N95- 23655

NUMERICAL STUDY OF BASE PRESSURE CHARACTERISTIC CURVE FOR A FOUR-ENGINE CLUSTERED NOZZLE CONFIGURATION

Ten-See Wang
Computational Fluid Dynamics Branch
NASA - Marshall Space Flight Center
Marshall Space Flight Center, AL 35812

Abstract

Excessive base heating has been a problem for many launch vehicles. For certain design such as the direct dump of turbine exhaust in the nozzle section and at the nozzle lip of the Space Transportation Systems Engine (STME), the potential burning of the turbine exhaust in the base region have caused a tremendous concern. Two conventional approaches have been considered for predicting the base environment: (1) empirical approach, and (2) experimental approach. The empirical approach uses a combination of data correlations and semi-theoretical calculations. It works best for linear problems, simple physics and geometry. However, it is highly suspecious when complex geometry and flow physics are involved, especially when the subject is out of historical database. The experimental approach is often used to establish database for engineering analysis. However, it is qualitative at best for base flow problems. Other criticisms include the inability to simulate forebody boundary layer correctly, the interference effect from tunnel wall, and the inability to scale all pertinent parameters. Furthermore, there is a contention that the information extrapolated from subscale tests with combustion is unconservative.

One potential alternative to the conventional methods is the computational fluid dynamics (CFD), which has none of the above restrictions and is becoming more feasible due to maturing algorithms and advancing computer technology. It provides more details of the flowfield and is only limited by the computer resources. However, it has its share of criticism as a predictive tool for base environment. One major concern is that CFD has not been extensively tested for base flow problems. It is therefore imperative that CFD be assessed and benchmarked satisfactorily for base flows.

In this study, the turbulent base flowfield of a experimental investigation for a four-engine clustered nozzle is numerically benchmarked using a pressure based CFD method. Since the cold air was the medium, accurate prediction of the base pressure distributions at high altitudes is the primary goal. Other factors which may influence the numerical results such as the effects of grid density, turbulence model, differencing scheme, and boundary conditions are also being addressed. Preliminary result of the computed base pressure agreed reasonably well with that of the measurement. Basic base flow features such as the reverse jet, wall jet, recompression shock, and static pressure field in plane of impingement have been captured.

Numerical Study of Base Pressure Characteristic Curve for a Four-Engine Clustered Nozzle Configuration

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

11th Workshop for CFD Applications in Rocket Propulsion
Main Session 9: Combustion - Nozzle/plume - Benchmark
MSFC, Alabama
April 21, 1993

OBJECTIVE

- ☆ To benchmark a cold flow experiment for a four-engine clustered nozzle base flowfield with a CFD model

Base environment predictive methods

★ The empirical approach

- works best for linear problems, simple physics, and simple geometry
- highly suspicious when complex geometry and complex physics such as base flows are involved
- especially when the subject is out of historical database

★ The experimental approach

- often used to establish a database for engineering analysis
- qualitative at best for base flow applications
- inability to simulate forebody boundary layers
- possible interference effect from tunnel wall
- inability to scale all pertinent parameters
- information extrapolated from subscale test with combustion is unconservative

Base environment predictive methods

☆ The CFD approach

- has none of the above restrictions
- is becoming more feasible due to maturing algorithms and advancing computer technology
- provides subtle details of flow physics
- is only limited by computer resources

CFD Methodology

- Non-Staggered Grid Pressure Based Method
- Curvilinear Transformed Navier-Stokes Equations
- Predictor plus Multi-Corrector Solution Procedure for Efficient Time Marching
- Second and Fourth-Order Central Plus Upwind Dissipation for the Convective Terms
- Two-Equation Turbulence Model

Parametric Study

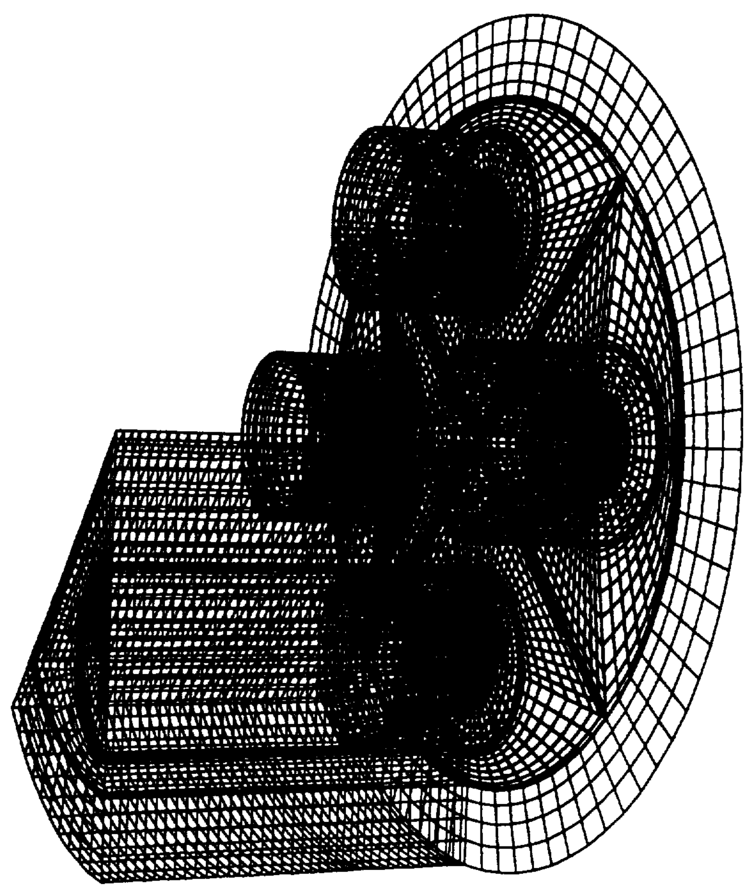
☆ Grid Resolution

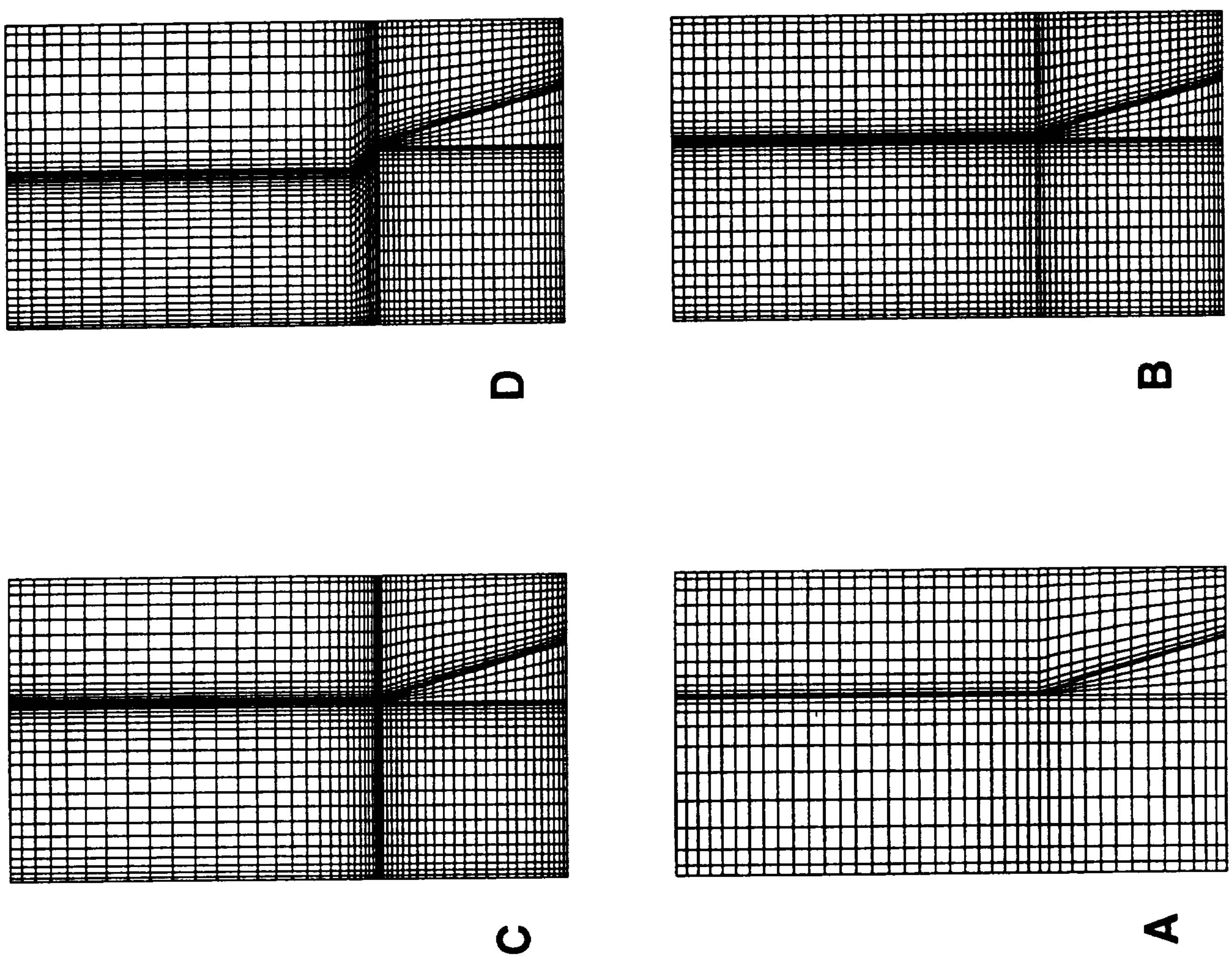
- four 2-zone 3D grid were generated
- Grid A: 34,030 points
- Grid B, C, and D: 113,202 points

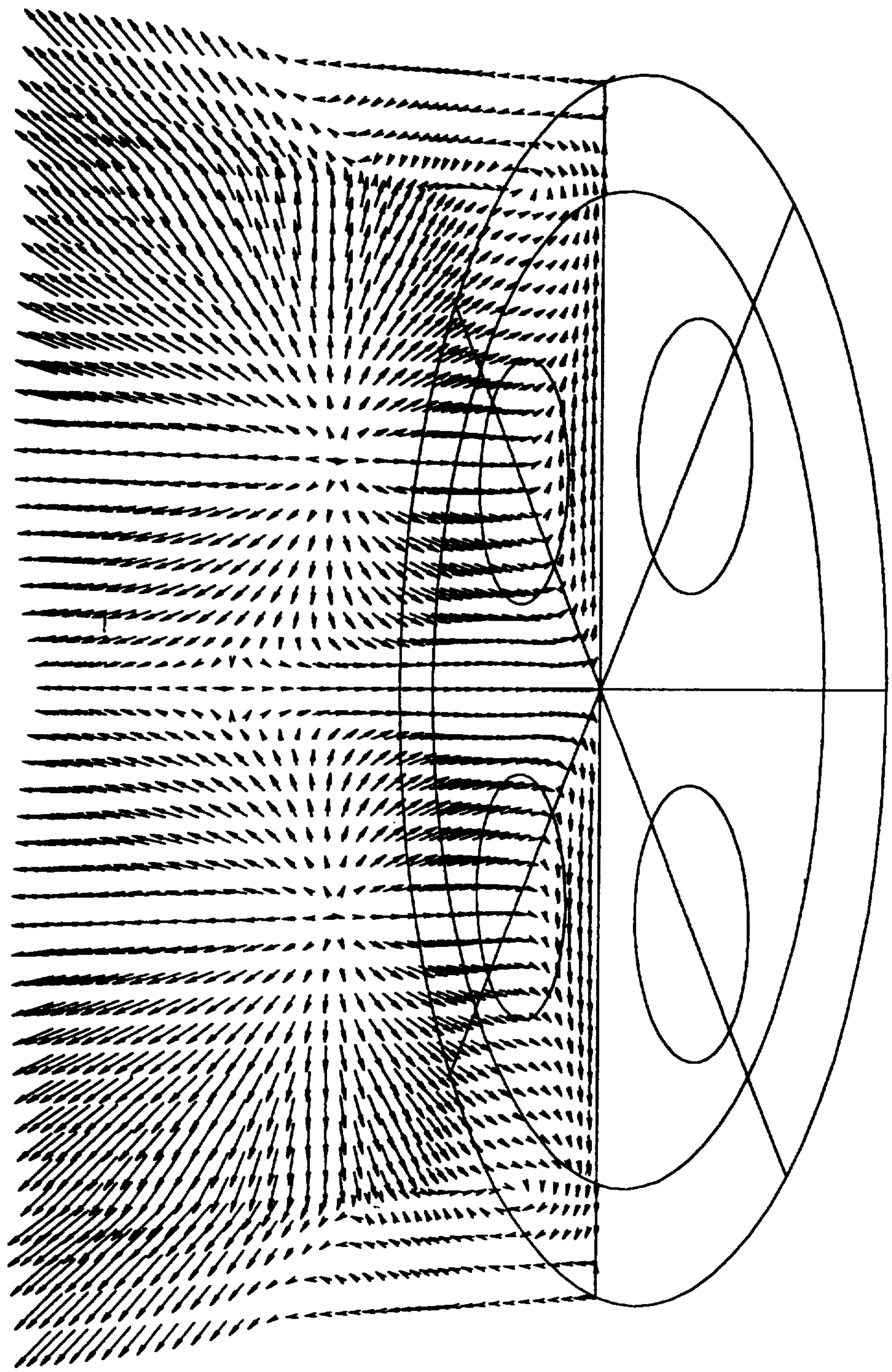
☆ Turbulence Model

☆ Inlet boundary condition

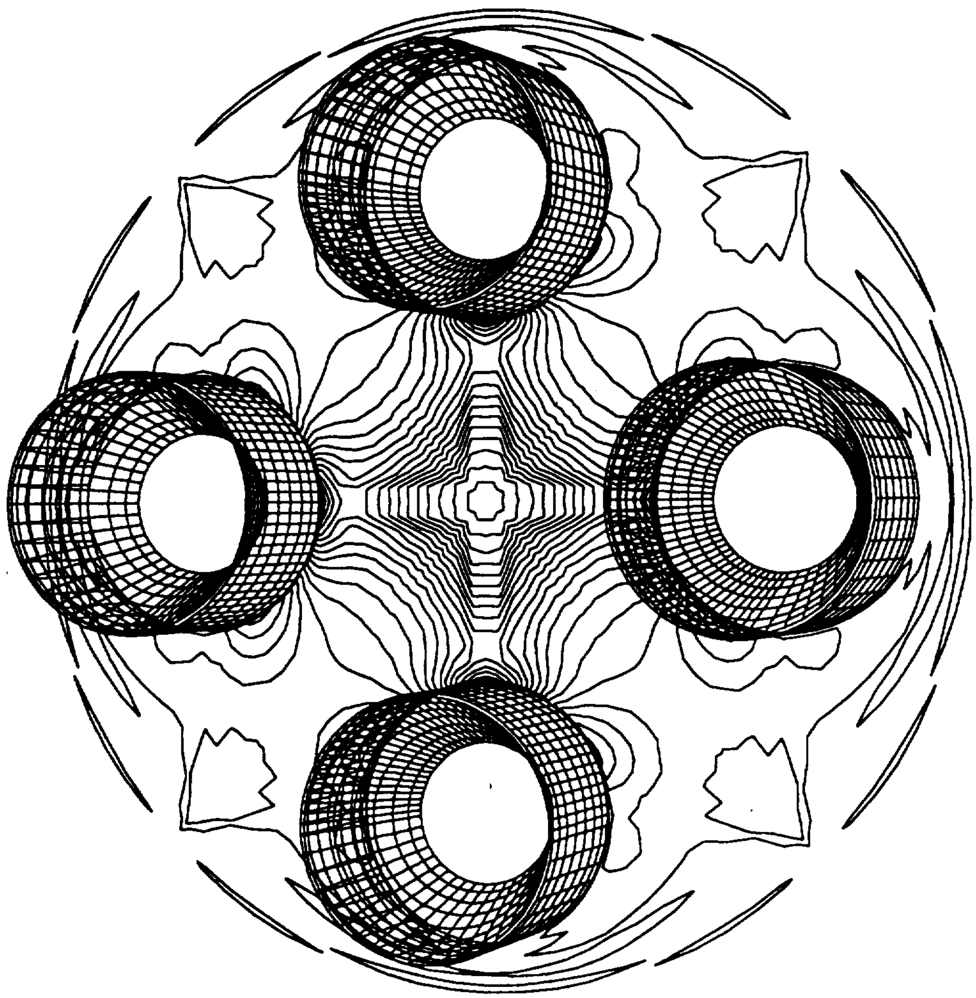
☆ Convective dissipation parameter

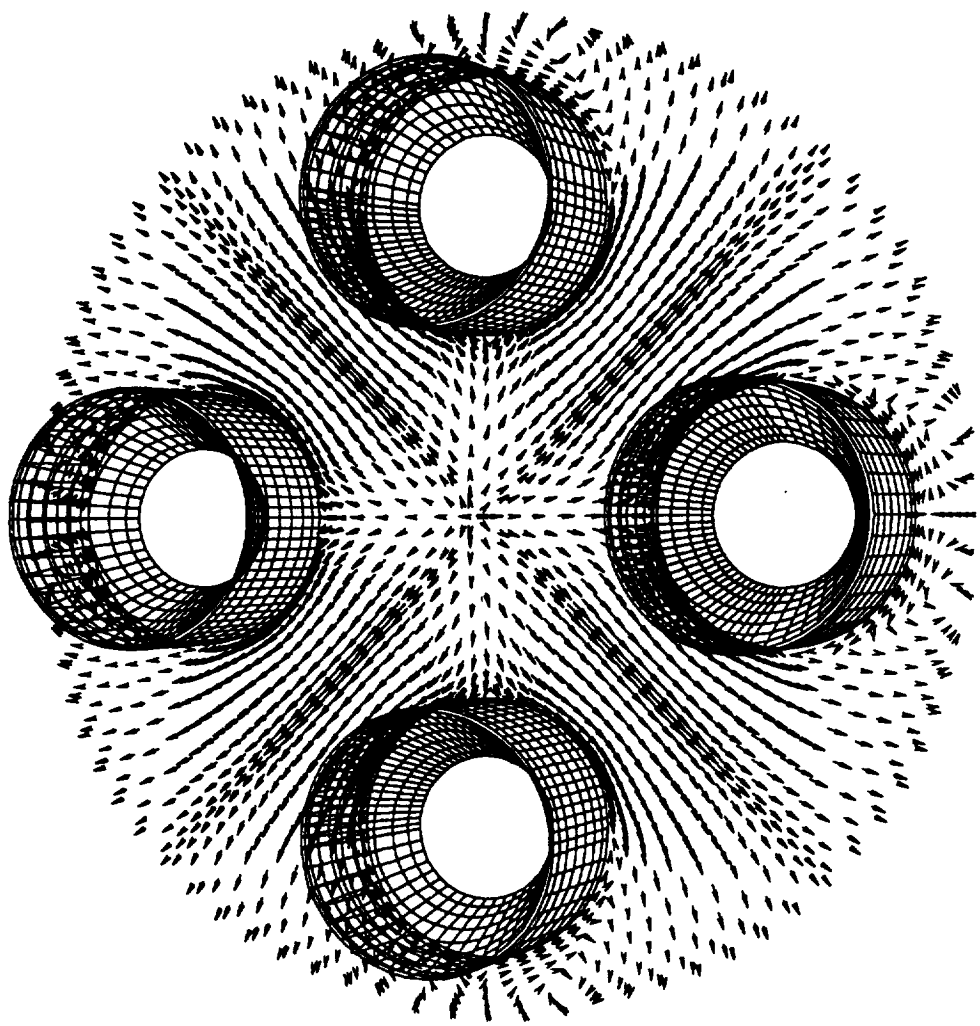


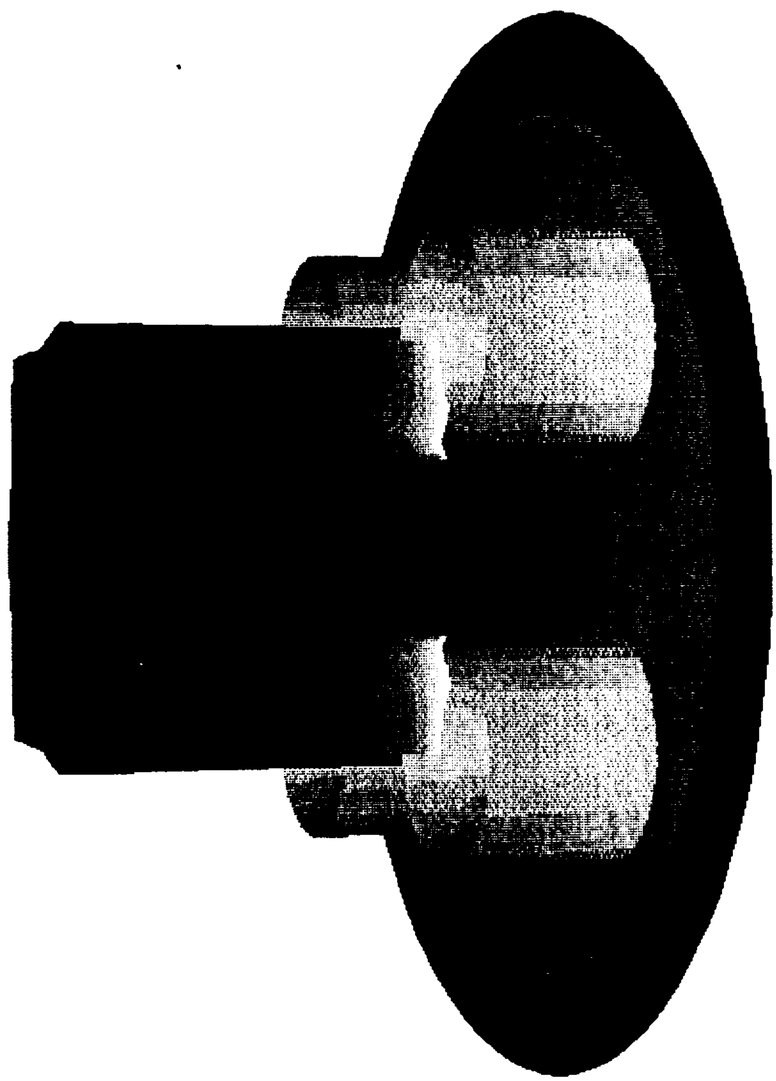

D
B
C
A



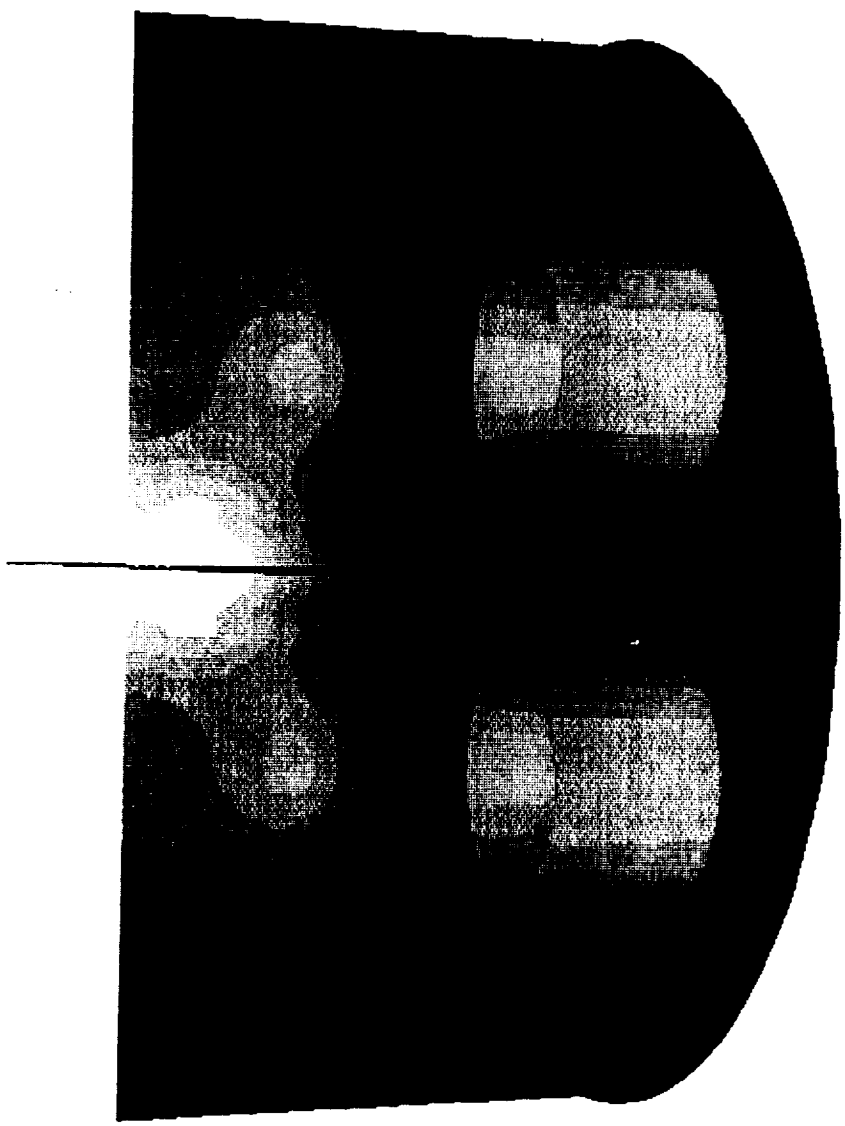
C-11.



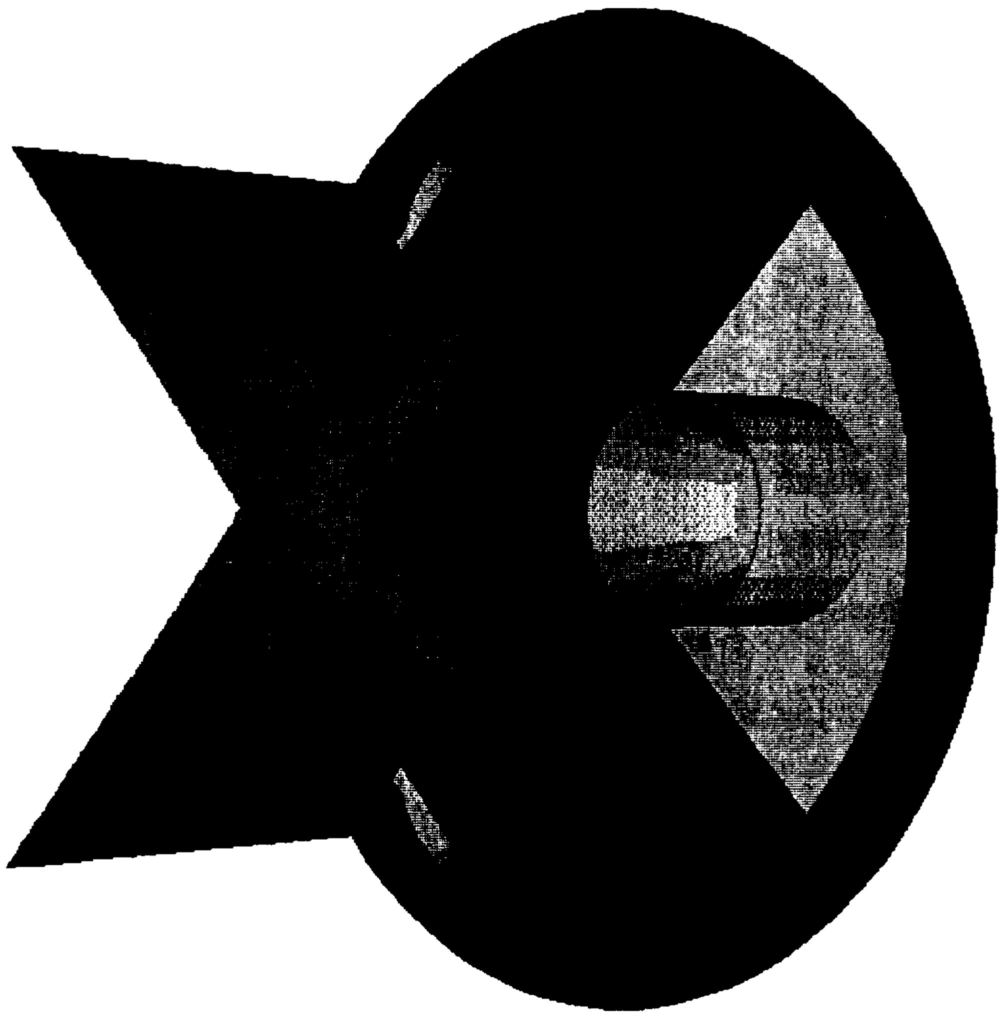




pwfmg4bw.1.img

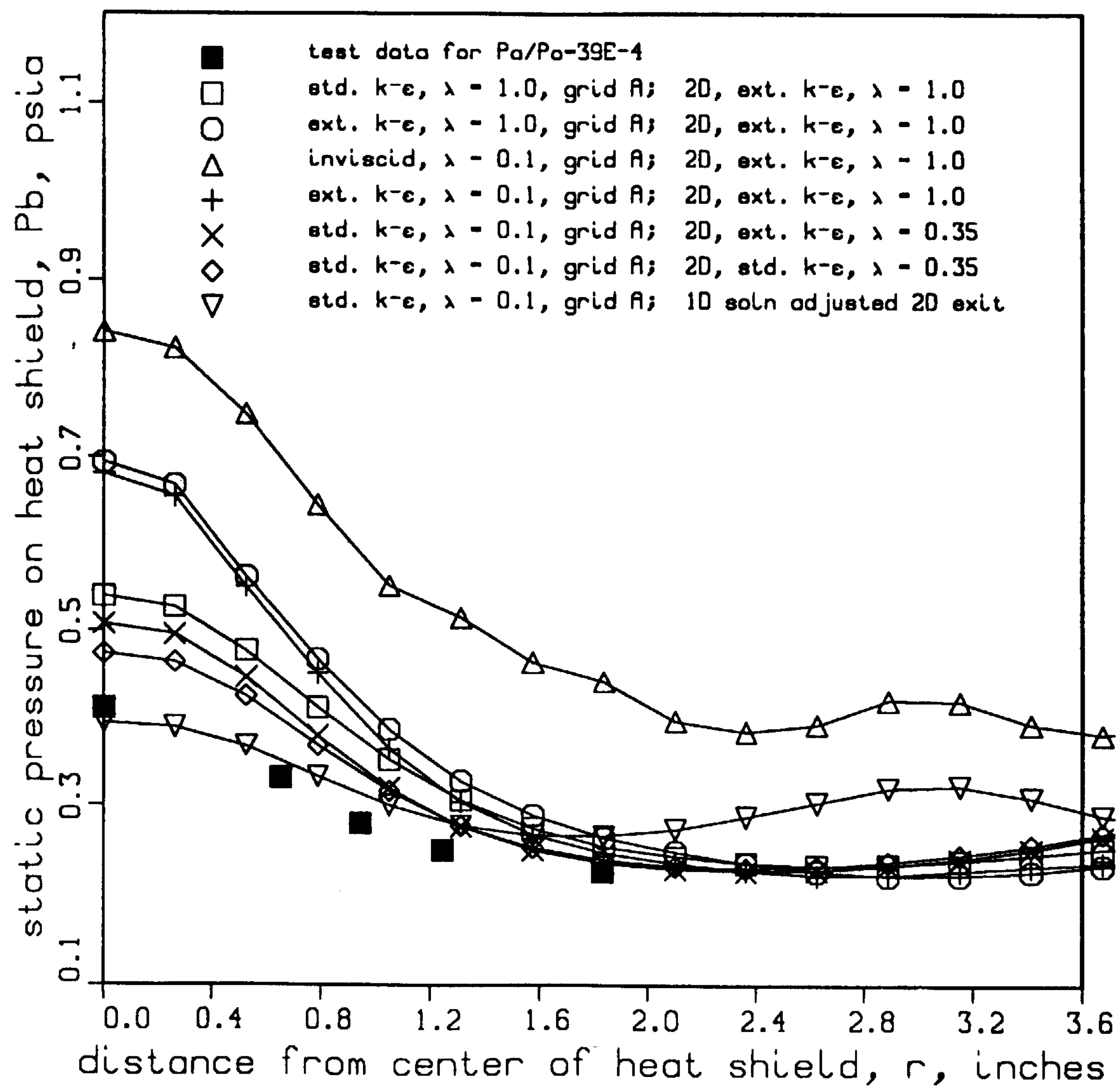


pmfmg4bw.2.img

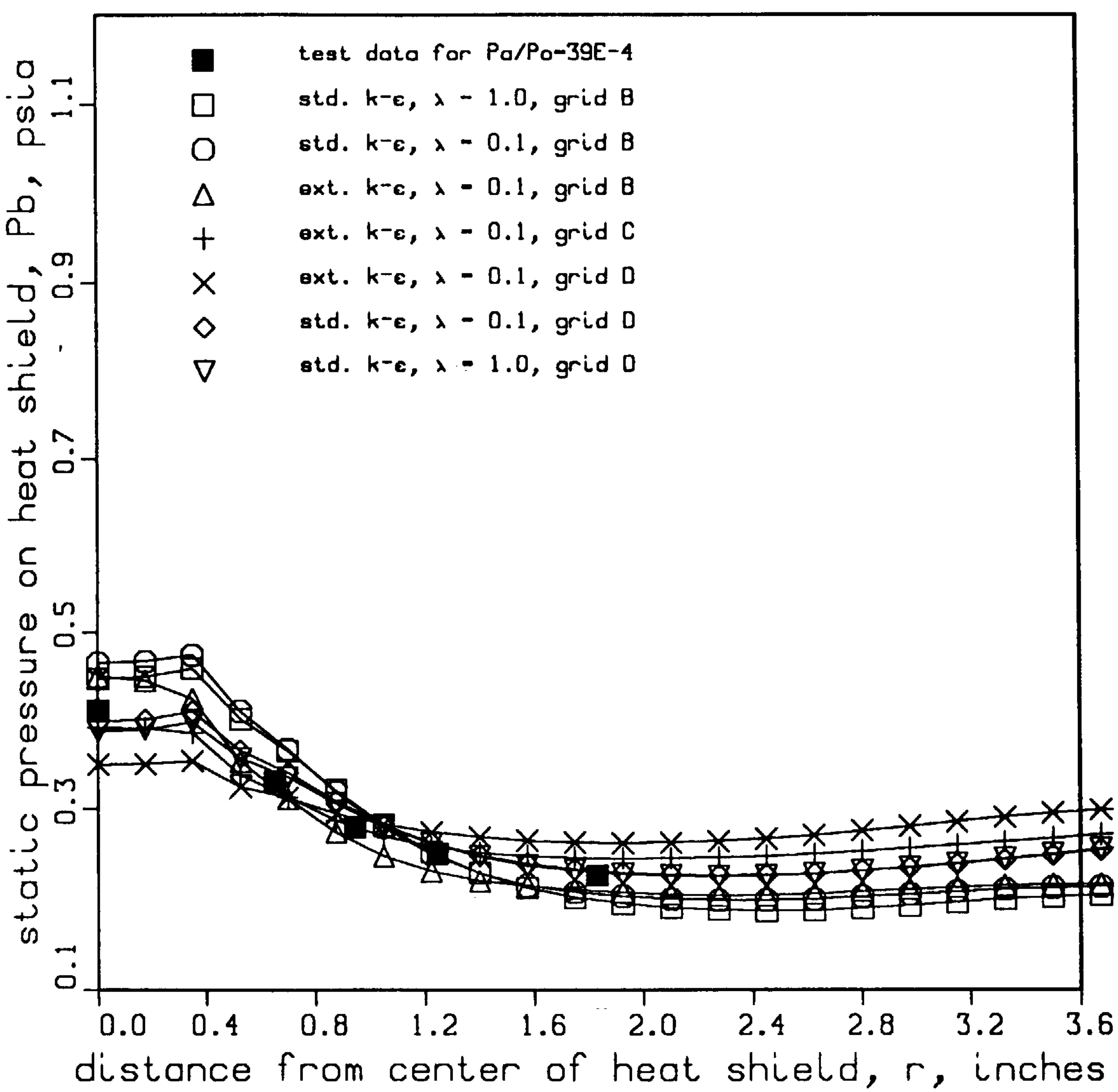


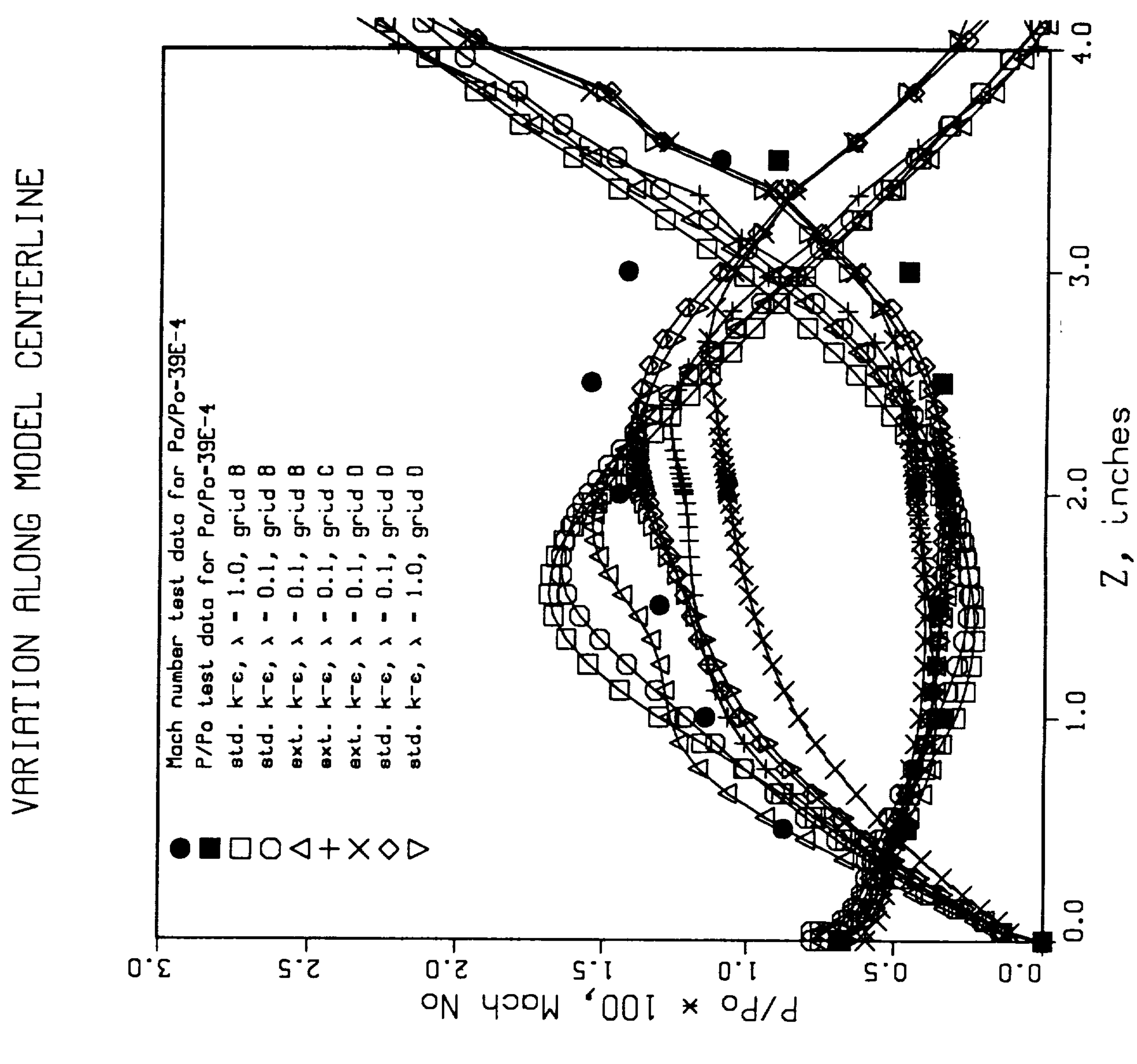
pmfmg4bw.3.img

RADIAL BASE PRESSURE DISTRIBUTION

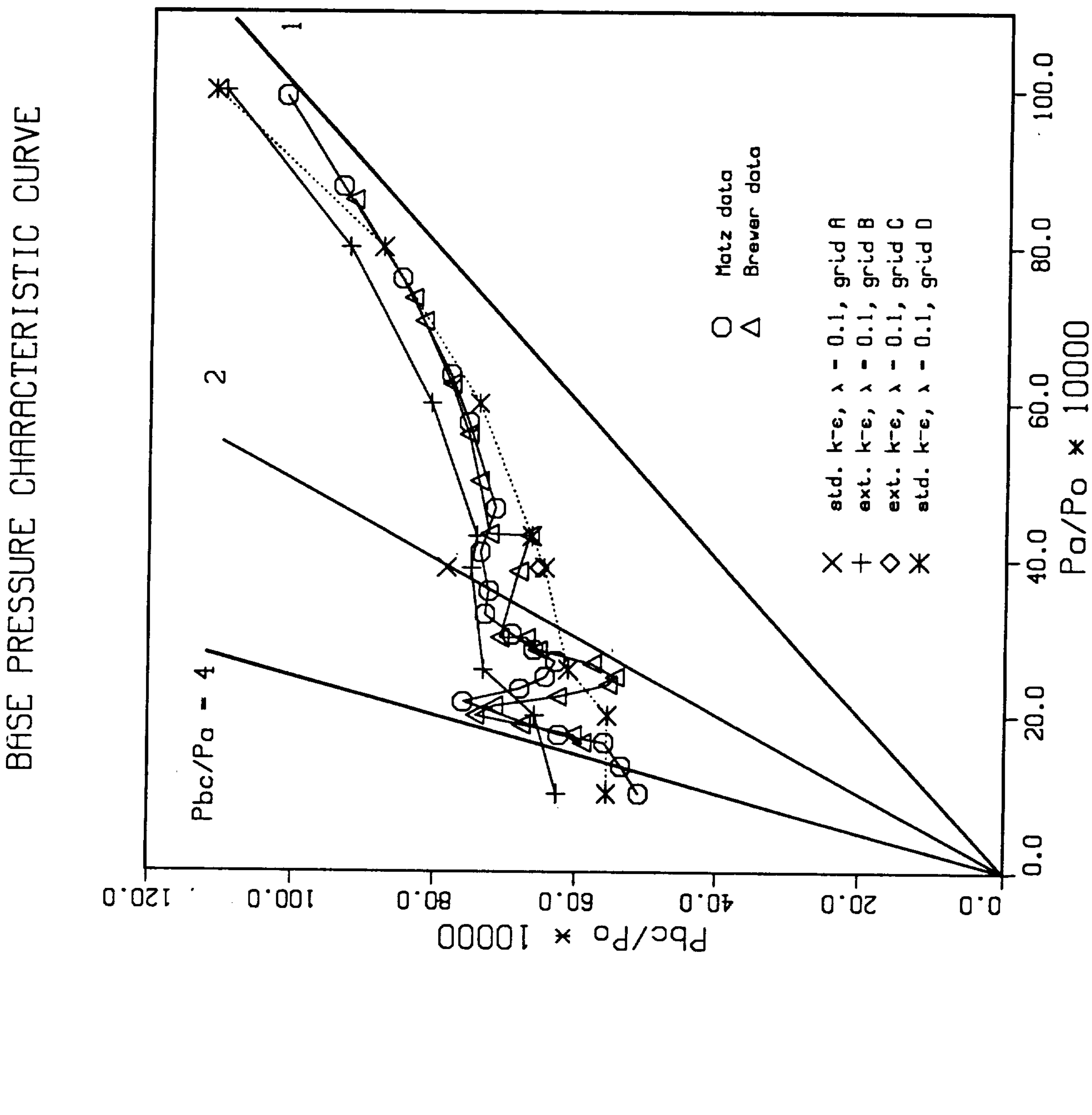


RADIAL BASE PRESSURE DISTRIBUTION





VARIATION ALONG MODEL CENTERLINE
Mach number test data for Pa/Po-39E-4
P/Po test data for Pa/Po-39E-4
std. k-ε, λ - 1.0, grid B
std. k-ε, λ - 0.1, grid B
ext. k-ε, λ - 0.1, grid B
ext. k-ε, λ - 0.1, grid C
ext. k-ε, λ - 0.1, grid D
std. k-ε, λ - 0.1, grid D
std. k-ε, λ - 1.0, grid D
P/Po × 100, Mach No
Z, inches
0.0
0.5
1.0
1.5
2.0
2.5
3.0
4.0



BASE PRESSURE CHARACTERISTIC CURVE
Pbc/Pa = 4
2
1
Matz data
Brewer data
std. k-ε, λ = 0.1, grid A
ext. k-ε, λ = 0.1, grid B
ext. k-ε, λ = 0.1, grid C
std. k-ε, λ = 0.1, grid D
Pa/Po * 10000
Pbc/Po * 10000
0.0
20.0
40.0
60.0
80.0
100.0
120.0

SUMMARY

- ☆ Qualitative base flow features such as the reverse jet, wall jet, recompression shock, and plume-plume impingement have been captured.
- ☆ Quantitative results such as the radial base flow distribution, Mach number and static pressure variations along model center line, and the base pressure characteristic curve agreed reasonable well with those of the experiment
- ☆ Parametric study indicated that the grid resolution and turbulence model are two important parameters which determine the accuracy of a base flow solution
- ☆ The potential of using CFD as a predictive tool for base environment prediction is demonstrated

Future work

- ☆ Grid adapted flowfield solution
- ☆ Hot flow multi-engine base flowfield benchmarking
- ☆ Combustion flow multi-engine base flowfield benchmarking
- ☆ Flight vehicle forbody and base environment simulation

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	July 1993	Conference Publication

4. TITLE AND SUBTITLE
Eleventh Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion—Part I

5. FUNDING NUMBERS

6. AUTHOR(S)
R.W. Williams, Compiler

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
George C. Marshall Space Flight Center
Marshall Space Flight Center, Alabama 35812

8. PERFORMING ORGANIZATION REPORT NUMBER
M-726

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA CP-3221

11. SUPPLEMENTARY NOTES
Prepared by Structures and Dynamics Laboratory, Science and Engineering Directorate.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Subject Category: 34
Unclassified—Unlimited

12b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

Conference publication includes 79 abstracts and presentations and 3 invited presentations given at the Eleventh Workshop for Computational Fluid Dynamic Applications in Rocket Propulsion held at George C. Marshall Space Flight Center, April 20–22, 1993. The purpose of the workshop is to discuss experimental and computational fluid dynamic activities in rocket propulsion. The workshop is an open meeting for government, industry, and academia. A broad number of topics are discussed including computational fluid dynamic methodology, liquid and solid rocket propulsion, turbomachinery, combustion, heat transfer, and grid generation.

14. SUBJECT TERMS spray, injector, computational fluid dynamics, rocket propulsion, liquid rocket, solid rocket, turbopump, turbomachinery, combustion, methodology, impeller, inducer, heat transfer, grid generation, nozzle, plume

15. NUMBER OF PAGES
970

16. PRICE CODE
A99

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	Unlimited