N91-10844

Johnson Space Center CFD Overview

C. P. Li
Advanced Programs Office
Johnson Space Center, Houston, Tx 77058

Recent applications and development of CFD technology have focused on flow problems that are critically important to the operation and design of space flight vehicles. The main effort is spent on the Space Shuttle in order to provide an understanding of the cryogenic fluid in the duct connecting the External Tank and the Main Engines, the subsonic flow surrounding the Orbiter during crew egress maneuvers, the transonic aerodynamic forces on the the Orbiter fuselage and wing, the high angle-of-attack abort flight, and the aerodynamic heating during entry. To provide in-depth analyses for such diverse problems within a timely schedule, matured panel codes and a state-of-the-art incompressible turbulent flow code were adapted. Collaboration with Ames Research Center has resulted in a Shuttle ascent aerodynamic code; and a viscous chemical nonequilibrium code is being developed for predicting Orbiter real-gas aerodynamics and finite-catalytic heating. The remaining activities are devoted to the prediction of the flow environment around the Aeroassist Flight Experiment vehicle at hypersonic speeds and high altitudes. A thermochemical nonequilibrium Navier-Stokes code has been developed on the basis of two- temperature and 11-species models for solving both the shock layer and near wake. After validating the code against wind-tunnel aerodynamic, pressure and heating data, the code is being used to supplement the ground test facilities in predicting a more realistic flight environment. CFD technology is being relied upon by other programs as well in the consideration of candidate configurations. A biconic cone entering the Martian atmosphere at moderate angles of attack will be analyzed for its stability and heating distribution for the proposed mission. Capabilities of simulating the low and medium lift-to-drag vehicles flowfield flying back from the Space Station have been demonstrated and will be enhanced to include winglets. The development of hypersonic CFD technology at JSC will continously emphasize the modeling of radiation and ablation in continuum flow regime, sufficient realism of geometry, and efficiency of computational methods.

CFD SUPPORT FOR VARIOUS PROGRAMS

- Shuttle (MY6.5, OSF)
- Orbiter (MY2.5, OSF)
- Aeroassist Flight Experiment (MY3.5, OAST)
- Mars Rover Sample Return Mission (MY1, SE)
- Crew Rescue Escape Vehicle (MY0.25, SS)
- High-Energy Aerobraking (MY0.5, OAST)

Simulation Codes and Computers

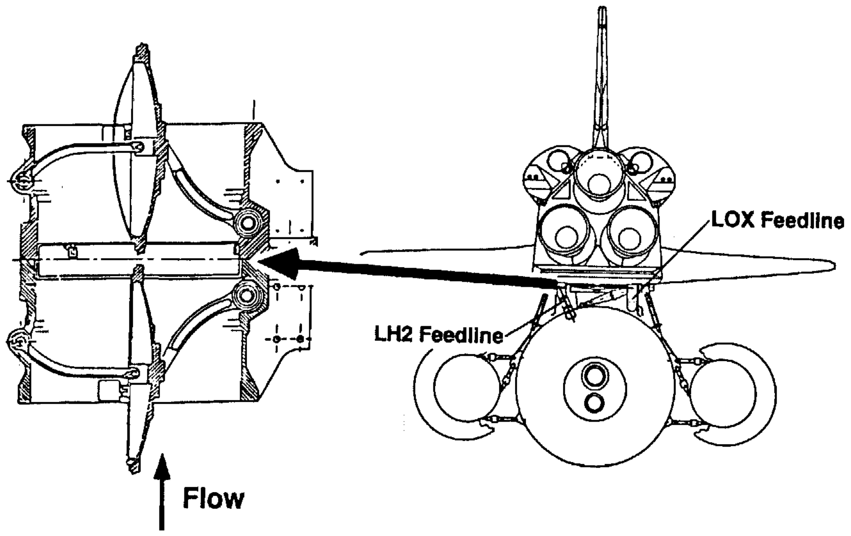
- PANAIR, VSAERO, QUADPLAN (potential flow, panel method)
- INS3D (incompressible Navier-Stokes code)
- F3D (compressible flow NS code)
- EAGLE, SVTGD3D (grid generation codes)
- U3D (upwind finite-volume implicit method)
- NOSIP, AFTB (shock-fitting NS and Parabolized NS codes)
- VRFNS (shock-fitting, chemically reactive NS codes)
- VRFLO (shock-fitting, thermochemical nonequilibrium NS code)
- VAX8650s, SCS40 or CX200 (code development)
- Cray XMP at MSFC and CRAY 2 at NAS (engineering application)
- Class IV computer (CY89)

OUTLINE OF PRESENTATION

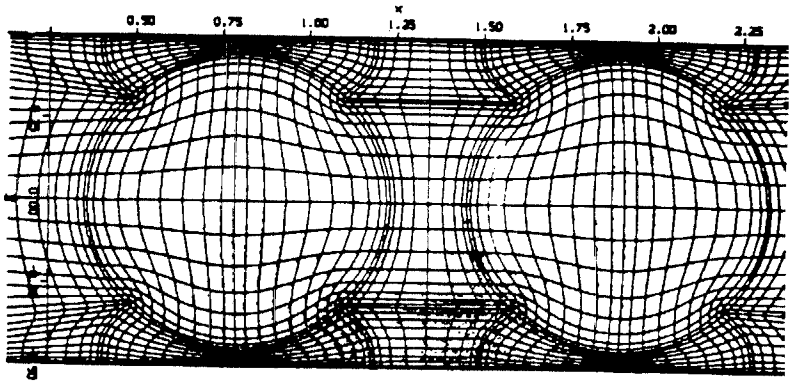
- Cryogenic Duct Flow Simulation by Kandula and Pearce
- Subsonic Orbiter Flow Computation by Slotnick
- Shuttle Debris Analyses by Gomez, Labbe, Martin
- Supersonic Orbiter Flow Computation and Comparison by Wey and Ma
- Hypersonic Orbiter Viscous and Reactive Flow Simulation by Li
- Biconic Cone Flow by Stuart
- AFE results by Gomez, McGary, Tam and Li
- CERV results

ANALYSIS OF ET/ORBITER DISCONNECT VALVES

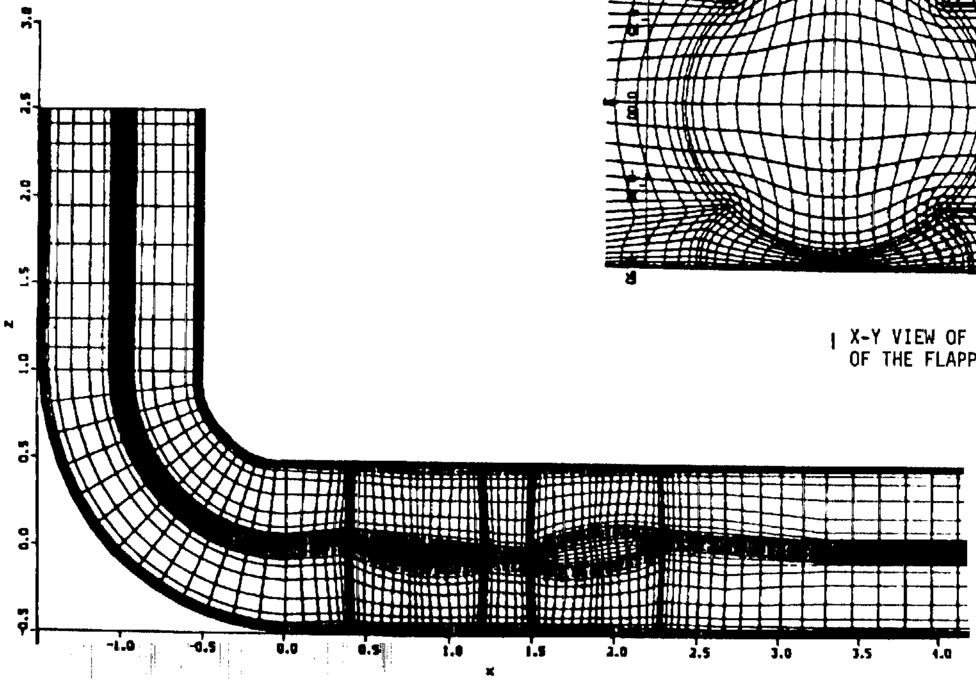
- Objective:
  - to predict and correlate the hydrodynamic stability of the flappers and pressure drop with test data
- Approach:
  - Adapted and modfied INS3D, SVTGD3D and INGRID
  - Compared with water test data



GRID IN THE DUCT WITH FLAPPER VALVES



X-Y VIEW OF THE GRID IN THE MIDPLANE
OF THE FLAPPERS



A PORTION OF X-Z VIEW OF THE GRID AT THE PLANE OF SYMMETRY

Crew Egress Aerodynamic Analysis

- Objective: Numerically simulate the external flowfield surrounding the various flight test vehicles so that an accurate assessment of the astronaut exit trajectories may be determined.

- Methodology: Use production panel code methods to compute aero characteristics.

  - Fast, efficient CFD tool
  - Appropriate for subsonic unobstructed external flow

- Simulations:

  - Space Shuttle Orbiter (VSAERO/PANAIR)
  - Convair C240 (QUADPAN) - Tractor Rocket Concept
  - Lockheed C-141B (QUADPAN) - Pole Concept

SPACE SHUTTLE LAUNCH VEHICLE (SSLV) CFD ANALYSIS STATUS & PLANS

Advanced Programs Office

Steven G. Labbe

3/3/89

NUMERICAL SIMULATION OF SSLV ASCENT AERO ENVIRONMENT

- RECENT ADVANCES IN CFD ENABLE FRESH LOOK AT SPACE SHUTTLE ASCENT AERODYNAMICS
- JOINT EFFORT BETWEEN ARC TEAM (J.L. STEGER, P.G. BUNING) & JSC TEAM (F. W. MARTIN)

TEAM OBJECTIVES

- ARC: Develop the Technology to Numerically Simulate Complex Launch Vehicle Geometry
- JSC: Employ CFD Technology to Gain Insight & Understanding of the Ascent Aero Loads Environment

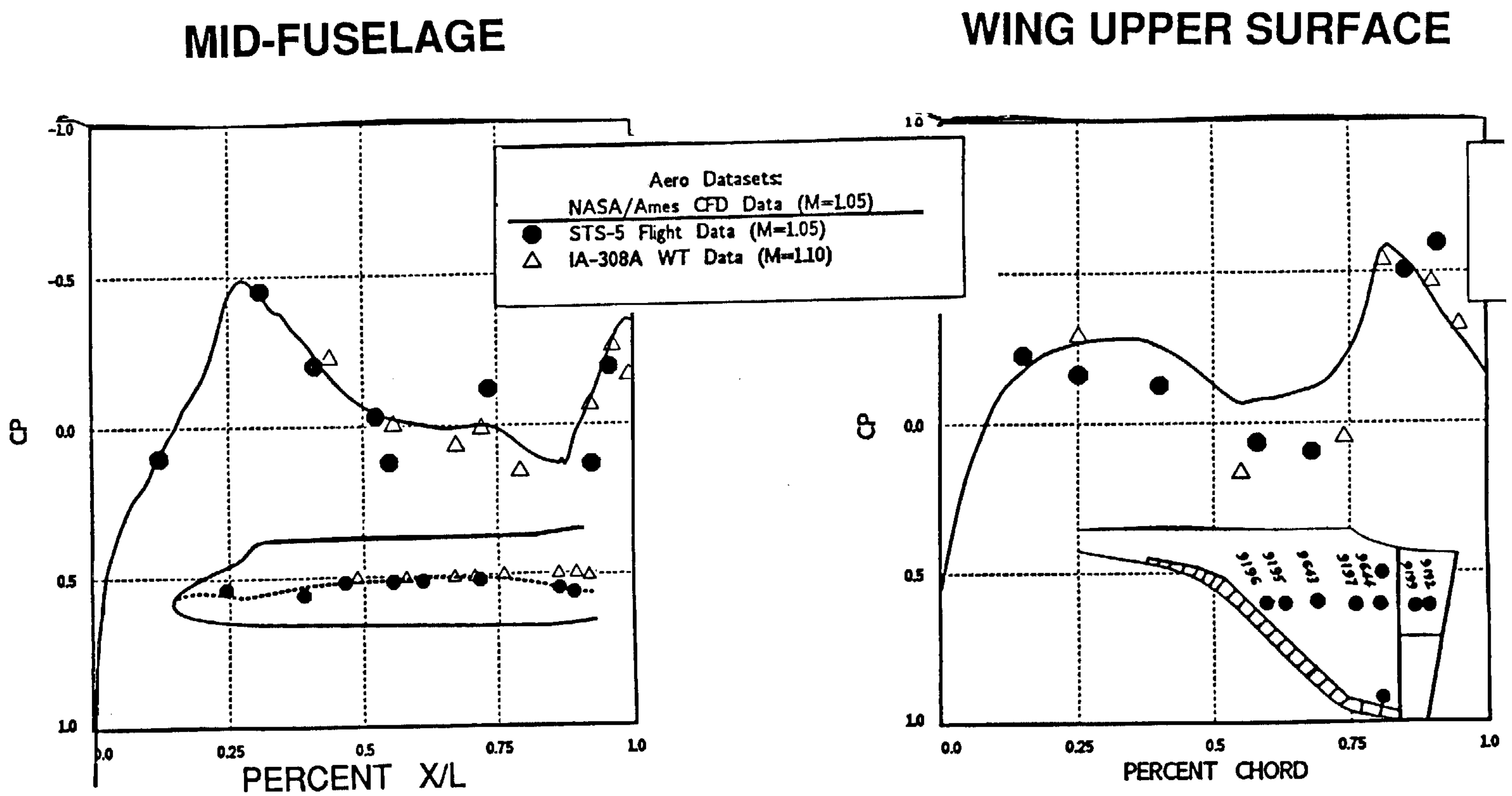
CFD ANALYSIS

- Complex SSLV Ascent Configuration Modelled Using "CHIMERA" Composite Grid Discretization Approach
- Overset Body-Conforming Grids Of Major Geometry Components - Communication by PEGASUS Code
- F3D Implicit Approx. Factored Finite- Difference Procedure - Solution of 3-D Thin-Layer NS Equs.

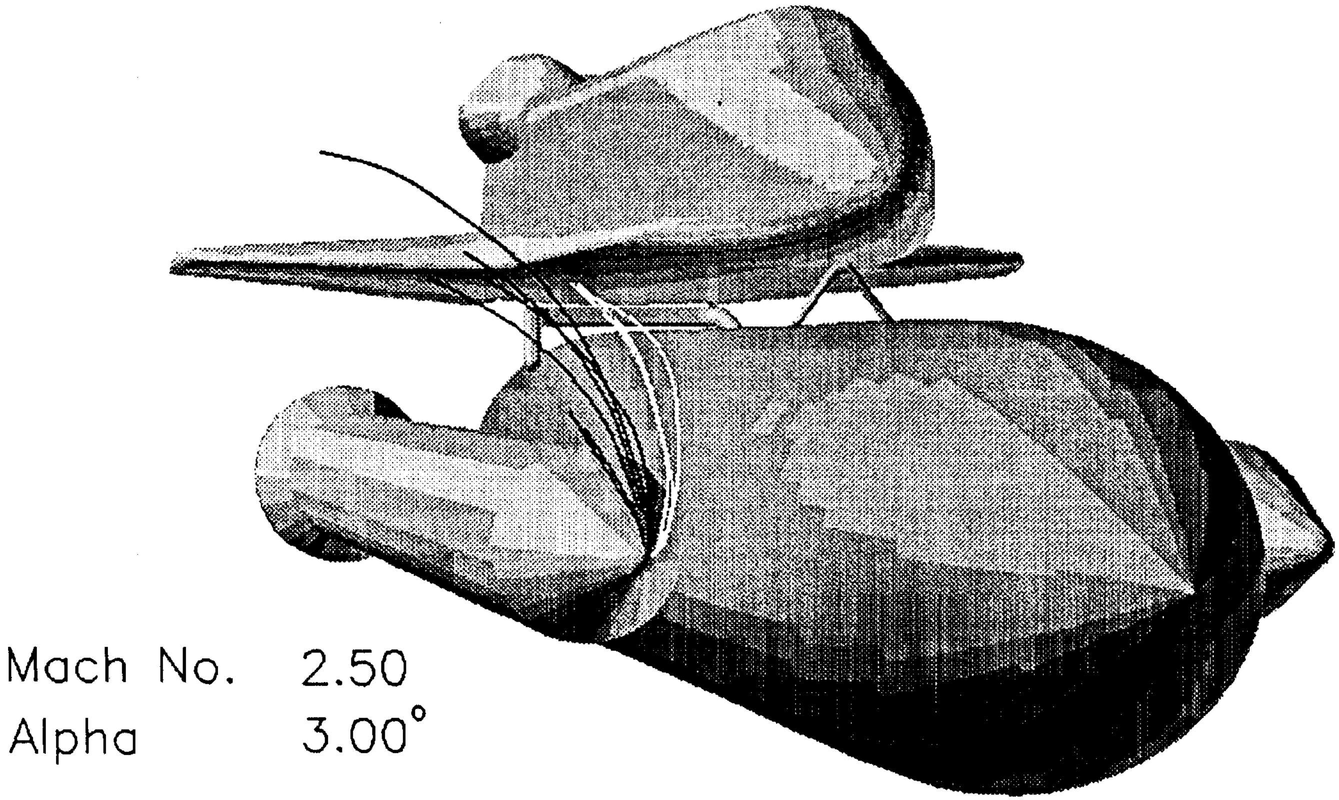
CFD ANALYSIS PROGRESS HAS BEEN STEADY & PROMISING

- EFFORTS CONCENTRATED ON FULLY UPDATING SSLV CONFIGURATION GEOMETRY DETAILS
  Highest Priority Given to Attach Hardware & LOX/Fuel Feedlines
  SRB/IEA Attach Ring and SRB Plume Simulations Incorporated -- Check Out Proceeding
- ABILITY TO ACCURATELY DETERMIN WING LOADS ←→ MODEL SSLV GEOMETRY DETAILS

AERO DATABASE COMPARISONS



STS-27 Ascent Debris Trajectory Simulation

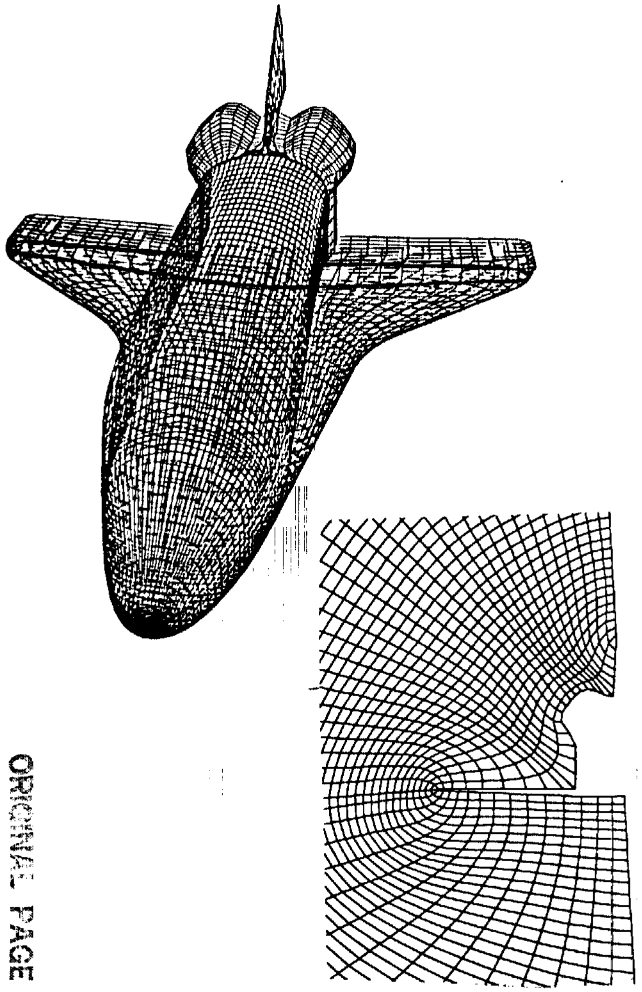


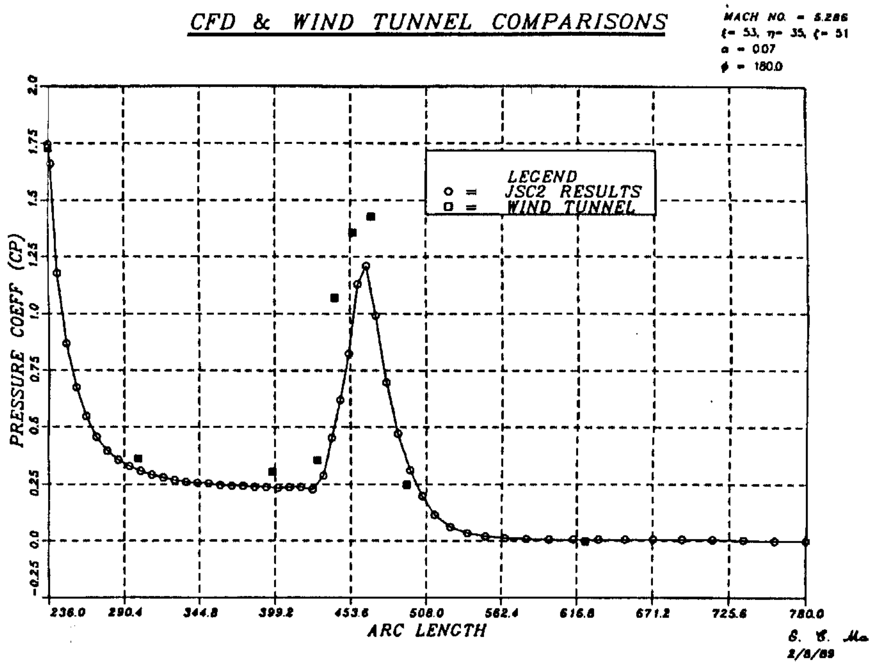
MSA-1 (SRB ablator) released @ 0° - 90° on nose cone

ORBITER AERODYNAMICS AT HIGH INCIDENCE ANGLES

- Objective:
  - To predict aerodynamics and flight characteristics for angles of attack up to 90 deg
- Approach:
  - Validate the U3D code with wind-tunnel data for angles of attack lower than 50 deg
  - Apply the multi-zonal U3D for higher angles-of-attack flow

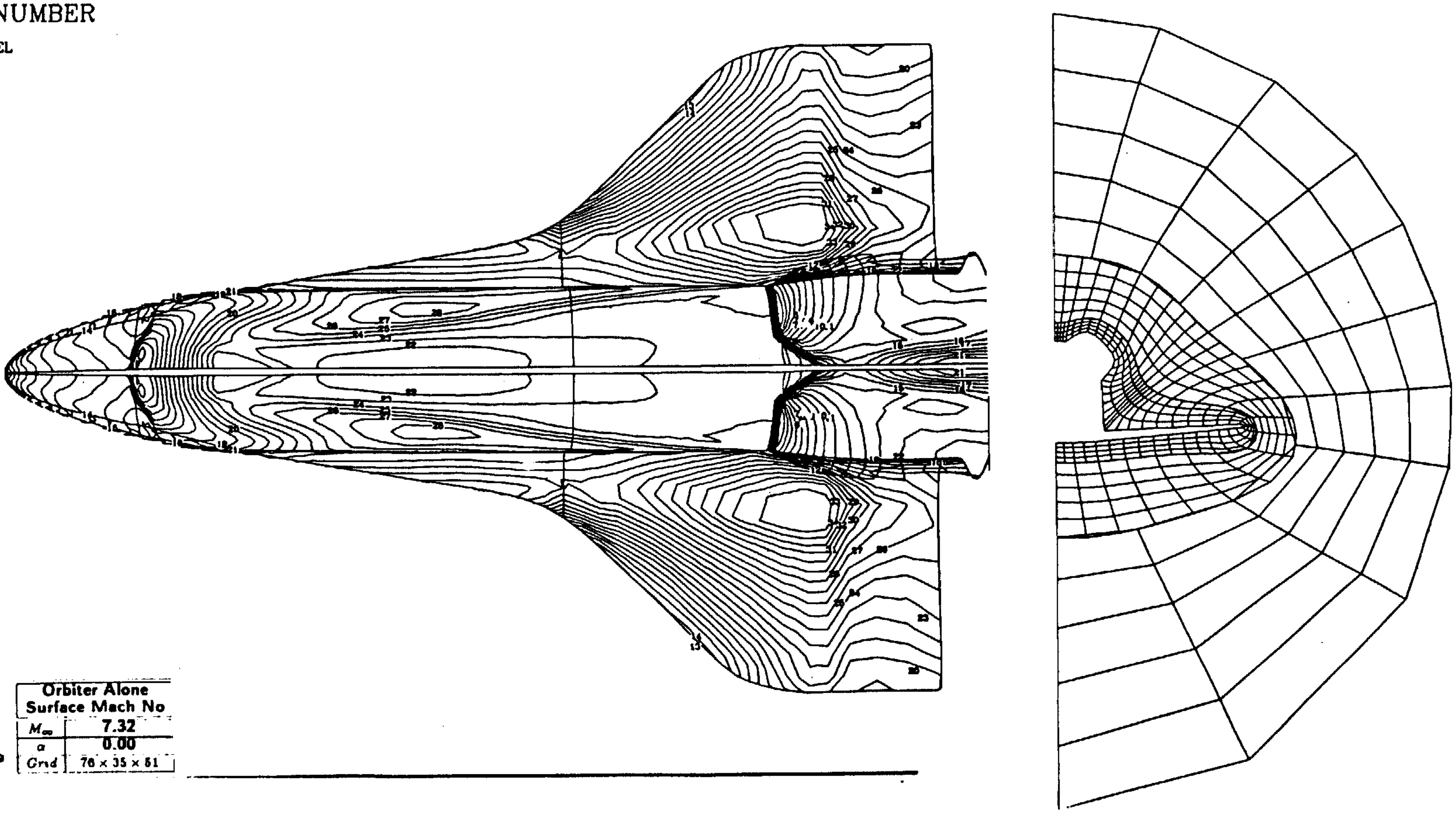
ORBITER GRID AND MACH 5.2 COMPARISON




CFD & WIND TUNNEL COMPARISONS

ORIGINAL PAGE IS
OF POOR QUALITY

MACH 7.32 FLOW RESULTS AND GRID



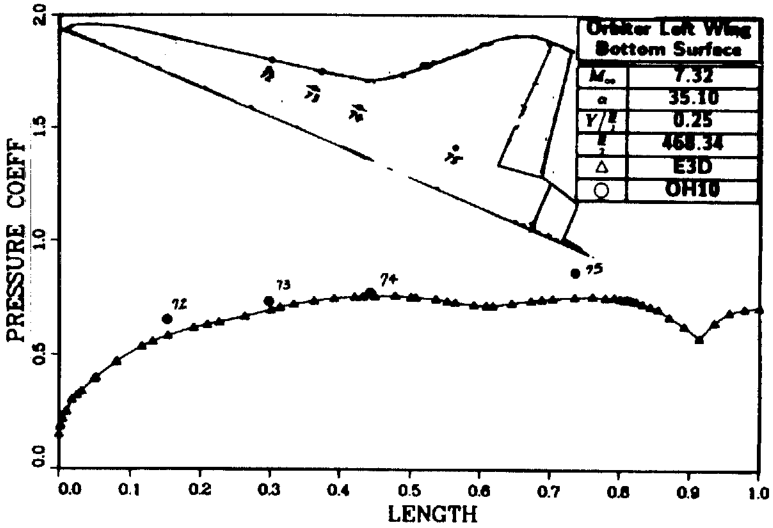
MACH NUMBER

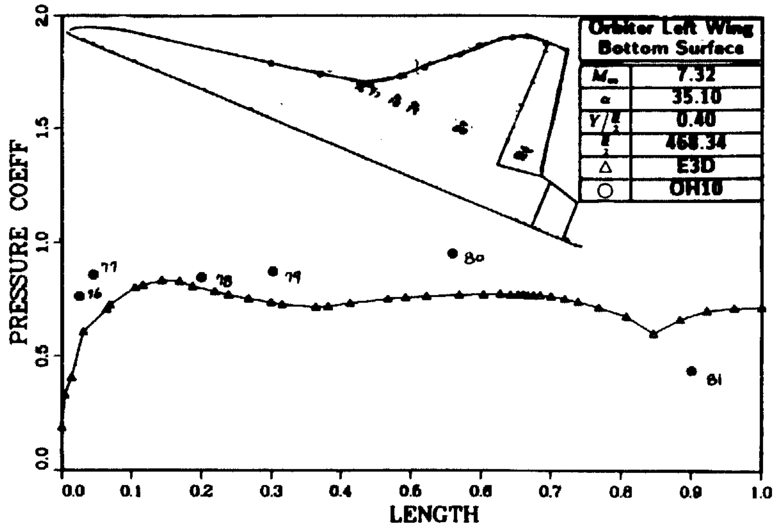
CONTOUR	LEVEL
1	0.164
2	0.361
3	0.558
4	0.755
5	0.953
6	1.150
7	1.347
8	1.544
9	1.742
10	1.939
11	2.136
12	2.333
13	2.530
14	2.728
15	2.925
16	3.122
17	3.319
18	3.517
19	3.714
20	3.911
21	4.108
22	4.306
23	4.503
24	4.700
25	4.897
26	5.095
27	5.292
28	5.489
29	5.686
30	5.884
31	6.081
32	6.278
33	6.475
34	6.672
35	6.870

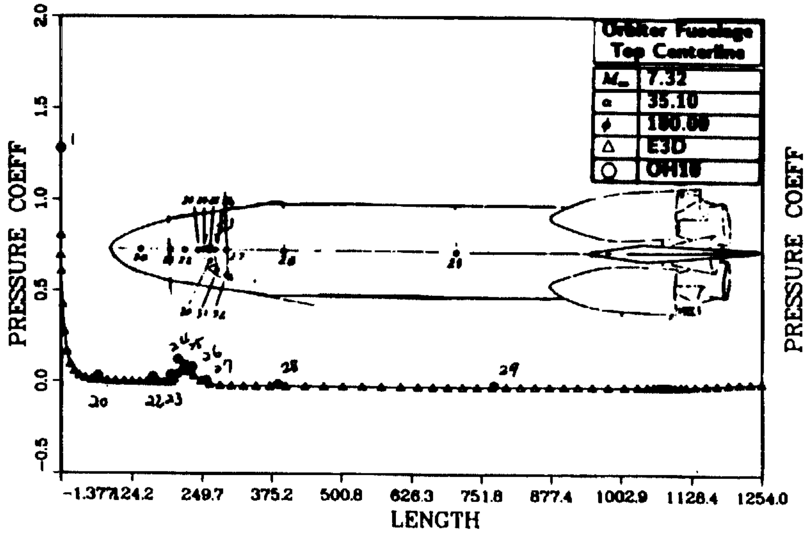
WEY 1/13/89

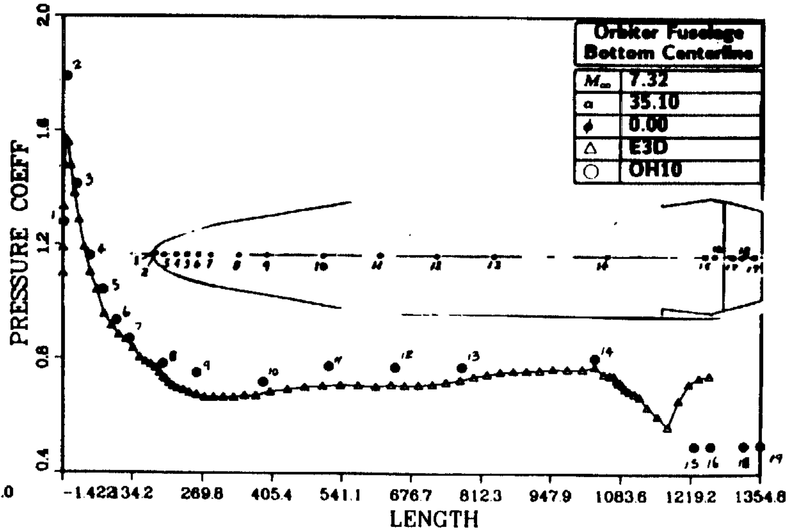
Orbiter Alone Surface Mach No	
M_∞	7.32
α	0.00
Grid	76 × 35 × 51

NUMERICAL INVESTIGATION OF CONTINGENCY ABORT









T. C. Wey/LESC

ORIGINAL PAGE IS
OF POOR QUALITY

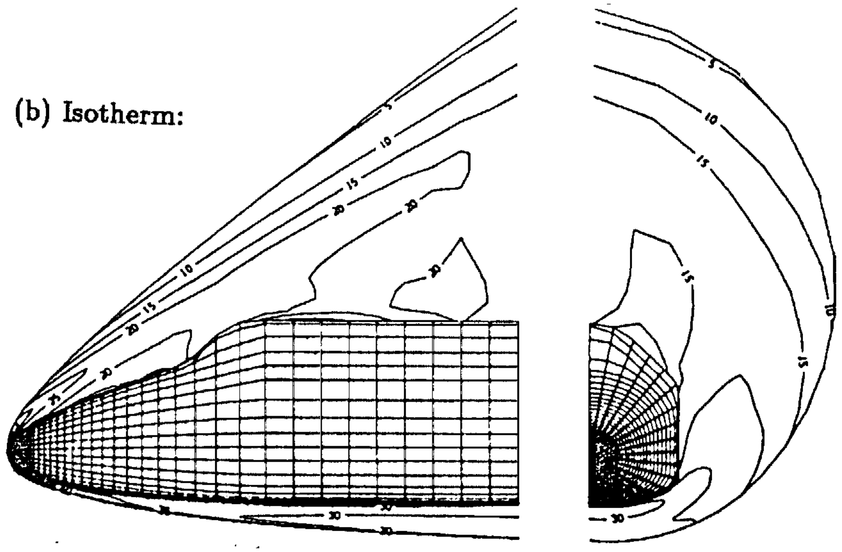
ORBITER ENTRY FLOW SIMULATION

- Objective:
  - To assess aerodynamic and heating issues for each flight
  - To validate the VRFNS code using flight data
- Approach:
  - Use a chemical nonequilibrium model and shock-fitting technique
  - Use a decouple technique for species
  - Compare the perfect-gas model results with wind-tunnel data

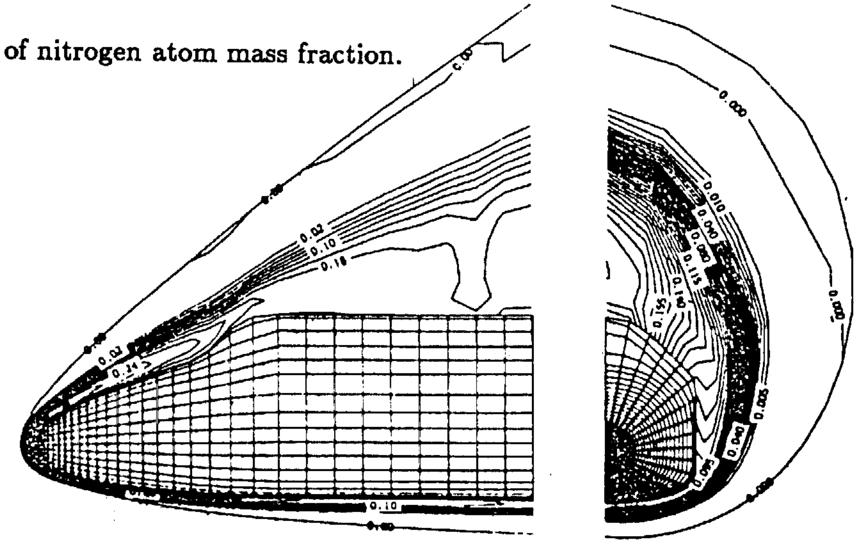
INVISCID CHEMICAL NONEQUILIBRIUM FLOW

Orbiter canopy

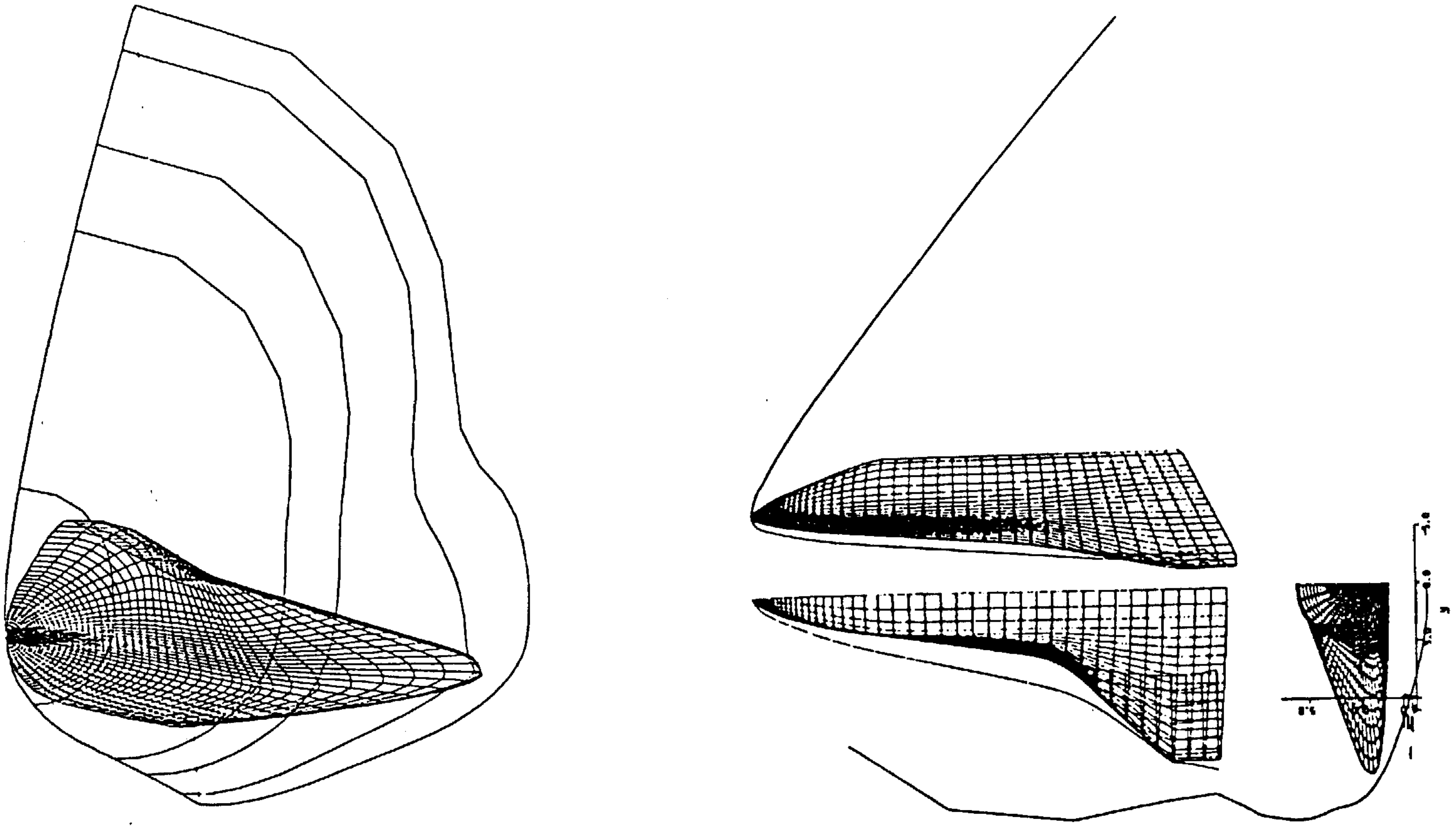
(b) Isotherm:



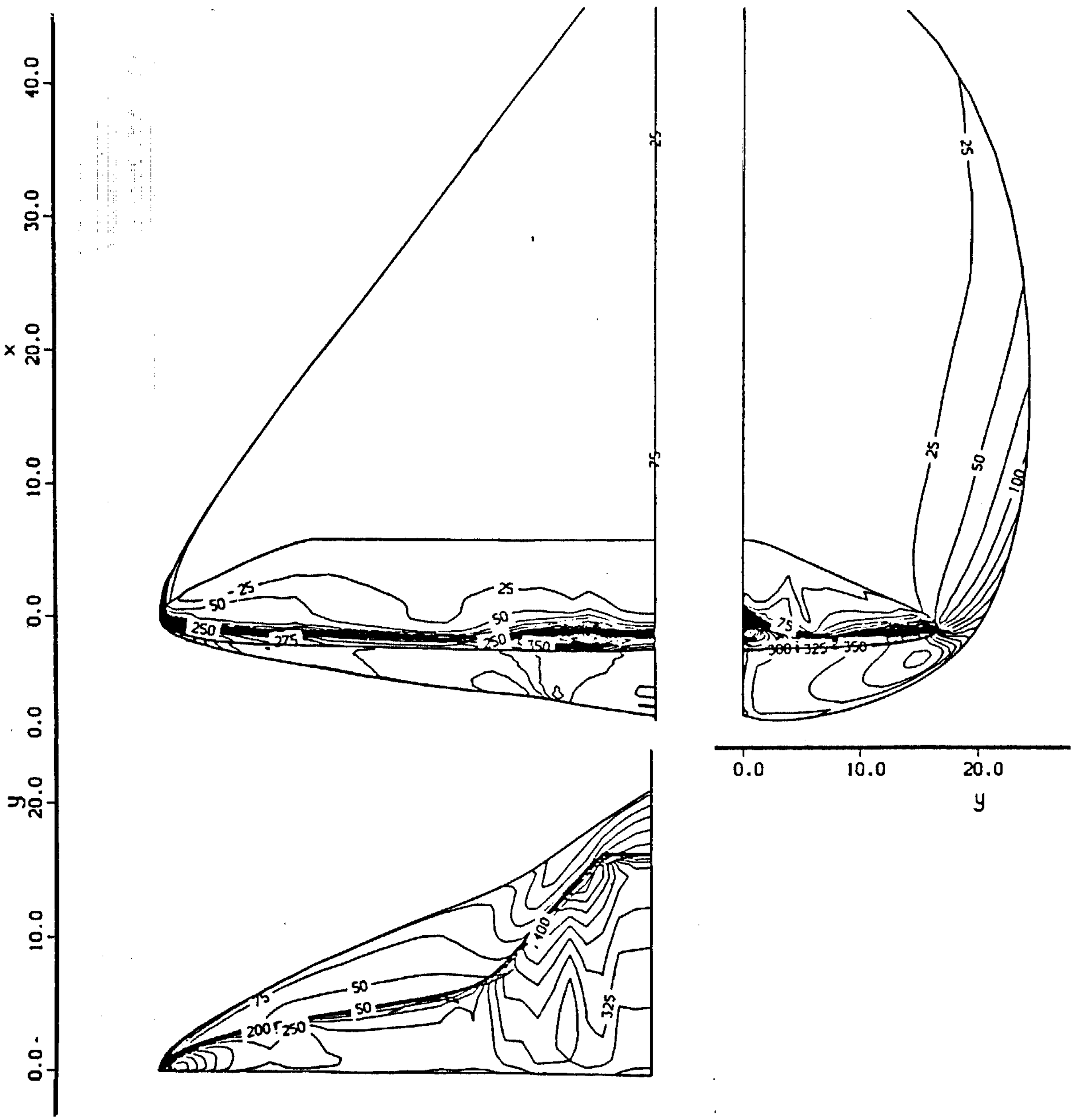
(c) Contours of nitrogen atom mass fraction.



BOW SHOCK SURROUNDING A SWEPT-WING VEHICLE
AT MACH 22 and AOA 40 WITH EQUILIBRIUM CHEMISTRY



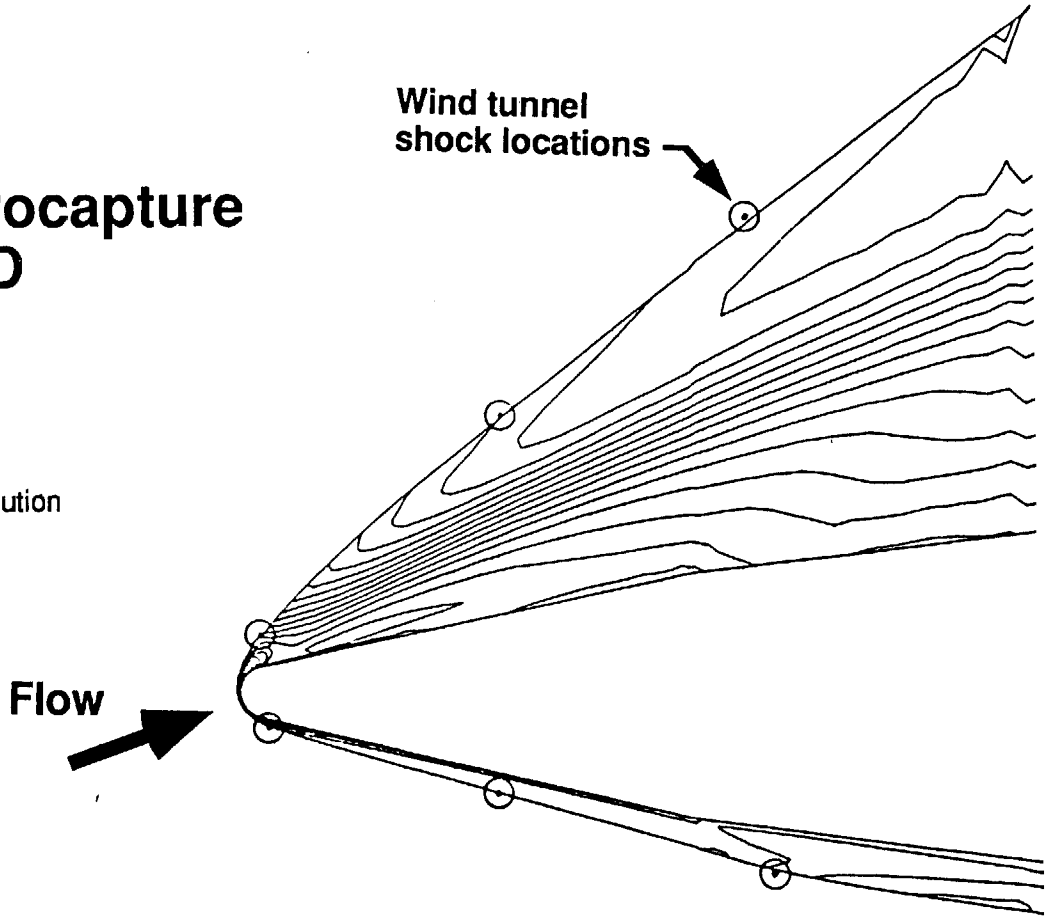
PROJECTED VIEWS OF TEMPERATURE CONTOURS



Comparison of shock shape with data

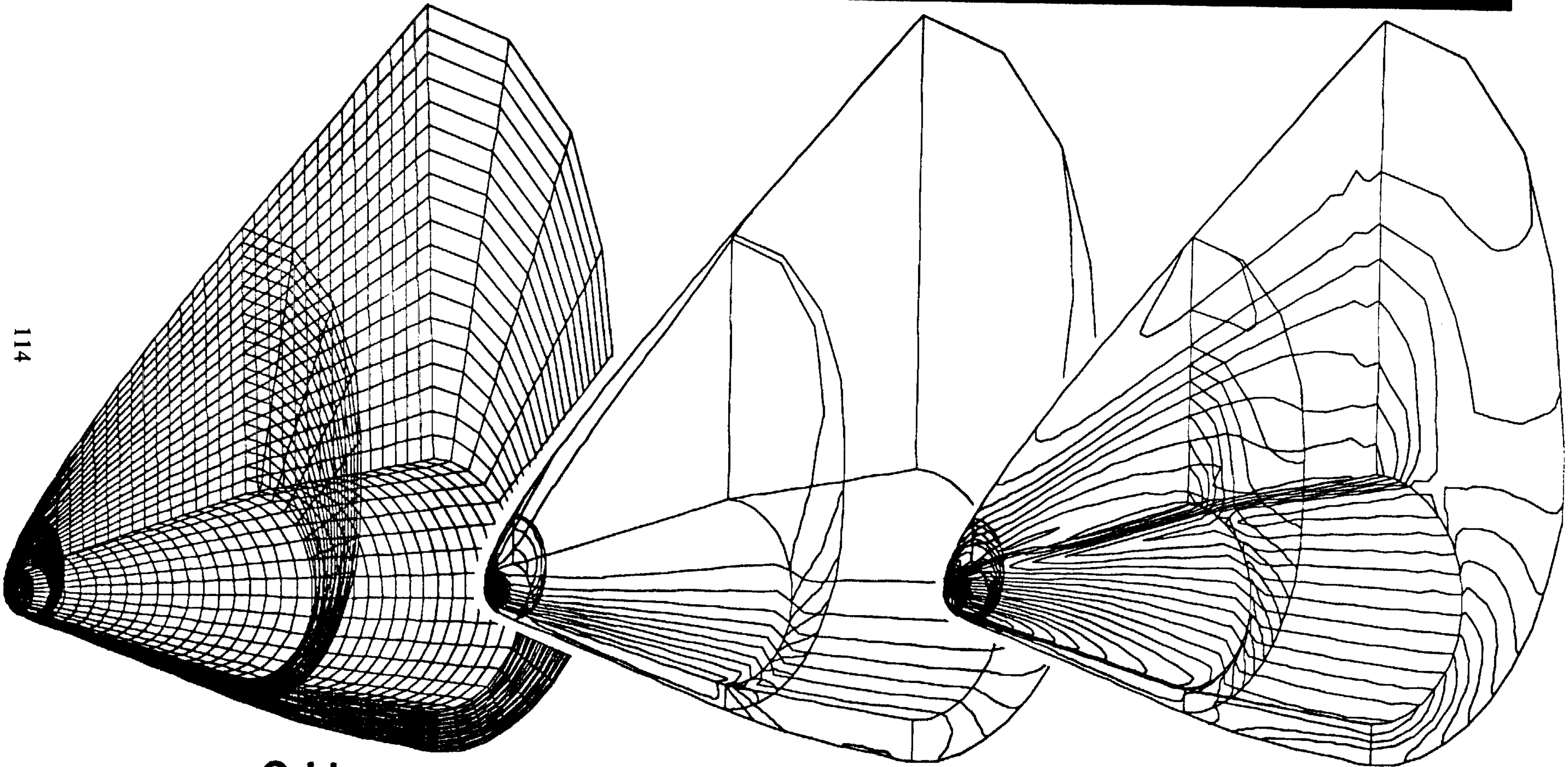
MRSR aerocapture
CFD

Mach 5.997
Alpha 27 deg
Air
Navier-Stokes Solution
45 x 19 x 29 grid
1200 iterations
CFL 2



Mach Number Contours

12.84°/7° Biconic
Mach 6.0   Alpha 27°

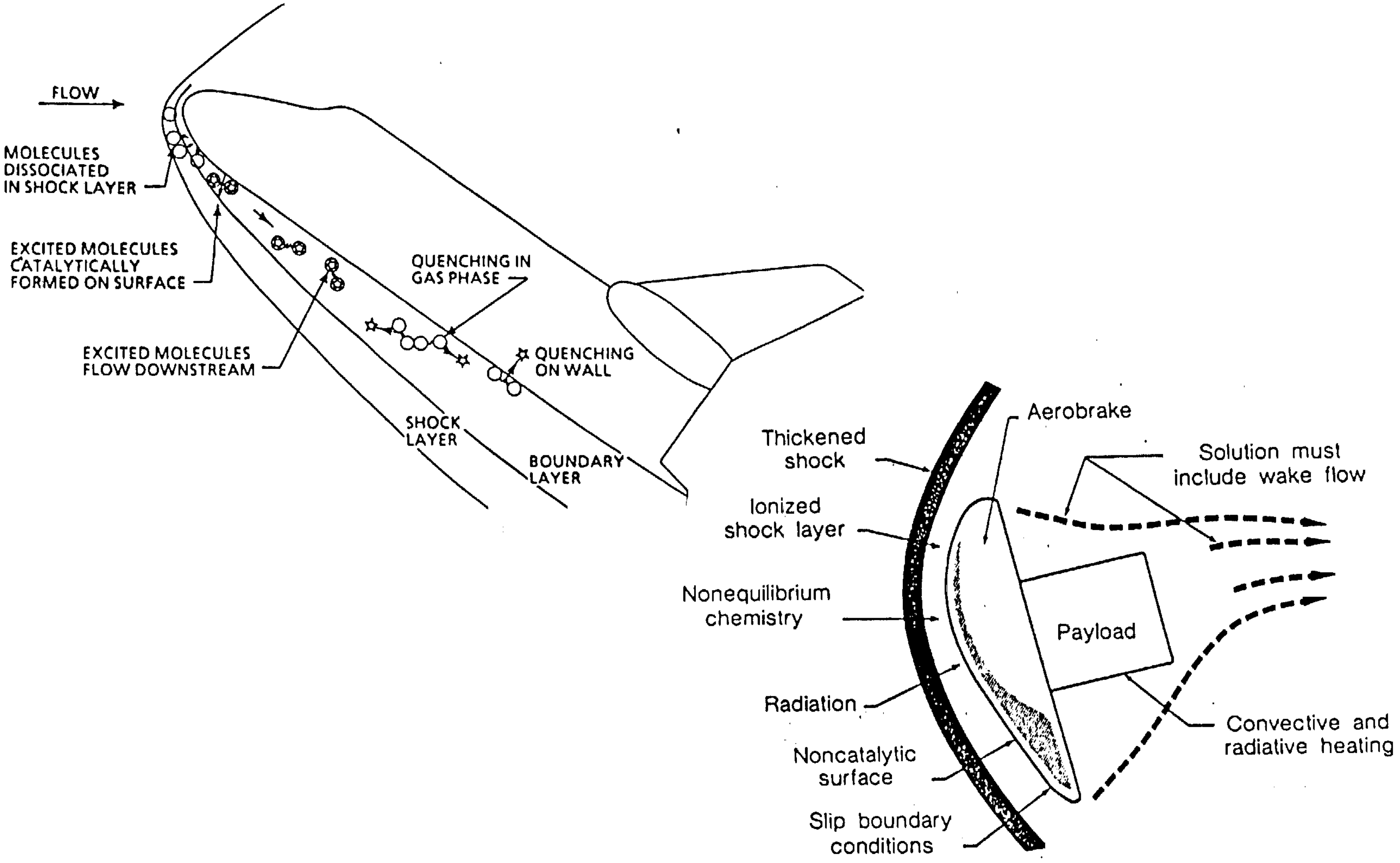


Grid   Pressure   Temperature

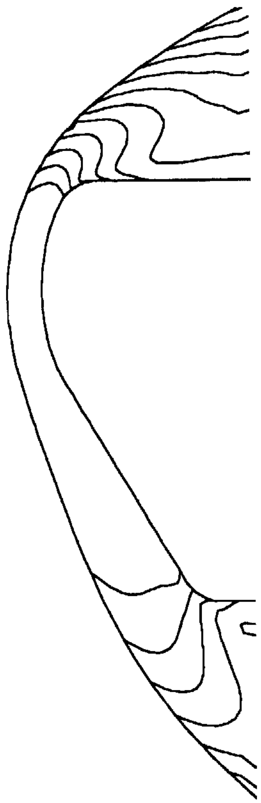
AFE FLOWFIELD SIMULATION

- Objectives:
  - To predict aerodynamic loads and heating distributions for flight conditions
  - To define flowfield environment using appropriate physical models
- Approaches:
  - Developed the VRFLO code for typical aerobraking vehicles
  - Calibrated the code using wind-tunnel data
  - Compared the two-temp and 11-species model results with RAMC data
  - Performed sensitivity studies of different models

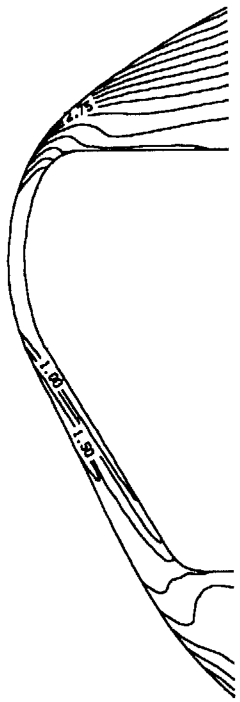
PHYSICAL MODELING FOR HYPERSONIC VEHICLES



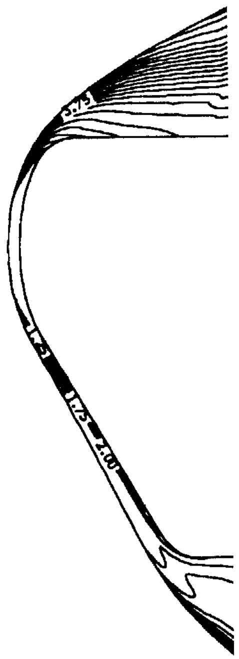
Comparison of Shock Shapes and Subsonic Regions for an AFE Configuration



Ideal Air
Mach 9.81

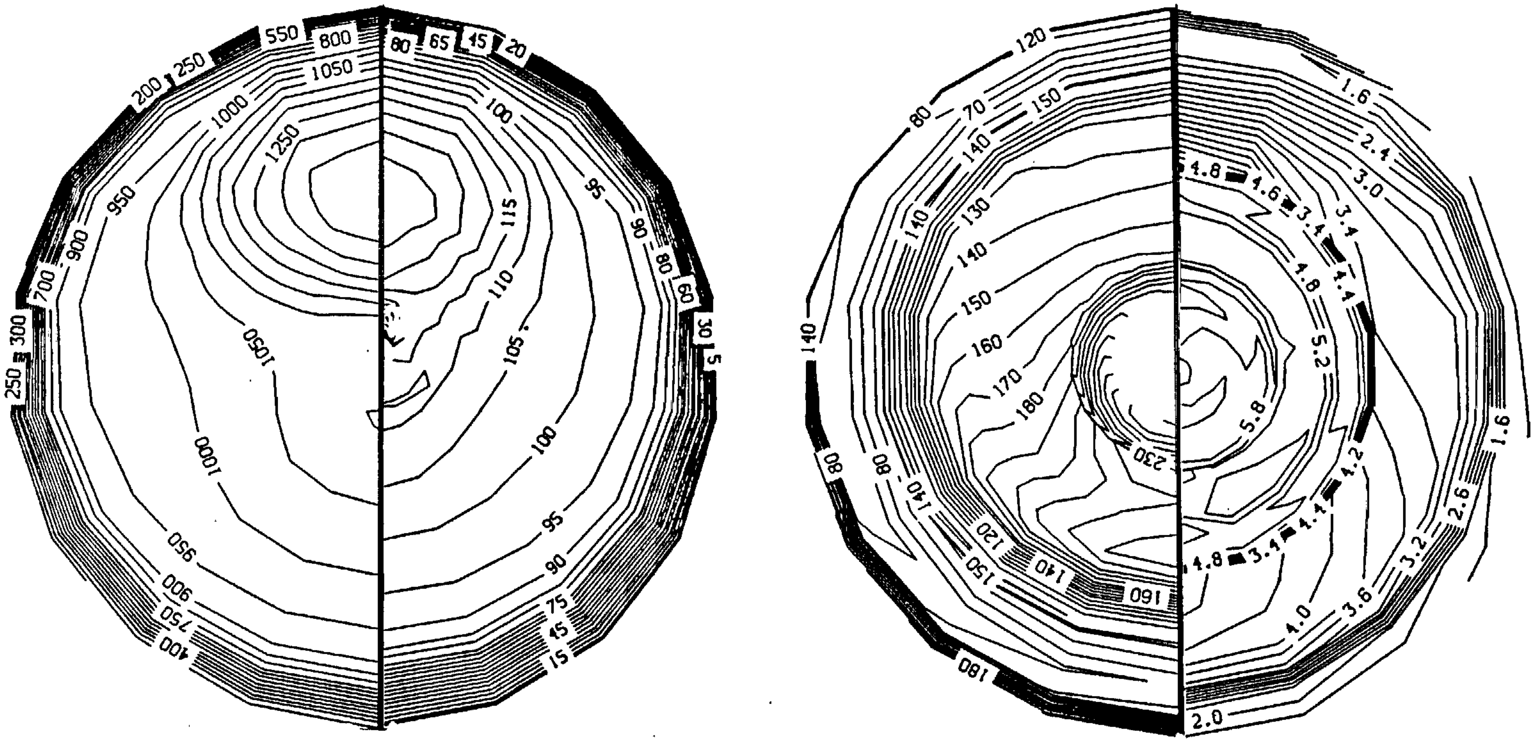


Ideal CF_4
Mach 6.29



Equilibrium Air
Mach 32.0

COMPARISON OF FLIGHT AND WIND-TUNNEL PRESSURES



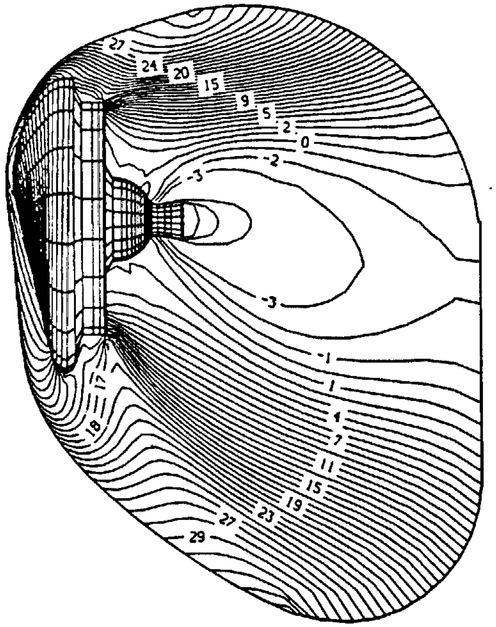
(a) Front view

(b) Rear view

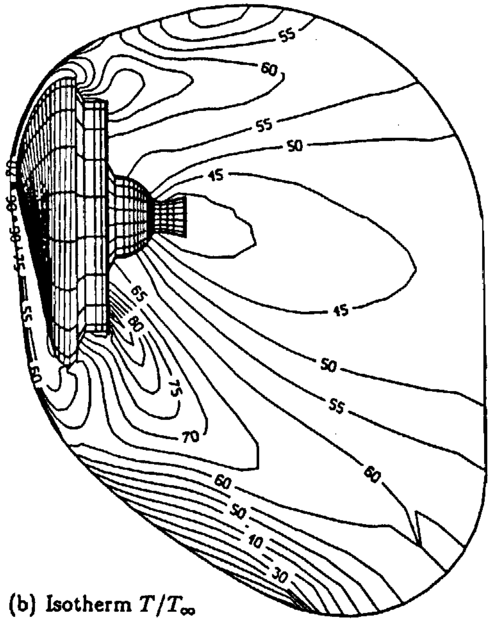
Comparison of flight and wind-tunnel wall pressure distributions (the left side corresponds to the flight case).

CONTOUR PLOTS OF AFE FLIGHT FLOWFIELD

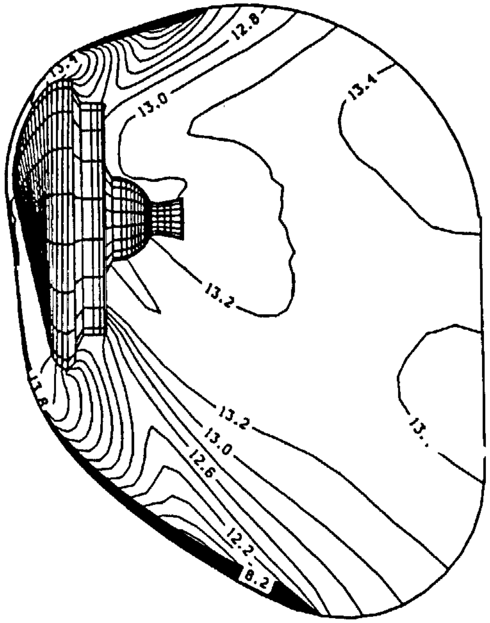
inviscid chemical nonequilibrium flow in the pitchplane



(c) z-componet velocity $w\sqrt{\rho_\infty/p_\infty}$



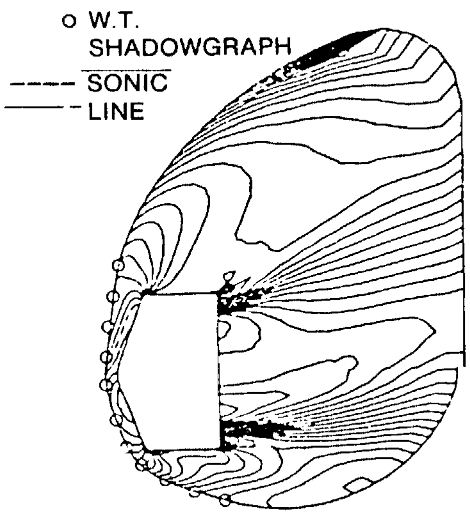
(b) Isotherm T/T_∞

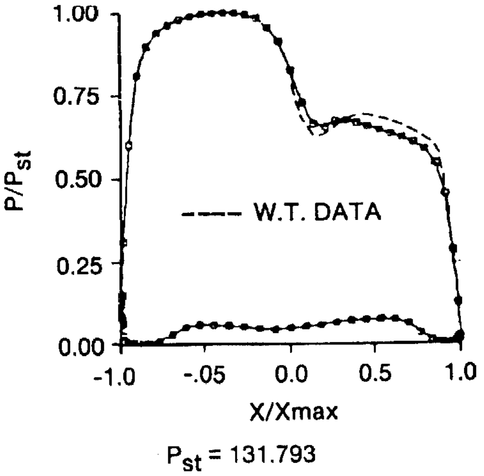


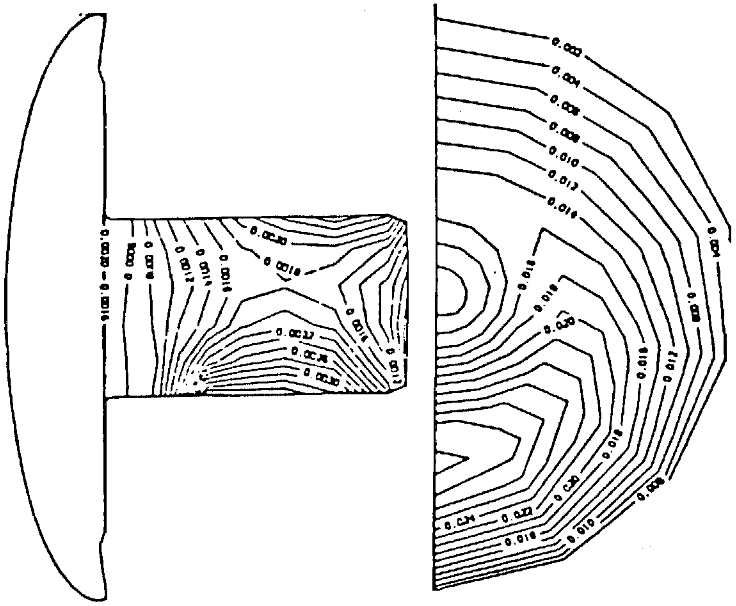
(b) N_e

WALL PRESSURES COMPARISON AND HEATING

LOW L/D CERV







TECHNOLOGY DEVELOPMEMT

- A multi-zonal Navier-Stokes code to predict the Orbiter flow at incidence up to 90 deg
- A thermochemical nonequilibrium PNS code for high L/D configurations
- An Euler code with adaptive, unstructured grid for complex geometry
- Radiation and ablation modeling and code implementation