2ND NASA CFD VALIDATION WORKSHOP

NASA

(NASA-TM-107972) 2ND NASA CFD
VALIDATION WORKSHOP (NASA) 320 p

N93-70575
--THRU--
N93-70583
Unclass

29/34 0116670

NASA Lewis Research Center
July 10-12, 1990
2ND NASA CFD VALIDATION WORKSHOP

NASA LEWIS RESEARCH CENTER
CLEVELAND, OHIO 44135

JULY 10-12, 1990

RAYMOND E. GAUGLER, CHAIRMAN
# TABLE OF CONTENTS

I. Workshop Summary

II. Pre-Workshop Activities
1. Invitation letter
2. Conclusions and recommendations of the Advisory Committee, May 1987
3. Summary of the 1987 NASA CFD Validation Workshop
4. List of attendees

III. Workshop Program

IV. Workshop Presentations
   Workshop Goals and Agenda ............... Ray Gaugler 16
   LaRC Summary ......................... Scott Kjelgaard 20
   ARC Summary .......................... Joe Marvin 45
   MSFC Summary ......................... Paul McConnaughey 70
   LeRC Summary .......................... Ray Gaugler 100
   ARC Technical presentations .......... William Lockman 126
                                      Paul McConnaughey 162
   LaRC Technical presentations ........ Scott Kjelgaard 182
   LeRC Technical presentations ........ Warren Hingst 197
                                      Michael Hathaway 232
   MSFC Technical presentation ........ Lisa Griffin 257

V. Small Working Group Reports
1. Subsonic
2. High-Speed
3. Hypersonic
1990 Workshop Summary

The 2nd NASA CFD Validation Workshop was held at Lewis Research Center on July 10-12, 1990.

The purpose of the workshop was to review NASA's progress in CFD validation since the first workshop (held at Ames in 1987) and to affirm the future direction of the NASA CFD validation program.

Forty-six people participated in the workshop. Of these, 25 were from NASA, 2 were from other government agencies, 13 were from industry, and 6 were from universities. A list of attendees is included in this report.

The first session, held on the morning of the first day, consisted of overviews of CFD validation research at each of the three OAET research centers and at Marshall Space Flight Center. The second session (afternoon of the first day) consisted of in-depth technical presentations of the best examples of CFD validation work at each center (including Marshall).

On the second day the workshop divided into three working groups to discuss CFD validation progress and needs in the subsonic, high-speed, and hypersonic speed ranges. The emphasis of the working groups was on propulsion. The subsonic, high-speed, and hypersonic working groups were led by Gordon Pickett (Pratt and Whitney), Joe Marvin (Ames), and Lou Povinelli (Lewis), respectively. At the end of the second day, each group leader reported to the workshop on his group’s findings and recommendations. The rosters of the working groups, and copies of the charts used in reporting out are included in this report.

It was apparent from the presentations that NASA's CFD Validation program has significantly changed the way experimental fluid dynamics research at NASA is performed - even research that is not funded out of the CFD Validation Program. Compared to before the validation program began, researchers today are finding it easier to obtain funding for instrumentation and tunnel time required for code validation. As a result, more experiments are producing data that is useful for code validation. The synergism between experiment and computation is producing better experiments and better computational methods.

**CFD Validation Coordinating Board Recommendations**

On the third and final day the NASA participants met to discuss the future of the NASA CFD validation program in light of what we had learned in the first two days. The Board, with input from other NASA participants, made the following general recommendations:

1. NASA has made significant progress in implementing the recommendations of the Ad Hoc Committee on CFD validation of the Aeronautics Advisory Committee and should continue to abide by them.
2. NASA has made significant progress in implementing the recommendations of the first NASA CFD Validation Workshop and should continue to follow them.

3. Changes in the work breakdown structure for FY91 and beyond must not adversely affect funding for CFD validation. In particular, since with the new WBS CFD validation will no longer have a funding stream separate from that for CFD Methods, NASA Headquarters and the research centers must ensure that the CFD validation programs is adequately funded.

4. NASA must maintain a highly visible CFD validation program to enhance advocacy for obtaining funding from other sources.

5. NASA should encourage applied programs to incorporate into their test plans methodology that allows for some level of code validation.

6. NASA should advocate to program offices that they provide advanced instrumentation to quantify flow environments.

7. NASA should encourage the continued participation of Marshall Space Flight Center in the validation program. The Board commends their synergistic role in working with the research centers.

8. The Coordinating Board should organize a third NASA CFD Validation workshop to be held at Langley Research Center in approximately three years.

The Board identified the following important propulsion validation needs that are not being adequately addressed:

1. supersonic mixing and combustion (2<M<6)
   a. lower speed: instrumentation is lacking
   b. higher speed: facilities are lacking

2. subsonic combustion

3. transonic afterbody propulsion integration

4. secondary flow systems in propulsion systems

5. supersonic/subsonic jet mixing for HSCT

6. swept-wing aircraft icing

The Board recognized that, since not all cognizant people were in attendance, this list may not be comprehensive.
Sample Invitation Letter
Dear:

I would like to invite you or your representative to participate in the Second NASA CFD Validation Workshop, to be hosted by the Lewis Research Center, July 10-11, 1990.

In 1986, NASA's Aeronautics Advisory Committee (AAC) formed an Ad Hoc Committee to review the CFD validation activities at the three NASA Research Centers. The Ad Hoc Committee was charged to: 1) classify CFD research to identify and characterize verification needs and, 2) assess ongoing and planned CFD verification experiments. The ad hoc committee conducted reviews at each of the three NASA research centers and reported their recommendations back to the AAC. In response to those recommendations, NASA conducted the First NASA CFD Validation Workshop, held in the summer of 1987 at the Ames Research Center, as part of the process to insure that the focus and future direction of the validation program was consistent with the anticipated needs of the aerospace community.

The time has now come for a follow-up second workshop. The purpose of this workshop will be to summarize progress in NASA CFD validation experiments, instrumentation, and facilities since the first workshop, and to affirm the future direction of NASA's CFD Validation Program.

It is planned that the first day of the workshop will consist of presentations summarizing the progress in CFD validation at the three research centers and at the Marshall Space Flight Center. The second day will consist of small working group meetings, focusing on specific topics. Since we will be meeting this year at the Lewis Research Center, the emphasis of the meeting will be on propulsion, focusing on the three principle speed ranges - hypersonic, high-speed, and subsonic.

Your active participation in this workshop will be of great value to NASA in insuring that the CFD Validation program continues to make progress in the areas of most critical need.

Please return the enclosed registration form as soon as possible. A detailed program, information on hotels and direction to NASA Lewis will be sent to registrants at a later date.

If you have any questions, please contact the Workshop Chairman, Dr. Raymond E. Gaugler, at (216) 433-5882.

Sincerely,

Neal T. Saunders
Director of Aeronautics
Conclusions and recommendations of the
Advisory Committee, May 1987
COMPUTATIONAL FLUID DYNAMICS VALIDATION

Prepared for

NASA'S AERONAUTICS ADVISORY COMMITTEE

by

THE AD HOC COMMITTEE ON CFD VALIDATION
OF THE AERONAUTICS ADVISORY COMMITTEE

May 1, 1987
CONCLUSIONS

Many excellent experimental programs are being conducted at the NASA research centers. The programs are generally attacking pertinent problems. However, only a few of the projects reviewed were considered by the committee to be CFD-validation projects. Further, adequate coordination among NASA centers was not evidenced in the reviews.

CFD validation is a relatively new concept that requires clearly defined and disciplined coupling between CFD and experiments. The committee has proposed the following definition of CFD validation, which emphasizes the computational and experimental discipline required and distinguishes validation from other experimentation related to CFD:

**CFD Code Validation** - Detailed surface- and flow-field comparisons with experimental data to verify the code's ability to accurately model the critical physics of the flow. Validation can occur only when the accuracy and limitations of the experimental data are known and thoroughly understood and when the accuracy and limitations of the code's numerical algorithms, grid-density effects, and physical basis are equally known and understood over a range of specified parameters.

The committee recognizes that four categories of experimentation are required for developing CFD capability:

A. Experiments designed to understand flow physics.
B. Experiments designed to develop physical models for CFD codes.
C. Experiments designed to calibrate CFD codes.
D. Experiments designed to validate CFD codes.

All four categories of tests are important and are necessary to build a mature CFD capability. Validation tests should be only a part of total test focus of NASA and should be formulated to provide specific data for validating CFD codes.

CFD validation is severely hampered in some areas by the lack of critical measurements under realistic conditions—especially at high Mach numbers. Measurements must be taken with adequate accuracy and resolution, and with redundant instruments, in order to evaluate CFD's capability to predict details and trends as well as to explore boundaries of application for specific codes. Specific instrumentation and facilities need to be developed when the state of the art is inadequate.

Equally important for validation is the mapping of the CFD code's sensitivity to numerical algorithms, grid density, and physical models. These effects must be known to the same degree as the experimental accuracy and resolution in order to understand the applicability of the code over a range of flow parameters.
RECOMMENDATIONS

It is recommended that

1. NASA classify its CFD related experiments into the above-defined categories A, B, C, and D. The classification provides a framework for future evaluation and allocation of resources.

2. NASA adopt a sharply defined program plan for CFD validation, consistent with the definitions proposed by the Committee in this report. In particular,

   o Projects falling under this program should be closely coordinated and adequately funded.

   o The limitations of the facilities, instrumentation, proposed models, and codes should be clearly specified at the onset of the projects, as should the critical data set required and the range of validity of the modeling assumptions.

   o The questions of such phenomenon as unsteadiness, three-dimensionality, transition, turbulence models, and the applicability of inviscid/interacting-viscous/Navier-Stokes models should influence the choices of experiments, code development, and application.

   o Duplication of experiments and code development should involve several centers.

   o A validated code should become a NASA or general code, not merely the code of a given center.

3. NASA pay careful attention to the instrumentation and facilities required to provide data of adequate accuracy and in the form and detail required for CFD validation. Close coordination must be maintained between centers.

4. NASA establish a Coordinating Board for CFD validation, consisting of key project leaders from each of the three NASA centers and chaired by NASA Headquarters. This board should have responsibility for defining, coordinating, and focusing the CFD validation effort.
Summary of the 1987 NASA CFD Validation Workshop
WORKSHOP SUMMARY

The first NASA CFD Validation workshop was held at the Ames Research Center on July 14-16, 1987 with 108 persons in attendance. After being welcomed by the Director of the Office of Aerophysics at the Ames Research Center, NASA Headquarters management presented introductory comments and expectations for the meeting. Recommendations from a recent review of CFD validation activities at three NASA Research Centers, conducted by the Ad Hoc Committee on CFD Validation of the Aeronautics Advisory Committee (AAC), were presented. Definitions developed by the Committee for code verification, code calibration and corresponding experiments were also introduced and discussed. NASA will use these definitions in conjunction with the CFD Validation program as a tool to manage and maintain a balanced program. For completeness, a copy of the AAC Ad Hoc Committee report is included herein.

The Ames, Langley, and Lewis Research Centers presented overviews of their CFD Validation activities with brief descriptions of their key experiments. These presentations covered the entire speed range and included external and internal aerodynamic flows, combustion, aerothermal loads, and aeroelasticity. Following the NASA presentations, other government installations, industry, and universities presented overviews of their CFD Validation activities and needs. The presentations and speakers are listed in the Workshop Program which is included herein along with copies of the presentations.

After these formal presentations, the workshop attendees were divided into six working groups. The working groups focused on the following key areas: Low Speed Aerodynamic Flows; High Speed Aerodynamic Flows-Commercial; High Speed Aerodynamic Flows-Military; Internal Flows; Hypersonics-Chemically Reacting External/Internal Flows; and Hypersonics-External Aerothermodynamics. Each working group was given a list of five issues to address in their discussions. The issues were: (1) identify key efforts (numerical and experimental) required to meet immediate modeling and validation needs; (2) identify near-term and far-term critical problem areas that require new or additional modeling and validation activities; (3) identify computational, experimental, facility and instrumentation, or other capabilities required to investigate these critical problem areas; (4) identify key modeling and validation projects that are potential cooperative/joint ventures; and (5) define, identify and/or propose standardized test cases for modeling and validation.

The working groups met for about 6 hours to discuss these issues and prepare summaries.

The formal workshop resumed with the various spokespersons presenting the results from their group discussions. These presentations have also been included herein. Several common recommendations were made. These, not in priority order, included: (1) provide closer cooperation between CFD developers and experimentalists at the outset of verification projects with a lasting commitment from both to see the projects through to completion; (2) provide detailed measurements of the flow field and boundary conditions in addition to model surface measurements and integral quantities; (3) provide improved or new non-intrusive measurement capabilities, especially for...
hypersonic or reacting flow conditions; (4) provide redundancy in both measurements and experiments whenever practical so as to clarify data accuracy and credibility; (5) provide dedicated large facilities for validation research and increase flight-based research activities; and (6) provide standardized test cases with accessible electronic data bases. The recommendation for standardized test cases received a strong consensus during the workshop. NASA will review the recommendations from the groups and determine a course of action to advocate their implementation.

The workshop concluded with the general feeling that the meeting was very successful. Attendees from the various research centers, universities and industry expressed a renewed understanding of each other's viewpoints on CFD validation. Experts in computational and experimental fluid dynamics were able to begin the synergistic interaction critical to successful validation activities in their one-on-one discussions. Many felt that this workshop should be the forerunner of future workshops on specific topic areas because the workshop atmosphere provides a forum for discussing both successes and failures which taken together often lead to more expeditious problem solutions.
2nd NASA CFD Validation Workshop
NASA Lewis Research Center
July 10-12, 1990

ATTENDEES

Dr. John Adamczyk
National Aeronautics & Space Administration
Lewis Research Center
21000 Brookpark Rd., M.S. 5-9
Cleveland, OH 44135

Mr. Kyung Ahn
National Aeronautics & Space Administration
Lewis Research Center
21000 Brookpark Rd., M.S. 301-2
Cleveland, OH 44135

Mr. Essam Atta
General Electric Aircraft Engines
1 Neumann Way MD/A323
Cincinnati, OH 45215

Dr. I. L. Bhaeley
General Dynamics, Fort Worth Division
P.O. Box 748, Mail Zone 2870
Fort Worth, TX 76101

Dr. William B. Compton
National Aeronautics & Space Administration
Langley Research Center
M.S. 280
Hampton, VA 23665-5225

Dr. Michael E. Crawford
University of Texas
Mechanical Engineering Dept.
Austin, TX 78712

Dr. Don Dietrich
General Electric Aircraft Engines
1 Neumann Way M.D. A-317
Cincinnati, OH 45215

Dr. J. Philip Drummond
National Aeronautics & Space Administration
Langley Research Center
M.S. 156
Hampton, VA 23665-5225

Mr. Kevin Early
General Electric Aircraft Engines
1 Neumann Way M.D. A-317
Cincinnati, OH 45215

Dr. Alan H. Epstein
Massachusetts Institute of Technology
Gas Turbine Laboratory
Dept. of A. & A., 31-266
Cambridge, MA 02139

Dr. Raymond E. Gaugler
National Aeronautics & Space Administration
Lewis Research Center
M.S. 5-11
Cleveland, OH 44135

Dr. Steve Gegg
Allison Gas Turbine Division of GMC
P.O. Box 420 (S-46)
Indianapolis, IN 46206

Professor F. B. Gessner
Department of Mechanical Engineering
University of Washington
M/C FU-10
Seattle, WA 98105

Ms. Lisa W. Griffin
National Aeronautics & Space Administration
George C. Marshall Space Flight Center
M.S. ED32
Marshall Space Flight Center, AL 35812

Dr. Michael Hathaway
National Aeronautics & Space Administration
Lewis Research Center
21000 Brookpark Rd., M.S. 5-11
Cleveland, OH 44135
Dr. Warren Hingst  
National Aeronautics & Space Administration  
Lewis Research Center  
21000 Brookpark Rd., M.S. 5-11  
Cleveland, OH 44135

Mr. Michael Holden  
CALSPAN Corp.  
4455 Genesee Street  
Buffalo, NY 14221

Dr. Mounir B. Ibrahim  
Cleveland State University  
Mechanical Engineering Dept.  
Cleveland, OH 44115

Mr. Anthony M. Ingraldi  
National Aeronautics & Space Administration  
Langley Research Center  
M.S. 280  
Hampton, VA 23665-5225

Mr. Scott O. Kjelgaard  
National Aeronautics & Space Administration  
Langley Research Center  
M.S. 347  
Hampton, VA 23665-5225

Mr. William K. Lockman  
National Aeronautics & Space Administration  
Ames Research Center  
M.S. 230-2  
Moffett Field, CA 94035-1000

Mr. John B. Malone  
National Aeronautics & Space Administration  
Langley Research Center  
M.S. 173  
Hampton, VA 23665-5225

Mr. Joseph G. Marvin  
National Aeronautics & Space Administration  
Ames Research Center  
M.S. 229-1  
Moffett Field, CA 94035

Dr. Paul McConnaughey  
National Aeronautics & Space Administration  
George C. Marshall Space Flight Center  
M.S. ED32  
Marshall Space Flight Center, AL 35812

Dr. Edward Mularz  
National Aeronautics & Space Administration  
Lewis Research Center  
21000 Brookpark Rd., M.S. 5-11  
Cleveland, OH 44135

Dr. Gordon F. Pickett  
Pratt & Whitney Aircraft Group  
400 Main St. M.S. 163-01  
East Hartford, CT 06108

Dr. Mark G. Potapczuk  
National Aeronautics & Space Administration  
Lewis Research Center  
M.S. 77-10  
Cleveland, OH 44135

Dr. Louis A. Povinelli  
National Aeronautics & Space Administration  
Lewis Research Center  
21000 Brookpark Rd., M.S. 5-11  
Cleveland, OH 44135

Dr. Lonnie Reid  
National Aeronautics & Space Administration  
Lewis Research Center  
21000 Brookpark Rd., M.S. 5-11  
Cleveland, OH 44135
# 2nd NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER
ADMINISTRATION BUILDING AUDITORIUM
JULY 10-12, 1990

## PROGRAM

**Tues., July 10, 1990**

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
<th>Presenter(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:30</td>
<td>Registration, Administration Building Auditorium</td>
<td></td>
</tr>
<tr>
<td>9:00</td>
<td>Welcome</td>
<td>Lonnie Reid</td>
</tr>
<tr>
<td>9:10</td>
<td>Workshop Goals and Agenda</td>
<td>Ray Gaugler</td>
</tr>
<tr>
<td>9:20</td>
<td>Overview of NASA CFD Validation Program</td>
<td>Ed Schairer</td>
</tr>
<tr>
<td>9:45</td>
<td>LaRC Summary</td>
<td>Scott Kjelgaard</td>
</tr>
<tr>
<td>10:15</td>
<td>ARC Summary</td>
<td>Joe Marvin</td>
</tr>
<tr>
<td>10:45</td>
<td>Break</td>
<td></td>
</tr>
<tr>
<td>11:00</td>
<td>MSFC Summary</td>
<td>Paul McConnaughey</td>
</tr>
<tr>
<td>11:45</td>
<td>LeRC Summary</td>
<td>Ray Gaugler</td>
</tr>
<tr>
<td>12:15</td>
<td>Lunch - On your own, Lewis Cafeteria</td>
<td></td>
</tr>
<tr>
<td>1:15</td>
<td>ARC Technical presentations</td>
<td>William Lockman</td>
</tr>
<tr>
<td>2:15</td>
<td>LaRC Technical presentations</td>
<td>Paul McConnaughey</td>
</tr>
<tr>
<td>3:15</td>
<td>Break</td>
<td>Scott Kjelgaard</td>
</tr>
<tr>
<td>3:30</td>
<td>LeRC Technical presentations</td>
<td>Warren Hingst</td>
</tr>
<tr>
<td>4:30</td>
<td>MSFC Technical presentation</td>
<td>Michael Hathaway</td>
</tr>
<tr>
<td>5:00</td>
<td>Social Hour, Hors D'Oeuvres &amp; Cash Bar, Ad. Building Foyer</td>
<td>Lisa Griffin</td>
</tr>
</tbody>
</table>

**Wed., July 11, 1990**

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
<th>Presenter(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:00</td>
<td>Instructions to working groups</td>
<td>Ray Gaugler</td>
</tr>
<tr>
<td>8:15</td>
<td>Meeting of working groups</td>
<td></td>
</tr>
<tr>
<td>12:15</td>
<td>Lunch - On your own, Lewis Cafeteria</td>
<td></td>
</tr>
<tr>
<td>1:15</td>
<td>Meeting of working groups</td>
<td></td>
</tr>
<tr>
<td>3:30</td>
<td>Working groups report</td>
<td></td>
</tr>
<tr>
<td>4:30</td>
<td>Adjourn</td>
<td></td>
</tr>
</tbody>
</table>

**Thurs., July 12, 1990**

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>8:00</td>
<td>Government caucus</td>
</tr>
<tr>
<td>12:00</td>
<td>Adjourn</td>
</tr>
</tbody>
</table>
2nd NASA CFD VALIDATION WORKSHOP

2ND NASA CFD VALIDATION WORKSHOP

JULY 10-12, 1990
GOALS

• SUMMARIZE PROGRESS SINCE '87 WORKSHOP

• AFFIRM FUTURE DIRECTION OF NASA'S CFD VALIDATION PROGRAM
Langley Research Center
CFD Code Validation
Program Overview

Scott O. Kjelgaard
Experimental Methods Branch
Fluid Mechanics Division

Second NASA CFD Validation Workshop
NASA - Lewis Research Center
July 10-12, 1990
Outline

- LaRC Approach to CFD Code Validation
- Experimental/CFD Perceptions
- CFD Code Validation Program Experiment Overview
- Experiment Highlights
- Concluding Remarks
CFD Code Validation

Objective: Conduct detailed benchmark experiments yielding high-quality archival data bases. Use these data bases to validate CFD codes and develop empirical models of complex fluid dynamic phenomena.

Approach:

Conduct cooperatively-designed experiments which provide:

- detailed flowfield and surface flow measurements required by code developers
- assessment of measurement errors
- redundant measurements with more than one instrument, in more than one facility
- documented results with archival storage of data for easy accessibility

Acquire dedicated, advanced instrumentation systems for the primary code validation facilities

Develop new instrumentation techniques when required
THE CFD'ERS VIEW
OF EXPERIMENTALISTS?

EARLY RIDE-QUALITY EXPERIMENTS
THE EXPERIMENTALIST'S VIEW?
EXPERIMENTALISTS
ATTEMPT FIRST MILKSHAKE
EXPERIMENTAL TECHNIQUES CLASS FOR CFD'ERS
CFD Code Validation Experiments

LaRC Code Validation Program

- 65° Swept Delta Wing
- 76° Swept Delta Wing
- Vertical Tail/Vortex Interaction Investigation
- Rearward Facing Step Experiment
- Particle Imaging Velocimeter (PIV)
- Doppler Global Velocimeter (DGV)
- Holocinematographic Velocimeter (HCV)
- LTPT Laser Velocimeter

Other Programs

- F-18 Mean Flowfield Measurements
- Transonic After-Body Experiment
- Supersonic Coaxial Jet
76° Delta Wing Experiment

Objective:
Develop a detailed data base for the flow over sharped edged delta wings with a leading-edge sweep of 76°. Use three component LV to obtain flowfield measurements in the burst vortex over the delta wing.

Approach:
Conduct experiments in various facilities documenting the force and moments, surface pressures and flowfield velocities throughout the angle-of-attack range.

Data acquired:
- Forces, Moments
- Static and Dynamic Surface Pressures
- Three component LV, 5-Hole Probe

Facility
- BART, 7x10, Vigyan
- BART
- BART
76° Delta Wing Experiments

Wind Tunnel Comparison

- Hummel, OTS, Re = 2.0e6
- BART, CTS, Re = 1.5e6
- OTS, Re = 1.5e6
- CFL3D, Re = 0.95e6

C_L

α
Vertical Tail/Vortex Interaction Investigation

Objective:

- Provide a fundamental understanding of the vortex-fin interaction process
- Determine the capability of CFD methods to predict the physics of the flowfield
- Provide a data base for developing improved tail buffet criteria
- Develop experimental measurement techniques
Vertical Tail/Vortex Interaction Investigation

**Approach:**

- Cooperative MCAIR-Langley MOA in place to study vortex flows and vortex-fin interactions
- Multi-disciplinary program involving structural dynamics and aerodynamics groups at MCAIR and Langley
- Experiments to be conducted using a simple delta wing models with twin vertical tails
Vertical Tail/Vortex Interaction

Experimental Set-up:
- 76° delta wing using Hummel planform
- 3 component LV measurements to determine the aerodynamic input forcing function to the vertical tails
- Unsteady surface pressure measurements on the rigid tail to be correlated with the velocity fluctuations in the vortex
- Dynamic response from the flexible tail

Measurement Techniques
- Unsteady Surface Pressures
- Tail Bending and Torsion
- Laser Light Sheet Flow Visualization
- 3 component Laser Velocimeter
65° Delta Wing Experiment

Objective:
Develop a detailed data base for the flow over rounded-edge delta wings with a leading-edge sweep of 65°. Measurements will be obtained throughout the Mach and Reynolds number range in LTPT, NTF, and 8' TPT at LaRC.

Low Turbulence Pressure Tunnel
- Static surface pressures
- Flow visualization
- 3 component LV

8-foot Transonic Pressure Tunnel
- Static surface pressures
- Flow visualization
- 3 component LV

National Transonic Facility
- Static surface pressures
- Forces, moments, buffet
- Vortex-core condensation

Model test envelopes

\[ R_e \]
Geometric Features of NTF Delta Wing

\[ \Lambda_{le} = 65^\circ, \, c_r \approx 2.14 \text{ ft}, \, b = 2.0 \text{ ft} \]

Interchangeable leading Edges
- \([r_{le}/c] s = 0.05, 0.15, 0.30\%
- 8\mu surface finish

Sting
- Symmetric
- \( d/b = 0.14 \)

Flat plate center wing
- \( t/c_r = 0.034 \)
- 8\mu surface finish
- Sharp trailing edge

Pressures:
- 183 orifices / configuration
- \( x/c_r = 0.2, 0.4, 0.6, 0.8, 0.95 \)
- every 10\% \( x/c_r \) along l.e.
- 0.010 inch diameter
Particle Imaging Velocimeter (PIV)

Objective:
Develop non-intrusive measurement techniques to obtain the data required for CFD code validation.

Approach:
Particle trajectories are tracked by double pulsing a laser light sheet and recording the images. Particles in the sheet form a double-spot pattern on the image when illuminated by the laser beam. The spacing and orientation of the spot-pairs provide an instantaneous measurement of the velocity in a two-dimensional plane in the flowfield.
Photograph of the Particle Image Velocimetry System
Photograph of the Particle Image Velocimetry Data Analysis
75° Delta Wing Investigation, $\alpha = 20.5^\circ$

Flow Visualization
- Surface using TiO2
- Off-body using scanning laser light sheet

Flowfield Surveys
- **Pitot pressure**
  \[ x/L = 0.3, 0.5, 0.7, 0.9, 1.1, \text{ Re } = 0.5, 1.0, 1.5 \text{ million} \]
  Each survey plane contains \( \approx 2800 \) data locations

- **Velocity (5-Hole Probe)**
  \[ x/L = 0.7, 0.9, 1.1; \text{ Re } = 0.5, 1.0, 1.5 \text{ million} \]
  Each survey plane contains \( \approx 3400 \) data locations

- **Velocity (LV)**
  \[ x/L = 0.9; \text{ Re } = 1.0 \text{ million} \]
  Data obtained to evaluate measurement errors of 5-hole probe
75° SWEPT DELTA WING
\[ \alpha = 20.5^\circ; \; R_e = 500,000 \]
COMPUTATIONAL GRID REFINEMENTS
75° Delta Wing, x/L = 0.7

65 X 65 (4225 Total) 4 Patches (7222 Total)

Single Dense Grid

Embedded Grid
STREAMWISE VELOCITY CONTOURS

$Re_L = 500,000; \alpha = 20.5^\circ; x/L = 0.7$

Experiment 2 Level Refinement

D88-1 & D88-4 Combined
Concluding Remarks

LaRC is dedicated to the development of a strong CFD Code Validation Program

- Ongoing experimental program documenting various fluid dynamic phenomena
- Experiments designed with experimental/computational cooperation
  - assessment of measurement errors
  - redundant measurements with more than one instrument, in more than one facility
  - documented results with archival storage of data for easy accessibility
- Acquisition of the required instrumentation for CFD code validation facilities
- Development of new instrumentation systems when required
OVERVIEW OF CFD FLOW MODELING AND VALIDATION
AMES RESEARCH CENTER

J. G. Marvin

Second NASA CFD Validation Workshop
Lewis Research Center
July 10-12, 1990
OUTLINE

Background
Experiments
Highlights
Instrumentation
Experiments
Appendix
Summaries And Key Results of Experiments
# STAGES OF CODE DEVELOPMENT AND CORRESPONDING EXPERIMENTS RELATED TO PAYOFF IN AERONAUTICAL APPLICATIONS

<table>
<thead>
<tr>
<th>Science discipline</th>
<th>Enabling technologies</th>
<th>Research codes</th>
<th>Pilot codes</th>
<th>Production codes</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>• Algorithms&lt;br&gt;• Grid generation&lt;br&gt;• Computer Power</td>
<td>• Technology integration&lt;br&gt;• Limited pioneering applications</td>
<td>• Wide range of applications</td>
<td>• Applied in design environment&lt;br&gt;• Cost effective</td>
</tr>
<tr>
<td>EFD</td>
<td>• Facilities&lt;br&gt;• Instrumentation&lt;br&gt;• Data acquisition</td>
<td>Building block experiments</td>
<td>Benchmark experiments</td>
<td>Design experiments</td>
</tr>
<tr>
<td></td>
<td></td>
<td>• Flow physics including numerical simulations&lt;br&gt;• Flow modeling</td>
<td>• Calibration&lt;br&gt;• Validation</td>
<td>• Configuration&lt;br&gt;• Performance&lt;br&gt;• System Integration</td>
</tr>
</tbody>
</table>
KEY PROGRAM ELEMENTS FOR FLOW MODELING AND CFD VALIDATION
AMES RESEARCH CENTER

Focus On 3-D Reynolds Averaged Navier Stokes Code Validation

Focus on Important CFD Applications

Involve CFD'ers, Modelers And Experimentalists

Design Experiments To Provide The Specific Data Necessary To Guide And Verify CFD Computations
FACILITY AND INSTRUMENTATION ENHANCEMENTS
1986-PRESENT

FACILITIES REACTIVATED FOR HYPERSONICS

3.5' Hypersonic Wind Tunnel
Ballistic Range
Combustion Driven Shock Tunnel
Electric Arc Shock Tube

INSTRUMENTATION ADVANCES/DEVELOPMENTS

New Near-Wall measurement capability using LDV
New LDV Systems for Hi Re Channel And 3.5' HWT
Pressure Sensitive Chemical Luminescence Paints
LIF For Temperature And density Measurements in Wind Tunnels
Holographic System For Ballistic Range Flow Field Density Measurements
LIF For Combustion Driven Shock Tunnel-Gas Species, Temperatures
# Flow Modeling and CFD Validation Experiments 1990

## Building Block Experiments

<table>
<thead>
<tr>
<th>EXPERIMENT</th>
<th>CONTACT</th>
<th>ISSUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>2) Rearward Step</td>
<td>Jovic</td>
<td>Flow Physics</td>
</tr>
<tr>
<td>3) 3-d Spin Flow</td>
<td>Driver</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>4) 3-d Wedge Flow</td>
<td>Johnson</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>5) Wing Tip Vortex</td>
<td>Zilliac</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>6) 3-d Supersonic Shock Interaction</td>
<td>Horstman</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>7) Supersonic Shock Vortex Interaction</td>
<td>Settles (Penn St.)</td>
<td>Flow Model</td>
</tr>
<tr>
<td>8) SSME Turn-Around Duct*</td>
<td>Monson</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>9) Coanda</td>
<td>Brown</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>10) Hypersonic Shock Interaction</td>
<td>Horstman</td>
<td>Turb. Model</td>
</tr>
<tr>
<td>11) Shock Tube</td>
<td>Sharma</td>
<td>Reaction Rates</td>
</tr>
</tbody>
</table>

## Benchmark Experiments

<table>
<thead>
<tr>
<th>EXPERIMENT</th>
<th>CONTACT</th>
<th>CODE</th>
</tr>
</thead>
<tbody>
<tr>
<td>12) Low Aspect Ratio Wing</td>
<td>Olsen</td>
<td>TNS</td>
</tr>
<tr>
<td>13) Airfoil w/wo Passive Shock Control</td>
<td>Mateer</td>
<td>TURF</td>
</tr>
<tr>
<td>14) Generic All Body*</td>
<td>Lockman</td>
<td>UPNS</td>
</tr>
<tr>
<td>15) Ballistic Range-Cone Real Gas</td>
<td>Strawa</td>
<td>TUFF/STUFF</td>
</tr>
<tr>
<td>16) NASP Nozzle</td>
<td>Lockman</td>
<td>F3D</td>
</tr>
<tr>
<td>17) STOVL</td>
<td>Van Dalsem</td>
<td>F3D</td>
</tr>
<tr>
<td>18) Transonic WT Wall Boundary Conditions</td>
<td>Roberts (Stanford)</td>
<td>TNS</td>
</tr>
</tbody>
</table>

* Chosen For Technical Presentation
KEY ACCOMPLISHMENTS

Instrumentation

Experiments
LASER DOPPLER VELOCIMETER FOR 3-D NEAR-WALL MEASUREMENTS

Run
○ 511, 4/90
△ 491, 4/90
▽ 471, 4/90
▽ 121, 3/90
△ 181, 4/90

— Spalart
— Karlsson and Johansson
— Barlow and Johnston

Laser beams from optics table
Traverse motion

To photodetector

Tunnel wall
Window
LDV FEASIBILITY STUDY IN A HYPERSONIC SHOCK-WAVE/TURBULENT-BOUNDARY LAYER INTERACTION

$M_a = 7$, $Re_{X_a} = 18 \times 10^6$
3.5 FOOT HWT LDV SYSTEM

COMPLERE INC.
LASER-INDUCED FLUORESCENCE (LIF)
MEASURES TEMPERATURE, DENSITY, SPECIES

EXPERIMENTAL CONFIGURATION

MOLECULAR ENERGY LEVELS AND RADIATIVE FREQUENCIES
TEMPERATURE AND DENSITY MEASUREMENTS IN AIR AT CONSTANT PRESSURE USING LIF/RAMAN SPECTROSCOPY

![Graph showing Raman density vs. LIF temperature for different pressures (P = 760 torr and P = 250 torr).]
KEY ACCOMPLISHMENTS

Instrumentation

Experiments
TOPOLOGY OF COHERENT STRUCTURES

Reθ = 2000   U∞ = 10 m/sec

Flow

x/h = 2.1  y/δ

x/h = 4.2  y/δ
ENSEMBLE AVERAGED QUANTITIES

\[ \frac{X}{h} = -2.6 \quad y^+ = 260 \]

\[ \frac{tU_\infty}{\delta} \]

0  20  40  60  80  100  120
VERIFICATION EXPERIMENT FOR TRANSONIC FLOW CODES

SUPERCRITICAL AIRFOIL WITH LDV

TEST CONDITIONS
- $0.73 < M < 0.80$
- $2 \times 10^6 < \text{Re} < 6 \times 10^6$
- $0.5 < \alpha < 1.5$

MEASUREMENTS
- PRESSURE ($P$, $P'$)
- FLOW FIELD ($U$, $V$, $u'$, $v'$, ...)
- OIL FLOW
- LIFT, DRAG

WING PRESSURES
$M = 0.78$, $\alpha = 1^\circ$

WAKE VELOCITY
$M = 0.78$, $\alpha = 1^\circ$, $x/c = 1.04$

EXPERIMENT
ADVANCED TURBULENCE MODEL
STANDARD TURBULENCE MODEL

MARVIN - 16
RANS TRANSONIC AIRFOIL PREDICTIONS  
TURBULENCE MODELING IMPROVEMENTS RAE 2822 AIRFOIL

- EXPERIMENT
- NONEQUILIBRIUM MODEL
- EQUILIBRIUM MODEL

ATTACHED CASE  
$M_\infty \approx 0.73, \alpha \approx 3^\circ$

SEPARATED CASE  
$M_\infty \approx 0.75, \alpha \approx 3^\circ$
EFFECT OF TURBULENCE MODEL ON WING PRESSURES
ONERA M-6 WING, M=0.84  α = 6

PRESSURE CONTOURS

BALDWIN-LOMAX

JOHNSON-KING

EXPERIMENT
289X65X49 JKM
289X65X49 BL

EXPERIMENT
289X65X49 JKM
289X65X49 BL

2Y/B=0.650

2Y/B=0.800

X/C

X/C
TRANSonic WING-BODY EXPERIMENT

WING-ALONE RESULTS

\( M_\infty = 0.8, \text{Re}_c = 9 \times 10^6 \)
LOW ASPECT RATIO WING/BODY EXPERIMENT

$\text{Re}_{\text{Cr}} = 12 \times 10^6 \quad \alpha = 5^\circ \quad M_\infty = 0.775$

$\eta = 0.3 \quad \eta = 0.5 \quad \eta = 0.7 \quad \eta = 0.8 \quad \eta = 0.9$

$C_p$ vs. $x/c$ for different values of $\eta$. The plots show the pressure coefficient ($C_p$) distribution along the chord length ($x/c$) for various values of the parameter $\eta$. The plots are used to analyze the aerodynamic characteristics of the wing at different stations along its chord.
REAL-GAS FORCE AND MOMENT MEASUREMENTS AND CFD CORRELATION FROM THE BALLISTIC RANGE

Re = 10^6, V_\infty = 5 km/sec, Mn = 14.5

- Measurement (Error: ± 3%)
- Computations
  - Perfect gas
  - Non-equilibrium air

Shadowgraph of blunt cone

Computed pressure contours

Flow Solvers: TUFF and STUFF
FINITE FRINGE INTERFEROGRAM

5 degree half-angle blunt cone
p = 63mm \hspace{1cm} v = 16,400 \text{ ft/sec}
NASP Nozzle/Afterbody Computations

Experimental Geometry

3-D Composite Grid Topology

Mach Contours in Symmetry & Outflow Planes

Pressure Contours & Particle Traces

S.M. Puffin & E. Venkatapathy
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BY

L.A. SCHUTZENHOFER, H.V. McCONNAUGHEY, P.K. McCONNAUGHEY

COMPUTATIONAL FLUID DYNAMICS BRANCH
AEROPHYSICS DIVISION
STRUCTURES AND DYNAMICS LABORATORY
NASA/MARSHALL SPACE FLIGHT CENTER

SECOND NASA CFD VALIDATION WORKSHOP
JULY 10–12, 1990
NASA LEWIS RESEARCH CENTER
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

OVERVIEW

- FOCUS OF MSFC CFD ACTIVITIES
- DESIGN APPLICATIONS
- VALIDATION ACTIVITIES
- CFD REQUIREMENTS
- CFD EXPECTATIONS
- SUMMARY
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

FOCUS OF MSFC CFD ACTIVITIES

● SUPPORT PROGRAM OFFICES
  - "QUICK TURNAROUND" APPLICATIONS; SENSITIVITY ANALYSIS; RETROFITTABLE DESIGN OPTIONS
  - INTERACT WITH HARDWARE CONTRACTORS IN DEVELOPMENT OF DESIGN ENVIRONMENTS
  - PROVIDE "SMART BUYER" CAPABILITY FOR LONG-TERM APPLICATIONS
  - DEVELOP SUBSYSTEMS CFD MODELS
  - FOCUS MSFC CFD ACTIVITIES/PROVIDE CENTERWIDE CFD SUPPORT

● FOCUS DEVELOPMENT OF CFD METHODOLOGY
  - INTERACT WITH ARC, LeRC, LaRC, AND OTHER RESEARCH ORGANIZATIONS TO FOCUS
    TECHNOLOGY DEVELOPMENT TOWARDS MSFC HARDWARE RELATED PROBLEMS
  - DEVELOP REQUIREMENTS FOR CFD CODE VERIFICATION
  - VERIFY CODES THROUGH BENCHMARK COMPARISONS
  - ADVANCE CFD TECHNOLOGY FOR APPLICATIONS

● DEVELOP ADVANCED HARDWARE TECHNOLOGY CONCEPTS
  - TURBINE STAGE
  - PUMP STAGE
  - NOZZLES, PREBURNERS, ETC.
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

DESIGN APPLICATIONS

• PROGRAM SUPPORTING ACCOMPLISHMENTS

• DESIGN APPLICATION EXAMPLES
  — ATD DISCHARGE VOLUTE SIDE LOAD ANALYSIS
  — MAIN INJECTOR LOX INLET FLOW INDUCED VIBRATION
  — NOZZLE/MAIN COMBUSTION CHAMBER (MCC) MISMATCH

• CURRENT PROGRAMS
## STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

### PROGRAM SUPPORTING ACCOMPLISHMENTS

<table>
<thead>
<tr>
<th></th>
<th>INHOUSE</th>
<th>CONT.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>SSME</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>● HPFTP TURBINE BLADES</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● TURBINE DISK CAVITIES</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● LOX PUMP BEARING INLET CAVITY</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>● LOX PUMP BEARINGS</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● FUEL PREBURNER</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● LOX PREBURNER</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● LOX MANIFOLD TEE (4000 Hz)</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>● HOT GAS MANIFOLD/MANIFOLD STRUTS</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>● PUMP COOLANT FLOW PATHS</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● NOZZLE/MCC MISMATCH</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>● HPOTP NOZZLE PLUG TRAJECTORIES</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>● TRANSIENT BEHAVIOR OF FUEL PREBURNER MANIFOLD</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>● BEARING DEFLECTOMETER (TTBE)</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● ENGINE 0212 BEARING CAGE DEBRIS</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● ENGINE 0209 MCC COOLANT LINER CRACKING</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● ENGINE 2019 HPFTP STATIC SEAL PARTICLE TRAJECTORIES</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td><strong>ATD</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>● TURBINE INLET TEMP. REDISTRIBUTION</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● TURBINE TEMP. PROFILE REDISTRIBUTION</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● ROTOR-STATOR INTERACTION</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● TURNAROUND DUCT AND HOT GAS MANIFOLD</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● BEARING ANALYSIS</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● LOX PUMP INLET SCROLL</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● FUEL PUMP INTERSTAGE CROSSOVER DUCTS</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● FUEL PUMP INLET SCROLL</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● LOX PUMP DISCHARGE VOLUTE</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>● SEALS</td>
<td></td>
<td></td>
</tr>
<tr>
<td>● HOT GAS MANIFOLD CHECKOUT CHAMBER</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● FUEL PUMP DISCHARGE VOLUTE</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● HPFTP AFT DISK CAVITY</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● HPOTP TURNAROUND DUCT VANE FAILURE</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>● UNSTEADY MULTISTAGE TURBINE LOADS</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>
## STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

### PROGRAM SUPPORTING ACCOMPLISHMENTS

<table>
<thead>
<tr>
<th>SRB</th>
<th>INHOUSE</th>
<th>CONT.</th>
</tr>
</thead>
<tbody>
<tr>
<td>• BORE FLOW</td>
<td></td>
<td></td>
</tr>
<tr>
<td>— CANTED NOZZLE</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>— BROKEN INHIBITOR</td>
<td></td>
<td>X</td>
</tr>
<tr>
<td>• FIELD JOINT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>— FLOW AND THERMAL TRANSIENT</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>— PRESSURIZATION TRANSIENT</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>• NOZZLE-TO-CASE JOINT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>— FLOW AND THERMAL TRANSIENT</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>— PRESSURIZATION TRANSIENT</td>
<td>X</td>
<td>X</td>
</tr>
<tr>
<td>• MNASA MOTOR</td>
<td></td>
<td></td>
</tr>
<tr>
<td>— WITH BLAST TUBE</td>
<td>X</td>
<td></td>
</tr>
<tr>
<td>— WITHOUT BLAST TUBE</td>
<td>X</td>
<td></td>
</tr>
</tbody>
</table>

| AFE                      |         |       |
| • AEROTHERMAL ENVIRONMENTS|       |       |
|   — DSMC                  | X       |       |
|   — NS                    | X       | X     |

| SPACE STATION            |         |       |
| • CONTAMINATION TRACKING  | X       |       |
| • ECLSS MODULE AND NODE   | X       | X     |

| ADVANCED PROGRAM DEVELOPMENT | EXTERNAL TANK GAMMA RAY IMAGING TELESCOPE | X |

---

5-9809-0-238
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

VISCOUS ANALYSIS OF THE ATD OXYGEN AND FUEL PUMP DISCHARGE VOLUTES

• OBJECTIVE: TO OBTAIN SIDELOADS DUE TO HIGH PRESSURE PUMP VOLUTES.

• JUSTIFICATION: ACCURATE SIDELOAD PREDICTION IS CRITICAL FOR SUCCESSFUL BEARING DESIGN. WATER-FLOW RIG DATA AVAILABLE FOR OXYGEN PUMP VOLUTE. NO DATA AVAILABLE FOR FUEL PUMP VOLUTE UNTIL 3RD HOTFIRE TEST SERIES OF TURBOPUMP.

• APPROACH: USED FDNS2D TO PERFORM INCOMPRESSIBLE, 2-D TURBULENT ANALYSIS OF VOLUTE GEOMETRY. JACOBIAN MODIFIED TO ACCOUNT FOR 3-D CROSS-SECTION AREA. USING 511 X 97 GRID POINTS FOR 5-VANED, DUAL EXIT LOX VOLUTE. USING 481 X 79 GRID FOR 13-VANED SINGLE EXIT FUEL VOLUTE.

• RESULTS/IMPACT: 2-D RESULTS: LOX VOLUTE ANALYSES MATCHED RESULTANT LOAD (700 LBS.) BUT NOT DIRECTION. FUEL VOLUTE RESULTS INDICATE VERY SMALL LOADS (< 100 LBS.). PSEUDO 3-D ANALYSIS STILL UNDERWAY.
LOX VOLUTE RESULTS
PRESSURE CONTOURS (psi)

CONTOUR LEVELS
2500.000
2550.000
2600.000
2650.000
2700.000
2750.000
2800.000
2850.000
2900.000
2950.000
3000.000
3050.000
3100.000
3150.000
3200.000
3250.000
3300.000
3350.000
3400.000
3450.000
3500.000
3550.000
3600.000
3650.000
3700.000
3750.000
3800.000
3850.000
3900.000
4000.000

STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

MAIN INJECTOR LOX INLET CFD RESULTS

A. MAIN INJECTION LOX INLET

FLIGHT CONFIGURATION Re = 10^6; S_l = .375 (3800 Hz)

FIX (BEVELED TRAINING EDGE) Re = 10^6; S_l = .445 (4600 Hz)

B. PRESSURE/LOAD VARIATION WITH TIME
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

NOZZLE/MCC MISMATCH FLOW FIELD ANALYSIS

OBJECTIVES

• DETERMINE TEMPERATURE IN G15 CAVITY FOR VARIOUS MISMATCH CONFIGURATIONS

• ASSESS SENSITIVITY OF CAVITY TEMPERATURE TO VARIATIONS IN MAXIMUM PROTRUSION, CIRCUMFERENTIAL DIFFERENTIALS IN PROTRUSION, AND GAP

• DEVELOP ACCEPT/REJECT CRITERIA FOR FUTURE ENGINE CONFIGURATIONS

APPROACH

• CFD SIMULATIONS OF 2D, CLOSED CAVITY FLOW (APPROXIMATE GEOMETRY)
  – ASSESS CAVITY PRESSURE AS A FUNCTION OF PROTRUSION
  – DEDUCE APPROXIMATE CIRCUMFERENTIAL FLOW AS A FUNCTION OF MAXIMUM PROTRUSION

• CFD SIMULATIONS OF 2D CAVITY WITH MASS FLOW OUT OF CAVITY BASE (EXACT GEOMETRY)
  – PERFORM PARAMETER SENSITIVITY STUDY
  – DETERMINE FOR WHAT PARAMETER VALUES THE CAVITY TEMPERATURE REACHES THE CRITICAL TEMPERATURE AT WHICH HEAT CONDUCTION AT NOZZLE LIP WILL CAUSE BLUEING OF SEAL

PROGRESS IMPACT

• RESULTS COMMUNICATED TO CHIEF ENGINEER FOR SSME AND TO HARDWARE CONTRACTOR

• RESULTS USED IN DEVELOPMENT OF ACCEPT/REJECT CRITERIA
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

NOZZLE/MCC MISMATCH CFD RESULTS

A. ENGINE CONFIGURATION

B. NOZZLE/MCC MISMATCH JOINT

C. PRESSURE/TEMPERATURE VARIATION WITH STEP HEIGHT
## STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

### CURRENT PROGRAMS

<table>
<thead>
<tr>
<th>Development Programs</th>
<th>CY 97</th>
<th>CY 98</th>
<th>CY 99</th>
<th>CY 00</th>
<th>CY 01</th>
<th>CY 02</th>
<th>CY 03</th>
<th>CY 04</th>
<th>CY 05</th>
<th>CY 06</th>
<th>CY 07</th>
<th>CY 08</th>
<th>CY 09</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Advanced Launch System</strong></td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td>✓</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>(STEP)</strong></td>
<td>Start</td>
<td>Start</td>
<td>Start</td>
<td>Start</td>
<td>CDR</td>
<td>CDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Advanced Solid Rocket Motor</strong></td>
<td>▼</td>
<td>PDR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td>CDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Aeroassist Flight Experiment</strong></td>
<td>PRR</td>
<td>PDR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>FLIGHT</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Space Station Freedom</strong></td>
<td>▼</td>
<td>PRR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td>CDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>ECLSS</strong></td>
<td>▼</td>
<td>PRR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Alternate Turbopump</strong></td>
<td>▼</td>
<td>PRR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Development</strong></td>
<td>▼</td>
<td>PRR</td>
<td>PDR</td>
<td>PDR</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
VALIDATION ACTIVITIES

● BENCHMARK VALIDATION PLAN
  - LEVEL I: FUNDAMENTAL FLOWS
  - LEVEL II: SUBCOMPONENTS
  - LEVEL III: INTERACTIVE COMPONENTS/SYSTEMS
  - END PRODUCTS

● CONSORTIUM
# STATUS OF MSFC CFD APPLICATION

## VALIDATION ACTIVITIES

### LEVEL I: FUNDAMENTAL FLOWS

<table>
<thead>
<tr>
<th>CODE</th>
<th>FLAT PLATE</th>
<th>CHANNEL FLOW</th>
<th>STRAIGHT PIPE</th>
<th>CIRCULAR PIPE BEND</th>
<th>RECT PIPE BEND</th>
<th>BACKWARD FACING STEP</th>
</tr>
</thead>
<tbody>
<tr>
<td>INS3D</td>
<td>-</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>INS3DLU</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>INS3DUP</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>CMINT</td>
<td>-</td>
<td>2</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>FDNS</td>
<td>3</td>
<td>3</td>
<td>4</td>
<td>4</td>
<td>2</td>
<td>3</td>
</tr>
<tr>
<td>REFLEQS</td>
<td>1</td>
<td>3</td>
<td>1</td>
<td>2</td>
<td>-</td>
<td>2</td>
</tr>
<tr>
<td>MAST</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>1</td>
<td>-</td>
<td>1</td>
</tr>
</tbody>
</table>

* NUMBER OF PEOPLE WORKING PROBLEM
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

- LEVEL 2 – SUBCOMPONENTS

  CASCADE
  DISK CAVITY
  SINGLE INJECTOR
  UNSTEADY FLOW OVER CYLINDER
  AXISYMMETRIC DIFFUSER
  CURVED DUCT WITH TURNING VANES
  HIGH CURVATURE DUCT WITH SEPERATION
  ROTATING DUCT
  ROTATING CURVED DUCT
  NOZZLE
  CYLINDER WITH BLOWING WALLS
  SLOTTED CYLINDER WITH BLOWING WALLS

- LEVEL 3 – INTERACTIVE COMPONENTS OR SYSTEMS

  HOT GAS MANIFOLD PILOT MODEL
  UNSTEADY TURBINE STAGE
  MAIN INJECTOR/NOZZLE/PLUME
  UNSTEADY PUMP STAGE
  INLET AND DISCHARGE VOLUTES
  AEROASSIST FLIGHT EXPERIMENT
  BEARING CAVITY MODEL
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

DODOCUMENTATION REQUIREMENTS

• OBJECTIVE: DESCRIBE PROBLEM AND THE FOCUS/EXPECTATION OF THE CFD COMPUTATION

• COMPUTATIONAL STRATEGY: PROBLEM DEFINITION, PROBLEM GOAL, BOUNDARY CONDITIONS, INITIALIZATION, USER ISSUES

• DATA BASE: DEFINITION OF GEOMETRY, Re, M, DATA SOURCE, ETC.

• COMPUTER REQUIREMENTS: CPU TIME, STORAGE REQUIREMENTS, SSD REQUIREMENTS

• LESSONS LEARNED: DESCRIPTION OF FALSE STARTS, GRID SENSITIVITY, INITIALIZATION, B.C. PROBLEMS.

• SUMMARY: STRENGTH/WEAKNESS IN COMPUTATION VS. DATA COMPARISON. EVALUATION OF CODE FOR POTENTIAL HARDWARE APPLICATION.

DOCUMENTATION SHOULD BE CONCISE.
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

BENCHMARK VALIDATION PLAN

• END PRODUCTS:
  - DIMENSIONLESS EVALUATION CRITERIA
  - ERROR ESTIMATION METHODOLOGY
  - ASSESSMENT OF CODES BASED UPON CRITERIA AND ERROR ESTIMATION

• FOCUS OF CRITERIA:
  - FLOW RANGES (MACH NUMBER, REYNOLDS NUMBER --)
  - CODE (SOLUTION ALGORITHM, CODE ARCHITECTURE, BOUNDARY CONDITIONS --)
  - FLOW PROCESS MODELS (TURBULENCE MODEL, MULTISPECIES --)
  - USER SPECIFIC EFFECTS (GRID GENERATING, INITIALIZATION --)
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

○ CONSORTIUM OBJECTIVES

○ FOCUS CFD APPLICATIONS IN PROPULSION
  ○ TECHNOLOGY ACQUISITION PHASE
    ○ DIRECT BASELINE PROGRAM TOWARDS IMPROVED ACCURACY, STABILITY, AND EFFICIENCY
  ○ LARGE SCALE SUBSYSTEM TECHNOLOGY VALIDATION
    ○ STIMULATE CFD VALIDATION TOWARDS PROPULSION FLOWS
    ○ DIRECT APPLICATIONS CODES TOWARD DESIGN TOOLS AND ADVANCED TECHNOLOGY HARDWARE CONCEPTS

○ IDENTIFY NATIONAL CFD PROPULSION REQUIREMENTS

○ STIMULATE A FORUM FOR GOVERNMENT, INDUSTRY, AND UNIVERSITY INTERACTIONS

○ ENCOURAGE INDUSTRY TO PARTICIPATE IN CFD DEVELOPMENT WITH IRAD FUNDS

○ PROVIDE SYNERGISM IN THE CFD COMMUNITY

○ PROVIDE PEER REVIEW OF CFD PROGRAMS
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM FOR CFD APPLICATIONS IN PROPULSION TECHNOLOGY

ORGANIZATIONAL STRUCTURE

NASA

PROJ. DIR. WORKING GROUP

MEMBER ORGANIZATIONS

CONSORTIUM FOR CFD COORDINATOR

COUNCIL

AGENCY PROJECT DIRECTOR
PRINCIPAL INVESTIGATOR

CONTRACTOR PROJECT DIRECTOR
PRINCIPAL INVESTIGATOR

UNIVERSITY PROJECT DIRECTOR
PRINCIPAL INVESTIGATOR

OTHER FUNDED PROJECTS
IR & D FUNDED PROJECTS
OTHER FUNDED PROJECTS
IR & D FUNDED PROJECTS
OTHER FUNDED PROJECTS
IR & D FUNDED PROJECTS
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

ADVANCED TECHNOLOGY HARDWARE

• CONSORTIUM TEAMS

FOCUS DEVELOPMENT OF CFD METHODOLOGY AND
DEVELOP ADVANCED TECHNOLOGY HARDWARE CONCEPTS

TURBOMACHINERY
- TURBINES
- PUMPS

COMPLEX FLOW PATHS
- DUCTS
- COOLANT FLOWS
- ROTATING CAVITIES

CASTINGS

COMBUSTION DRIVEN FLOWS

SOLID ROCKET MOTORS
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

● CONSORTIUM TEAM VALIDATION ACTIVITIES

- TURBINE TEAM
  - CASCADES
  - ROTOR STATOR
  - SSME

- PUMP TEAM
  - ROCKETDYNE INDUCER

- COMPLEX FLOW PATHS
  - DISK CAVITIES
  - HGM PILOT MODEL
  - DUCT FLOWS

- SRM TEAM
  - CSD CYLINDER
  - SLOTTED CYLINDER
  - SRM PILOT MODEL

- COMBUSTION TEAM
  - H_{2} MIXING
  - DUMP COMBUSTION
  - NOZZLE FLOW
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

TURBINE STAGE TECHNOLOGY TEAM

OBJECTIVE: DEVELOPMENT, ENHANCEMENT, VALIDATION, AND DEMONSTRATION OF CFD TOOLS FOR TURBINE STAGE DESIGN

APPROACH:

- ENHANCE EXISTING DATA BASE (COLD FLOW RIG AND HOT-FIRE SYSTEM TESTS), DEVELOP/ENHANCE CFD MODELS AND CODES, EXPERIMENTALLY DEMONSTRATE CFD-PREDICTED DESIGN IMPROVEMENTS

- ASSEMBLE TURBINE TECHNOLOGY EXPERTS REPRESENTING DIFFERENT AREAS OF EXPERTISE. POOL TECHNICAL RESOURCES AND EXPERIENCE BASES TO BRING VARIOUS PERSPECTIVES AND ABILITIES TO BEAR ON TURBINE STAGE ISSUES AND ON ACCOMPLISHMENT OF TEAM OBJECTIVE

- DEVELOP AND IMPLEMENT PLAN WHICH COORDINATES MSFC-SUPPORTED TURBINE STAGE ANALYSIS AND TEST ACTIVITIES

- IDENTIFY MILESTONES, DELIVERABLE PRODUCTS, AND TARGET DATES

- IDENTIFY ADDITIONAL ACTIVITIES REQUIRED TO ACCOMPLISH TEAM OBJECTIVE

- INTERACT THROUGH REGULAR MEETINGS TO STATUS, CROSS-FERTILIZE, CRITIQUE, AND/OR DIRECT EACH ACTIVITY
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM

• TURBINE STAGE TECHNOLOGY TEAM

STATUS: 5 MEETINGS HELD TO DATE—4/28/89, 8/23/89, 11/1/89, 1/17/90, 4/19/90

MEMBERSHIP:
• NASA MARSHALL SPACE FLIGHT CENTER (MSFC)
• NASA AMES RESEARCH CENTER (ARC)
• NASA LEWIS RESEARCH CENTER (LeRC)
• ROCKETDYNE (RDYN)
• PRATT & WHITNEY (P&W)
• AEROJET
• GENERAL ELECTRIC COMPANY
• UNITED TECHNOLOGIES RESEARCH CENTER (UTRC)
• CALSPAN—UNIVERSITY OF BUFFALO RESEARCH CENTER (CUBRC)
• ROTO DATA, INC.
• SCIENTIFIC RESEARCH ASSOCIATES (SRA)
• UNIVERSITY OF ALABAMA—HUNTSVILLE (UAH)
• PENNSYLVANIA STATE UNIVERSITY (PSU)
• MISSISSIPPI STATE UNIVERSITY (MSU)
# STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

## CONSORTIUM - TURBINE TEAM

### KEY MILESTONES

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CODE DEVELOPMENT,</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>ENHANCEMENT, AND VALIDATION</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotor3 Validation</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stage 2 Development</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(2D Unsteady, Multiblade, Multistage)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Improved Turbulence Modeling</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stage 3 Development</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(3D Unsteady, Multiblade, Multistage)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Improved Deterministic Stress Modeling</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3D Multistage development with deterministic stress modeling</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>DATA</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>UTIA Heat Transfer</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P&amp;W Unsteady Aerodynamics</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>SSME Aerodynamics &amp; Heat Transfer</td>
<td>4</td>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>P&amp;W Hel Streak</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>ADVANCED CONCEPT DEVELOPMENT</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline Design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Baseline Rig Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Concept Design</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Concept Rig Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Advanced Concept Hot Fire Test</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**KEY:**

1. Validation against steady aerodynamic and heat transfer data
2. Validation against unsteady aerodynamic data
3. Validation against SSME or advanced concept data
4. Steady and unsteady aerodynamic data, unsteady heat transfer data
5. ATD steady aerodynamic data
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CONSORTIUM - TURBINE TEAM

- BASELINE TURBINE MIDSPAN CONTOURS/CALCULATED STREAMLINES

1st VANE  1st BLADE  2nd VANE  2nd BLADE

NOTE: ROW TO ROW SCALES ARE DIFFERENT
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CFD REQUIREMENTS

• GRID GENERATION
  – ELECTRONIC SURFACE GENERATORS COMPATIBLE WITH CFD CODES
  – INTERACTIVE GRID PACKAGES WITH INTERROGATION SCHEMES
  – ADAPTIVE GRID PACKAGES

• EVALUATION CRITERIA AND ERROR ANALYSIS METHODOLOGY
  – STANDARDIZED DIMENSIONLESS PARAMETERS

• VALIDATION EXPERIMENTS

• CODES
  – BENCHMARKING USING STANDARDIZED METHODS
  – MULTIBLOCKING OR ZONAL STRATEGY

• FLOW PROCESS MODELS
  – TURBULENCE AND TRANSITION
  – MULTISPECIES
  – CHEMISTRY WITH AND WITHOUT REACTIONS
  – MULTIPHASE
  – ATOMIZATION
  – EVAPORATION
  – MIXING

• POST PROCESSING
  – DEMONSTRATION OF CONSERVATION PROPERTIES
  – ERROR ANALYSIS

• GRAPHICS
  – INTERACTIVE
  – THREE DIMENSIONAL FLOW DESCRIPTIONS
  – ANIMATION
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

CFD EXPECTATIONS

• DIRECT HARDWARE DESIGN UTILIZING CFD
  — PROVIDE INITIAL IMPACT IN DESIGN
  — PERFORM DESIGN OPTIMIZATION STUDIES
  — DEVELOP ADVANCED TECHNOLOGY HARDWARE CONCEPTS

• BENCHMARKED/VALIDATED CODES
  — LAMINAR FLOWS
  — TURBULENT FLOWS $u_1, p$
  — ACOUSTIC PROBLEMS
  — CERTAIN CLASS OF UNSTEADY PROBLEMS

• GENERAL METHODOLOGY FOR ERROR ESTIMATION AND EVALUATION CRITERIA FOR CODES

• USER FRIENDLY CODES
  — B.S. LEVEL ENGINEER 2-3 YRS EXPERIENCE
  — GUIDELINES FOR CLASSES OF PROBLEMS
  — CAD/CAM/CAE; GEOMETRY GRID GENERATION
  — GENERALIZED BOUNDARY CONDITIONS
  — ALGORITHM/GRAID OPTIMIZATION FOR SOLUTION EFFICIENCY
  — ARTIFICIAL INTELLIGENCE/EXPERT SYSTEMS
  — MODULAR CODES

• FLOW ADAPTIVE GRIDS FOR CURRENT CLASS OF PROBLEMS

• MULTIPLE SCALE AND/OR ZONAL TURBULENCE MODELS, MULTIPHASE, MULTISPECIES, COMBUSTION FLOW PROCESS ENGINEERING MODELS EVOLVED FROM EXPERIMENTS AND CFD ANALYSIS
STATUS OF MSFC CFD APPLICATION AND VALIDATION ACTIVITIES

SUMMARY

- CFD BEING APPLIED IN THE HARDWARE DESIGN PROCESS AT MSFC
  - FOCUS ON TEAM STRATEGY; RIGHT PROBLEM
  - QUICK TURN-AROUND; SENSITIVITY ANALYSIS
  - RETROFITTABLE DESIGN OPTIONS

- VALIDATION ACTIVITIES PROVIDING COMMUNITY SYNERGISM
  - LEVEL I BENCHMARK PLAN IN PLACE / IN PROGRESS
  - CONSORTIUM TEAMS INTEGRATE ADVANCED HARDWARE DESIGN AND VALIDATION ACTIVITIES
    - FOCUS CFD GOALS AND DEFINE REQUIREMENTS
    - INTEGRATE ACTIVITIES, POOL RESOURCES, AND TRANSFER TECHNOLOGY BETWEEN GOVERNMENT, CONTRACTORS, AND UNIVERSITIES

- CFD EXPECTATIONS ARE APPLICATIONS-ORIENTED AND REQUIRE SUSTAINED TECHNOLOGY TRANSFER FROM RESEARCH CENTERS
SUMMARY OF LEWIS RESEARCH CENTER VALIDATION EFFORTS

RAYMOND E. GAUGLER
CHIEF, TURBOMACHINERY FLOW PHYSICS BRANCH
CLOSELY COUPLED EXPERIMENTAL AND COMPUTATIONAL RESEARCH

NUMERICAL CODE DEVELOPMENT

MODELING OF PHYSICS
VALIDATION OF COMPUTATIONS

EXPERIMENTATION

UNDERSTANDING OF FLOW PHYSICS
ACCURATE PREDICTIVE CODES
LINEAR TRANSONIC CASCADE

INSTRUMENTATION - PRESSURE TAPS
\[(y/a)^n + (z/b)^n = 1\]

AR410 TRANSITION DUCT
HEAT TRANSFER COEFFICIENTS ALONG DUCT CENTERLINE

HEAT TRANSFER COEFFICIENT, \( h \) (W/M\(^2\) K)

STEADY-STATE PREDICTION
TRANSIENT PREDICTION

STEADY-STATE DATA
SINGLE CRYSTAL EXP.
RUN 1
RUN 2
RUN 3
RUN 4

DOUBLE CRYSTAL EXPERIMENT
TRANSIENT DATA

INLET X/D EXIT
EXPERIMENTAL AND PREDICTED HEAT TRANSFER COEFFICIENTS
LOW SPEED CENTRIFUGAL COMPRESSOR

BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES

CD-84-15176
TURBOMACHINERY BLADE ROW INTERACTIONS

MULTI-STAGE COMPRESSOR FLOW PHYSICS
MULTISTAGE COMPRESSOR FLOW PHYSICS PROGRAM

ERBNET

CFD Development
Navier-Stokes
Avg. Passage

Closure Validation

High-Speed Interactive Computing

Guidance

Experiments
Rotor/Stator Multistage

Adv. Msmt. Tech
3D Velocity State Variables

Validated Methods for Practical Computation of Flows in Multistage Compressors
TURBOMACHINERY BLADE ROW INTERACTIONS

MOTIVATION –

Current Designs Based On Time–Averaged Axisymmetric Analyses

Actual Flow Field Highly Non–Axisymmetric And Time Dependent

OBJECTIVE

Experimentally Validated Numerical Analysis Of Flows In Multi–Stage Turbomachinery

Improved Understanding of Flow Physics Of Blade–Row Interactions
LOW SPEED MULTISTAGE AXIAL-FLOW COMPRESSOR
BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES

4 ft. diam. AXIAL-FLOW COMPRESSOR ASSEMBLY
MULTISTAGE COMPRESSOR FLOW PHYSICS PROGRAM

SUMMARY

- A SYNERGISTIC PROGRAM PULLING TOGETHER CFD, EXPERIMENTS, AND HIGH-SPEED INTERACTIVE COMPUTING TO PROVIDE A NEEDED CAPABILITY

- CFD - UTILIZE AVERAGE-PASSAGE METHOD FOR MULTI-STAGE SIMULATION AND TIME ACCURATE METHODS TO SUPPORT CLOSURE MODELING

- EXPERIMENTS - INITIAL RESULTS FROM HIGH-SPEED, SINGLE STAGES - CLOSURE MODELING AND VALIDATION/CALIBRATION FROM EXISTING HIGH-SPEED MULTISTAGE AND NEWLY REQUIRED LOW SPEED MULTISTAGE

- MEASUREMENT TECH. - INSTRUMENTATION ADVANCES REQUIRED FOR 3D VELOCITY AND SCALAR MEASUREMENTS IN COMPRESSIBLE FLOWS

- HIGH-SPEED INTERACTIVE COMPUTING - REQUIRED FOR TIMELY CFD AND EXPERIMENTAL INTERACTION
LeRC/IFMD INLET DUCT AND NOZZLE HIGH SPEED VALIDATION EXPERIMENTS

• VALIDATION WORKSHOP EXPERIMENTS
  - CROSSING SHOCKS/BOUNDARY-LAYER INTERACTION
  - LeRC/UNIVERSITY UNSTEADY SHOCK/BOUNDARY-LAYER INTERACTION

• ADDITIONAL VALIDATION EXPERIMENTS
  - HIGH SPEED MIXING
  - TRANSITION DUCTS
  - VORTEX GENERATORS
EXPERIMENTS IN 3-D FLUID MECHANICS

HYPERMIXING

TRAN. DUCT

VORTEX GEN

CROSSING SHOCK

QUANTITATIVE DATA

QUALITATIVE DATA

UNSTEADY SHOCK

FLOW
EXPERIMENTS IN 3-D FLUID MECHANICS

HYPERMIXING

TRAN. DUCT

VOlTEX GEN

CROSSING SHOCK

UNSTEADY SHOCK

CFD VALIDATION
FLOW PHYSICS
EXPERIMENTAL AND COMPUTATIONAL SURFACE AND FLOW-FIELD RESULTS FOR AN ALL-BODY HYPersonic Aircraft

WILLIAM K. LOCKMAN
NASA Ames Research Center
Moffett Field, CA 94035-1000

SCOTT L. LAWRENCE
NASA Ames Research Center
Moffett Field, CA 94035-1000

JOSEPH W. CLEARY
EloRET Institute
Sunnyvale, CA 94087

Second NASA CFD Validation Workshop
NASA Lewis Research Center
July 10-11, 1990
OUTLINE

OBJECTIVE

EXPERIMENT
- Model
- Test Conditions
- Measurements

UPS CODE (Upwind PNS Solver)

RESULTS (Experimental & Computational)
- Flow Visualization
- Surface Pressures
- Surface Convective Heat Transfer
- Pitot-Pressure Flow-Field Surveys

CONCLUDING REMARKS
ALL-BODY HYPersonic TEST PROGRAM FOR CFD CODE VALIDATION

OBJECTIVE: Establish benchmark experimental data base for generic hypersonic vehicle shape for validation and/or calibration of advanced CFD computer codes

MOTIVATION: Need for extensive hypersonic data to fully validate CFD codes to be used for NASP & other hypersonic vehicles

APPROACH: Conduct comprehensive test program for generic all-body hypersonic aircraft model in Ames 3.5-ft Hypersonic Wind Tunnel to obtain pertinent surface and flow-field data over broad range of test conditions
ALL-BODY HYPersonic Aircraft Experiment
Ames 3.5-ft HWT

MODEL:

AMES GENERIC ALL-BODY HYPersonic MODEL:
  ● Delta Planform ($\Lambda = 75^\circ$)
    ● Forebody — Elliptic Cone ($a/b = 4$)
    ● Afterbody — Elliptic Cross Sections with Sharp Trailing Edge
  ● Sharp or Blunt Nose Tip
  ● With or Without Control Surfaces (Tested without to date)
    ● Canard
    ● Combination Horizontal/Vertical Tails
ALL-BODY HYPERSONIC AIRCRAFT MODEL
W/O CONTROL SURFACES

Elliptical Cross Sections

\[ R_1 = 0.0208 L \]
\[ R_2 = 0.00428 L \]

L = 0.914 m (3 ft)

Forebody – Elliptic Cone (a/b = 4) with Sharp or Blunt Nose Tip
Afterbody – Elliptical Cross Sections with Sharp Trailing Edge
ALL-BODY HYPERSONIC AIRCRAFT MODEL
IN NASA/AMES 3.5 -FT HWT
W/O CONTROL SURFACES; LENGTH = 3 FT
ALL-BODY HYPERSONIC AIRCRAFT EXPERIMENT
Ames 3.5-ft HWT

TEST CONDITIONS:
- \( M_\infty = 5, 7, \text{ & } 10 \)
- \( Re_{\infty, L} = 1.5 \times 10^6 \text{ to } 25 \times 10^6 \) (Laminar to Turbulent Flows)
- \( \alpha = 0^\circ \text{ to } 15^\circ \) (Attached & Separated Flows)

MEASUREMENTS:
- Flow Visualization
  - Shadowgraphs
  - Surface Oil-Flow Patterns (Skin-Friction Lines)
- Surface Pressures
- Surface Convective Heat Transfer (Selected Areas)
- Flow-Field Surveys
  - Pitot-Pressure Probes
  - Laser Doppler Velocimetry (To be done)
    - Mean Velocities
    - Turbulence Quantities
CHARACTERISTICS OF ALGORITHM:
• Second-order accurate and upwind in crossflow directions
• First-order accurate in streamwise (marching) direction
• Implicit
• Finite Volume

ADVANTAGES OF UPWIND SCHEMES:
• Shock waves are captured sharply and without oscillation
• User specification of smoothing parameters is not required

PRESENT ASSUMPTIONS:
• Laminar flow
• Turbulent flow
  — Boundary-layer transition specified
  — Baldwin-Lomax turbulence model
• Perfect gas (Used for this study)
• Equilibrium or nonequilibrium air
ALL-BODY MODEL SHADOWGRAPH

\[ \alpha = 15^\circ; \quad M_\infty = 7.4; \quad Re_{\infty,L} = 15 \times 10^6 \]
FOREBODY SHOCK-WAVE ANGLE

$M_\infty = 7.4; \text{Re}_{\infty,L} = 15 \times 10^6$

$\Theta_s = \alpha + 3.83^\circ$

EXPERIMENT

UPS CODE; TURBULENT

TANGENT-CONE METHOD; INVISCID

TANGENT-WEDGE METHOD; INVISCID
SURFACE OIL-FLOW PATTERN

$\alpha = 15^\circ; M_\infty = 7.4; \text{Re}_\infty L = 15 \times 10$

WINDWARD
SURFACE OIL-FLOW PATTERN

$\alpha = 15^\circ; M_\infty = 7.4; \text{Re}_{\infty, L} = 15 \times 10^6$

LEEWARD
PARTICLE TRACES FOR LEEWARD SURFACE BY UPS CODE
(Simulated Surface Oil-flow Patterns; \( x/L = 0.0 \) to 0.9)
\[ \alpha = 15^\circ; M_\infty = 7.4; \text{Re}_{\infty,L} = 15 \times 10^6 \]
MACH NUMBER CONTOURS FOR CROSS-FLOW PLANE BY UPS CODE

\[ x/L = 0.6 \]

\[ \alpha = 15^\circ; \, M_\infty = 7.4; \, Re_{\infty,L} = 15 \times 10^6 \]
EFFECT OF ANGLE OF ATTACK ON CENTERLINE SURFACE PRESSURES

$M_\infty = 7.4; \text{Re}_{\infty,L} = 15 \times 10^6$

WINDWARD

EXPERIMENT; $\alpha$
- $0^\circ$
- $10^\circ$
- $5^\circ$
- $15^\circ$

UPS CODE

TURBULENT

AXIAL STATION, $x/L$
EFFECT OF ANGLE OF ATTACK ON CENTERLINE SURFACE PRESSURES

$M_\infty = 7.4; \text{Re}_\infty, L = 15 \times 10^6$

LEEWARD

EXPERIMENT; $\alpha$
- $0^\circ$
- $10^\circ$
- $5^\circ$
- $15^\circ$

UPS CODE

TURBULENT

PRESSURE RATIO, $p/p_\infty$

AXIAL STATION, $x/L$

$0$ $0.1$ $0.2$ $0.3$ $0.4$ $0.5$ $0.6$ $0.7$ $0.8$ $0.9$ $1.0$

$0$ $0.5$ $1.0$ $1.5$ $2.0$

$2/3 L$
SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

$\alpha = 0^\circ; M_\infty = 7.4; Re_{\infty,L} = 15 \times 10^6$

**Experiment:**
- $x/L = 0.6$ (Forebody)
- $x/L = 0.8$ (Afterbody)

**UPS Code:**
- Turbulent
SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

\[ \alpha = 15^\circ; \quad M_\infty = 7.4; \quad Re_\infty,L = 15 \times 10^6 \]

WINDWARD

\[ x/L = 0.6 \text{ (FOREBODY)} \]

EXPERIMENT; \( x/L \)
- \( 0.6 \) (FOREBODY)
- \( 0.8 \) (AFTERBODY)

UPS CODE
- TURBULENT

SPANWISE STATION, \( y/y_{LE} \)

PRESSURE RATIO, \( p/p_\infty \)
SPANWISE SURFACE-PRESSURE DISTRIBUTIONS FOR FOREBODY AND AFTERBODY

$\alpha = 15^\circ; M_\infty = 7.4; Re_{\infty,L} = 15 \times 10^6$

LEEWARD

![Graph showing spanwise surface-pressure distributions for forebody and afterbody.](attachment:image.png)

- EXPERIMENT: $x/L$
  - $0.6$ (FOREBODY)
  - $0.8$ (AFTERBODY)

- UPS CODE
  - TURBULENT
LEADING-EDGE SURFACE Pressures

$\alpha = 0^\circ; M_\infty = 7.4; Re_{\infty,L} = 15 \times 10^6$

![Graph showing pressure ratio vs. axial station with experiment, UPS code turbulent, inviscid method tangent-cone, and swept-cylinder lines.](chart.png)
EFFECT OF REYNOLDS NUMBER ON CENTERLINE HEAT TRANSFER

$M_{\infty} = 7.4; \quad H_w/H_t = 0.4$

$\alpha = 0^\circ$

$\alpha = 15^\circ$; WINDWARD

$\mathbf{q_s} = \text{Stagnation-point heat-transfer rate for reference sphere (R = 0.01L)}$
CENTERLINE HEAT TRANSFER

\[ \alpha = 0^\circ; \ M_\infty = 7.4; \ \text{Re}_\infty, L = 15 \times 10^6; \ H_w/H_t = 0.4 \]

\[ St = \frac{q/((\rho u)_\infty (H_t - H_w))}{\text{TURBULENT}} \]
CENTERLINE HEAT TRANSFER

\( \alpha = 5^\circ; \ M_\infty = 7.4; \ Re_{\infty,L} = 15 \times 10^6; \ H_w/H_t = 0.4 \)

\[ St = \frac{q}{(\rho u)_\infty (H_t - H_w)} \]

STANTON NUMBER, \( St \)

EXPERIMENT

- WINDWARD
- LEEWARD

UPS CODE

- TURBULENT

AXIAL STATION, \( x/L \)
CENTERLINE HEAT TRANSFER

\[ \alpha = 10^\circ; \quad M_\infty = 7.4; \quad Re_{\infty,L} = 15 \times 10^6; \quad H_W/H_t = 0.4 \]

Stanton Number, St

\[ St = \frac{q}{(\rho u)_\infty (H_t - H_W)} \]

Windward

Leeeward

EXPERIMENT

- WINDWARD
- LEEWARD

UPS CODE

Turbulent
CENTERLINE HEAT TRANSFER

\[ \alpha = 15^\circ; \ M_\infty = 7.4; \ \text{Re}_\infty, L = 15 \times 10^6; \ H_w/H_t = 0.4 \]

\[ St = \frac{q}{(\rho u)_\infty (H_t - H_w)} \]

![Graph showing Stanton number vs. axial station for windward and leeward sides with experimental and UPS code data.]
EFFECT OF ANGLE OF ATTACK ON CENTERLINE HEAT TRANSFER

$M_\infty = 7.4; \text{Re}_{\infty,L} = 15 \times 10^6; H_w/H_t = 0.4$

$St = \frac{q}{(\rho u)_\infty (H_t - H_w)}$

WINDWARD

LEEWARD

STANTON NUMBER, $St$

EXPERIMENT; $\alpha$

- $0^\circ$
- $5^\circ$
- $10^\circ$
- $15^\circ$

UPS CODE

TURBULENT
SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

\( \alpha = 0^\circ; \ M_\infty = 7.4; \ Re_{\infty,L} = 15 \times 10^6; \ H_w/H_t = 0.4 \)

\[
St = \frac{q}{(\rho u)_\infty (H_t - H_w)}
\]

![Graph showing spanwise heat-transfer distributions with Stanton number (St) on the y-axis and spanwise station (y/y_{LE}) on the x-axis. The graph compares experimental data with the UPS code for turbulent flow at x/L = 0.5 and x/L = 0.6.]
SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

\[ \alpha = 5^\circ; \ M_\infty = 7.4; \ Re_{\infty,L} = 15 \times 10^6; \ H_w/H_t = 0.4 \]

\[ St = \frac{q}{(\rho u)_\infty (H_t - H_w)} \]

[Graph showing Stanton number (St) variation with spanwise station (y/y_{LE})]

\[ \text{x/L} \]

- EXPERIMENT
- UPS CODE, TURBULENT

- 0.5
- 0.6

[Legend showing data points for windward and leeward sides]
SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

\[ \alpha = 10^\circ; \quad M_\infty = 7.4; \quad Re_{\infty, L} = 15 \times 10^6; \quad H_W/H_t = 0.4 \]

\[ St = \frac{q}{(\rho u)_\infty (H_t - H_w)} \]

![Diagram showing spanwise heat-transfer distributions for a forebody with annotations for experimental and UPS code turbulent results for different spanwise stations.](image)
SPANWISE HEAT-TRANSFER DISTRIBUTIONS FOR FOREBODY

\[ \alpha = 15^\circ; \ M_\infty = 7.4; \ \text{Re}_\infty, L = 15 \times 10^6; \ \frac{H_w}{H_t} = 0.4 \]

\[ \text{St} = \frac{q}{(\rho u)_\infty (H_t - H_w)} \]

Windward

Leeward

SPANWISE STATION, \( \frac{y}{y_{LE}} \)

STANTON NUMBER, \( \text{St} \)
COMPARISONS OF EXPERIMENTAL PITOT-PRESSURE DISTRIBUTIONS WITH INVISCID MODEL OF FLOW
Afterbody Centerline; $x/L = 0.8; \ M_\infty = 7.4; \ Re_\infty, L = 15 \times 10^6$
EXPERIMENTAL AND COMPUTATIONAL PITOT-PRESSURE DISTRIBUTIONS AT VARIOUS AFTERBODY STATIONS

Centerline; $\alpha = 0^\circ$; $M_\infty = 7.4$; $Re_\infty, L = 15 \times 10^6$

ENTIRE SHOCK LAYER

<table>
<thead>
<tr>
<th>EXPERIMENT; $x/L$</th>
<th>UPS CODE</th>
</tr>
</thead>
<tbody>
<tr>
<td>$0.7$</td>
<td></td>
</tr>
<tr>
<td>$0.8$</td>
<td></td>
</tr>
<tr>
<td>$0.9$</td>
<td></td>
</tr>
</tbody>
</table>

TURBULENT

BOUNDARY-LAYER EDGE
(Shadowgraph)

NORMALIZED DISTANCE FROM SURFACE, $z_s/L$

PITOT-PRESSURE RATIO, $p_{t,2}/(p_{t,2})_\infty$
EXPERIMENTAL AND COMPUTATIONAL PITOT-PRESSURE DISTRIBUTIONS AT VARIOUS AFTERBODY STATIONS

Centerline; \( \alpha = 0^\circ; M_{\infty} = 7.4; \text{Re}_{\infty,L} = 15 \times 10^6 \)
EFFECT OF ANGLE OF ATTACK ON
PITOT-PRESSURE DISTRIBUTIONS AT AFTERBODY STATION
Centerline; \( x/L = 0.8; \) \( M_\infty = 7.4; \) \( \text{Re}_{\infty,L} = 15 \times 10^6 \)

**ENTIRE SHOCK LAYER**

- **LEEWARD**
  - EXPERIMENT; \( \alpha \)
    - \( \nabla \) \(-15^\circ\)
    - \( \triangle \) \(15^\circ\)
    - \( \square \) \(-10^\circ\)
    - \( \diamond \) \(10^\circ\)
    - \( \bigcirc \) \(-5^\circ\)
    - \( \bigtriangleup \) \(5^\circ\)
    - \( \bigcirc \) \(0^\circ\)

- **WINDWARD**

**BOUNDARY-LAYER EDGE**
(Shadowgraph)

**PITOT-PRESSURE RATIO, \( p_{t,2}/(p_{t,2})_\infty \)**

NORMALIZED DISTANCE FROM SURFACE, \( z_s/L \)
EFFECT OF ANGLE OF ATTACK ON PITOT-PRESSURE DISTRIBUTIONS AT AFTERBODY STATION
Centerline; x/L = 0.8; M_∞ = 7.4; Re_∞,L = 15 \times 10^6
CONCLUDING REMARKS

DESCRIBED EXPT. IN AMES 3.5-FT HWT WITH ALL-BODY MODEL

OUTLINED UPS CODE (Upwind PNS Solver)

PRESENTED EXPERIMENTAL & COMPUTATIONAL RESULTS:

- Flow Visualization (Shadowgraphs & Oil-Flow Patterns)
- Surface Pressure Distributions
- Surface Convective Heat-Transfer Distributions
- Afterbody Pitot-Pressure Surveys

OBSERVATIONS MADE:

- Significant changes from forebody (conical) to afterbody (nonconical) flows
- Complex leeward flow at angle of attack with cross-flow separation and vortices
- Generally good agreement between experimental and UPS code results for:
  - Shock-Wave Angles
  - Surface Pressures (some differences at higher angles of attack and near leading edge)
  - Surface Heat Transfer (some differences for afterbody and leeward flows)
  - Afterbody Pitot-Pressure Surveys (some differences for leeward flow at higher angles of attack and for inner region of viscous layer)
COMPARISON OF EXPERIMENT WITH CALCULATIONS USING CURVATURE-CORRECTED TWO EQUATION TURBULENCE MODELS FOR A TWO-DIMENSIONAL U-DUCT

D.J. MONSON AND H.L. SEEKMILLER
NASA-AMES RESEARCH CENTER, MOFFETT FIELD, CA

P.K. MCCONNAUGHEY
NASA-MARSHALL SPACE FLIGHT CENTER, HUNTSVILLE, AL

Y.S. CHEN
SECA, INC., HUNTSVILLE, AL

2ND NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER, JULY 10-12, 1990
AIAA 90-1484
Comparison of Experiment with Calculations Using Curvature-Corrected Zero and Two Equation Turbulence Models for a Two-Dimensional U-Duct
D.J. Monson and H.L. Seegmiller
NASA-Ames Research Center, Moffett Field, CA
P.K. McConnaughey
NASA-Marshall Space Flight Center, Huntsville, AL
Y.S. Chen
SECA Inc., Huntsville, AL
COMPARISON OF EXPERIMENT WITH CALCULATIONS USING CURVATURE-CORRECTED ZERO AND TWO EQUATION TURBULENCE MODELS FOR A TWO-DIMENSIONAL U-DUCT

D. J. Monson* and H. L. Seegmiller†
NASA Ames Research Center, Moffett Field, CA 94035

P. K. McConnaughey‡
NASA Marshall Space Flight Center, Huntsville, AL 35812

and

Y. S. Chen*
SECA Inc., Huntsville, AL 35805

Abstract

In this paper experimental measurements are compared with Navier-Stokes calculations using seven different turbulence models for the internal flow in a two-dimensional U-duct. The configuration is representative of many internal flows of engineering interest that experience strong curvature, such as that in the turnaround duct of the Space Shuttle Main Engine powerhead. A previous paper showed that application of a simple mixing length model to predict this flow gives poor agreement with important features of the experiment. In an effort to improve the agreement, this paper tests several versions of the two-equation k-ε turbulence model including the standard version, an extended version with a production range time scale, and a version that includes curvature time scales. Each are tested in their high and low Reynolds number formulations. Calculations using these new models and the original mixing length model are compared here with measurements of mean and turbulence velocities, static pressure and skin friction in the U-duct at two Reynolds numbers. The comparisons show that only the low Reynolds number version of the extended k-ε model does a reasonable job of predicting the important features of this flow at both Reynolds numbers tested.

Nomenclature

\[ A^+ = 26 \]
\[ AR = \text{aspect ratio} \]
\[ C_f = \text{skin friction coeff.} = \tau_w/\rho U_{ref}^2 \]
\[ C_p = \text{static pressure coeff.} = (p - p_{ref})/\rho U_{ref}^2 \]
\[ H = \text{channel height} \]
\[ k = \text{turbulent kinetic energy} \approx \frac{1}{2}(\langle u' \rangle^2 + \langle v' \rangle^2) \]
\[ l = \text{mixing length} \]

\[ M = \text{Mach number} \]
\[ p = \text{static pressure} \]
\[ p_t = \text{total pressure} \]
\[ q = \text{dynamic pressure} \]
\[ R_c = \text{streamline radius of curvature} \]
\[ Re = \text{Reynolds number based on} \ H \text{and} U_{ref} \]
\[ r = \text{radial dist. from center of curvature} \]
\[ s = \text{down. dist. from channel entrance on duct c/l} \]
\[ U, V = \text{longitudinal, vertical mean velocities} \]
\[ u, v = \text{longitudinal, vertical inst. velocities} \]
\[ u', v' = \text{long., vert. inst. turb. vel. fluctuations} \]
\[ u''v'' = \text{inst. turbulent Reynolds stress} \]
\[ u_r = \text{wall friction velocity} = \sqrt{\tau_w/\rho} \]
\[ u^+ = \text{dim. velocity} = U/u_r \]
\[ x, y = \text{x, y coordinates} \]
\[ y_i = \text{distance from wall} \ i \text{in mixing length equation} \]
\[ y_i^+ = \text{dim. wall variable for wall} \ i = y_i u_r/\nu \]
\[ \delta = \text{boundary layer thickness} \]
\[ e = \text{turbulent energy dissipation rate} \]
\[ \theta = \text{angle into bend measured from bend entrance} \]
\[ K = \text{von Karman's constant} = 0.4 \]
\[ \nu = \text{fluid kinematic viscosity} \]
\[ \nu_t = \text{turbulent or eddy kinematic viscosity} \]
\[ \rho = \text{fluid density} \]
\[ \tau_w = \text{turbulent shear stress} = -\rho <uv> \]
\[ \omega = \text{mean wall shear stress or skin friction} \]
\[ \langle > = \text{RMS time average} \]

Subscripts

i, o = inner, outer walls
r, \theta = radial, tangential dir. in cyl. coord.
ref = ref. conditions
s, n = parallel, normal dir. in stream. coord.
I. Introduction

Computational fluid dynamics (CFD) is coming to play an increasingly major role in the initial design or verification of the internal flow in rocket engine components. For example, a three-dimensional incompressible Navier-Stokes code (INS3D) was recently used to guide a possible redesign of the hot gas manifold for the Space Shuttle Main Engine (SSME) powerhead. Changing from a three-elliptical-duct to a new two-elliptical-duct configuration greatly improved flow through the new powerhead as confirmed by experiment.1,2

In spite of the above success, before CFD can be applied widely and with sufficient confidence for future rocket engine design, the codes must be calibrated and verified by comparing them against well-documented experimental data. Most engine components operate at very high pressures and flow Reynolds numbers (Re's), so the flows are fully turbulent. Thus the accuracy of the codes is critically dependent on the turbulence models that they contain. Unfortunately, most turbulence models have been developed and verified only for flows with very mild extra strain rates. In contrast, the flows in rocket engine components may be subjected to very large extra strain rates or other perturbations. It is clear that knowledge on the structure of internal shear layers subject to such effects is crucial for successful numerical simulation of these flows. The accuracy of any turbulence model proposed for internal flow calculations should first be evaluated by comparing calculations using it with well-documented internal flow experiments containing one or more of the expected elements.

To date, few experimental studies of such internal flows have been reported. There are at least two reasons for this. First, CFD has only recently been applied to flows in rocket engines, so there has been little need for code verification experiments. Second, detailed measurements in rocket engines is usually quite difficult because of limited probe and/or optical access. One of the more widely-studied extra strain effects on internal flows is that arising from streamline curvature. There have been many such studies on the effects of mild curvature, and Bradshaw' gives a comprehensive review of them. However, information on the effects of strong curvature and other large extra strain rates is very much needed.

Four recent studies have begun to examine both experimentally and theoretically some of the important effects present in strongly-curved internal flows. These studies investigate the internal flow in 180° turnaround or U-ducts (TAD's), where the radius ratio of bend centerline radius to duct height is of order unity. Such a geometry closely simulates many of the important features of the flow in the TAD of the SSME powerhead, such as the presence of strong curvature and pressure gradients, unsteady separation and interacting shear layers on opposite walls.

In one study, Sharma et al.3 measured the flow using hot wires in an axisymmetric TAD air tunnel with \( M = 0.1 \) and \( Re = 10^6 \). (For reference, the flow in the SSME TAD is at \( M = 0.1 \) and \( Re = 10^7 \).) Chang and Kwak4 calculated this flow using INS3D with a simple mixing length turbulence model. Both experiment and theory indicated a small amount of separation on the inner (convex) wall near the end of the bend. Poor agreement was found for the velocities near the outer (concave) wall and for the turbulent shear stresses throughout the bend, however.

In a second study, Sandborn5 measured the flow using a one-component laser Doppler velocimeter (LDV) in a two-dimensional (2D) TAD water tunnel with \( U_{\infty} \approx 2 \) m/s, and \( Re = 7 \times 10^4 \) to \( 5 \times 10^5 \). The results showed no inner wall separation at the lowest \( Re \) and greater separation than did Sharma et al.3 at the highest \( Re \). Chen and Sandborn6 calculated this flow using a Navier-Stokes (NS) solver and the two-equation \( k-\varepsilon \) turbulence model both in its standard high \( Re \) form7, and also with a curvature correction.9 Few conclusions about the accuracy of the models could be drawn, however, since the calculations proceeded only to the 180° bend location. To that point the flow field is mainly pressure-driven and turbulence models are of less importance.

In a third study, Avva et al.10 calculated the TAD flow of Sandborn5 at a \( Re = 9 \times 10^4 \) using the standard \( k-\varepsilon \) model in both its high and low \( Re \) versions. At that \( Re \), Sandborn's data showed only a small amount of separation on the inner wall at the bend exit. The low \( Re \) \( k-\varepsilon \) model predicted this, whereas the high \( Re \) model predicted no separation. In contrast, and to the puzzle of the authors, the high \( Re \) model predicted the measured static pressure very well and the low \( Re \) one underpredicted it.

Finally, in a fourth study, Monson et al.11 measured the flow using a two-component LDV in a 2D TAD air tunnel with \( M = 0.1 \) and \( Re = 10^5 \) and \( 10^6 \). The measurements showed a small amount of inner wall separation at the lower \( Re \), and a much larger amount at the higher \( Re \). The flow was calculated using INS3D and a simple mixing length model. Poor agreement was found with the experiment in several respects. The static pressure drop in the bend was badly underpredicted, no separation was predicted at the higher \( Re \), and the computed velocities near both walls were in error through most of the bend and further downstream. It appeared that the turbulence model was not reproducing the experiment very well, but no conclusions as to the cause of the failure could be drawn from the comparisons that were made.

Because of the generally poor agreement found by the above studies between theory and experiment for TAD-type flows, it is clear that improved turbulence models will be required to accurately compute them.
ther downstream are much too low. For the outer wall (Fig. 9b), the model indicates separation at the bend entrance, whereas none was measured. The model predicts very low $C_f$ levels throughout the bend (as do all the models). Downstream $C_f$ is also underestimated. The net result of the low $C_f$ for the m.l. model is a prediction of too high a static pressure on the inner wall downstream of the bend (Fig. 10a). Note that the outer-wall pressure in the bend (Fig. 10b) is well-predicted by this and all of the other models. At $Re = 10^6$, the m.l. model underpredicts $C_f$ on both walls (Fig.'s 9c, 9d) to an even greater extent than at the lower $Re$. Also, attached flow is indicated whereas the experiment shows increased separation. The net result is, once again, too high a static pressure estimate downstream of the bend (Fig. 10c). It thus appears as though this zero-equation model, and others that employ the van Driest inner-scale mixing length (Eqn. 3), are not well-suited to calculating this TAD-type of flow.

Turning to the $C_f$ results for the two standard $k-\epsilon$ models at $Re = 10^5$ in Fig. 9a, one can see that the low $Re$ version (std. CH) shows early separation and later reattachment as compared to the experiment. (Note that the “waviness” of $C_f$ predicted by the CH model and other ITW models was also seen by Patel et al.\textsuperscript{17} for b.l. flow in the presence of adverse pressure gradients.) A large overshoot in $C_f$ then occurs. This is typical behavior of the standard Chien low $Re$ model in reattachment regions.\textsuperscript{30} The high $Re$ model predicts separation length better, but undershoots measured $C_f$ further downstream. For the outer wall (Fig. 9b), the CH model agrees quite well with measured downstream values, whereas the high $Re$ model overpredicts the experiment. The net result for static pressure for these models (Fig. 10a), is that the CH version predicts too low a pressure downstream of the bend, and the WF version agrees with experiment. For $Re = 10^6$, both standard models indicate almost no separation (Fig.9c), which is opposite the experimental $Re$ trend. Downstream of the bend, the WF version predicts $C_f$ well on the inner wall, and the ITW version agrees with experiment on the outer wall. The net result is that both versions predict inner-wall pressure (Fig. 10c) slightly higher than the measurements. It may be concluded that, although the standard $k-\epsilon$ models predict the TAD flow better than the m.l. model, neither version consistently predicts measured $C_f$, $C_p$ and extent of separation at both $Re$ tested. Recent calculations (not included in this paper) of this TAD flow using the “LB1\textsuperscript{17}” low $Re$ model rather than the CH model in std. $k-\epsilon$ show much improved $C_f$ and $C_p$ predictions. The prediction of separation extent is not improved, however.

The next models to be considered for their $C_f$ and $C_p$ behavior are the two versions of the curvature-corrected $k-\epsilon$ model. Recall that the goal of these models is to decrease TKE and TSS on the convex wall to be more consistent with experimental observations. It is obvious from the $C_f$ data in Fig.’s 9a and 9c that the models grossly overcorrect for such effects. Much too large an extent of separation is predicted at both $Re$. Following reattachment, both models also suffer from the same problems as did the standard models (i.e., an overshoot of $C_f$ for the ITW version and an undershoot for the WF version). The net result is a prediction of too much downstream pressure loss for both versions at both $Re$ (Fig.’s 10a and 10c). These results indicate that curvature corrections developed for flows with small curvature don’t always apply to flows with large curvature like the present TAD-type of flow. Thus, the curvature $k-\epsilon$ model of Park and Chung\textsuperscript{25} cannot be recommended for this flow.

Finally, consider the skin friction and pressure predictions of the extended $k-\epsilon$ models. At $Re = 10^5$, the ITW version indicates slightly more separation than the experiment Fig.9a), but predicts the downstream recovery of $C_f$ very well. The WF version shows much larger separation and an undershoot of downstream $C_f$. For the outer wall (Fig. 9b), the ITW model once again predicts downstream $C_f$ and the WF model shows high values. For $C_p$ (Fig. 10a), the ITW model predicts pressure exactly and the WF model shows too much pressure loss. For $Re = 10^6$, the ITW model predicts extent of separation and downstream $C_f$ almost exactly (Fig. 9c). (This is the only model tested that doesn’t show decreased separation at this $Re$.) The WF version once again shows very slow reattachment and downstream $C_f$ recovery. Both versions of this model show excellent agreement with measured pressure at this $Re$ (Fig.10d). The conclusion can be reached that the extended $k-\epsilon$ model in its low $Re$ version is the only one tested in this study that predicts most of the important measured features of this TAD-type of flow at both $Re$. The high $Re$ version using the wall function formulation of Ref. 20 does not do as well, however. Perhaps a more sophisticated wall function treatment of the separated region which includes a wake-like parameter\textsuperscript{20} in the “law-of-the-wall”, and/or better choice of near-wall grid spacing ‘n certain regions of the TAD flow, would provide improved results.

Overall, the extended $k-\epsilon$ model together with the “LB1” low $Re$ model overcomes the two main shortcomings of the standard CH $k-\epsilon$ model for this TAD flow: 1) It reduces TKE and TSS on the convex wall so that separation is predicted at both $Re$; and 2) It eliminates the large $C_f$ overshoot downstream of reattachment. Even though it is not developed specifically for curved flows, the extended LB $k-\epsilon$ model seems to capture most of the important features of this flow and yet retains complete generality. Other turbulence models that reduce the level of turbulent stresses near the convex wall in the bend could perhaps do equally well when combined with the “LB1” low $Re$ model.
V. Summary and Conclusions

A joint experimental and computational study of the flow in a two-dimensional U-duct with very strong curvature has been carried out. The significant conclusions of this study are as follows:

1. The experiment shows: a) significant turbulence enhancement on the outer (concave) wall consistent with a previously-observed curvature instability mechanism; b) almost total destruction of turbulence on the inner (convex) wall consistent with previous studies of flows with much less curvature; c) separation on the inner wall at the bend exit that increases with Reynolds number; and d) extreme core turbulence downstream of the bend that creates linear "plug-flow" type velocity profiles.

2. The mixing length model of Patankar et al.\textsuperscript{22} and the standard and curvature-corrected $k - \epsilon$ models used with the low Reynolds number model of Chien\textsuperscript{16}, failed to predict this flow and the trends with Reynolds number.

3. The extended $k - \epsilon$ turbulence model of Chen and Kim\textsuperscript{24}, combined with the low Reynolds number model of Lam and Bremhorst\textsuperscript{19}, was the only turbulence model of those evaluated that predicted measured extent of separation, skin friction and static pressure throughout all regions of the flow at both low and high Reynolds number. The good prediction of skin friction downstream of the bend on the concave wall side may mean that modeling the details of that complex flow in the bend itself may not be required.

4. Application of wall functions as formulated by Launder and Spalding\textsuperscript{20} to the above extended model produced poor results. More sophisticated treatment of the separated region and better choice of near-wall grid spacing in some regions of the flow may be required to improve predictions using wall functions.

References


Fig. 1 Coordinate system for Ames HRC I turnaround duct.

Fig. 2a Inlet longitudinal velocity in turnaround duct, $M_{ref} = 0.1$, $x/H = -4$.

Fig. 2b Inlet turbulent kinetic energy.

Fig. 3a Longitudinal velocity in turnaround duct, $Re = 10^5$, $\theta = 0^\circ$.

Fig. 3b $Re = 10^5$. 

12
Fig. 4a Longitudinal velocity in turnaround duct, \( Re = 10^6 \), \( \theta = 90^\circ \), (legend as in Fig. 3a).

Fig. 4b Longitudinal velocity in turnaround duct, \( Re = 10^6 \).

Fig. 4c Turbulent kinetic energy.

Fig. 4d Turbulent shear stress.

Fig. 4e Turbulent shear stress near convex wall.

Fig. 5a Longitudinal velocity in turnaround duct, \( Re = 10^6 \), \( \theta = 180^\circ \), (legend as in Fig. 3a).
Fig. 5b $Re = 10^5$.

Fig. 5c Turbulent kinetic energy.

Fig. 6a Longitudinal velocity in turnaround duct, $Re = 10^5$, $z/H = 2$, (legend as in Fig. 3a).

Fig. 6b $Re = 10^5$.

Fig. 6c Turbulent kinetic energy.

Fig. 6d Turbulent shear stress.
Fig. 7a Longitudinal velocity in turnaround duct, \( Re = 10^8 \), \( z/H = 4 \), (legend as in Fig. 3a).

Fig. 8a Longitudinal velocity in turnaround duct, \( Re = 10^8 \), \( z/H = 12 \), (legend as in Fig. 3a).

Fig. 8b \( Re = 10^6 \).

Fig. 8c Turbulent kinetic energy.
Fig. 9d Outer wall.

Fig. 10a Static pressure coefficient on turnaround duct inner wall, \( Re = 10^4 \), (legend as in Fig. 3a).

Fig. 10b Outer wall.
Fig. 10c Inner wall, Re = 10^6.

Fig. 10d Outer wall.
Flow over a Rearward Facing Step

Scott O. Kjelgaard
Experimental Methods Branch
Fluid Mechanics Division

Second NASA CFD Validation Workshop
NASA - Lewis Research Center
July 10-12, 1990
Outline

- Objective/Approach
- Experimental Set-up
- Global Flowfield Measurements
- Detailed Profiles
- Concluding Remarks
Langley Investigation

Instrumentation Techniques used:
- 3-component Laser Velocimeter
- 2-component hot wire
- Pitot pressure surveys
- 4th LV component for measuring freestream velocity

Measurements to be made:
- Mean Velocity
- Fluctuating Velocities
- Spectral Information

Measurement locations:
- Global surveys - 700 pt surveys on grid (0.5 inch spacing)
- Detailed surveys at x/H = -3, -2, -1, ..., 8, 9 (0.06 inch spacing)
- Spectral data at x/H = 0, 7
Schematic Of Rearward Facing Step Experiment

Duct made of .5 in SAR Plexiglas with glass top

Fiber optic linked 4th LV component measures freestream velocity

5 hp blower

Three-component LV traverse system
- 1 meter cube
- 10 micron resolution
LV System Description

Three-component Laser Velocimeter

- 5 watt Argon ion laser
- color separation (514.5,496.5,476.5 nm wavelengths)
- sample volume size ≈ 100 microns
- 0.8 micron polystyrene latex seed particle
- data acquired in coincidence
- orthogonal transmit optics
- receive optics backscatter at 45 degrees

Fourth-component Laser Velocimeter

- fiber-optic link using 488 nm wavelength
Mean Flowfield Characteristics

Cross-flow Velocity Vectors

Streamlines

x/H

July 1990
Rearward Facing Step Investigation

U COLOR CONTOUR PLOT

V COLOR CONTOUR PLOT

W COLOR CONTOUR PLOT
Mean Velocity Profile at $x/H = 3$
Normal Stress Profile at x/H = 3

\[ z, \text{ inches} \]

\[ \text{Velocity}^2/ U_\infty^2 \]
Shear Stress Profile at $x/H = 3$

$z$, inches

$\text{Velocity}^2/ U_\infty^2$

- $uv$
- $-uw$
- $vw$
Rearward Facing Step Investigation

ref. vector = 0.500

Mean u

ref. vector = 0.250

Mean v

ref. vector = 0.250

Mean w
Rearward Facing Step Investigation

Mean \( uu \)

Mean \( vv \)

Mean \( ww \)
Rearward Facing Step Investigation

ref. vector = 0.015

-Mean uw

Mean uv

Mean vw
Concluding Remarks

An investigation of the flow over a rearward facing step is currently underway.

- A facility was constructed to meet the requirements of the DNS
- Three-component LV data has been acquired throughout the flow documenting
  - Mean Velocities
  - Full Reynolds stress tensor
- Preliminary review of the data by CFD personnel has expanded the scope to include additional Reynolds numbers and ER
- Next phase of the test should begin Jan. 91
LeRC/IFMD INLET DUCT AND NOZZLE HIGH SPEED VALIDATION EXPERIMENTS

• VALIDATION WORKSHOP EXPERIMENTS
  - CROSSING SHOCKS/BOUNDARY-LAYER INTERACTION
  - LeRC/UNIVERSITY UNSTEADY SHOCK/BOUNDARY-LAYER INTERACTION
  - VORTEX GENERATORS

• ADDITIONAL VALIDATION EXPERIMENTS
  - HIGH SPEED MIXING
  - TRANSITION DUCTS
EXPERIMENTS IN 3-D FLUID MECHANICS

HYPERMIXING       TRAN. DUCT       VORTEX GEN

CROSSING SHOCK    UNSTEADY SHOCK

QUANTITATIVE DATA
QUALITATIVE DATA
NASP-HYPERMIXING CONCEPTS
EXPERIMENTAL INVESTIGATION IN 1×1 SWT

TUNNEL FLOW

INJECTOR FLOW

EXPERIMENTAL CONFIGURATION
BASELINE MODEL

OIL FLOW VISUALIZATION

CD-89-44432
NASP-HYPERMIXING CONCEPTS
TRACE GAS MEASUREMENTS

MASS FRACTION—MACH 1.6
MASS FRACTION—MACH 3.0
NASP-HYPERMIXING CONCEPTS
MACH NUMBER CONTOURS

TUNNEL MACH NUMBER - 1.6
TUNNEL MACH NUMBER - 3.0
\[(y/a)^\eta + (z/b)^\eta = 1\]

CIRCULAR-TO-RECTANGULAR TRANSITION DUCT
Turbulence kinetic energy contours

\[(k/U_p^2) \times 10^4\]
FIGURE 1 - Details of the test section.
FIGURE 2 - Details of the vortex generator mounting station.
Vortex array embedded in a boundary layer. Top picture is the data taken at Mach .2. Bottom picture shows a model of the data composed of Oseen vortices with vortex core locations, circulations, and peak vorticities corresponding to the data results.
• UNIVERSITY OF TEXAS AT AUSTIN
  MACH 5 (DOLLING)

• PRINCETON UNIVERSITY
  MACH 3 (SMITS)

• LeRC 1X1 SWT
  MACH 3 AND 5 (BARNHART, HINGST)

• FLIGHT TESTING?
Dynamics of Shock Wave
Turbulent Boundary Layer Interactions
Induced by Cylinders and Blunt Fins

Principal Investigator: D. S. Dolling, Aerospace Engineering and Engineering Mechanics, The University of Texas at Austin

Graduate Research Asst.: Leon Brusniak

Sponsor: NASA Lewis Research Center (Warren Hingst)
CROSS-CORRELATION

Stations 1 and 2

Stations 2 and 3

Stations 3 and 4

Stations 1 and 4
Ensemble-Averaged Wall Pressure Distribution on Centerline Upstream of Cylinder (D = 1/2")

Open Symbols:  Shock-forward, $\gamma \approx 0.2$
Closed Symbols:  Shock-back, $\gamma \approx 0.8$
Surface Flow Pattern

RMS Distribution on Centerline
Flow Conditions:
\[ M_{ae} = 2.9 \]
\[ T_0 = 265 \, ^\circ\text{OK} \]
\[ U_e = 575 \, \text{m/s} \]
\[ Re_e/m = 6.5 \times 10^7 \]
Blunt fin diameter: 0.75"
Boundary layer thickness: 1.1"

Visualization:
* Nd:YAG laser with a fourth harmonic generator operating at 266 nm (far UV), 4 ns pulses at 10 Hz.
* Images obtained with an intensified, UV sensitive CID camera.
* Flow is from right to left
Rayleigh scattering

* In air flows: small ice clusters (30 nm) can dominate the Rayleigh signal. Yields high intensity signals in regions of low temperature, and signal dropout in regions of high temperature. A strong shock will appear as the boundary between a bright region of low density fluid and a dark region of high density fluid (the reverse of the expected result).

* In nitrogen flows: the Rayleigh signal is now dominated by scattering from nitrogen molecules. A shock will appear as the boundary between a darker region of low density fluid and a brighter region of high density fluid (as expected).
Rayleigh Scattering:

* scattering by particles much smaller than the wavelength of light.

* The intensity of the signal is proportional to the density of the scattering centers - gives the possibility of quantitative density maps.

* Can reveal instantaneous turbulence and shock structure in a plane.

* Sensitivity proportional to the fourth power of frequency, so operation at short wavelengths very beneficial.
Blunt Fin Interaction Static Pressures

Mach Number
- 2.0
- 2.5
- 3.0
- 3.5
- 4.0

\[ \frac{P}{P_0} \text{ vs. } \frac{X}{D} \]
CROSS SHOCK NORMALIZED STATIC PRESSURE

DISTANCE FROM CENTERLINE (in inches)

DISTANCE FROM THE START OF THE TEST SECTION (in inches)

$\alpha = 10^\circ$, $M = 3.5$
CENTERLINE STATIC PRESSURE

\[ \begin{align*}
\alpha &= 4 \text{ deg.} \\
\alpha &= 6 \text{ deg.} \\
\alpha &= 8 \text{ deg.} \\
\alpha &= 10 \text{ deg.} \\
\alpha &= 12 \text{ deg.}
\end{align*} \]

NORMALIZED STATIC PRESSURE

DISTANCE FROM THE START OF THE TEST SECTION (in inches)

\[ M = 3.5 \]
NASA LOW SPEED CENTRIFUGAL COMPRESSOR

SECOND NASA CFD VALIDATION WORKSHOP

Lewis Research Center
July 10-11, 1990

MICHAEL D. HATHAWAY
NASA LOW SPEED CENTRIFUGAL COMPRESSOR

- OBJECTIVES
- COMPRESSOR PROGRAM
- DESCRIPTION OF FACILITY AND COMPRESSOR
- INSTRUMENTATION
- CFD EFFORT & IMPACT ON RESEARCH PROGRAM
- CFD/EXPERIMENT PRELIMINARY COMPARISONS
- SUMMARY
RESEARCH OBJECTIVES

- Improve understanding of flow in centrifugal compressors

- Provide data for modeling of various flow phenomena

- Acquire "benchmark" data for 3-D viscous code validation
CENTRIFUGAL COMPRESSOR PROGRAM

COMPUTATIONAL METHODS DEVELOPMENT
- FULL 3-D VISCOUS
- AVERAGE PASSAGE

HIGH-SPEED CENTRIFUGAL COMPRESSOR EXPERIMENTS

LOW-SPEED CENTRIFUGAL COMPRESSOR EXPERIMENTS
LOW SPEED CENTRIFUGAL COMPRESSOR

BENCHMARK EXPERIMENTS FOR COMPUTATIONAL CODES
### NASA LOW SPEED CENTRIFUGAL COMPRESSOR

#### DESIGN CHARACTERISTICS

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air Flow</td>
<td>66 lbm/sec</td>
</tr>
<tr>
<td>Rotative Speed</td>
<td>1920 RPM</td>
</tr>
<tr>
<td>Inlet Pressure</td>
<td>14.7 psia</td>
</tr>
<tr>
<td>Pressure Ratio</td>
<td>1.18</td>
</tr>
<tr>
<td>Outlet Temp.</td>
<td>85 F</td>
</tr>
<tr>
<td>Exhaust Altitude</td>
<td></td>
</tr>
</tbody>
</table>
LOW SPEED CENTRIFUGAL COMPRESSOR

COMPRESSOR INSTRUMENTATION

TRACING PROBES

TRANSUDER TOWER

FLOW

ROTATING FEEDTHROUGH
NASA LOW SPEED CENTRIFUGAL COMPRESSOR INSTRUMENTATION

- 465 STATIC PRESSURES (208 ON ROTOR)
- 83 TOTAL PRESSURES (75 ON ROTOR)
- 51 TOTAL TEMPERATURES (27 ON ROTOR)
- 8 FLOW ANGLE PROBES
- 28 STRAIN GAUGES
- 6 PROXIMITY PROBES
- LASER ANEMOMETRY
CENTRIFUGAL COMPRESSOR PROGRAM

COMPUTATIONAL METHODS DEVELOPMENT
- FULL 3-D VISCOUS AVERAGE PASSAGE

HIGH-SPEED CENTRIFUGAL COMPRESSOR EXPERIMENTS

LOW-SPEED CENTRIFUGAL COMPRESSOR EXPERIMENTS
OBJECTIVE:

Perform CFD analysis in order to provide guidance for experimental measurements:
- develop a "feel" for the flow physics
- aid in planning location/extent of measurements
- on-line assessment of CFD limitations

APPROACH:

3D steady Navier-Stokes analysis of impeller only
- VPI&SU grant
- in-house

3D steady Average-Passage analysis of impeller+diffuser

STATUS:

VPI&SU impeller analysis complete
In-house impeller analysis in progress
Average-passage stage analysis in progress
3D NAVIER STOKES CODE
ANALYSIS OF LARGE LOW-SPEED CENTRIFUGAL IMPELLER

PRESSURE DRIVEN SECONDARY FLOWS
NEAR BLADE SUCTION SURFACE

Radial Distance

Axial Distance
3D NAVIER STOKES CODE
ANALYSIS OF LARGE LOW-SPEED CENTRIFUGAL IMPELLER

PRESSURE DRIVEN TIP CLEARANCE FLOW

Diagram showing the flow analysis of a large low-speed centrifugal impeller with pressure-driven tip clearance flow.
LOW SPEED CENTRIFUGAL COMPRESSOR DIFFUSER SHROUD
66 lbm/sec
LOW SPEED CENTRIFUGAL COMPRESSOR

3-D NS CODE PREDICTION V.S. EXPERIMENT

3-D NAVIER STOKES ANALYSIS
PARTICLE TRACES

IMPELLER EXIT
RADIAL VELOCITY DISTRIBUTION

\[ V_r \text{ (fps)} \]

\[ R = 37.745 \text{ in} \]

\[ R = 32.000 \text{ in} \]

\[ \circ \text{ DATA} \]

\[ \text{CFD} \]
LOW SPEED CENTRIFUGAL COMPRESSOR
DIFFUSER HUB
66 lbm/sec
LeRC COMPRESSOR FLOW PHYSICS
EXPERIMENT SCHEDULE

<table>
<thead>
<tr>
<th></th>
<th>90</th>
<th>91</th>
<th>92</th>
<th>93</th>
<th>94</th>
<th>95</th>
</tr>
</thead>
</table>

Low Speed Compressor Facility - IFMD

Centrifugal

Rotor only

Full stage

Multistage Axial

4-stage axial compressor

Small High Speed Compressor Facility - PSD

Centrifugal compressor

3-stage axial compressor

LSA-363
SUMMARY

- LOW SPEED CENTRIFUGAL COMPRESSOR FACILITY
  ✓ Improve understanding of complex flows
  ✓ Data for flow physics modeling
  ✓ "Benchmark" data for code assessment

- CFD ANALYSIS
  ✓ Develop feel for flow physics
  ✓ Aid in planning location/extent of measurements
  ✓ On-line assessment of CFD limitations

- HARDWARE MODIFICATIONS
  ✓ 2% Tip clearance
  ✓ Diffuser wedge plate

- MILESTONES
  ✓ Laser anemometer surveys LSCC, mid Sept.
  ✓ High speed centrifugal, LFA surveys, 1990
  ✓ Low speed axial compressor, May 1991
  ✓ Impeller/Diffuser LSCC, 1993
MSFC CFD VALIDATIONS - CURVED DUCT AND TURBOMACHINERY FLOWFIELD ANALYSES

SECOND NASA CFD VALIDATION WORKSHOP
NASA LEWIS RESEARCH CENTER

PREPARED BY: LISA W. GRIFFIN, ROBERT WILLIAMS, AND PAUL MCCONNAUGHEY
COMPUTATIONAL FLUID DYNAMICS BRANCH
AEROPHYSICS DIVISION
STRUCTURES AND DYNAMICS LABORATORY
OVERVIEW

- DUCT AND PIPE BEND FLOW CODE VALIDATION
- 90° DUCT BEND
  - OBJECTIVES
  - APPLICATION CONFIGURATION
  - CODE METHODOLOGY
  - RESULTS
  - COMPUTER RESOURCE REQUIREMENTS
  - CONCLUSIONS
- TURBOMACHINERY CODE VALIDATION
- TURBINE CASCADES
  - OBJECTIVES
  - APPROACH
  - APPLICATIONS CONFIGURATIONS
  - CODE METHODOLOGY
  - RESULTS
  - COMPUTER RESOURCE REQUIREMENTS
  - CONCLUSIONS
## Duct and Pipe Bend Flow Code Validation

<table>
<thead>
<tr>
<th>CODE</th>
<th>Application</th>
<th>Principal Investigators</th>
</tr>
</thead>
</table>
| INS3D    | * 90° Duct Bend  
S-Shaped Duct Bend  
180° Turnaround Duct | Paul McConnaughey,  
Joni Cornelison |
| INS3DLU  | * 90° Duct Bend  
90° Pipe Bend | Robert Williams |
| FDNS3D   | * 90° Duct Bend  
90° Pipe Bend | Robert Williams |
OBJECTIVE

ASSESS CODE CAPABILITY TO PREDICT COMPLEX SECONDARY FLOWS AND ASSOCIATED AXIAL FLOW VARIATION
APPLICATION CONFIGURATION

- NINETY DEGREE DUCT BEND WITH CONSTANT, SQUARE CROSS SECTION (TAYLOR, WHITELAW, AND YANNESKIS)
  - RADIUS RATIO - 2.3
  - REYNOLDS NUMBER - 790
  - DEAN NUMBER - 368
- LASER-DOPPLER VELOCIMETER DATA
  - STREAMWISE AND GAPWISE VELOCITY COMPONENTS
  - 8 STREAMWISE STATIONS
  - 50 MEASUREMENTS PER STATION
CODE METHODOLOGY

- **INS3D**
  - Incompressible Navier-Stokes Code
  - Beam-Warming / Briley McDonald Algorithm
  - Artificial Compressibility
  - 2nd or 4th Order Numerical Dissipation Model

- **INS3DLU**
  - Incompressible Navier-Stokes Code
  - Lower-Upper Symmetric-Gauss-Sidel Implicit Scheme
  - Artificial Compressibility
  - 3rd Order Numerical Dissipation Model

- **FDNS3D**
  - Unsteady Reynolds-Averaged Navier-Stokes Code
  - Pressure-Based Predictor-Corrector PISO Type Algorithm
  - Variable Order Upwind Dissipation
  - K-ε Turbulence Model for Turbulent Flows
  - Equilibrium/Finite Rate Chemistry for Reactive Flows
Streamwise Velocity Contours at the Plane 0.25 H Past the 90° Bend Exit.
Gapwise Velocity Contours at the Plane 0.25 H Past the 90° Bend Exit.
Predicted Radial Flow in 90° Bend Compared with the Data of Taylor et al.
Predicted Axial Flow In 90° Bend Compared with the Data of Taylor et al.⁹
Contour Interval of Plots is 0.08.
STREAMWISE VELOCITY PROFILES - INS3DLU

0.25 H Past the 90° Bend Exit
GAPWISE VELOCITY PROFILES - INS3DLU

22,500 GRID POINTS

49,000 GRID POINTS

85,000 GRID POINTS

0.25 H Past the 90° Bend Exit
Streamline Velocity Contours - INS3DUL

x = -.25 H

Theta = 60 degrees

x = .25 H

x = 2.5 H
GAPWISE VELOCITY PROFILES - FDNS3D

0.25 H Past the 90° Bend Exit
COMPUTER RESOURCE REQUIREMENTS

* ALL CALCULATIONS WERE RUN ON THE CRAY XMP-416 AT MSFC

- INS3D
  - 17 WORDS/GGRID POINT OF MEMORY
  - 25.0 MICRO SEC/GGRID POINT/ITERATION OF CPU TIME
  - REQUIRED 3.0 MW OF MEMORY AND 1.2 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 1000 ITERATIONS

- INS3DLU
  - 74 WORDS/GGRID POINT OF MEMORY
  - 11.5 MICRO SEC/GGRID POINT/ITERATION OF CPU TIME
  - REQUIRED 6.1 MW OF MEMORY AND .14 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 500 ITERATIONS

- FDNS3D
  - 46 WORDS/GGRID POINT OF MEMORY
  - 127.4 MICRO SEC/GGRID POINT/ITERATION OF CPU TIME
  - REQUIRED 3.7 MW OF MEMORY AND 3.0 CPU HOURS FOR THE CALCULATION WITH THE DENSEST GRID IN THIS STUDY BASED ON 1000 ITERATIONS
SUMMARY/CONCLUSIONS

• VALIDATION OF INS3D, INS3DLU, AND FDNS3D FOR LAMINAR FLOW THROUGH A 90° DUCT BEND HAS BEEN COMPLETED

• AGREEMENT BETWEEN PREDICTED AND MEASURED AXIAL FLOW WAS GOOD FOR EACH CODE (PROVIDED THE GRIDS WERE DENSE ENOUGH TO ADEQUATELY RESOLVE ALL FLOW COMPONENTS).

• FOR ADEQUATE FLOW RESOLUTION, INS3DLU AND FDNS3D REQUIRED GRIDS OF 85,000 GRID POINTS. INS3D REQUIRED 100,000 TO 175,000 GRID POINTS.

• INS3DLU SHOWS A MARKED IMPROVEMENT OVER INS3D IN COMPUTATIONAL SPEED. FDNS3D, A MORE GENERAL CODE, IS THE LEAST EFFICIENT IN TERMS OF COMPUTATIONAL SPEED OF THE THREE CODES.
<table>
<thead>
<tr>
<th>CODE</th>
<th>APPLICATION</th>
<th>PRINCIPAL INVESTIGATORS</th>
</tr>
</thead>
<tbody>
<tr>
<td>MTSBL/STAN5</td>
<td>UTRC LSRR FIRST STAGE</td>
<td>HELEN MCCONNAUGHEY</td>
</tr>
<tr>
<td></td>
<td>SSME HPFTP (FPL) FIRST STAGE</td>
<td>JOE RUF</td>
</tr>
<tr>
<td></td>
<td>SSME HPFTP TTA FIRST STAGE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(MSFC AND CALSPAN TESTS)</td>
<td></td>
</tr>
<tr>
<td>FDNS3D</td>
<td>SSME HPFTP TTA FIRST STAGE</td>
<td>HELEN MCCONNAUGHEY</td>
</tr>
<tr>
<td></td>
<td>(MSFC TESTS)</td>
<td></td>
</tr>
<tr>
<td>ROTOR1</td>
<td>UTRC LSRR FIRST STAGE</td>
<td>LISA GRIFFIN</td>
</tr>
<tr>
<td></td>
<td>SSME HPFTP (FPL) FIRST STAGE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SSME HPFTP TTA FIRST STAGE</td>
<td>LISA GRIFFIN</td>
</tr>
<tr>
<td></td>
<td>(MSFC TESTS)</td>
<td>HELEN MCCONNAUGHEY</td>
</tr>
<tr>
<td>ROTOR3</td>
<td>KOPPER'S CASCADE</td>
<td>LISA GRIFFIN/PRATT &amp; WHITNEY</td>
</tr>
<tr>
<td></td>
<td>HODSON'S CASCADE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>* UTRC LSRR FIRST STATOR</td>
<td>LISA GRIFFIN/PRATT &amp; WHITNEY</td>
</tr>
<tr>
<td></td>
<td>* LANGSTON'S CASCADE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>GGGT FIRST STATOR</td>
<td>LISA GRIFFIN/PRATT &amp; WHITNEY</td>
</tr>
<tr>
<td></td>
<td>SSME HPFTP TTA FIRST STAGE</td>
<td></td>
</tr>
<tr>
<td></td>
<td>(CALSPAN TESTS)</td>
<td>LISA GRIFFIN</td>
</tr>
<tr>
<td></td>
<td>UTRC LSRR FIRST ROTOR</td>
<td>LISA GRIFFIN</td>
</tr>
<tr>
<td></td>
<td>HEAT TRANSFER</td>
<td>LISA GRIFFIN</td>
</tr>
</tbody>
</table>
INTRODUCTION/OBJECTIVES

- OVERALL OBJECTIVE - VERIFICATION OF THE UNSTEADY, 3D, VISCOUS FLOW PREDICTION CAPABILITIES OF THE ROTOR3 CODE

- PHASE I OBJECTIVE - VERIFICATION OF THE PERFORMANCE AND HEAT LOAD PREDICTION CAPABILITIES OF ROTOR3 THROUGH COMPARISON WITH BENCHMARK DATA

- PHASE II OBJECTIVE - VERIFICATION OF THE UNSTEADY FLOW PREDICTION CAPABILITIES OF ROTOR3 THROUGH COMPARISON WITH UNSTEADY FLOW DATA

- PHASE III OBJECTIVE - PREDICTION OF THE FLOWFIELD IN A ROCKET ENGINE TURBINE (SSME OR ADVANCED TURBINE) WITH ROTOR3 AND COMPARISON OF THE PREDICTION WITH EXPERIMENTAL DATA
APPROACH

• MODIFICATIONS WERE MADE TO ROTOR3 TO
  • ENHANCE THE CODE'S CAPABILITIES
  • DECREASE THE AMOUNT OF RUN TIME NECESSARY FOR PHASE I CODE VERIFICATION

• CODE MODIFICATIONS
  • CREATION OF CASCADE VERSION OF ROTOR3 WITH WHICH TO VERIFY THE CODE'S PREDICTIVE CAPABILITIES IN STEADY FLOWS FOR WHICH DETAILED DATA IS AVAILABLE
  • INCORPORATION OF INLET BOUNDARY LAYER SPECIFICATION
  • INCORPORATION OF HEAT TRANSFER PREDICTION CAPABILITY
  • INCORPORATION OF TRANSITION LOCATION SPECIFICATION
APPLICATION CONFIGURATION

• UNITED TECHNOLOGIES RESEARCH CENTER LARGE SCALE ROTATING RIG (LSRR) FIRST STATOR
  • 3D ANNULAR CONFIGURATION
  • COMPARISON BETWEEN PREDICTIONS AND DATA USED TO VERIFY HEAT TRANSFER, SECONDARY FLOW, AND PERFORMANCE PREDICTION CAPABILITY

• LANGSTON'S CASCADE
  • 3D PLANAR CONFIGURATION
  • TWO DIFFERENT INLET BOUNDARY LAYER PROFILES
  • COMPARISONS BETWEEN PREDICTIONS AND DATA USED TO ASSESS CODE SENSITIVITY TO INLET BOUNDARY CONDITIONS
SOLUTION METHODOLOGY

• 3D UNSTEADY, THIN-LAYER, COMPRESSIBLE NAVIER-STOKES

• FACTORED, ITERATIVE, IMPLICIT ALGORITHM, THIRD-ORDER ACCURATE UPWIND DIFFERENCING SCHEME

• MODIFIED BALDWIN-LOMAX TURBULENCE MODEL, FULLY TURBULENT FLOW ASSUMED

• ADIABATIC SURFACES ASSUMED

• UNIFORM INLET CONDITIONS

• OVERLAID AND PATCHED O- AND H-TYPE GRIDS, ROTOR H-GRID SLIDES PAST STATOR H-GRID AND OUTER CASING
TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF THE LSRR FIRST STATOR

MEASURED

COARSE GRID/UNIFORM INLET

COARSE GRID/PRESCRIBED INLET BOUNDARY LAYER

REFINED GRID/PRESCRIBED INLET BOUNDARY LAYER
GAP AVERAGED TOTAL PRESSURE LOSS DOWNSTREAM OF THE LSSR FIRST STATOR

\[ C_{PTL} = \frac{P_{ts} - P_{to}}{\frac{1}{2} \rho o w^2} \]

- Coarse Grid/Uniform Inlet
- Coarse Grid/Specified Boundary Layer
- Refined Grid/Specified Boundary Layer
- Experimental Data
Stanton Numbers At The LSRR First Stator Midspan

\[ q^* = \frac{1}{2} \rho_c U_e C_p (T_a - T_w) \]

- Prediction
- Experimental Data
PREDICTED STANTON NUMBER CONTOURS FOR THE LSRR FIRST STATOR

PRESsure surface

SUCTION SURFACE

EffeCts of Secondary Flow

LeadinG edge

Trolling edge

Hub

Casing

3.4

2.8

1.2

1.0

2.2

2.2

3.2

3.25

3.20

3.15

3.10

3.05

3.00

2.95

2.90

2.85

2.80

2.75

2.70

2.65

2.60

2.55

2.50

2.45

2.40

2.35

2.30

2.25

2.20

2.15

2.10

2.05

2.00

2.00
TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF LANGSTON'S CASCADE, THIN INLET BOUNDARY LAYER
TOTAL PRESSURE LOSS THROUGH LANGSTON'S CASCADE, THIN INLET BOUNDARY LAYER

\[ C_{\text{PTL}} = \left( \frac{P_{T_1} - P_{T_2}}{\frac{1}{2} \rho u_0^2} \right) \]

- PREDICTION
- DATA

\[ z/b \]
TOTAL PRESSURE LOSS CONTOURS DOWNSTREAM OF LANGSTON'S CASCADE, NOMINAL INLET BOUNDARY LAYER

MIDSPAN

ENDWALL
TOTAL PRESSURE LOSS THROUGH LANGSTON'S CASCADE, NOMINAL INLET
BOUNDARY LAYER

\[ C_{PTL} = \left( \frac{P_T - P_0}{\frac{1}{2} \rho_0 u_0^2} \right) \]

- PREDICTION
- DATA

\[ Z/b \]

0.0 0.1 0.2 0.3 0.4 0.5
RESOURCE REQUIREMENTS

- CALCULATIONS RUN ON THE CRAY X-MP 416 AND CRAY X-MP-280

- CALCULATION REQUIREMENTS DEPENDENT UPON INLET CONDITION AND GRID DENSITY

- LSRR CALCULATIONS STARTED FROM FREESTREAM

- LSRR CALCULATION WITH REFINED GRID AND SPECIFIED INLET BOUNDARY LAYER PROFILE REQUIRED 20 CPU HOURS AND 4.0 X 10\(^8\) WORDS OF CORE MEMORY PLUS 6.4 X 10\(^6\) WORDS OF SSD

- LANGSTON’S CASCADE, THIN INLET BOUNDARY LAYER AND FINE GRID CASE, REQUIRED 4.0 X 10\(^6\) WORDS OF CORE MEMORY AND 3.7 X 10\(^6\) WORDS OF SSD. A CONVERGED SOLUTION REQUIRED 16 CPU HOURS WHEN STARTED FROM FREESTREAM CONDITIONS AND 7 CPU HOURS WHEN STARTED FROM AN EULER SOLUTION.

- LANGSTON CASCADE, NOMINAL INLET BOUNDARY LAYER AND FINE GRID CASE, REQUIRED 4.0 X 10\(^6\) WORDS OF CORE MEMORY PLUS 5.8 X 10\(^6\) WORDS OF SSD. A CONVERGED SOLUTION REQUIRED 12 CPU HOURS WHEN STARTED FROM AN EULER SOLUTION.
DISCUSSION SUMMARY

• ACCURACY OF TOTAL PRESSURE LOSS PREDICTION TIED TO GRID DENSITY AND INLET BOUNDARY CONDITION
  - OVERPREDICTION OF MIDSPAN LOSS FOR EACH CASE. OVERPREDICTION INCREASES AS SECONDARY FLOW INCREASES (TURBULENT MIXING?)
  - UNDERPREDICTION OF CASING BOUNDARY LAYER FOR ANNULAR CASCADE (CURVATURE EFFECTS?)

• PREDICTED MIDSPAN HEAT TRANSFER SHOWS EXCELLENT AGREEMENT WITH DATA. COMPUTED STANTON NUMBER CONTOURS DISPLAY FEATURES CONSISTENT WITH THE FLOW

• HIGH RESOLUTION SMOOTH RESULTS TIED TO VERY FINE SPATIAL RESOLUTION AND, CONSEQUENTLY, LONG RUN TIMES. IMPROVED INITIALIZATION, SUCH AS STARTING THE CALCULATION WITH AN EULER SOLUTION, CAN REDUCE RUN TIME CONSIDERABLY.
<table>
<thead>
<tr>
<th>Name</th>
<th>Affiliation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gordon Pickett</td>
<td>Pratt &amp; Whitney</td>
</tr>
<tr>
<td>Robert Simoneau</td>
<td>NASA Lewis</td>
</tr>
<tr>
<td>Scott Kjelgaard</td>
<td>Nasa Langley</td>
</tr>
<tr>
<td>Hugh Shin</td>
<td>GE Aircraft Engines</td>
</tr>
<tr>
<td>Don Dietrich</td>
<td>GE Aircraft Engines</td>
</tr>
<tr>
<td>Wei Tseng</td>
<td>Naval Air Development Center</td>
</tr>
<tr>
<td>Michael Crawford</td>
<td>U. of Texas, Austin</td>
</tr>
<tr>
<td>Terry Simon</td>
<td>U. of Minnesota</td>
</tr>
<tr>
<td>Mounir Ibrahim</td>
<td>Cleveland State U.</td>
</tr>
<tr>
<td>Ted Reyhner</td>
<td>Boeing Commercial Airplanes</td>
</tr>
<tr>
<td>Ed Schairer</td>
<td>NASA Headquarters</td>
</tr>
<tr>
<td>Mark Potapczuk</td>
<td>NASA Lewis</td>
</tr>
<tr>
<td>Lisa Griffin</td>
<td>NASA Marshall</td>
</tr>
<tr>
<td>Fred Gessner</td>
<td>U. of Washington</td>
</tr>
<tr>
<td>Reda Mankbadi</td>
<td>NASA Lewis</td>
</tr>
<tr>
<td>Michael Hathaway</td>
<td>NASA Lewis</td>
</tr>
</tbody>
</table>
- Our group accepted the "braunley" definition of validation.

- Validation programs
  - Backward step - A
  - Transition duct - B
  - Turn around duct - C
  - Transonic cascade - D

Improvements:

(A) Duplicate geometry
Unsteady Meas.

(B) Needs stronger CFD input

(C) Unsteady Meas

(D) Duplicate geometry (#3)

All = Better Initial Conditions

L.S.-1 7/11/90

293
- **ADDITIONAL BENEFIT**

  VALIDATION DEFINITION PROVIDED
  A GOOD VISION FOR OTHER CFD PROGRAMS
  (CAT. A, B, C)

  EXAMPLES:

  - CENTRIFUGAL COMPRESSOR PROG.
  - GENERIC GAS GENERATOR TURBINE

  ⇒ ULTIMATE GOAL IS VALIDATION
  OF CODES (i.e. CAT. D)

  ⇒ INTERMEDIATE GOAL HAS TO BE
  CALIBRATION FOR COMPLEX (ENGINEERING)
  TYPE FLOWS

  -... AND THAT'S O.K.
- Progress to date on CFD related programs is good
  - Instrumentation development
  - Increased emphasis on B.C.'s

Future needs - General

- Improved teaming
  - Experimentor
  - CFD'or
  - Fluid flow modeler

- Clearer understanding and/or quantification of validation

- Improve "rewards" at NASA for validation activity
FUTURE NEEDS — SPECIFIC

- Flows with transition affecting:
  - Boundary layers
  - Large scale flow features

[Minority position:
  "Unsteady flow" should replace
  "transition"
]

- Reacting flows for combustors
- Roughness effects
- Review inventory of existing programs directed towards CFD validation
  - Separation bubble exp. LUTEC
  - Compressor cascade (Penn State)
  - Host programs

- High "rates-of-strain" flow fields

*! Calibration — validation!! *
# Working Group Registration

## High Speed

<table>
<thead>
<tr>
<th>Name</th>
<th>Affiliation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Joe Marvin</td>
<td>NASA Ames</td>
</tr>
<tr>
<td>Jerry Wood</td>
<td>NASA Lewis</td>
</tr>
<tr>
<td>Tony Ingraldi</td>
<td>NASA Langley</td>
</tr>
<tr>
<td>Kevin Early</td>
<td>GE Aircraft Engines</td>
</tr>
<tr>
<td>Walt Sturek</td>
<td>Army Ballistic Research Laboratory</td>
</tr>
<tr>
<td>Bill Compton</td>
<td>NASA Langley</td>
</tr>
<tr>
<td>Alan Epstein</td>
<td>MIT</td>
</tr>
<tr>
<td>John Malone</td>
<td>NASA Langley</td>
</tr>
<tr>
<td>August Verhoff</td>
<td>McDonnell Aircraft Company</td>
</tr>
<tr>
<td>Essam Atta</td>
<td>GE Aircraft Engines</td>
</tr>
<tr>
<td>Frank Spaid</td>
<td>McDonnell Douglas Research Laboratory</td>
</tr>
<tr>
<td>Warren Hingst</td>
<td>NASA Lewis</td>
</tr>
</tbody>
</table>
General Observations/Comments

- CFD/FD Cooperation
  essential
  needs continued emphasis
- Strong need for standard data sets - NASA
  Flow Phenomena - critical now!
- Components
  format for archiving
  Hard Copy - classic test cases
  floppy/electronic - near term
- NASA responded positively to
  Bradley Committee recommendations
  to community: a small step, however
  Reporting of validation effort is
  recommended on periodic basis
CFD Validation - Hi Speed

Focus: Hi Speed Civil Transport
transonic → supersonic

CFD Requirements - Propulsion
Inlet
Nozzles
Turbo machinery
Afterbody
Inlets

Key Physics Issues
- Current program
- Shock-Boundary layer Interactions
- 3-d compression
- Glancing crossing

Needed -
- Boundary layer control exp's.
- Bleed, blowing...
- Unsteadiness
Nozzles

Transitions Sections

Current
round to rectangular
cross flow initial conditions

Needed
upstream conditions representing
Vorticity
unsteady flow (15%)
Thermal gradients
measurement of downstream boundary conditions

Nozzles
Current - not much
needed - jet mixing at representative
scale... Noise, shock, noise
Rotating Machinery

UNRESOLVED PROBLEMS

1. Transition & Reattachment
2. Shock - B.L. Interaction
   \( M = 1 \sim 3 \)
3. Rotating B.L.'s
4. Tip Leakage Vortices
5. Interaction of Vortices & Airfoils
6. Film Cooling
7. Unsteady 3-D Blade Row Interactions
8. Heat transfer

General Comment:
1. Most often Inlet B.L. not well characterized in exp.
2. Outlet B.C. difficult to specify
3. Geometry must be well specified

- addressed in current program
<table>
<thead>
<tr>
<th>Name</th>
<th>Affiliation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Louis A. Povinelli</td>
<td>NASA Lewis</td>
</tr>
<tr>
<td>Irwin E. Vas</td>
<td>Boeing Defense and Space Group</td>
</tr>
<tr>
<td>Michael S. Holden</td>
<td>Calspan Corp.</td>
</tr>
<tr>
<td>William K. Lockman</td>
<td>NASA Ames</td>
</tr>
<tr>
<td>I. L. Bhatley</td>
<td>General Dynamics, Fort Worth</td>
</tr>
<tr>
<td>J. Phillip Drummond</td>
<td>NASA Langley</td>
</tr>
<tr>
<td>Mani Subramanian</td>
<td>GE Aircraft Engines</td>
</tr>
<tr>
<td>Charles R. McClinton</td>
<td>NASA Langley</td>
</tr>
<tr>
<td>Eli Reshotko</td>
<td>Case-Western Reserve U.</td>
</tr>
</tbody>
</table>
Hypersonics

Procedure

- Look at CFD validation experiments currently underway related to hypersonic and re-entry vehicles.

- Identify new or additional requirements or needs.

- Provide recommendations for CFD validation activity.
Discussion broken into following areas:

- Vehicle Forebody
- Generic building block (aero) exper.
- Inlet
- Combustors
- Nozzles
- Propulsion System Integration

Additional considerations:
- Non-air-breathing propulsion vehicles.
Vehicle Forebody

All-body configuration at ARC (elliptic delta wing) has produced primarily code calibration data.

McDonnell-Douglas Generic Option 2 (Blended Wing Body) revealed significant cross-flow on underside near inlet plane, not captured by codes.
Vehicle Forebody Tests

- Experiments in number of facilities in generic set of elliptic cones with ellipsoidal bluntness to provide validation data including:
  - 3D flow field (and boundary layers)
  - Transition (3D) (Multiple mechanisms)
  - Real gas effects
  - Tunnel flow quality & availability
  - Illustrating non-intrusive measurements.
Building Block (Aero) Experiments
(Shock-wave BLI)

- 2D Compression ramp
- Glancing & crossing shock interactions (3D)
- 2D Oblique/Wall layer interaction
  - Film cooling
    - Transpiration Cooling
- Flow Insteadiness & Separation
Building Block Experiments

- Appropriate level of effort
- Insufficient detailed measurements related to turbulence modeling (fluctuating flow field measurements)
  - Reynolds stress components
  - Density fluctuations
  - Pressure fluctuations
  - Velocity fluctuations
Inlets

- Review current data base
- Consider "generic" inlet for validation experiment. (M10 underway)
- Current data base on cowl heating adequate, requires improved modeling for codes - in particular for transitional shear layers.
- Experimental study of inlet restart for guidance in modeling.
Inlets

- Mach 5 produced limited validation data; primarily calibration data.
- Mach 18. Sidewall inlet produced some flow field data (limited).
Combustors

- Experiments identified to study:
  - Wall jets (film cooling)
  - Co-axial jets (w/o reaction)
  - Swept ramp injectors
  - Vected jet injectors
  - High Mach, hi-enthalpy mixing & combustion
  - Effect of divergence on turbulence.
Combustor

- Only limited experiments funded.
- Limited diagnostic capabilities for reacting (and fluctuating) flows.
  - Turbulent
- Need measurements of:
  - Species (mean & fluctuating)
  - Temp
  - Velocity
- Structure of mixing process.
- Generic combustor shape needs to be defined.
Nozzles

1. Run current codes against existing nozzle data.

2. Grumman
   - LARC
   - AAC
   - 8 x 6 AAC
Integrated System (Experiments) is a design issue related to performance parameters (primarily).

Not a NASA cash, but rather industrial/NASA.

Proper codes (validated) should work pretty well.
Non-Air breathing Hypersonics

- Considers continuum & non-continuum
- High drag, C/D < 2
- Concerns with thermal protection, radiation, b.c. transition, real gas effects
- Suggest separate group to discuss validation issues