determined by finding a point on the nozzle axis of symmetry where the torque vanishes. A negative value indicates that it is upstream of the reference point, whereas a positive value indicates that it is downstream of the reference point.
PARTICLE TRACES

RSHM with nozzle inclined at 4 degrees.
PARTICLE TRACES

ASAN at 19 sec. burn time with nozzle gimbaled 4 degrees.
PARTICLE TRACES

RASM with nozzle gimbaled at 4 degrees.
PARTICLE TRACES
ASRM at 19 sec. burn time with nozzle sinballed 4 degrees.
PARTICLE TRACES
ARSM with nozzle gimbaled at 4 degrees.
A simple model for calculating the hinge moment about a pivot point that is shifted relative to the nominal location:

\[ T_z = T_{pz} + y F_N - x F_A \]

where, using a body-fixed coordinate system,

- \( T_z \) = torque about the new pivot point;
- \( T_{pz} \) = torque about the nominal pivot point;
- \( y \) = axial displacement of the new pivot point;
- \( x \) = normal displacement of the new pivot point;
- \( F_N \) = normal load;
- \( F_A \) = axial load.
An Analysis of the Flow Field in the Region of the ASRM Field Joints

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

Abstract

The flow field in the region of a solid rocket motor field joint is very important since fluid dynamic and mechanical propellant stresses can couple to cause a motor failure at a joint. This paper presents an examination of the flow field in the region of the ASRM field joints. The analyses were performed as a first step in assessing the design of the ASRM forward and aft field joints in order to assure the proper operation of the motor prior to further development or test firing.

The analyses discussed are a first step in the process of a full analysis of the ASRM field joints. The first step involves the analysis of both the forward and aft motor field joints at the 0 and 19 second motor burn back times. The zero second burn back time has the potential for causing the greatest possible fluid dynamic induced stresses at the joints. This is because the port flow Mach number and dynamic pressure decrease as the motor burns, thus reducing the stresses at the joints. Initial analyses have also been performed on the inhibitor stub left protruding into the port flow field at the field joint caused by propellant burn back at the 19 second burn back time. The analyses discussed are for non-deformed propellant grains. Analyses of the field joints deformed from cure shrinkage, thermal cool down and gravity loading will be included at a later time. Also a coupled fluid dynamic/mechanical stress analysis will be performed in conjunction with NASA/MSFC mechanical stress analysts in order to assess any adverse dynamic mechanical effects of the flow field on the propellant grain shape.

The analyses presented in this paper have been performed by employing a two-dimensional axisymmetric assumption. Fluent/BFC, a three dimensional full Navier-Stokes flow field code, has been used to make the numerical calculations. This code utilizes a staggered grid formulation along with the SIMPLER numerical algorithm. Wall functions are used to determine the character of the laminar sublayer flow and a standard $\kappa-\varepsilon$ turbulence model is used to close the fluid dynamic equations.

The analyses performed to this date verify that the ASRM field joint design operates properly. The fluid dynamic stresses at the field joints are small due to the inherent design of the field joints. A problem observed in some other solid rocket motors is that large fluid dynamic stresses are generated at the motor joint on the downstream propellant grain due to forward facing step geometries. The design of the ASRM field joints are such that this is not a problem as shown by the analyses. Also, the analyses of the inhibitor stub left protruding into the port flow from normal propellant burn back show that more information is necessary to complete these analyses. These analyses were performed as parametric analyses in relation to the height of the inhibitor stub left protruding into the motor port. A better estimate of the amount of the inhibitor stub remaining at later burn times must be determined since the height which the inhibitor stub protrudes into the port flow drastically affects the fluid dynamic induced stresses on the propellant grain at the field joints.
AN ANALYSIS OF THE FLOW FIELD IN THE REGION OF THE ASRM FIELD JOINTS

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

Tenth Annual CFD Working Group Meeting
Session 7
NASA/MSFC

April 29, 1992
OBJECTIVES

1) DETERMINE SLOT/PORT FLOW INTERACTIONS FOR ASRM FWD AND AFT FIELD JOINT DESIGNS

2) PERFORM PRELIMINARY CFD ANALYSES OF INITIAL GRAIN CONFIGURATIONS AT THE FORWARD AND AFT FIELD JOINTS TO DETERMINE PROPELLANT GRAIN PRESSURE LOADS AND IDENTIFY POTENTIAL EARLY DESIGN PROBLEMS
CFD METHODOLOGY

- Governing equations are the 3-D ensemble-averaged Navier Stokes equations in conservation form

- Closure of the equations by the standard two-equation $\kappa-\varepsilon$ model of turbulence

- Wall functions used to determine near wall gradients

- Discretization method
  - Governing equations are written in component form using contravariant velocity components
  - This allows the use of a boundary fitted curvilinear coordinate system
  - Numerical method is finite volume based
  - Staggered grid storage system is used
  - Convection and diffusion fluxes are approximated using a power-law scheme
  - Time derivatives are calculated using a fully implicit first order scheme

- Pressure-velocity coupling is accomplished by using the SIMPLER algorithm

- Solver uses linearized block implicit scheme
THE ASRM

Overview

- HTPB PROPELLANT
- FORWARD DOME
- CENTER SEGMENT
- IGNITER ASSEMBLY
- FORWARD GRAIN FIN AREA
- ET ATTACH RING
- STIFFENER RINGS
- AFT SEGMENT
- EPDM INSULATION
- TWO BOLTED FIELD JOINTS
- NOZZLE ASSEMBLY

- CARBON PHENOLIC EXIT CONE
- D&AC HOUSING
OVERVIEW OF THE GENERAL SLOT GEOMETRY

- Propellant Surface
- Inhibitor Surface
- Symmetry Boundary

Step Height

Motor Field Joint Slots

Inhibitor Stub Height
ASRM FIELD JOINT CONFIGURATIONS ANALYZED

- 0 SECOND BURN TIME MOTOR CONFIGURATION
  - FORWARD SLOT
  - AFT SLOT

- 19 SECOND BURN TIME MOTOR CONFIGURATION
  - FORWARD SLOT
    INHIBITOR STUB HEIGHT, 3.9 INCHES
    INHIBITOR STUB HEIGHT, 0 INCHES
  - AFT SLOT
    INHIBITOR STUB HEIGHT, 3.9 INCHES
    INHIBITOR STUB HEIGHT, 0 INCHES
<table>
<thead>
<tr>
<th>ASAM FORWARD SLOT/Ø SECOND BURN/175X35</th>
<th>FLUENT/BFC V3.02</th>
</tr>
</thead>
<tbody>
<tr>
<td>Finite-Difference Grid</td>
<td>2D Domain</td>
</tr>
<tr>
<td>creare.x</td>
<td>Steady State</td>
</tr>
</tbody>
</table>

670
<table>
<thead>
<tr>
<th>Parameter</th>
<th>AFT Slot</th>
<th>Forward Slot</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Static Pressure</td>
<td>821.9 psia</td>
<td>855.9 psia</td>
</tr>
<tr>
<td>Average Port Velocity</td>
<td>177.5 ft/s</td>
<td>877.5 ft/s</td>
</tr>
<tr>
<td>Stagnation Temperature at the Inlet</td>
<td>6345 °R</td>
<td>6345 °R</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.128</td>
<td>1.128</td>
</tr>
<tr>
<td>Propellant Injection Velocity</td>
<td>13.467 ft/s</td>
<td>9.9837 ft/s</td>
</tr>
<tr>
<td>Molecular Weight</td>
<td>29.489</td>
<td>29.489</td>
</tr>
<tr>
<td>Mass Flow Rate (Inlet)</td>
<td>8682 lbm/s</td>
<td>6178 lbm/s</td>
</tr>
<tr>
<td>CFD Calculated Mass Flow Rate (Inlet)</td>
<td>8562 lbm/s</td>
<td>6103 lbm/s</td>
</tr>
</tbody>
</table>
ASRM Fwd Slot Undeformed Grain Port Pressures On the Surface and at the Motor Centerline

Pressure (psia) 820

Axial Distance (inches)

- Wall
- Centerline

0 Second Burn Time

Bottom of Slot
Downstream Edge of Slot
Upstream Edge of Slot
KEY

Minimum =
0.00E+00
0.00E+00
4.68E-02
9.36E-02
1.40E-01
1.87E-01
2.34E-01
2.81E-01
3.28E-01
3.74E-01
4.21E-01
4.68E-01
5.15E-01
5.62E-01
6.09E-01
6.55E-01
7.02E-01

Maximum =
7.02E-01

ASRM AFT SLOT AT 0 SECONDS BURN TIME
Fluent/BFC V3.02
Raster Plot of MACH-NUMBER
2D Domain
Steady State
ASRM Aft Slot Undeformed Grain Port Pressures On the Surface and at the Motor Centerline

![Graph showing pressure variations with axial distance.](https://example.com/graph.png)
## ASRM Motor Field Joint Boundary Conditions
19 Second Burn Time Configuration

### Aft Slot

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Static Pressure</td>
<td>861.3 psia</td>
</tr>
<tr>
<td>Average Port Velocity</td>
<td>746.25 ft/s</td>
</tr>
<tr>
<td>Stagnation Temperature at the Inlet</td>
<td>6317.6 °R</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.128</td>
</tr>
<tr>
<td>Propellant Injection Velocity</td>
<td>10.135 ft/s</td>
</tr>
<tr>
<td>Molecular Weight</td>
<td>29.295</td>
</tr>
<tr>
<td>Mass Flow Rate (Inlet)</td>
<td>8846 lbm/s</td>
</tr>
<tr>
<td>CFD Calculated Mass Flow Rate (Inlet)</td>
<td>8824 lbm/s</td>
</tr>
</tbody>
</table>

### Forward Slot

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Static Pressure</td>
<td>886.0 psia</td>
</tr>
<tr>
<td>Average Port Velocity</td>
<td>521.2 ft/s</td>
</tr>
<tr>
<td>Stagnation Temperature at the Inlet</td>
<td>6317.6 °R</td>
</tr>
<tr>
<td>Ratio of Specific Heats</td>
<td>1.128</td>
</tr>
<tr>
<td>Propellant Injection Velocity</td>
<td>9.956 ft/s</td>
</tr>
<tr>
<td>Molecular Weight</td>
<td>29.295</td>
</tr>
<tr>
<td>Mass Flow Rate (Inlet)</td>
<td>5963 lbm/s</td>
</tr>
<tr>
<td>CFD Calculated Mass Flow Rate (Inlet)</td>
<td>5944 lbm/s</td>
</tr>
<tr>
<td>KEY</td>
<td></td>
</tr>
<tr>
<td>-----</td>
<td></td>
</tr>
<tr>
<td>Minimum = 5.61E+06</td>
<td></td>
</tr>
<tr>
<td>5.61E+06</td>
<td></td>
</tr>
<tr>
<td>5.63E+06</td>
<td></td>
</tr>
<tr>
<td>5.65E+06</td>
<td></td>
</tr>
<tr>
<td>5.68E+06</td>
<td></td>
</tr>
<tr>
<td>5.70E+06</td>
<td></td>
</tr>
<tr>
<td>5.72E+06</td>
<td></td>
</tr>
<tr>
<td>5.74E+06</td>
<td></td>
</tr>
<tr>
<td>5.77E+06</td>
<td></td>
</tr>
<tr>
<td>5.79E+06</td>
<td></td>
</tr>
<tr>
<td>5.81E+06</td>
<td></td>
</tr>
<tr>
<td>5.83E+06</td>
<td></td>
</tr>
<tr>
<td>5.86E+06</td>
<td></td>
</tr>
<tr>
<td>5.88E+06</td>
<td></td>
</tr>
<tr>
<td>5.90E+06</td>
<td></td>
</tr>
<tr>
<td>5.92E+06</td>
<td></td>
</tr>
<tr>
<td>5.95E+06</td>
<td></td>
</tr>
</tbody>
</table>

**ASAM AFT SLOT/READJUSTED GRID HALF-HEIGHT INHIB.**

**FLUENT/BFC V3.02**

**Raster Plot of PRESSURE**

2D Domain

Steady State
ASRM Fwd Slot Undeformed Grain Port Pressures On the Surface and at the Motor Centerline

19 Second Burn Time
3.87 Inch Inhibitor Height

Axial Distance (inches)

Pressure (psia) 860
890
880
870
860
850
840
830
800 850 900 950 1000 1050 1100

Upstream Edge of Slot
Bottom of Slot
Downstream Edge of Slot
ASRM Aft Slot Undeformed Grain Port Pressures On the Surface and at the Motor Centerline

19 Second Burn Time
3.87 Inch Inhibitor Height

Bottom of Slot
Upstream Edge of Slot
Downstream Edge of Slot
ASRM Aft Slot Undeformed Grain Port Pressures On the Surface and at the Motor Centerline

![Graph showing pressure (psia) vs. axial distance (inches) for different points: Upstream Edge of Slot, Bottom of Slot, Downstream Edge of Slot, and Centerline. The graph includes a note for 19 Second Burn Time, No Inhibitor Height.](image-url)
CONCLUSIONS

1) CFD ANALYSES HAVE BEEN COMPLETED FOR THE AFT AND FORWARD SLOTS AT 0 AND 19 SECOND BURN TIMES.

2) THE PRESSURE LOADS AT 0 SECOND BURN TIME ARE SMALL.

3) THE PRESSURE LOADS ON THE PROPELLANT GRAIN AT THE MOTOR JOINTS AT 19 SECONDS BURN TIME IS SIGNIFICANTLY AFFECTED BY THE INHIBITOR HEIGHT AND ORIENTATION.

4) THE SLOT REGION ANALYSES ARE BEING EXTENDED TO INCLUDE ACTUAL DEFORMED GRAIN AND ERODED INHIBITOR GEOMETRIES.

5) INTERACTIVE CFD/STRUCTURAL ANALYSES ARE REQUIRED TO PROVIDE REALISTIC ASSESSMENT OF THE SLOT/PORT FLOW INTERACTIONS AND RESULTING PROPELLANT LOADS.
Effect of Including Variable Gas Properties and Entrained Particles in the Flow Analysis of the ASRM Nozzle

Curtis D. Clayton, Ph.D.
Aerojet ASRM Division
Iuka, Mississippi

ABSTRACT

CFD analyses of solid rocket motors typically use constant fluid properties throughout the flow domain. While this may be an acceptable approximation inside the motor chamber, it is probably not a good approach for the expansion that occurs in the nozzle.

As the flow expands from 900 psi chamber pressure, the temperature decreases by 35%, viscosity and thermal conductivity are reduced by similar amounts (25%), and the specific heat ($C_p$) and specific heat ratio ($\gamma$) change by 4% and 1%, respectively. While the change in $\gamma$ appears to be small, its effect is significant because of its use as an exponent in the isentropic expansion equations.

The objective of this study is to determine the effect of using constant gas properties for the analysis of the ASRM nozzle and to gain more understanding concerning those types of analysis which might require this additional complexity.

Kinetics data for viscosity, thermal conductivity, specific heat ($C_p$), and specific heat ratio ($\gamma$) are extracted from the Solid Propellant Rocket Motor Performance Prediction Computer Program (SPP) and tabulated as a function of temperature. These tables are added to the Aerovisc CFD code in place of the constant gas property values.

The results of a CFD analysis of the ASRM 48" motor with constant gas properties will be compared with an analysis which uses variable gas properties. Mach number, surface pressure, and torque plots will be presented. A full scale ASRM analysis using SPP with and without particle flow will also be presented.
Effect of Including Variable Gas Properties and Entrained Particles in the Flow Analysis of the ASRM Nozzle

Curtis D. Clayton, Ph.D.

APRIL 28 -30, 1992
Overview

- CFD analyses frequently assume that the fluid behaves as an ideal gas with constant properties.
- Combustion gases in solid rocket motors are not ideal.
  - Chemical and phase changes
  - Entrained particles in the flow
  - Extreme temperature changes effect gas properties
- This study compares several analyses with and without these effects to determine their impact on the results.
Variable Gas Properties

- Performed Aerovisc CFD analysis of 48" motor with a scaled ASRM nozzle.
- Case 1 was run with constant gas properties typical of chamber conditions.
- Case 2 used local temperatures to determine:
  - Specific heats - $C_p$ and $C_v$
  - Viscosity, $\mu$
  - Thermal heat transfer coefficient, $K$
- Values were obtained from the SPP kinetics module.
Variation of Cp with Temperature

- **Cp** with Frozen Properties
- Cp - Equilibrium Expansion
- Cp - Kinetic Expansion

Temperature (R)

Cp (ft-lbf/lbm-R)

300 400 500 600 700 800

3500 4000 4500 5000 5500 6000 6500
## Effect of Temperature Changes on Thermodynamic Properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Change for 2000 R Temperature Change (4000-6000 R)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cp</td>
<td>3.5 %</td>
</tr>
<tr>
<td>Cv</td>
<td>3.6 %</td>
</tr>
<tr>
<td>Conductivity</td>
<td>33 %</td>
</tr>
<tr>
<td>Viscosity</td>
<td>27 %</td>
</tr>
</tbody>
</table>
Nozzle Exit Plane
48" Motor Nozzle Exit Plane
Mach Number

Variable Gas Properties
Constant Gas Properties

Mach Number

Radial Distance (Inches)
Nozzle Wall Profile
Effect of Pressure Differences

- Fictitious torque created by applying different pressure profiles to different halves of the nozzle.
- Net torque 2.32 K in-lbf
  - 1.3% of torque due to variable properties
  - 342 K in-lbf if scaled to ASRM
- This would be a significant error if it were present in the gimbaled nozzle analysis.
48" Motor Nozzle Torque

Torque per Unit Length (lbf)

Axial Distance (inches)

Constant Gas Properties

Variable Gas Properties

Aerovisc CFD Analysis
Entrained Particle Flow

- Compared SPP analysis with and without particles in full scale ASRM.
- SPP - Solid Rocket Motor Performance Program
  - Industry standard for motor performance
  - Inviscid, axisymmetric, real gas flow
  - Method of Characteristics
- Three particle groups with diameters of 6.3\(\mu\), 11.7\(\mu\), and 21.8\(\mu\).
48" Motor Nozzle Exit Plane
Mach Number

- CFD with Variable Gas Properties
- CFD with Constant Gas Properties
- SPP without Particles
- SPP with Particles

Radial Distance (inches)
Effect of Particles on Wall Pressure in Full Scale ASRM Nozzle

[Graph showing the effect of particles on wall pressure in an ASRM nozzle. The graph compares pressure (psia) against axial distance (inches) with and without particles.]
Effect of Particles on Torque in Full Scale ASRM Nozzle

Torque per unit Length (K lbf)

Axial Distance (inches)
Effect of Pressure Differences

- Fictitious torque created by applying different pressure profiles to different halves of the nozzle.
- Net torque 142 K in-lbf
  - 0.39% of torque due to particles
  - Net torque would have been greater if pressure profiles had not crossed
- This would be a significant error if it were present in the gimbaled nozzle analysis.
Torque Differences With and Without Particles
Flow Field Effects

- Particles slow the flow along the center line.
  - 33% mass fraction
  - Less actual gas per unit volume to expand
  - Velocity difference adds to drag

- Using an effective R does not correct for these problems because the particles are not uniformly distributed.

- When the nozzle is gimbaled the low-flow, particle entrained region comes closer to the wall.
ASRM Nozzle Exit Plane
Mach Number

Mach Number

Without Particles

With Particles

Radial Distance (inches)
Particle laden flow retards velocity in center.
Flow along wall is nearly equal to non-particle values.
ASRM Gimbaled Nozzle

※ Particles influence flow near nozzle wall.
※ Flow decreases along lower wall; increases on upper wall.
Summary

- Actual gas properties and characteristics do affect the flow field.
  - Especially compressible flows
- Particles have a significant impact on the flow field.
- Torque calculations magnify these effects.
A TWO-PHASE RESTRICTED EQUILIBRIUM MODEL FOR COMBUSTION OF METALIZED SOLID PROPELLANTS†

J. S. Sabnis*, F. J. de Jong and H. J. Gibeling
Scientific Research Associates, Inc.
Glastonbury, Connecticut

ABSTRACT

An Eulerian-Lagrangian two-phase approach has been adopted to model the multi-phase reacting internal flow in a solid rocket with a metalized propellant. An Eulerian description has been used to analyze the motion of the continuous phase which includes the gas as well as the small (micron-sized) particulates, while a Lagrangian description is used for the analysis of the discrete phase which consists of the larger particulates in the motor chamber. The particulates consist of Al and Al₂O₃, such that the particulate composition is 100% Al at injection from the propellant surface with Al₂O₃ fraction increasing due to combustion along the particle trajectory. An empirical model is used to compute the combustion rate for agglomerates while the continuous phase chemistry is treated using chemical equilibrium. The computer code was used to simulate the reacting flow in a solid rocket motor with an AP/HTPB/Al propellant. The computed results show the existence of an extended combustion zone in the chamber rather than a thin reaction region. The presence of the extended combustion zone results in the chamber flow field and chemical being far from isothermal (as would be predicted by a surface combustion assumption). The temperature in the chamber increases from about 2600 K at the propellant surface to about 3350 K in the core. Similarly the chemical composition and the density of the propellant gas also show spatially non-uniform distribution in the chamber. The analysis developed under the present effort provides a more sophisticated tool for solid rocket internal flow predictions than is presently available, and can be useful in studying apparent anomalies and improving the simple correlations currently in use. The code can be used in the analysis of combustion efficiency, thermal load in the internal insulation, plume radiation, etc.

† This work was supported by Phillips Laboratory, Edwards AFB, under Contract F04611-86-C-0096
* Currently at United Technologies Research Center, East Hartford, CT
OBJECTIVE

DEVELOPMENT OF A PREDICTIVE TOOL FOR REACTING MULTI-PHASE FLOW IN SOLID ROCKETS WITH METALIZED PROPELLANTS

MOTIVATION

• MOTOR PERFORMANCE AND DESIGN
  - STABILITY ANALYSIS
  - EROSI VE BURNING MODELING

• NOZZLE PERFORMANCE PREDICTON

• INSULATOR DESIGN

PREVIOUS ANALYSES

• POTENTIAL FLOW MODELS

• ROTATIONAL INVISCID FLOW MODEL
  - INADEQUATE WHEN VISCIOUS EFFECTS BECOME SIGNIFICANT

• INVISCID FLOW MODELS COUPLED WITH ITERATIVE BOUNDARY LAYER CORRECTIONS
  - REQUIRE DIVISION OF FLOW INTO INVISCID CORE AND BOUNDARY LAYER

• NAVIER-STOKES ANALYSES

• TWO-PHASE NAVIER-STOKES ANALYSES
CHARACTERISTICS OF METALIZED SOLID PROPELLANT COMBUSTION

- MULTI-PHASE REACTING FLOW WITH POLY-DISPERSED PARTICLES WHICH UNDERGO CHANGES IN SIZE, TEMPERATURE AND COMPOSITION

- DISPARATE TIME SCALES ASSOCIATED WITH THE DROPLET COMBUSTION AND THE GAS PHASE CHEMISTRY

- SPATIAL VARIATION OF GAS COMPOSITION DUE TO
  - SPECIES TRANSPORT
  - COMBUSTION OF ALUMINUM DROPLETS
  - CHANGE OF THERMODYNAMIC STATE

- GAS COMPOSITION DETERMINES MIXTURE MOLECULAR WEIGHT, SPECIFIC HEAT ETC.
TWO PHASE REACTING FLOW MODEL FOR SOLID ROCKET INTERNAL FLOWS

- CONTINUOUS PHASE CONSISTING OF PRODUCTS OF COMBUSTION FROM AP, BINDER AND REACTED ALUMINUM.

- DISCRETE PHASE CONSISTING OF DROPLETS CONTAINING UNREACTED ALUMINUM AND \( \text{Al}_2\text{O}_3 \) CAPS.

- KINETIC TIME SCALES FOR CONTINUOUS PHASE REACTIONS SIGNIFICANTLY SMALLER THAN FLUID DYNAMIC TIME SCALES. HENCE CHEMICAL EQUILIBRIUM ASSUMED FOR CONTINUOUS PHASE ANALYSIS.

- COMBUSTION OF ALUMINUM DROPLET LIMITED BY AVAILABILITY OF OXIDIZING SPECIES AT THE DROPLET. HENCE MASS TRANSPORT CONTROLLED COMBUSTION OF ALUMINUM DROPLET ASSUMED.

- AN EULERIAN-LAGRANGIAN ANALYSIS ADOPTED TO SIMULATE THE MULTI-PHASE REACTING FLOW.

CONTINUOUS PHASE ANALYSIS

- CONTINUITY EQUATION
  \[
  \frac{\partial (\rho \Phi)}{\partial t} + \nabla \cdot (\rho \Phi \textbf{U}) = m_v
  \]

- MOMENTUM EQUATION
  \[
  \frac{\partial (\rho \Phi \textbf{U})}{\partial t} + \nabla \cdot (\rho \Phi \textbf{U} \textbf{U}) = \nabla (\rho \Phi) + \nabla \cdot \textbf{F}_t + m_v \textbf{U}_p + \textbf{F}_D
  \]

- ENERGY EQUATION
  \[
  \frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho h \textbf{U}) = \alpha \frac{\partial p}{\partial t} + \alpha \Phi + \nabla \cdot \textbf{q} + q_v
  \]
  \[
  - \textbf{U}_R \cdot \textbf{F}_D + m_v \left( h_v + \frac{1}{2} \textbf{U}_R \cdot \textbf{U}_R \right)
  \]
CONTINUOUS PHASE ENERGY EQUATION

MIXTURE ENTHALPY

\[ h = \sum_{i=1}^{n} y_i^s h_i \]

\[ h_i = h_i^0 + \sum_{j=1}^{5} a_{ij} T^j \]

HEAT FLUX VECTOR

\[ q = \kappa \nabla T - \sum_{i=1}^{n} \rho D_i \nabla Y_i^S \]

IF TURBULENT AND LAMINAR LEWIS NUMBERS ARE ASSUMED TO BE UNITY, THIS CAN BE SIMPLIFIED TO

\[ q = -\left( \frac{\mu_T}{P_{T_2}} + \frac{\mu_T}{P_{T_T}} \right) \nabla h \]

CONTINUOUS PHASE MASS TRANSPORT ANALYSIS

\[ \frac{\partial \left( \rho Y_i^s \right)}{\partial t} + \nabla \cdot \left( \rho U Y_i^s \right) = \nabla \cdot \left( \rho D_i \nabla Y_i^s \right) + w_i + m_{v_i} \]

NOTES:

1. \( \sum_{i=1}^{n} w_i = 0 \)

2. \( m_{v_i} = 0 \) FOR \( i > 2 \) (\( i = 1 \Rightarrow \text{Al} ; i = 2 \Rightarrow \text{Al}_2\text{O}_3 \))

DEFINE

\( \alpha_{M} = \text{MASS FRACTION OF ELEMENT } k \text{ IN SPECIES} \)

\( Y_k = \sum_{i=1}^{n} \alpha_{ij} Y_i^s = \text{MASS FRACTION OF ELEMENT } k \text{ IN GAS} \)

\( \alpha_{k1} = \alpha_{k2} = 0 \) FOR \( k > 2 \) (\( k = 1 \Rightarrow \text{Al} ; k = 2 \Rightarrow \text{O} \))

ASSUME

\( D_i = D ; i = 1,2,...,n \)
CONTINUOUS PHASE MASS TRANSPORT ANALYSIS (cont'd)

\[ \frac{\partial (\rho Y_k)}{\partial t} + \nabla \cdot (\rho U Y_k) = \nabla \cdot (\rho D \nabla Y_k) + \sum_{i=1}^{\alpha} \alpha_{ki} m_v \]

BOUNDARY CONDITIONS

- PROPELLANT SURFACE: \( Y_k \) SPECIFIED BY PROPELLANT COMPOSITION
- INERT SURFACE: \( \frac{\partial Y_k}{\partial n} = 0 \)
- IF PROPELLANT COMPOSITION IS UNIFORM THEN ELEMENTAL TRANSPORT EQUATIONS FOR \( k = 3, \ldots, L \) ARE PROPORTIONAL TO EACH OTHER AND NEED NOT BE SOLVED
- IF \( m_{v1} \sim m_{v2} \) THEN ELEMENTAL TRANSPORT EQUATIONS FOR \( A1 \) AND 0 ARE PROPORTIONAL AND ONLY ONE NEEDS TO BE SOLVED.

CONTINUOUS PHASE CHEMISTRY ANALYSIS

- ELEMENTAL MASS FRACTIONS AND TWO STATE VARIABLES DEFINE THE THERMODYNAMIC STATE.
- COMPUTE SPECIES MASS FRACTIONS USING SUITABLE CURVE FITS OBTAINED FROM EQUILIBRIUM CODE.
- COMPUTE TEMPERATURE FROM MIXTURE COMPOSITION AND ENTHALPY USING POLYNOMIAL COEFFICIENTS (ITERATION REQUIRED).
- COMPUTE MIXTURE MOLECULAR WEIGHT AND SPECIFIC HEAT.
DISCRETE PHASE ANALYSIS

- COMPUTATIONAL PARTICLES USED TO REPRESENT COLLECTION OF DROPLETS CONTAINING ALUMINUM AND ALUMINUM OXIDE

- DISTRIBUTION OF PARTICLES IN THE DOMAIN COMPUTED USING LAGRANGIAN ANALYSIS IN COORDINATE SPACE

- SOURCE TERMS FOR EULERIAN ANALYSIS COMPUTED FROM THE COMBUSTION RATE, DRAG FORCE AND HEAT TRANSFER FOR THE PARTICLES

\[
F_p = m \ddot{x}_p \\
\dot{x}_p = \int_{t_0}^{t} \frac{F_p}{m} \, dt + \dot{x}_p \bigg|_{t_0}
\]

- COORDINATE TRANSFORMATION

\[
y' = y'(x_1, x_2, x_3) \\
\dot{y}_p = J \dot{x}_p \quad \text{where} \quad J = \begin{vmatrix} \frac{\partial y'}{\partial x_1} \\ \vdots \\ \frac{\partial y'}{\partial x_3} \end{vmatrix}
\]

- INTEGRATION YIELDS

\[
\Delta y_p = \frac{1}{2} \Delta t^2 J \frac{F_p}{m} + \Delta t \dot{x}_p \bigg|_{t_0}
\]
DISCRETE PHASE DROPLET COMBUSTION MODEL

- IDEALIZED ANALYSIS DIFFUSION CONTROLLED COMBUSTION OF DROPLET WITH SURFACE REACTION YIELDS

\[
\dot{m}_b = 2\pi D_p \rho D \ln(1 + \chi Y_{0,\infty})
\]

WHERE

\( \chi = \text{STOICHIOMETRIC FUEL TO OXIDIZER MASS RATIO} \)

\( Y_{0,\infty} = \text{MASS FRACTION OF OXIDIZER IN FAR FIELD} \)

- FOR MASS TRANSPORT CONTROLLED COMBUSTION OF ALUMINUM DROPLET CALCULATE BURNING RATE FROM

\[
\dot{m}_{Al} = \pi \rho_{Al} \frac{k}{n} D_p^{n-1}
\]

- REDUCES TO RESULTS OF THE IDEALIZED ANALYSIS FOR \( n=2 \) AND APPROPRIATE EXPRESSION FOR \( k \).

DISCRETE PHASE DROPLET COMBUSTION MODEL (Cont'd)

- REDUCES TO HERMSEN MODEL WITH

\[
\begin{align*}
 n & = 1.8 \\
 k & = 8.3314 \times 10^{-5} \cdot A_k^{0.9} \cdot \rho_c^{0.27} \cdot R_k \\
 R_k & = 2.7 \\
 A_k & = 100\sum X_i \quad ; \quad i = CO_2, H_2O, O_2, OH, O
\end{align*}
\]

- CAN ACCOUNT FOR EFFECTS OF FORCED CONVECTION, DROPLET-DROPLET INTERACTION, VARIATION OF OXIDIZER CONCENTRATION ALONG TRAJECTORY ETC. ON BURNING RATE WITH REDUCED EMPIRICISM.

FOR EXAMPLE

\[
\begin{align*}
 k &= 1 + 0.24 \cdot R_\theta^{1/2} \cdot Sc^{1/3} \\
 k &= 1 \left( \sum_{i=1}^{n} X_i Y_i^p \right)
\end{align*}
\]

IN GENERAL

\[
k = k(D, Sh, \alpha, X_i, Y_i^p)
\]
APPLICATION CASE STUDY

GEOMETRY

PROPELLANT DATA

COMPOSITION

<table>
<thead>
<tr>
<th>Component</th>
<th>Percentage</th>
</tr>
</thead>
<tbody>
<tr>
<td>AP</td>
<td>71.0%</td>
</tr>
<tr>
<td>BINDER (HTPB)</td>
<td>14.0%</td>
</tr>
<tr>
<td>AI</td>
<td>15.0%</td>
</tr>
</tbody>
</table>

DENSITY

1794.6 kg/m³ (112.0 lbm/ft³)

BURN RATE

9.0678 x 10⁻³ m/s (0.357 in/s)

APPLICATION CASE STUDY (Continued)

CASE I - SURFACE COMBUSTION SIMULATION

• ALL ALUMINUM ASSUMED TO BURN AT PROPELLANT SURFACE
  • FLAME TEMPERATURE = 3435 K

• 20% Al₂O₃ ASSUMED TO BE IN CAPS
  • GAS PHASE ELEMENTAL ALUMINUM MASS FRACTION
    GIVEN BY \((0.8 \times 0.15) / (1 - 0.2 \times 0.15 \times 10² / 54) = 0.1272\)

CASE II - DISTRIBUTED COMBUSTION SIMULATION

• ALL ALUMINUM INJECTED AS DROPLETS IN DISCRETE PHASE
  • FLAME TEMPERATURE = 2592.5 K

• LOG-NORMAL SIZE DISTRIBUTION WITH MEAN DIAMETER = 150μm
  AND \(\log_{10}\) STANDARD DEVIATION = 0.2

• PARTICLES ASSUMED TO RETAIN 20% OF REACTED ALUMINUM
  IN FORM OF Al₂O₃ CAPS
COMPUTATIONAL DOMAIN AND GRID

PRESSURE CONTOURS
($P_{\text{max}} = 6.6 \text{ MPa}, P_{\text{min}} = 0.4 \text{ MPa}, \Delta P = 0.2 \text{ MPa}$)
TEMPERATURE CONTOURS
\(T_{\text{max}} = 3750K, T_{\text{min}} = 2350K, \Delta T = 100 \text{ K}\)

DENSITY CONTOURS
\(\rho_{\text{max}} = 7.7 \text{kg/m}^3, \rho_{\text{min}} = 0.3 \text{kg/m}^3, \Delta \rho = 0.2 \text{kg/m}^3\)
SUMMARY

- A TWO-PHASE DISTRIBUTED COMBUSTION MODEL DEVELOPED TO SIMULATE COMBUSTION OF METALIZED SOLID PROPELLANTS
- CALCULATED RESULTS SHOW EXISTENCE OF AN EXTENDED COMBUSTION REGION IN THE MOTOR CHAMBER
  - SIGNIFICANT SPATIAL VARIATION IN TEMPERATURE, COMPOSITION AND DENSITY IN THE CHAMBER
- EXPERIMENTAL DATA NEEDED FOR INITIAL PARTICLE SIZE DISTRIBUTION AND FRACTION OF METAL THAT BURNS AT SURFACE FOR FURTHER CODE VALIDATION
- CODE CAN BE EFFECTIVELY USED IN PARAMETRIC STUDIES AT PRESENT
- PRESENT APPROACH CAN BE READILY MODIFIED TO STUDY EFFECTS SUCH AS RADIATION AND PARTICLE SIZE CHANGES DUE TO BREAKUP AND COALESCENCE
Conference publication includes 59 abstracts and presentations and three invited presentations given at the Tenth Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion held at George C. Marshall Space Flight Center, April 28–30, 1992. The purpose of the workshop is to discuss experimental and computational fluid dynamic activities in rocket propulsion. The workshop is an open meeting for government, industry, and academia. A broad number of topics are discussed including computational fluid dynamic methodology, liquid and solid rocket propulsion, turbomachinery, combustion, heat transfer, and grid generation.