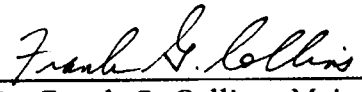
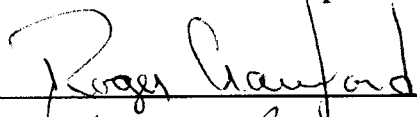
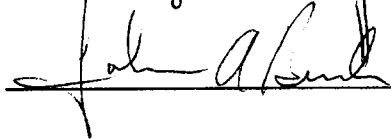


To the Graduate Council:

I am submitting herewith a thesis written by Thomas P. Dye entitled "An Experimental and Computational Investigation of the Flowfield about an Aeroassisted Space Transportation Vehicle at Mach 10". I have examined the final copy of this thesis for form and content and recommend that it be accepted in partial fulfillment of the requirements for the degree of Master of Science, with a major in Aerospace Engineering.


Dr. Frank G. Collins, Major Professor

We have read this thesis
and recommend its acceptance:

Accepted for the Council:


Associate Vice Chancellor
and Dean of the Graduate School

STATEMENT OF PERMISSION TO USE

In presenting this thesis in partial fulfillment of the requirements for a Master's degree at The University of Tennessee, Knoxville, I agree that the Library shall make it available to borrowers under the rules of the Library. Brief quotations from this thesis are allowable without special permission, provided that accurate acknowledgment of the source is made.

Permission for extensive quotation from or reproduction of this thesis may be granted by my major professor, or in his absence, by the Head of Interlibrary Services when, in the opinion of either, the proposed use of the material is for scholarly purposes. Any copying or use of the material in this thesis for financial gain shall not be allowed without my written permission.

Signature Thomas P. Dye

Date December 1, 1993

**AN EXPERIMENTAL AND COMPUTATIONAL INVESTIGATION
OF THE FLOWFIELD ABOUT AN AEROASSISTED SPACE
TRANSPORTATION VEHICLE AT MACH 10**

A Thesis

Presented for the

Master of Science

Degree

The University of Tennessee, Knoxville

Thomas P. Dye

December 1993

DEDICATION

This thesis is dedicated to my wife, Cindy, whose love and support made this work possible. In addition, this thesis is dedicated to my parents, Lou and Barbara Dye, who always supported my many interests and education. I owe my success to you.

ACKNOWLEDGMENTS

I wish to thank my major professor, Dr. Frank Collins, and the other members of my committee, Dr. John Benek and Dr. Roger Crawford, for their help and advice. This work would not have been possible without the support I received from the Arnold Engineering Development Center (AEDC) and NASA Langley Research Center. At AEDC, I wish to thank my supervisor, Mr. Albert Boudreau for his support and encouragement. I also thank Mr. Keith Jordan for his help in building the Computational Fluid Dynamics grid and Dr. Robert Nichols for his help in running the code and answering my many questions. I also thank Mr. Allen Buck for his help in solving many computer related problems. In addition, I wish to thank Mr. Russell Sorrells, Mr. William Strike, and Mr. Robert Spinetti for their advice. At NASA Langley, I wish to thank Mr. Charles Miller for supporting my work over the last 5 years. In addition, I thank Ms. Shiela Wright for her help in data reduction and Mr. Michael DiFulvio for his help in obtaining the heat transfer data. I also wish to thank Dr. Scott Holland and Mr. Hubert Senter for their advice.

ABSTRACT

A combined experimental and computational investigation of the flowfield about an Aeroassisted Space Transportation Vehicle (ASTV) has been performed. The configuration examined was a 70° sphere cone with a cylindrical afterbody at nominal Mach 10, perfect gas conditions. The experimental portion of the investigation was performed in the NASA Langley 31-Inch Mach 10 Tunnel at zero degrees angle of attack. Reynolds number behind the normal shock, based on the forebody diameter was nominally 14,950. Measurements of surface pressure and heat transfer were made on the ASTV forebody, base, and on a cylinder in the near wake. The computational portion of the investigation was performed using the TUFF (Three-Dimensional, Finite-Volume Flow Solver with Fully Coupled Chemistry) code assuming laminar, axisymmetric flow. Pressure and heat transfer distributions on the cylinder were influenced by the impingement of a free shear layer which originated at the forebody corner. The flowfield in the forebody base was influenced by a region of supersonic recirculation upstream of the free shear layer. Experimental and computational surface pressure distributions were in agreement within the experimental uncertainty on the forebody and near the end of the cylinder. Experimental and computational heat transfer distributions were in qualitative agreement on the forebody, but a large disagreement was observed on the cylinder. Previous studies substantiated the experimental results and the disagreement was thought to be due to several factors associated with the computational model.

TABLE OF CONTENTS

CHAPTER	PAGE
1. Introduction.....	1
1.1 Background.....	1
1.2 Description of Present Work.....	4
1.3 Blunt Body Near-Wake Flowfields.....	5
2. Experimental Methods.....	7
2.1 Test Facility.....	7
2.2 Test Conditions.....	8
2.3 Models.....	10
2.4 Instrumentation.....	12
2.5 Data Reduction and Uncertainty.....	15
3. Computational Methods.....	21
3.1 Overview.....	21
3.2 Grid Structure.....	22
3.3 Conditions.....	23
3.4 Computer and Analysis Tools.....	23
4. Results and Discussion.....	25
4.1 Experimental Surface Pressure.....	25
4.2 Experimental Heat Transfer.....	30
4.3 TUFF.....	33
5. Summary and Recommendations.....	43
List of References.....	48
Appendices.....	53

CHAPTER	PAGE
Appendix A - Figures.....	54
Appendix B - Uncertainty Analysis for Pressure Measurements.....	89
Vita.....	93

LIST OF FIGURES

FIGURE	PAGE
1. Typical Flight Profile for ASTV Orbit Transfer Mission.....	55
2. Aeroassist Flight Experiment (AFE) Vehicle.....	56
3. Axisymmetric ASTV Concepts.....	57
4. ASTV Configuration for the Present Study.....	58
5. Simplified Model of Near-Wake Flow of Hypersonic Blunt Body.....	59
6. Diagram of the NASA Langley 31-Inch Mach 10 Tunnel.....	60
7. Photograph of the NASA Langley 31-Inch Mach 10 Tunnel.....	61
8. Pressure Model Details.....	62
9. Photograph of Forebody and Cylinder Pressure Models.....	64
10. Photograph of Cylinder Pressure Model Installed in Wind Tunnel.....	65
11. Heat Transfer Model Details.....	66
12. Photograph of Heat Transfer Model.....	67
13. Heating Rate Time Histories Indicating Range of Signal-to-Noise Ratio.....	69
14. TUFF Grid Structure.....	71
15. Experimental Pressure Distributions at $\theta=0^\circ$	74
16. Experimental Pressure Distributions at $\theta=0^\circ$ and $\theta=90^\circ$	75
17. Experimental Pressure Distributions - Data Repeatability at $\theta=0^\circ$	76
18. Experimental Heat Transfer Distributions at $\theta=0^\circ$	77
19. Experimental Heat Transfer Distributions at $\theta=0^\circ$ and $\theta=90^\circ$	78
20. Experimental Heat Transfer Distributions - Data Repeatability at $\theta=0^\circ$	79
21. Computed Heat Transfer Anomaly on Forebody.....	80

FIGURE	PAGE
22. Comparison of Experimental and Computed Pressure Distributions at $\theta=0^\circ$	81
23. Comparison of Experimental and Computed Heat Transfer Distributions at $\theta=0^\circ$	82
24. Computed Mach Contours for the ASTV Flowfield.....	83
25. Computed Static Pressure Contours for the ASTV Flowfield.....	84
26. Computed Sonic Line for the ASTV Flowfield.....	86
27. Computed Velocity Vectors Colored by Mach Number for the ASTV Flowfield.....	87

LIST OF SYMBOLS

c	model material heat capacity, BTU/lb _m °R
h	enthalpy, BTU/lb _m
i	grid index, streamwise direction
j	grid index, normal direction
k	model material thermal conductivity, BTU/ft·sec·°R
m	mass, lb _m
p	pressure, psia
q	dynamic pressure, psia
$\dot{q}$	heat flux, BTU/ft ² ·sec
r	radius, inches
s	distance along surface, inches
x	horizontal distance, inches
y	vertical distance, inches
t	time, seconds
t ₉₅	95th percentile for two-tailed Student's "t" distribution
B	bias limit
C _D	drag coefficient
C _h	heat transfer coefficient, lb _m /ft ² ·sec
C _p	pressure coefficient
J	Jacobian matrix
Pr	Prandtl number
R	model forebody radius, 2.0 inches
S	precision index

T	temperature, °R
U	uncertainty
Q	arbitrary dependent variable
$\hat{Q}$	arbitrary independent variable
α	angle-of-attack, degrees
β	thermal product, $\sqrt{\rho c k}$, BTU/ft ² ·°R·sec ^{1/2}
θ	model rotation angle, degrees
Γ	velocity gradient
ϕ	model instrumentation array location, degrees
γ	test gas ratio of specific heats
μ	test gas viscosity, slug/ft·sec
ρ	density, slug/ft ³
ξ	grid index in computational space, streamwise direction
ψ	angle between free shear layer and horizontal, degrees

subscripts

aw	adiabatic wall conditions
b	body
c	corner
cyl	cylinder
f	fillet
n	nose
o	initial conditions
ref	reference conditions
surf	surface conditions
t	stagnation conditions
w	wall

∞	freestream conditions
2	post-normal shock conditions

CHAPTER 1

Introduction

1.1 Background

Aeroassist has been considered since the early 1960's as a means to achieve a weight savings in space transportation missions. Aeroassist has been defined as the use of aerodynamic forces to produce an orbital plane change with an expenditure of energy significantly smaller than that associated with an extra-atmospheric propulsive maneuver (Walberg, 1982). The major advantage of using aeroassist as compared to an all propulsive maneuver is the savings in propellant weight which would otherwise be required to accomplish the desired maneuver. For some missions, this weight savings has been estimated to allow nearly double the payload for the same size vehicle (Jones, 1987). Aeroassist maneuvers can generally be described by one of three mission scenarios: 1) synergetic plane change in which a combination of aerodynamic and propulsive forces are used to accomplish the maneuver, 2) planetary aerocapture which involves a single pass into a planet's atmosphere to achieve orbit from deep space, and 3) orbit transfer involving single or multiple atmospheric entries to alter the vehicles altitude or inclination. A wide variety of Aeroassisted Space Transportation Vehicle (ASTV) configurations have been considered for these scenarios including lifting bodies, relatively slender bi-conic shapes, ballutes, and very blunt cones (Walberg, 1982). During the 1980s work primarily focused on the development of orbital transfer vehicles (OTVs) to return to low Earth orbit from a higher orbit (such as geosynchronous orbit) with a very small fuel

expenditure. More recently, NASA's Space Exploration Initiative (SEI) has turned interest toward both orbit transfer and aerocapture vehicle concepts to support a manned Mars mission (NASA 1992). ASTVs characterized by a large frontal area and high drag coefficient, and therefore very low values of the ballistic parameter ($m/C_D A$), have received the most attention for the orbit transfer mission since they appear to minimize the weight associated with the aeroassist system (Jones, 1987).

Aeroassist involves very high altitude, high velocity flight which presents many challenges in the design and use of an ASTV. A typical ASTV orbit transfer mission flight profile is shown in Figure 1*. An understanding of nonequilibrium flow, radiative transfer, wall catalysis, base flow, and aerodynamic performance as experienced by large-angle, blunt cones is critical to the successful design of an OTV (Jones, 1987). Understanding of the base flow is particularly critical since payloads will most likely be carried in the base region of OTVs and the payloads must be designed to survive the aeroassist maneuver. During the 1980's, NASA planned a flight experiment to better understand the flight environment of an OTV. This experiment was known as the Aeroassist Flight Experiment (AFE) (Jones, 1987). The AFE vehicle was a blunted elliptic cone, raked off at 73° to the vehicle centerline, as shown in Figure 2.

Wind tunnel and Computational Fluid Dynamics (CFD) research has also played a key role in the development of ASTVs. Although conventional hypersonic wind tunnels provide a very limited simulation of the ASTV environment, they have proven to be important tools in determining

*All figures may be found in Appendix A

convective heat transfer rates, pressure distributions, aerodynamics, and flowfield structure. In addition, conventional wind tunnel investigations have been identified as an important element in the calibration of CFD codes (Gnoffo, 1989). The majority of recent wind tunnel and computational research has focused on the AFE configuration. An investigation of the experimental pressure and heating distributions on the AFE forebody was conducted, including comparison to Euler and Navier-Stokes solutions (Micol, 1989). Similar studies were conducted using tetrafluoromethane as the test medium to simulate high-temperature effects (vibration, dissociation, and ionization) on AFE aerodynamics and shock shape (Micol, 1987; Wells, 1990). The base flowfield was also examined by determining surface flow and heat transfer distributions on a cylinder in the near-wake of the AFE (Wells 1990). CFD investigations have examined the entire AFE flowfield at flight conditions (Gnoffo, 1987), studied the near-wake flow (Gnoffo and Greene, 1991; Venkatapathy, et al, 1991), and predicted AFE performance in rarefied flow (Celenligil, et al, 1989). These are just a few examples of the rather large database that exists for the AFE configuration.

Axisymmetric ASTV concepts have been more recently examined as candidates for the Mars Environmental Survey (MESUR) probe (Ressler, et al, 1992). An ASTV based on an axisymmetric design is shown in Figure 3. An extensive database exists for these types of configurations, although not nearly as detailed as for the AFE. One axisymmetric configuration that has received the most attention is a 70° blunted sphere-cone with a bluntness ratio (nose-to-body radius, r_n/r_b) of 0.5, a shape similar to the Viking aeroshell (Knox and Best, 1972). An investigation of the pressure and heating distributions on the forebody was conducted with comparison to Euler, Navier-

Stokes, and axisymmetric analog methods (Weilmuenster and Hamilton, 1986; Gnoffo, 1986). Investigations of the heat transfer to the forebody and base region have also been conducted as part of larger parametric studies (Shih and Gay, 1984; Shimshi, 1992). Computational investigations have also included the entire flowfield (Li, 1986). Although many of the computational investigations on axisymmetric ASTVs have included the near-wake region, most of the experimental studies lack detailed near-wake measurements. Larger bluntness ratio (0.7 to 1.0) axisymmetric ASTVs have been examined as candidates for the SEI missions of Earth-return from the Moon (Gnoffo, et al, 1991) and the MESUR probe. However, detailed experimental and computational studies in the near-wake of these larger bluntness ratio configurations have been limited.

1.2 Description of Present Work

Because of recent interest and the limited study of larger bluntness ratio ASTV configurations, an investigation was conducted to provide further insight into the flowfields of these shapes. There were three objectives to this work: 1) determine pressure and heating distributions on a larger bluntness ratio ASTV configuration, including the near-wake, 2) obtain detailed, high-quality experimental data, and 3) understand the near-wake flow structure. These objectives support both ASTV development and CFD code validation. This effort combined both experimental and computational methods to obtain the necessary data. The configuration examined was a 70° axisymmetric cone with a bluntness ratio of 1.0 with a cylindrical afterbody. A drawing of the configuration is shown in Figure 4. Figure 4 also includes the nomenclature

that will be used in describing the various components of the configuration. The configuration has a rounded corner with $r_c/r_b=0.125$ and was examined at nominal Mach 10, perfect gas conditions at zero angle of attack. Although these conditions do not represent ASTV flight conditions, they provide a simplified case for understanding the flow structure and a basis for parametric studies. The experimental portion of the study was conducted in the NASA Langley 31-Inch Mach 10 Tunnel with air as the test medium. Surface pressure and heat transfer measurements were obtained on the model forebody, base, and on a cylinder in the near-wake using arrays of static pressure ports and thin-film resistance gages. Reynolds number behind the normal shock, based on the model forebody diameter was nominally 14,950. The computational portion of the investigation was performed using an algorithm solving the thin-layer Navier-Stokes equations, referred to as TUFF (Three-Dimensional, Upwind-Differenced, Finite-Volume Flow Solver with Fully Coupled Chemistry) (Molvick, 1989). Laminar, axisymmetric flow was assumed and flow conditions matched the wind tunnel freestream values.

1.3 Blunt Body Near-Wake Flowfields

Hypersonic flow over axisymmetric blunt cones is relatively well understood and has been addressed in detail in many textbooks. However, the near-wake flow of such bodies is not as well understood. A brief review of the basic character of these flows is provided as background for the reader.

The literature available on hypersonic blunt body near-wake flows reveals that some controversy exists on their exact structure under various conditions (Berger, 1971). A generally accepted model of these flows was

introduced in conjunction with research on the AFE and is shown in Figure 5 (Wells, 1990). The major features are an inner flow that recirculates in the base region and an outer flow that continues downstream. A dividing streamline (Chapman, 1949) separates this inner and outer flow and terminates at a rear stagnation point. This dividing streamline is embedded in a viscous shear layer that originates as part of the boundary layer that separates from the corner of the forebody. Flow from the opposite sides of the forebody meet near the rear stagnation point. Recompression waves coalesce to form a recompression shock, and the shear layer flow outside the dividing streamline continues downstream through a "neck". Flow inside the dividing streamline turns upstream and recirculates in the base region. The presence of a solid surface in the near-wake, such as a cylinder, obviously alters this flow model. These flows are similar to the flow over a backward facing step. Studies with backward facing steps reveal the same primary features of an inner recirculating flow, free shear layer, outer flow, and a recompression shock (Hama, 1966).

In the case for an ASTV, location of the free shear layer in the near-wake roughly defines the usable volume behind the forebody for placement of a payload. Impingement of the shear layer causes high, localized heating with magnitudes depending upon the impingement angle and the payload radius at the impingement location (Gnoffo, 1991).

CHAPTER 2

Experimental Methods

2.1 Test Facility

The facility used for the present study was the NASA Langley 31-Inch Mach 10 Tunnel. The characteristics and performance of this facility are well documented (Penaranda and Freda, 1985; Miller, 1990). This facility was formerly known as the Continuous Flow Hypersonic Tunnel (CFHT) and was originally designed to run in a blowdown start, continuous flow operation. Since the mid-1970's this facility has operated in blowdown mode only because of energy conservation measures. Dry air test gas is supplied from a storage system with a volume of 875 ft³ and rated for a maximum pressure of 4400 psia. The air is heated by a 12.5 MW electrical resistance heater to a nominal temperature of 1850 °R to prevent liquefaction in the 31" by 31" square test section, as shown in the diagram in Figure 6. The maximum reservoir (stagnation) pressure of the facility is nominally 1500 psia and run times up to 60 seconds can be achieved. The 31-Inch Mach 10 Tunnel is the only hypersonic facility in the United States to have a three-dimensional contoured nozzle (Beckwith and Miller, 1990). This nozzle allows the test section to be free of the centerline disturbance which is characteristic of axisymmetric contoured nozzles. Variation of Pitot pressure across the center 7" of the test core has been determined to be within $\pm 1\%$ of the mean. Because of its uniform flow the facility has been identified as particularly well suited for CFD

code calibration studies (Miller, 1984). A photograph of the facility is presented in Figure 7.

Models are supported on a hydraulically operated, sidewall mounted injection system. This system is capable of injecting the model to the tunnel centerline in less than 0.6 seconds. This fast model injection time is important for heat transfer tests in order to maintain the validity of the assumption of one-dimensional heat transfer used in the data reduction. Prior to injection the model is stored in an enclosure which is isolated from the flow by a sliding door. This enclosure can be rotated to provide access to the model before or after a run.

Freestream conditions are calculated from the measured values of the reservoir pressure and temperature during each run and a curve fit (from an unpublished calibration) used to determine the Pitot pressure. Imperfect-gas effects in the reservoir are accounted for in the calculation.

2.2 Test Conditions

The tests were conducted at a nominal freestream Mach number of 10. The nominal freestream unit Reynolds number was 0.54 million per foot and the nominal angle of attack was zero degrees. The nominal post-normal shock Reynolds number, based on the model forebody diameter, was 14,950. The nominal sideslip angle was also zero degrees. The calculated flow conditions for each run are listed in Table 1.

As mentioned in the introduction, the near perfect gas conditions of the wind tunnel do not match the flight conditions of an operational ASTV. The AFE trajectory, for example, encompassed Mach numbers from 36 to 18 and

Table 1. Run Schedule and Calculated Flow Conditions for Wind Tunnel Tests

Test 241 Forebody Pressure Model

Run	α , deg	θ , deg	$Re_{2,D}$	M_∞	U_∞ , ft/sec	p_∞ , psia	T_∞ , °R	T_t , °R	p_t , psia
4	-0.10	0	14,940	9.76	4695	0.00951	96.2	1846	358
9	-0.13	0	14,950	9.76	4687	0.00948	95.8	1840	357
19	0.08	90	14,983	9.76	4673	0.00944	95.2	1830	355

Test 238 Cylinder Pressure Model

Run	α , deg	θ , deg	$Re_{2,D}$	M_∞	U_∞ , ft/sec	p_∞ , psia	T_∞ , °R	T_t , °R	p_t , psia
7	-0.15	0	14,947	9.76	4697	0.00952	96.2	1847	358
10	-0.14	0	14,913	9.76	4654	0.00931	94.4	1817	350
19	-0.07	90	14,687	9.76	4679	0.00934	95.4	1834	351

Test 281 Heat Transfer Model

Run	α , deg	θ , deg	$Re_{2,D}$	M_∞	U_∞ , ft/sec	p_∞ , psia	T_∞ , °R	T_t , °R	p_t , psia
4	*	0	15,406	9.79	4617	0.00941	92.4	1790	359
5	*	90	14,923	9.78	4633	0.00919	93.2	1801	349
7	*	0	14,833	9.78	4632	0.00914	92.1	1800	347

* α data not available, but believed to be within $\pm 0.1^\circ$

post-normal shock Reynolds numbers from 10 (at entry) to 10,000 (at perigee) (Wells, 1987). The present test conditions were selected to match previous research on the AFE (Wells, 1990).

2.3 Models

Three models were used in the present study: one each for forebody pressure measurements, cylinder pressure measurements, and convective heat transfer measurements over all surfaces. Separate forebody and cylinder pressure models were used since a single model could not accommodate all of the pressure tubing through the support sting. All models had the same external dimensions.

Forebody Pressure Model The forebody pressure model was machined from 15-5 PH stainless steel, heat treated to 1025 °F with a forebody wall thickness of 0.20 inches. It had 31 orifices on the windward forebody and shoulder with 4 additional orifices on the base. Orifice diameters were 0.040 inches on the forebody and shoulder and 0.060 inches on the base. The orifices were distributed along two rays emanating from the forebody center point, along $\phi=0^\circ$ and $\phi=180^\circ$ lines. The forebody orifices were spaced nominally 0.150 inches apart along the surface and the base orifices were spaced nominally 0.200 inches apart. All pressure tubing was routed through the cylinder and exited through the support sting. A diagram showing the model details is shown in Figure 8 and includes the forebody coordinates for reference. The model was machined within a contour tolerance of ± 0.003 inches with a 32 surface finish (microinches rms).

Cylinder Pressure Model The cylinder pressure model had 29 orifices on the cylinder and 6 additional orifices in the base region. Orifice diameters were 0.077 inches along the cylinder and 0.060 inches on the base. The orifices were distributed along the $\phi=0^\circ$ and $\phi=180^\circ$ rays, similar to the forebody model. Orifices at the forebody center point, corners, and on the base were common with the forebody pressure model. The cylinder orifices were spaced 0.300 inches apart and the base orifices were spaced 0.200 inches apart. A diagram showing the model details is also shown in Figure 8. All other external specifications for this model were the same as the forebody pressure model. A photograph of both the forebody and cylinder pressure models is shown in Figure 9. Both models were installed on the tunnel centerline. A photograph of the cylinder pressure model installed in the facility is shown in Figure 10. All models were installed on the tunnel centerline with the nose at the center of rotation of the pitch sector. This allowed the model to remain in the most uniform region of the flow during runs where the angle of attack was varied. This location resulted in a distance of approximately 6 inches between the end of the cylinder and the leading edge of the conical portion of the strut head shown in Figure 10. This distance was believed to be sufficient to prevent interference from the strut head, although a study of interference effects was not conducted.

Heat Transfer Model The forebody and cylinder surfaces of the heat transfer model were fabricated from Macor[™], a machinable glass ceramic and registered trademark of Corning Glass Works. The model was supported by a sting made from 15-5 PH stainless steel, heat treated to 1025 °F. Thin-film resistance gages were applied by a sputtering process (Miller, 1981) to

surfaces that installed within the forebody and on the support sting. These inserts allowed easy replacement of damaged gages between tests. The forebody inserts were cemented in place with a high-temperature rubber compound that was also used to seal all gaps on the model. The cylinder inserts were held in place with screws. The model was instrumented with 111 thin-film gages located along the $\phi=0^\circ$ and $\phi=180^\circ$ rays, similar to the instrumentation arrays on the pressure models. All gages on the model were spaced nominally 0.150 inches apart. A diagram showing the model details is shown in Figure 11. The model was machined within a contour tolerance of ± 0.003 inches with a 16 surface finish on the forebody and a 63 surface finish on all other surfaces. The difference in surface finish between the heat transfer model and pressure models was not expected to have a significant difference on the data. A photograph of the heat transfer model is shown in Figure 12.

2.4 Instrumentation

Pressure The surface pressures were measured with electronically scanned pressure (ESP) silicon sensors (ESP model 708B, manufactured by Pressure Systems, Inc.). Each ESP module contained 32 differential transducers and was located behind the tunnel injection plate, as close to the model as possible to minimize the tubing length and, hence, settling time. The modules were placed in a thermally isolated box to maintain the modules at a constant temperature. A thermocouple placed in the box indicated that the average ESP module temperature varied by no more than 2 °R during each

run. Each module contained internal multiplexing and amplification for scanning at a high data rate. The modules were calibrated prior to each run. The orifices for the forebody pressure model were connected to modules rated for a maximum measured pressure of 5.0 psi (full-scale) in the nose and conical region and for 0.36 psi on the shoulder, corner, and base as shown in Figure 8a. The orifices for the cylinder pressure model were connected to modules rated for 0.36 psi, except for the orifice at the nose which was connected to a 5.0 psi module (see Figure 8b).

Calibration of the ESP system was performed by applying three known pressures to each transducer and measuring the voltage output. The resulting pressure-voltage relationship was fitted with a second-order polynomial that was usually linear over the range of each module. A constant vacuum reference was provided to each module. Each calibration pressure for the 5.0 psi modules was measured by a digiquartz calibration standard, a highly accurate vibrating quartz pressure standard manufactured by Paroscientific, Inc. For the 0.36 psi module, a precision vacuum/pressure sensor incorporating a capacitive diaphragm transducer manufactured by MKS, Inc. was used to measure the calibration pressures. The calibration pressure was measured at the supply point which was approximately 6 ft away from the ESP modules. The reference pressure was measured approximately 3 ft from the ESP modules.

Line losses, leaks, or outgassing in the stainless steel tubing connecting the calibration pressure sources to the ESP modules may have introduced a bias in the calibration, especially for the 0.36 psi module due to the very low pressures. These effects may also have been present for the reference

pressure tubing and could have introduced an error if the reference pressure at the ESP module changed from the calibration value during to the run. This was a strong possibility since flexing of these tubes was required during model injection. An error introduced in the calibration would have shifted the level of the measurement from its true value. Due to the significantly higher pressures associated with the calibration of the 5.0 psi module, these effects were considered negligible. Tests conducted after the present study (Holland, et al, 1991) accounted for these effects by measuring the calibration pressure very close to the ESP modules and through additional pre-test checkout.

Heat Transfer Thin-film resistance gages were used to measure the convective heat transfer rate on the model surface. Each gage had a palladium sensing element in a serpentine pattern which measured approximately 0.040 inches by 0.050 inches and was approximately 1000 Å thick. Each gage was covered with a layer of Al_2O_3 approximately 5000 Å thick to provide protection against dust, rubbing, or particulates in the flow. Each gage's resistance was nominally 100 Ω at room temperature. The resistance of each gage varied with temperature as determined from a calibration in a well-stirred oil bath at temperatures from 535 °R to 685 °R. During a run a constant current was applied though each gage and the voltage was recorded as the temperature changed on the model surface. Thus, the resistance of each gage was known and the surface temperature could be determined. A current of either 1 mA or 4 mA could be supplied depending on the expected heating level. Larger current values were not used to avoid ohmic heating of the gages. Gain output signals could be amplified by factors of 2, 5, 10, 20, 50, and 100 to a maximum of 1000 mV, depending upon expected heating levels. In the present study, constant-current and amplifier combinations were selected in an attempt to

provide nominal mid-range signal output. A 38 Hz filter was included in the gage circuit to cutoff higher frequency-induced oscillations.

2.5 Data Reduction and Uncertainty

Pressure Instrumentation Data Reduction The calibration coefficients for each pressure transducer were stored in the Hewlett-Packard 9000 Series 300 data acquisition computer. The pressure was calculated from the measured voltage using the calibration coefficients. Data was acquired from the ESP modules at a rate of 4 samples/second for the forebody test and 2 samples/second for the cylinder test. Final reduced pressure coefficients included in the present work represent a pressure value averaged over 0.25 to 0.5 seconds depending upon the sampling rate. The data represent steady state values due to the low sampling rate.

Pressure Instrumentation Uncertainty A particular concern in making low pressure measurements is the instrumentation settling (or lag) time. The required settling time primarily depends on the measured pressure, transducer volume, and the length and diameter of the tubing from the orifice to the transducer. Settled data was obtained by injecting the model into the flow and waiting approximately 15 seconds for the pressures to reach a constant value prior to initiating data acquisition. Data acquisition typically occurred over a period of 10 seconds. Settling time for orifices on the forebody was within 1 second while orifices on the base required up to 10 seconds. Base pressures were estimated to be settled to within 1% of the measured value (for

the 0.36 psi module). This was determined by comparing the data to settled values obtained from a checkout run where data was acquired for 60 seconds.

The uncertainty in the experimental pressure coefficients was determined from instrumentation specifications and the Taylor's series expansion method (Abernethy and Thompson, 1973). A detailed discussion of the pressure measurement uncertainty is presented in Appendix B. The uncertainty of the pressure coefficients on the forebody was determined to be within $\pm 2.0\%$, except at $s/R=1.125$ (the corner) where the uncertainty was within $\pm 7.0\%$. On the cylinder the uncertainty in pressure coefficient was determined to be within $\pm 7.0\%$ for $x/R < 1.85$ and within $\pm 2.1\%$ for $x/R > 1.85$. The uncertainty in pressure coefficient on the base was determined to be within $\pm 7.0\%$.

Thin-film Gage Data Reduction Heat transfer rates were determined from the measured temperature-time histories using a technique which approximates the gage voltage-time history by a piecewise liner expression and assumes a thick wall with one-dimensional heat transfer (Cook and Felderman, 1966). This method has been shown to be very accurate in determining the heat flux for high data sampling rates (50 samples/sec) as was the case for the present experiments (Miller, 1981). The Macor substrate thermal property ($\beta = \sqrt{\rho c k}$) was assumed to be constant at 0.098 BTU/ft²·sec^{1/2}·°R.

For each run a "time zero" was selected to be 0.06 seconds prior to a detectable signal increase resulting from model injection into the tunnel flow. The baseline signal was taken to be the average signal over a 0.5 second interval preceding time zero. Data was acquired at a rate of 50 samples/second

and each channel was filtered at 38 Hz. The heating rate was determined over a 3 second run time beginning at time zero, and a least-squares curve fit was applied to the heat flux-time history for 1.3 seconds beginning 0.5 seconds after time zero (immediately after the model became stationary on the tunnel centerline). The signal-to-noise ratio during a run depended on the signal strength, which varied with gage location. Examples of heat flux-time histories are presented in Figure 13, where Figure 13a shows the result of a high heating signal and Figure 13b shows the result of a very low heating signal. Notice that the change in surface temperature (ΔT_{surf}) is significantly higher for the high heating signal case. In addition, note the rise in heat flux with time in Figure 13c, a mid-range heating signal. The increase in heat flux in Figure 13c indicates that conduction effects may be present and that the one-dimensional heat transfer assumption may have been violated. This is not surprising since the output in Figure 13c is from a corner gage where the surface was highly three-dimensional and exposed to heating from the shoulder, corner, and base. The data presented in this work represent values selected at the beginning of the heat flux-time history curve fit in an attempt to avoid these conduction effects.

The fast-response, surface mounted thin-film gages make it possible to infer heating rates even with ΔT_{surf} as low as that indicated in Figure 13b. Although each gage's output was filtered, the large fluctuations shown in Figures 14b and c may be the result of flow unsteadiness. The corner and base flow might be expected to be unsteady since these regions are dominated by free shear layer detachment and a recirculation region, respectively. Further analysis of the flow unsteadiness was not included as part of the present study.

The computed heat transfer rates were reduced to heat transfer coefficient form and non-dimensionalized by a reference heat transfer coefficient. The heat transfer coefficient was determined from the following equation:

$$C_h = \frac{\dot{q}}{h_{aw} - h_w}$$

where $h_{aw} = h_{t,2}$ was assumed for the hemispherical nose of the model. The stagnation point heat transfer coefficient for a sphere was used as the reference value. The reference sphere radius was 2 inches, which was the radius of the model forebody. A correlation of numerical results for undissociated airflow resulted in the following equation for the reference heat flux (Fay and Riddell, 1958):

$$\dot{q}_{ref} = \frac{0.76}{Pr_w^{0.6}} (\rho_{t,2} \mu_{t,2})^{0.4} (\rho_w \mu_w)^{0.1} (h_{t,2} - h_w) \Gamma^{0.5}$$

where for air $Pr_w = 0.71$, and Γ is the wall velocity gradient approximated by the Newtonian expression,

$$\Gamma = \frac{1}{r_{ref}} \sqrt{\frac{2(p_{t,2} - p_\infty)}{\rho_{t,2}}}$$

and the subscript w denotes quantities at the wall where the temperature was set to 540 °R. The quantities ρ_w and $\rho_{t,2}$ were determined from the ideal gas law (using the stagnation pressure) and the viscosity was determined from

Sutherland's formula (Ames Research Staff, 1953). The reference heating rate was computed as part of the calculation of facility freestream properties.

Heat Transfer Uncertainty The possible sources for uncertainty in the heat transfer rate inferred from the gages have been addressed in detail by previous studies (Miller, 1981). In addition, other studies have addressed the effect on the heating rate resulting from variations in Macor thermal properties (Miller, et al, 1985). Based on this past work, the primary contributors to the overall uncertainty in the present tests were the uncertainty in the thermal properties of the substrate, conduction effects, and the poor signal-to-noise ratio on the base. A more accurate determination of the thermal properties of the substrate was used in the data reduction as recommended by previous work to account for some of this uncertainty (Miller, et al, 1985). In addition, each gage's resistance was measured and compared with the calibration resistance at a nominal temperature of 540 °R prior to each run to ensure that each gage was functioning properly. Gages that had a resistance significantly different than the calibrated value were not included in the data reduction.

Estimation of uncertainty for the thin-film gages was approached in a more subjective manner than for the pressure measurements and relied on past research and results of similar tests. This approach was necessary due to the complexity of the data reduction procedure and the lack of a specified precision for the thin-film gages. Further research into the uncertainty of thin-film measurements is necessary before a quantitative uncertainty value can be quoted with confidence. Based on past research and similar tests, the estimated uncertainty for the non-dimensionalized heat transfer coefficient on the forebody nose and conical section was determined to be within $\pm 8\%$.

The uncertainty for the forebody shoulder, corner, base, and cylinder was determined to be within $\pm 12\%$.

CHAPTER 3

Computational Methods

3.1 Overview

The code used in the present study was the Three-Dimensional, Upwind-Differenced, Finite-Volume Flow Solver with Fully Coupled Chemistry, also referred to as TUFF (Molvick, 1989).

The TUFF code employs a cell-centered, finite-volume approach to ensure the scheme is fully conservative. Upwind differencing based on the Chakravarthy-Osher flux vector extrapolation (Chakravarthy and Osher, 1985) is used in a thin-layer Navier-Stokes time-marching scheme. Recall that the thin-layer approximation to the Navier-Stokes equations is made by neglecting all viscous transport terms except those normal to a viscous boundary. The numerical scheme is made implicit by fully linearizing all of the fluxes and source terms and by using a modified Newton iteration to eliminate any linearization and approximate factorization errors. Although the code has equilibrium, frozen, and nonequilibrium models for air, only the perfect gas model was used for the present results. In addition, the flow was assumed to be axisymmetric and fully laminar.

The code required a total of 28,631 iterations to converge. A small local time step, based on a CFL number of 0.001, was used at the start of the solution to improve stability. The time step was then increased as the solution progressed until CFL=1.0. Convergence was assumed after the maximum residual in the governing equations had decreased by five orders of magnitude

and no discernible differences in the heat transfer results were observed between iterations.

3.2 Grid Structure

A mesh was generated over a one-half plane of the ASTV model configuration using the EAGLE interactive grid generation program (Dietz and Evans, 1990). The coordinates used for the forebody surface were the same as those used to fabricate the wind tunnel models. The cylinder, base-cylinder fillet, and the rearward facing step on the cylinder were included in the numerical model. The mesh was divided into nine domains for simplicity. This allowed increased grid density in strong gradient regions such as at the forebody shoulder and corner, in the vicinity of the free shear layer, and downstream of the rearward facing step on the cylinder. The grid is shown in Figure 14a. Each domain in Figure 14a is shown in a different color. An enlarged view of the forebody and near-wake grid is shown in Figure 14b. Figure 14c shows the size of each domain. The grid shown in Figure 14a consisted of a total of 37,000 points. Domain 5 was increased in size and Domain 9 was added after the solution had started due to movement of the bow shock beyond the original boundary. Determination of normal spacing of the grid at a viscous boundary was based on the "y+" parameter, discussed in more detail in Chapter 4. The normal spacing at the walls was typically 0.00004 inches, resulting in $y^+ < 0.1$.

3.3 Conditions

The freestream conditions for the code were the same as for Run 7 of the cylinder model pressure test (Test 238), as shown in Table 1. The wall temperature was assumed to be constant at 360 °R. This was different than the experimental wall temperature, which was nominally 540 °R. This difference in wall temperature was not intentional. During difficulties in starting the code the wall temperature was lowered to facilitate numerical stability. The wall temperature was to be raised to the correct value of 540 °R after starting of the solution. Unfortunately, the wall temperature was not increased and the discrepancy was not discovered until the solution had converged. Conditions for Domain 5 were initialized by interpolating from freestream to stagnation conditions between the freestream and wall boundaries. Domains 1-4, 6, and 7 were initialized by interpolating from zero density and velocity in the base region to freestream conditions (all at constant pressure) at the exit of Domain 6. Domain 8 was initialized in a similar manner due to the recirculation expected after the rearward facing step.

3.4 Computer and Analysis Tools

The code was run on a Convex 3840 vector computer located at the Arnold Engineering Development Center, Arnold Air Force Base, Tennessee. Post-processing of the code output to determine surface pressure and heat transfer coefficients was performed on a Silicon Graphics Iris workstation.

Flowfield contour and vector plots were produced using the PLOT3D program (Walatka and Buning, 1989).

CHAPTER 4

Results and Discussion

4.1 Experimental Surface Pressure

Static pressure distributions for the forebody, cylinder, and base are presented in Figure 15 in terms of the local pressure coefficient as a function of non-dimensional distance along each surface. The model orientation is $\theta=0^\circ$.

Forebody Pressure The forebody pressure distribution shown in Figure 15a demonstrates the expansion of the flow from the stagnation point over the spherical nose and around the shoulder of the forebody. A slight overexpansion of the flow from the nose to the conical section was observed near $s/R=0.3$. This behavior is typical of slender cone flowfields where the flow will overexpand past the nose and then recompress so that the pressure approaches the sharp cone value further downstream. The flowfield for the present configuration was anticipated to be significantly different, however, since the forebody flow was likely dominated by a much larger subsonic region which encompassed a large portion of the forebody. This would place the location of subsonic to supersonic flow transition, also referred to as the "sonic line", on the shoulder. The flow continued to expand over the conical section due to the influence of the strong favorable pressure gradient at the shoulder. The measured stagnation pressure agreed with the facility Pitot pressure (calculated as part of the flow conditions) to within 0.1%. The pressure distributions from the $\phi=0^\circ$ and $\phi=180^\circ$ instrumentation rays on the

forebody agreed within the experimental uncertainty except for the orifices on the corners ($s/R=1.125$), indicating good flow symmetry. The corner orifice on the $\phi=180^\circ$ ray measured a pressure coefficient of 0.0155 and the corner orifice on the $\phi=0^\circ$ ray measured 0.0114, a disagreement of 36% and indicating a slight positive angle of attack. The indicated angle of attack during the run was -0.10° which is within the experimental uncertainty of the setting. These differences could also have been due to small differences in the location of the orifices.

The pressure instrumentation was checked prior to the run by exposing the model to a quiescent low pressure environment and comparing the measured pressures on the model. The model was injected into the tunnel when it was at a low pressure (nominally 0.1 psia) prior to the facility operating and data was acquired as though a normal run had occurred. Each orifice should have ideally measured the same pressure within the experimental uncertainty. During this checkout the pressure measured by the two corner orifices agreed to within 0.4% at a nominal pressure of 0.074 psia. This indicated that the disagreement during the run was likely due to flow phenomena as opposed to an instrumentation problem. However, leaks or outgassing within the orifice tubing cannot be ruled out since the pressure during the run was substantially lower than during the instrumentation check. An instrumentation check at very low pressures (near the tunnel freestream static value) would provide an improved indication of those orifices subject to leaks or the effects of outgassing.

If the flow within the shock layer is assumed to expand isentropically from the stagnation region, it will become supersonic when the pressure

coefficient becomes 0.5175 times the stagnation point value. Based on this criteria, supersonic flow occurred when $C_p < 0.9509$ as indicated in Figure 15a. This occurred at approximately $s/R=1.0$, which was on the shoulder for the present configuration. This confirms the anticipated location of the sonic line, as discussed earlier.

Cylinder Pressure Figure 15b shows the pressure coefficient distribution for the cylinder. The pressure coefficient increased with distance along the cylinder indicating influence of the free shear layer on the cylinder. Although the pressure increased less after $x/R=2.5$, a distinct peak in pressure was not observed as would be expected with an impingement (flow stagnation) point. Although there was some disagreement between several orifices on the $\phi=0^\circ$ and $\phi=180^\circ$ instrumentation rays (particularly near $x/R=1.0$), the measurements were within the experimental uncertainty indicating good flow symmetry. The higher pressure on the $\phi=0^\circ$ ray near $x/R=1.0$ and the generally lower pressures for that ray for $x/R>2.0$ indicated that the model was at a slight positive angle of attack. These differences should not be over emphasized, however, since they fall within the experimental uncertainty band of the data. The peak pressure coefficient on the cylinder surface was determined to be 4.4% of the forebody stagnation point value at a location greater than $x/R=2.5$.

Base Pressure Measured pressure coefficients on the forebody base are shown in Figure 15c. The measured pressure was relatively constant on the base and the magnitude was near the minimum value on the cylinder. The $\phi=0^\circ$ and $\phi=180^\circ$ distributions agree within the experimental uncertainty except for the orifices at $y/R=0.575$. The pressure measured by these two orifices agreed within 0.3% during the instrumentation check at a pressure of

0.058 psia. Therefore, the difference observed appeared to be due to the flowfield indicating slightly asymmetric flow in the base region. However, problems such as leaks and outgassing cannot be ruled out since the instrumentation check occurred at a higher pressure than was measured during the run. The average base pressure coefficient was determined to be 0.83% of the forebody stagnation point value.

Pressure Test Compatibility As mentioned earlier, two separate tests were performed to obtain the pressure data. The data obtained from the common orifices of the two models were compared to determine if the tests occurred at similar conditions and if the data could be presented as a single set. The stagnation point pressure coefficient for the two runs shown in Figure 15 differed by 0.85%. The average base pressure coefficient differed by 12.1%, an amount just beyond the experimental uncertainty. The corner pressure coefficients were 78% greater for the cylinder pressure model. During the instrumentation check the corner orifices for the cylinder pressure model measured pressures that were 7.4% greater than the quiescent tunnel pressure. This indicated the presence of a leak or a bad transducer for these orifices. The agreement of the stagnation and base orifices between the two tests indicated that the conditions for the two tests were similar and that the data can be considered part of a single simulation. The large disagreement between the corner pressures may be due to instrumentation problems or other factors such as orifice placement and/or model contour fidelity.

Axisymmetric Flow Analysis During each test the model was rotated to $\theta=90^\circ$ (with the model yaw set to zero) to further examine the possibility of asymmetric flow. If the model had been at a slight angle of attack during the

runs at $\theta=0^\circ$ the effects of asymmetric flow would be most evident. By rotating the model to $\theta=90^\circ$ the effect of angle of attack should be minimized.

Figure 16 presents a comparison of the pressure data obtained at model rotations of $\theta=0^\circ$ and $\theta=90^\circ$. The forebody pressure coefficients agreed within the experimental uncertainty for both model rotations as shown in Figure 16a, except at the corners. The corner pressure coefficients for the $\theta=90^\circ$ run disagreed by 19% and, as with the $\theta=0^\circ$ data, the $\phi=180^\circ$ orifice measured higher. In addition, both corner orifices were over 34% higher for the $\theta=90^\circ$ run. The results on the model forebody indicated that the disagreement between the corner orifices was most likely due to instrumentation problems and not asymmetric flow (due to angle of attack) for the $\theta=0^\circ$ model rotation. The results shown in Figure 16b support this conclusion. The measurements from both model rotations on the cylinder agree within the experimental uncertainty and the slightly higher pressures observed near $x/R=1.0$ for the $\phi=0^\circ$ ray were repeated for the $\theta=90^\circ$ run. The base pressure coefficients shown in Figure 16c were similar for both model rotations, supporting the conclusion that the flow was symmetric. The differences in pressure coefficient at the common orifices between the two models at the $\theta=90^\circ$ model rotation were of similar magnitude to that discussed for Figure 15.

Data Repeatability Figure 17 shows the repeatability of the pressure data at the $\theta=0^\circ$ model rotation. On the forebody (Figure 17a), the data was repeatable within the experimental uncertainty. Again, the pressure coefficient for the $\phi=180^\circ$ ray at the corner was higher for the repeat run (Run 9) and the data for both corner orifices for Run 9 were at a higher level than for Run 4. The cylinder measurements (Figure 17b) indicated a bias in

the data for the repeat run. The data for the repeat run (Run 7) was shifted substantially lower, but the two runs agreed qualitatively. Note that the shift in the data appears to be largest at the lower pressures. This would coincide with a shift in the low-end calibration pressure supplied to the ESP module, a value near freestream static pressure. This low-end point would be the most likely calibration pressure to be affected by line losses, leaks, or outgassing. The bias indicated in the cylinder data also was observed in the base region (Figure 17c), although the agreement between the $\phi=0^\circ$ and $\phi=180^\circ$ rays was improved closer to the cylinder ($y/R < 0.7$) for the repeat run. The bias in the data for the cylinder and base region reduces the confidence in the accuracy of the low pressure data on these surfaces, however, the agreement shown in Figure 16 substantiated the data for Run 10 (also shown in Figure 15). Runs with the cylinder pressure model at a rotation of $\theta=45^\circ$ also substantiated the data for Run 10 (this data is not included in the present work).

4.2 Experimental Heat Transfer

Heat transfer distributions for the forebody, cylinder, and base are presented in Figure 18 in terms of the local heat transfer coefficient ratio as a function of distance along each surface. The heat transfer coefficients were non-dimensionalized by the Fay-Riddell stagnation point heat transfer coefficient, as described in Chapter 2.

Forebody Heat Transfer The forebody heat transfer distribution is shown in Figure 18a. The model orientation is at $\theta=0^\circ$ and data for the $\phi=0^\circ$ and $\phi=180^\circ$ instrumentation rays are presented. The stagnation point heat transfer

coefficient was measured to be 14% higher than the Fay-Riddell value. The heat transfer was reduced as the flow moved over the nose and cone section. This reduced heating was expected since the boundary layer growth was greatest in this region, reducing the temperature gradient at the wall. Flow expansion in this region was only moderate and did not counteract the reduced heating caused by growth of the boundary layer. This trend was reversed on the shoulder. Heating increased as the shoulder was approached due to the rapidly expanding flow which reduced the boundary layer thickness, increasing the temperature gradient and heat transfer rate. This type of heat transfer distribution has been well documented and observed on the AFE configuration (Micol, 1989) and on a similar configuration at $\alpha=10^\circ$ (Gnoffo, et al, 1991). The distributions for the $\phi=0^\circ$ and $\phi=180^\circ$ instrumentation rays agreed within the experimental uncertainty, indicating good flow symmetry. The gage near $s/R=0.2$ appeared to read slightly low, although pre-test calibrations and instrumentation checks indicated that the gage was in proper working condition. The corner gages ($s/R=1.05$) agreed to within $\pm 5\%$ for all runs indicating good flow symmetry.

Cylinder Heat Transfer Heat transfer distributions for the cylinder are presented in Figure 18b. The heating on the cylinder increased with distance and reached a peak value at $x/R>2.5$. This trend indicated influence of the free shear layer on the cylinder. A distinct peak in heating was not observed. A large disagreement was observed between the $\phi=0^\circ$ and $\phi=180^\circ$ instrumentation rays after $x/R=2.5$. The $\phi=0^\circ$ ray did not indicate a maximum in heating, while the $\phi=180^\circ$ ray indicated that peak heating occurred at $x/R=2.6$. Although this disagreement was within the experimental uncertainty, the trends indicated

that the model was at a slight positive angle of attack. Before and after the run, however, the angle of attack of the model was verified to be zero with an inclinometer. This disagreement could also be caused by differences in the shoulder and corner contours on the forebody. Recall that the instrumented shoulders and corners consisted of inserts that were cemented into the forebody. Misalignment of these inserts could also have caused the differences in the cylinder heating distributions. The maximum heating on the cylinder was determined to be 19.2% of the stagnation point value and occurred at a location of $x/R=2.69$.

Base Heat Transfer Base heating distributions are shown in Figure 18c. In general, heating increased with distance from the cylinder. This may have been caused by increased shear at the wall caused by a recirculation in the base region. The agreement between the $\phi=0^\circ$ and $\phi=180^\circ$ rays was within the experimental uncertainty except for the gage at $y/R=0.575$. The maximum heating in the base region was determined to be 1.4% of the stagnation point value and occurred at a location of $y/R=0.95$.

Axisymmetric Flow Analysis The heat transfer model was also rotated to $\theta=90^\circ$ (with the model yaw set to zero) to further examine the possibility of asymmetric flow. Forebody heating for both the $\theta=0^\circ$ and $\theta=90^\circ$ model rotations are shown in Figure 19a. The data agreed within the experimental uncertainty, indicating symmetric flow. The cylinder heating distribution in Figure 19b for the $\theta=90^\circ$ case was lower for $x/R>2.0$. If the model were to have a slight angle of attack, the heating for the $\theta=90^\circ$ case would be expected to be lower since a crossflow over the cylinder gages would have reduced the heat transfer. However, if the model was at slight angle of attack the agreement

would be expected to be better than shown in Figure 19b between the $\phi=0^\circ$ and $\phi=180^\circ$ rays for the $\theta=90^\circ$ case. Again, it is important to note that the data presented in Figure 19b agreed within the experimental uncertainty, so it was difficult to draw firm conclusions regarding the model angle of attack. The data in the base region also agreed within the experimental uncertainty as shown in Figure 19c. Note the improved agreement for the gage at $y/R=0.575$ for the $\theta=90^\circ$ case.

Data Repeatability Figure 20 shows the repeatability of the heat transfer data at the $\theta=0^\circ$ rotation. Good repeatability was observed for all surfaces. Data repeatability was within $\pm 3\%$ for the forebody and cylinder and was within the experimental uncertainty for the base.

4.3 TUFF

Anomaly Analysis During an initial check of the results of the calculated heat transfer on the forebody, it was observed that large fluctuations in the heat transfer distribution were present near the stagnation point. This initial result is shown in Figure 21 by the dashed line. This phenomena has been observed on similar configurations (Grasso and Gnoffo, 1990) and has been attributed to a coordinate singularity along the stagnation streamline (Blottner, 1988). The stagnation streamline lies along the body axis and is a boundary of the computational domain, located at $\xi=0$ and $j=1$. Although the physical variables are well behaved in this area, the Jacobian $J=C/\xi+\dots$ goes to infinity and results in a coordinate singularity (Blottner, 1988). The Jacobian on the axis is obtained through extrapolation to overcome this

singularity. This extrapolation results in a considerable error since a finite value is obtained for the Jacobian instead of an infinite value. The dependent variable $Q = J\hat{Q}$ at the singularity is not obtained from the governing equations but from the interpolation of Q 's near the body axis. These Q 's are used with the Jacobians to obtain the dependent $\hat{Q}$ variables. The $\hat{Q}$ variables are small but not the correct zero value. Methods to overcome this problem are localized grid restructuring (Grasso and Gnoffo, 1990) and use of a modified difference scheme which expands a derivative in the x-momentum equation (Blottner, 1988) which essentially separates the Jacobian from the finite differences of the variables in the problem terms. In the present study this problem was addressed by increasing the entropy smoothing parameters referred to as "epc" and "epu" in the code. These parameters, also known as the entropy fix, slightly alter the coefficient of numerical viscosity in the first-order upwind flux function to enable the function to satisfy an entropy inequality (Yee, 1987). The magnitude of epc and epu can be viewed as a measure of the amount of numerical dissipation for the flux. The case of epc=epu=0 is the least dissipative, while larger epc's and epu's result in greater dissipation and smearing of discontinuities. Typically these two parameters are set to 0.1, as they were initially for the present study. After observation of the fluctuations in the heating on the forebody, epc and epu were increased to 0.5. The heat transfer distribution for the increased entropy smoothing is shown by the solid line in Figure 21. The increased entropy smoothing resulted in an improved distribution in the stagnation region and had a minimal effect on the remainder of the heat transfer and pressure distributions. After adequate results were obtained in the stagnation region

an attempt was made to return the entropy smoothing parameters to the original levels. This resulted in numerical instability.

Comparison with Pressure Data Calculated pressure distributions on the forebody, cylinder, and base are shown in Figure 22 along with the experimental results from Figure 15. The computed forebody pressure distribution (Figure 8a) agreed with the experimental results within $\pm 3\%$ of the measured values in the nose and cone region ($s/R < 0.6$). The difference between the computation and experiment increased from $\pm 3\%$ at $s/R = 0.675$ to $\pm 90\%$ at the corners.

Figure 22b presents the computed pressure distribution on the cylinder surface. The computed results agreed qualitatively with the experimental results with the peak pressure on the cylinder at $x/R = 2.56$. The computed pressure agreed with the experimental data within the experimental uncertainty for $x/R > 1.5$. However, for $x/R < 1.5$ the agreement was within $\pm 60\%$. The computed pressure for $x/R < 1.5$ appeared to agree more favorably with the experimental data from Run 7 in Figure 3b. Recall that this data was thought to be in error due to a bias in the pressure instrumentation.

The computed pressure distribution for the base region is shown in Figure 22c. The computed pressure had a slight peak in the region of $y/R = 0.6$ to 0.8 . This peak indicated that the base may have been influenced by a recirculation region. Other than the slight peak, the computed pressure remained relatively constant, similar to the experimental results. However, the computed pressure was substantially lower than the experimental data. The agreement between the experiment and computation was within approximately $\pm 40\%$ of the mean experimental value.

The rather large disagreement between the computation and experiment for the low pressure regions on the cylinder, base, and corners may have been due to inadequate grid resolution. The disagreement on the forebody may have been due to inadequate resolution of the rapidly expanding flow in the shoulder region, as indicated by the larger disagreement after $s/R=0.8$. This would affect the resolution of the strong normal and streamwise gradients in the free shear layer that forms off the corner, and in turn, would alter the cylinder and base flowfield since these areas are influenced by the shear layer. Thus, proper modeling of the forebody expansion appeared to be an important element in obtaining accurate computed surface pressure and heat transfer distributions on the cylinder and base. In addition, the grid structure in the near-wake domains may also have been inadequate for computing the shear layer gradients.

Comparison with Heat Transfer Data Computed heat transfer distributions on the forebody, cylinder, and base are shown in Figure 23 along with the experimental results from Figure 18. Figure 23a shows the computed heating on the forebody. The computed heat transfer distribution was substantially lower than the experimental data and agreed within $\pm 20\%$, except for the corner where the code was greater than the experiment by 130%. The computation also indicated a reduction in heating in the cone region of the forebody ($s/R= 0.3$ to 0.7). The cause of this phenomenon was not determined. Also note that the computation resulted in higher heating for the flow expansion over the corner ($s/R>1.0$). The experimental stagnation point heating would be expected to be higher than the reference value, as indicated by previous tests on a similar shape with $r_n/r_b=0.5$ (Shih and Gay 1984). The results of these previous tests showed that $C_{h}/C_{h, \text{ref}} = 1.28$ at the stagnation

point. The present experimental results were also higher than the reference value ($C_{h,ref}/C_{h,ref} = 1.14$), but were lower than the Shih and Gay's results, as expected, due to the larger bluntness ratio. Note that the computed results were below the reference value at the stagnation point. This indicated that the experimental heating data shown in Figure 23 was at the appropriate level.

Figure 23b shows the computed heating on the cylinder. The computational results were significantly lower than the experiment over the last half of the cylinder. However, the qualitative aspects of the computed distribution agree well with experiment. The computation resulted in an initial rise in heating on the cylinder at $x/R < 0.5$, where the experimental data indicated that the initial increase in heating occurred at $x/R = 0.5$. Both methods indicated an increase in heating at the end of the cylinder ($\phi = 0^\circ$ ray for the experiment).

Computed base heating is shown in Figure 23c. The computation and experiment were in general agreement; both indicated an increase in heat transfer toward the shoulder of the forebody (increasing y/R). The computation resulted in high localized heating in the region $y/R = 0.6$ to 0.8 . This coincided with the higher computed pressure in this region, supporting the hypothesis that the region may have been influenced by a recirculation. The experimental data indicated a more constant heating distribution than the computation. The disagreement between the two methods was beyond the experimental uncertainty with a maximum disagreement of 50% occurring at $y/R = 0.95$.

The difference in the experimental and computed heating may have been due to the incorrect wall temperature used in the code. Recall that the

code was run with a slightly lower wall temperature than occurred for the experiment. The wall temperature affects the boundary layer growth, and in turn, the free shear layer thickness which influences the cylinder. The effect of this difference in wall temperature should be fully investigated before reaching a final conclusion as to the cause of the disagreements in Figure 23.

The increased smoothing applied to the code may also have been a factor in the disagreement. The effect of the increased smoothing would be expected to be greatest on the forebody where the gradients were largest. Other areas of large gradients, such as within the free shear layer, may also have been affected.

The difference between the experimental and computed heating may also have been due to inadequate normal spacing of the grid near the surface. Thus, the temperature gradient at the surface may not have been accurately modeled. An indicator of proper grid structure at the wall is the y^+ parameter which is based on the thickness of the laminar sublayer (Schlichting, 1968). Typically, values of $y^+ < 1$ are considered proper for adequate modeling of the wall temperature gradient, with $y^+ < 0.2$ considered ideal. The y^+ parameter was determined to be less than 0.1 for the present results on the forebody.

Use of the thin-layer Navier-Stokes equations as opposed to the full equations may also be considered as a source for the disagreement in Figure 23. Previous research has shown that significant differences in stagnation point heating occur on the AFE vehicle depending on which set of equations are used (Gnoffo and Greene, 1987). However, use of the thin-layer equations were shown to result in a higher stagnation point heating value than the full Navier-Stokes equations. Since the computed heating was below the measured

values (and the measured values were determined to be at the appropriate level), the thin-layer approximation would not be expected to cause the observed differences.

Another possible source of the disagreement in Figure 23 is the grid resolution of the expanding flow over the corner, as discussed earlier. The computed heating in Figure 23a supports this hypothesis since the decrease in heating in the shoulder region ($s/R > 0.8$) occurred much later for the computation than the experiment. This may also have led to the disagreement on the cylinder since modeling of the corner expansion would be expected to affect the free shear layer.

The experimental heating results on the cylinder may also indicate a transitional or turbulent free shear layer or cylinder boundary layer. Recall that the flow was modeled as fully laminar. A similar heat transfer test on the AFE showed good agreement with a fully laminar computation on a cylinder in the near-wake at identical freestream conditions (Wells, 1990). In the AFE test, the model forebody diameter was 3.67 inches, slightly smaller than the present configuration. The AFE results indicated that fully laminar flow was a reasonable assumption for this investigation. However, the large disparity shown in Figure 23b indicates that further investigation into the state of the free shear layer and cylinder boundary layer is warranted.

Flow Structure A comparison of the forebody shock stand-off distance between the experimental and computational methods was considered an important first check of the CFD solution. The experimental shock stand-off distance was measured from a shadowgraph video, and was determined to be 12% of the forebody diameter. The computational shock stand-off distance was measured from a Mach contour plot (Figure 24) and was determined to be 10%

of the forebody diameter. The difference could be due to an error in the measurement of the experimental shock stand-off distance since it was made from a rather poor quality image. Note that the bow shock in Figure 24 was very well resolved and that these differences do not appear to be due to inadequate grid resolution. The smaller shock stand-off distance for the computation would be expected to cause higher forebody heating due to increased density in the shock layer resulting in better agreement with the experiment. This indicated that the discrepancy in shock stand-off distance was not the cause of the disagreement between the computation and experiment. The discrepancy in shock stand-off distance may have been due to an error in the measurement of the experimental shock location.

Figures 25 and 26 show the Mach and pressure contours, respectively, for the entire computed flowfield. Figure 25 is presented in two parts to provide good resolution of both the forebody and near-wake flowfields. The calculated sonic line for the flowfield is shown in Figure 26. As discussed earlier, the sonic line was located on the forebody shoulder. This location agreed well with the estimate from the experimental data as shown in Figure 15a. As shown in Figure 24, lines of constant Mach number were parabolic in shape within the shock layer. In addition, the free shear layer was observed to impinge near the end of the cylinder. This impingement corresponded with the location of increased pressure and heating at the end of the cylinder. Recall that the experimental data did not reveal a distinct peak in pressure or heating, indicating that the shear layer impingement may have occurred over a wide area or downstream of the instrumentation. The pressure in the vicinity of the shear layer (comparing Figures 24 and 25b) appeared to be

constant except very near the cylinder, as expected. Also note the presence of a recirculation region upstream of the shear layer impingement. The recirculation resulted from the flow inside the dividing streamline having to turn upstream into the base region, as discussed earlier. A supersonic core was also embedded in the recirculation region. Recompression waves coalesced into a weak shock at the end of the cylinder as shown in Figure 25b. These waves resulted from the flow outside the dividing streamline having to turn parallel to the cylinder surface.

Figure 26 shows the recirculation region more clearly with velocity vectors colored by Mach number. This recirculation region has been observed previously on a similar configuration, although without the presence of the cylinder (Gnoffo, 1991). Gnoffo observed for a smaller bluntness ratio configuration that the wake drives the recirculating flow toward the base at Mach 1.3, a shock forms in front of the base, the flow stagnates, and then accelerates outward to the free shear layer. Obviously, the presence of the cylinder changes this flow structure. The computational results indicated that the flow gradually turned toward the shear layer and impinged upon the base (as indicated by the increased heating in this region) without the presence of a shock. The smooth increase in pressure along the base surface further supports this hypothesis. Note the recirculation drives the flow toward the base at Mach 1.2, as seen in Figure 26. The location of this supersonic region corresponded with the region of lowest pressure on the cylinder surface in Figure 15.

The computed location of the rear stagnation point was determined by observing the magnitude of the velocity at one grid point above the cylinder surface. This point also indicated the vicinity of the dividing streamline. This

method did not necessarily locate the point of maximum heating or pressure. Previous studies have observed that maximum heating occurs downstream of the location of the rear stagnation point and depends upon angle of attack and post-normal shock Reynolds number (Wells, 1989). The rear stagnation point was determined to be at $x/R=0.87$ on the cylinder. This point can be used to calculate the shear layer deflection angle measured downward from the horizontal at the shoulder, as shown in Figure 5. The shear layer deflection angle was determined to be 16.3° . Another method to determine the deflection angle is based on the position of the sonic line within the free shear layer (Gnoffo, 1991). This method yielded a deflection angle of 19.5° , based on the mean angle, or 11.4° , based on the initial angle at the forebody corner. These results were substantiated by previous results on a similar configuration ($r_n/r_b=0.83$) where the shear layer deflection angle was determined to be 14.3° , based on location of the sonic line (Gnoffo, 1991).

CHAPTER 5

Summary And Recommendations

A combined experimental and computational investigation of the flowfield about an Aeroassisted Space Transportation Vehicle (ASTV) has been performed. The configuration examined was a 70° sphere cone with a cylindrical afterbody at nominal Mach 10, perfect gas conditions. The configuration had a bluntness ratio (nose-to-body radius ratio) of 1.0. The experimental portion of the investigation was performed in the NASA Langley 31-Inch Mach 10 Tunnel in air at zero degrees angle of attack and yaw. Detailed measurements of surface pressure and heat transfer were obtained on the ASTV forebody, base, and on a cylinder in the near-wake using arrays of static ports and thin-film resistance gages. Reynolds number behind the normal shock, based on the model forebody diameter, was nominally 14,950. The computational portion of the investigation was performed using the TUFF (Three-Dimensional, Finite-Volume Flow Solver with Fully Coupled Chemistry). Laminar, axisymmetric flow was assumed and flow conditions matched the wind tunnel freestream properties.

The measured forebody stagnation pressure agreed to within 0.1% of the facility Pitot pressure. In general, pressure increased with distance along the cylinder, indicating influence of a free shear layer formed by the separation of the boundary layer off the forebody corner. The maximum pressure coefficient on the cylinder was determined to be 4.4% of the stagnation pressure value at a location greater than 2.5 body radii along the cylinder from the base. A distinct peak in pressure along the cylinder was not

observed. The average base pressure coefficient was determined to be 0.83% of the stagnation point value. The measurements at the forebody corners revealed several discrepancies which were believed to be due to instrumentation problems. A detailed analysis of the pressure data uncertainty was also performed.

The stagnation point heat transfer coefficient was determined to be 14% higher than the Fay-Riddell reference value. This level was substantiated by comparison with previous tests on a similar configuration. In general, heating increased with distance along the cylinder and a distinct peak was not observed. Again, this distribution indicated the influence of the free shear layer. The maximum heating on the cylinder was determined to be 19.2% of the stagnation point value and occurred at a distance of 2.69 body radii along the cylinder. The maximum base heating was determined to be 1.4% of the stagnation point value. The heating distribution on the forebody followed a trend observed in previous investigations of similar shapes. A disagreement was observed between instrumentation arrays along the top and bottom of the cylinder. This disagreement indicated the model was at a slight angle of attack. However, the data agreed within the experimental uncertainty indicating symmetric flow.

The computed pressure on the forebody and downstream portion of the cylinder agreed within the experimental uncertainty with the wind tunnel data. However, the two methods did not agree well in the lower pressure regions of the base and upstream portion of the cylinder. The computational results were substantially lower than the experimental heating data on the forebody and cylinder, although the results were in qualitative agreement. The computation provided significant insight into the flowfield structure. The

free shear layer was observed to impinge on the cylinder. The experimental data indicated that this impingement occurred over a wide area or downstream of the instrumentation. A recirculation region with a supersonic core was observed in the base region. This recirculation was observed to drive the flow toward the base at Mach 1.2. In addition, the shear layer deflection angle was observed to be 16.3° as determined from the location of the stagnation point on the cylinder. The sonic line was observed to occur on the forebody shoulder, in qualitative agreement with the experimental pressure data.

The differences between the experimental and computational results were believed to be due to several factors. The model wall temperature for the computation was slightly lower than the experiment. This may have led to inadequate modeling of the boundary layer thickness by the code which, in turn, may have affected the forebody and cylinder heating rates. The increased smoothing applied to the code to resolve an anomaly in the stagnation point heating may also have contributed to the differences. The higher smoothing may have dissipated regions of large gradients on the forebody and within the free shear layer. The forebody shock stand-off distance was observed to be smaller for the computation than for the experiment. In addition, the rapid expansion of the flow over the forebody corner may not have been adequately resolved by the grid which could have led to the disagreement on the forebody shoulder and cylinder. The rather large disagreement in the heat transfer distributions on the cylinder may also have been due to transition from laminar to turbulent flow in the free shear layer or cylinder boundary layer during the experiment. However, previous experimental and computational investigations on a similar shape with a

cylinder in the near-wake indicated that the assumption of fully laminar flow was valid.

The following recommendations were formulated to aid in the resolution of the differences between the experimental and computational results: 1) continue the current TUFF solution, except increase the wall temperature to 540°R, 2) perform a grid refinement study using a method to generate a refined grid through redistribution of grid points based on gradients in the flowfield, 3) perform a transition study with the current code (possibly using two cases, starting transition at the forebody corner and at the cylinder impingement point), 4) perform a detailed analysis of the experimental heat transfer data uncertainty, and 5) perform a check of the model instrumentation at very low pressures (near tunnel freestream static pressure) to identify leaks or effects of outgassing.

LIST OF REFERENCES

LIST OF REFERENCES

- Ames Research Staff, "Equations, Tables, and Charts for Compressible Flow", NACA TN-1135, 1953.
- Abernethy, R.B., and Thompson, J.W., "Handbook: Uncertainty in Gas Turbine Measurements", AEDC TR-73-5, February 1973.
- Berger, Stanley, A., Laminar Wakes, American Elsevier Publishing Company, Inc. 1971.
- Beckwith, I.E. and Miller, C.G., "Aerothermodynamics and Transition in High-Speed Wind Tunnels at NASA Langley", Annu. Rev. Fluid Mech., 1990, Vol. 22, pp. 419-439.
- Blottner, Frederick G., "Verification of a Navier-Stokes Code for Solving the Hypersonic Blunt Body Problem", SAND-88-0917C, Sandia National Laboratories, November 1988.
- Celenligil, M.C., Moss, J.N., and Bird, G.A., "Direct Simulation of Three-Dimensional Flow About the AFE Vehicle at High Altitudes", Rarefied Gas Dynamics: Theoretical and Computational Techniques, Vol. 118, Muntz, E.P., Weaver, D.P., and Campbell, D.H., editors, Washington, D.C. 1989.
- Chakravarthy, S.R. and Osher, S., "A New Class of High Accuracy TVD Schemes for Hyperbolic Conservation Laws", AIAA 85-0363, January, 1985.
- Chapman, Dean, R., "Laminar Mixing of a Compressible fluid", NACA Report 958, 1949.
- Cook, W.J. and Felderman, E.J., "Reduction of Data from Thin-Film Heat Transfer Gages: A Concise Numerical Technique", AIAA Journal, Vol. 4, pp. 561-562, March 1966.
- Dietz, W.E. and Evans, S.B., "Interactive EAGLE: An Interactive Surface Mesh and Three-dimensional Grid Generation System - Version 1.0, User's Guide", AEDC TR-90-25, December 1990.

- Fay, J.A., and Riddell, F.R., "Theory of Stagnation Point Heat Transfer in Dissociated Air", J. Aeronaut. Sci., Vol. 25, No. 2, February 1958, pp. 73-85, 121.
- Gnoffo, Peter, A., "Application of Program LAURA to Three-Dimensional AOTV Flowfields", AIAA 86-0565, January 1986.
- Gnoffo, P.A. and Greene, F.A., "A Computational Study of the Flowfield Surrounding the Aeroassist Flight Experiment Vehicle", AIAA 87-1575, June 1987.
- Gnoffo, Peter, A., "A Code Calibration Program in Support of the Aeroassist Flight Experiment", AIAA 89-1673, June 1989.
- Gnoffo, Peter, A., Price, Joseph, M., and Braun, Robert, D., "On the Computation of Near Wake, Aerobrake Flowfields", AIAA 91-1371, June 1991.
- Grasso, Francesco and Gnoffo, Peter A., "A Numerical Study of Hypersonic Stagnation Heat Transfer Predictions at a Coordinate Singularity", Proceedings from the Eighth GAMM-Conference on Numerical Methods in Fluid Mechanics, Pieter Wessling, editor, Braunschweig, 1990 (ISBN 3-528-07629-1).
- Hama, Francis, R., "Experimental Investigations of Wedge Base Pressure and Lip Shock", Technical Report 32-1033, Jet Propulsion Laboratory, California Institute of Technology, December 1966.
- Holland, S.D., Hodge, J.S., and Perkins, J.N., "Wind Tunnel Blockage Study of a Generic Three-Dimensional Sidewall Compression Scramjet Inlet at Mach 10", AIAA 91-0294, January 1991.
- Jones, Jim, J., "The Rationale for an Aeroassist Flight Experiment", AIAA 87-1508, June 1987.
- Knox, E.C. and Best, Jr., J.T., "Pressure Test on the Viking Lander Capsule at Mach Number 10", AEDC TR-72-185, December 1972.
- Li, C.P., "Computation of Three-Dimensional Flow about Aerobrake Configurations", AIAA 86-0566, January 1986.

- Micol, J.R., "Simulation of Real-Gas Effects on Pressure Distributions for a Proposed Aerassist Flight Experiment Vehicle and Comparison to Prediction", AIAA 87-2368, August 1987.
- Micol, J.R., "Experimental and Predicted Pressure and Heating Distributions for an Aeroassist Flight Experiment Vehicle in Air at Mach 10", AIAA 89-1731, June 1989.
- Miller, Charles, G. III, "Comparison of Thin-Film Resistance Heat-Transfer Gages With Thin-Skin Transient Calorimeter Gages in Conventional Hypersonic Wind Tunnels", NASA TM 83197, December 1981.
- Miller, C.G., "Experimental and Predicted Heating Distributions for Biconics at Incidence in Air at Mach 10", NASA TP-2334, November 1984.
- Miller, Charles G. III, Micol, John R., Gnoffo, Peter A., "Laminar Heat-Transfer Distributions on Biconics at Incidence in Hypersonic-Hypervelocity Flows", NASA TP-2213, January 1985.
- Miller, C.G., "Langley Hypersonic Aerodynamic/Aerothermodynamic Testing Capabilities - Present and Future", AIAA 90-1376, June 1990.
- Molvick, Gregory Alan, "A Computational Model for the Prediction of Hypersonic, Reacting Flows", Ph.D. Dissertation, The Pennsylvania State University, December 1989.
- Penaranda, F.E. and Freda, M.S., "Wind Tunnels. Aeronautical Facilities Catalogue - Volume 1", NASA RP-1132, January 1985.
- Ressler, D.E., Shivananda, T.P., Zabrensky, E.F., and Cruz, M.I., "Drag and Heat Transfer Calculations about a MESUR Aerobrake Vehicle", AIAA 92-2684, June 1992.
- Schlichting, H., Boundary Layer Theory, McGraw-Hill Book Company, New York, 1968.
- Shih, Peter Kwang-Tien and Gay, Archie, "Low Lift-to-Drag Aerobrake Heat-Transfer Test at Mach 10", AIAA 84-0309, January 1984.

- Shimshi, Jason Paul, "An Investigation of Aerodynamic Heating to Spherically Blunted Cones at Angle of Attack", Masters Thesis, The George Washington University, August 1992.
- Venkatapathy, E., Palmer, G., and Prabhu, D.K., "AFE Base Flow Computations", AIAA 91-1372, June 1991.
- Weilmuenster, K.J. and Hamilton II, H.H., "A Comparison of Computed and Experimental Surface Pressure and Heating on 70° Sphere Cones at Angles of Attack to 20°", AIAA 86-0567, January 1986.
- Walberg, Gerald, D., "A Review of Aeroassisted Orbit Transfer", AIAA 82-1378, August 1982.
- Walatka, Pamela P., and Buning, Pieter G., "PLOT3D User's Manual", NASA TM 101067, 1989.
- Wells, William, L., "Wind Tunnel Preflight Test Program for Aeroassist Flight Experiment", AIAA 87-2367, December 1987.
- Wells, William, L., "Measured and Predicted Aerodynamic Coefficients and Shock Shapes for Aeroassist Flight Experiment (AFE) Configuration", NASA TP-2956, January 1990.
- Wells, William, L., "Surface Flow and Heating Distributions on a Cylinder in Near Wake of Aeroassist Flight Experiment (AFE) Configuration at Incidence in Mach 10 Air", NASA TP-2954, January 1990.

APPENDICES

APPENDIX A

Figures

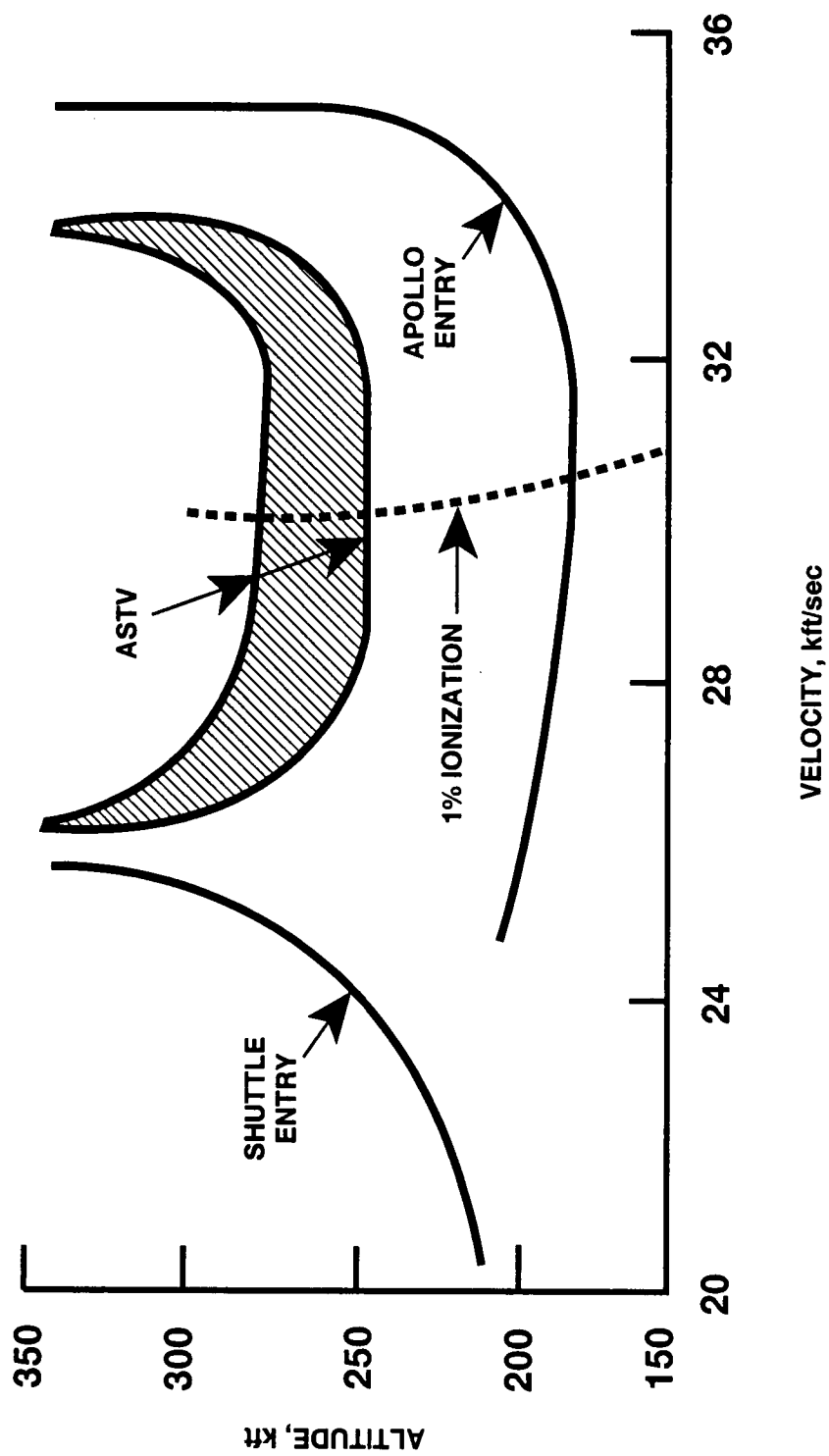


Figure 1. Typical Flight Profile for ASTV Orbit Transfer Mission

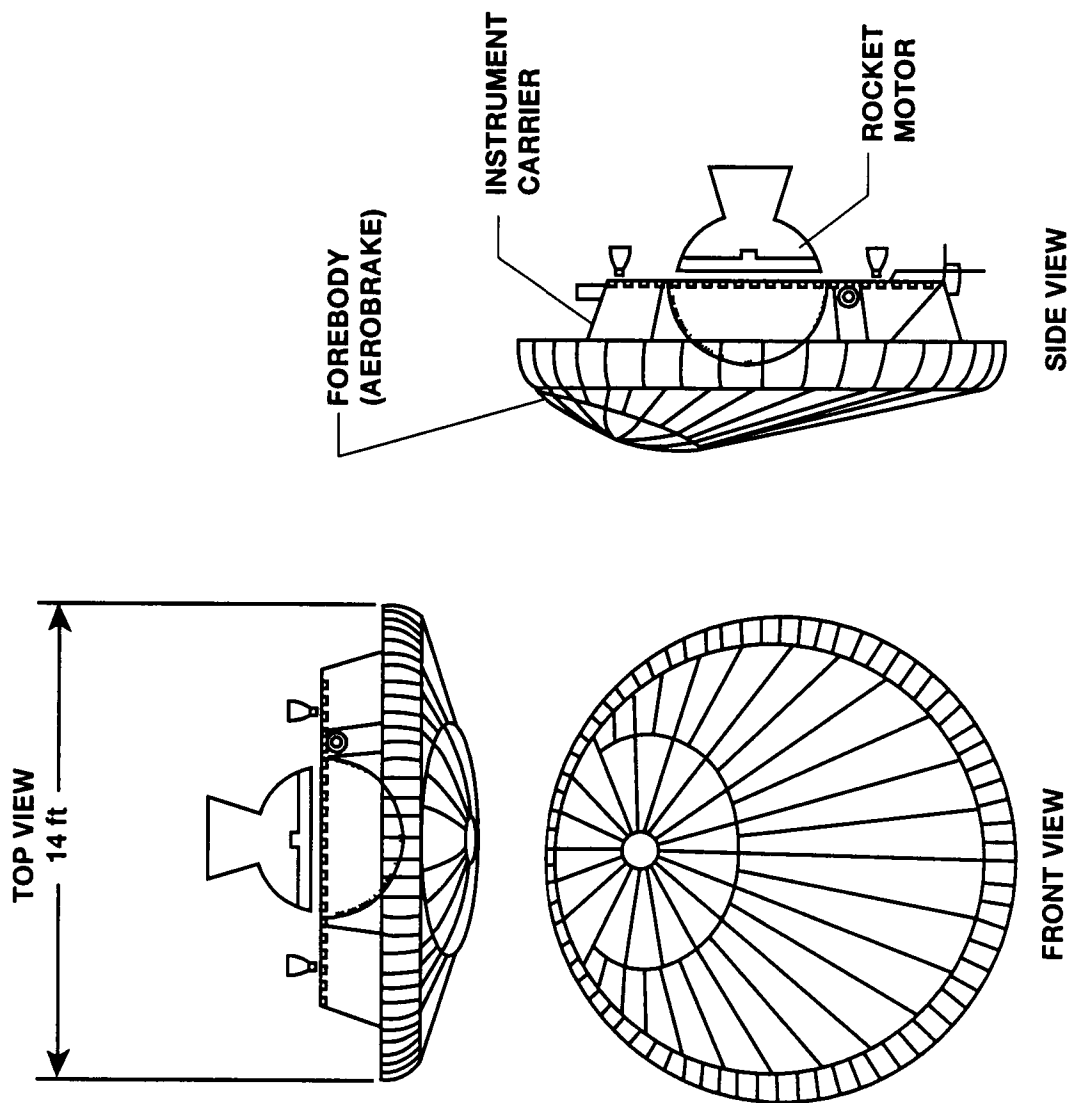


Figure 2. Aeroassist Flight Experiment (AFE) Vehicle

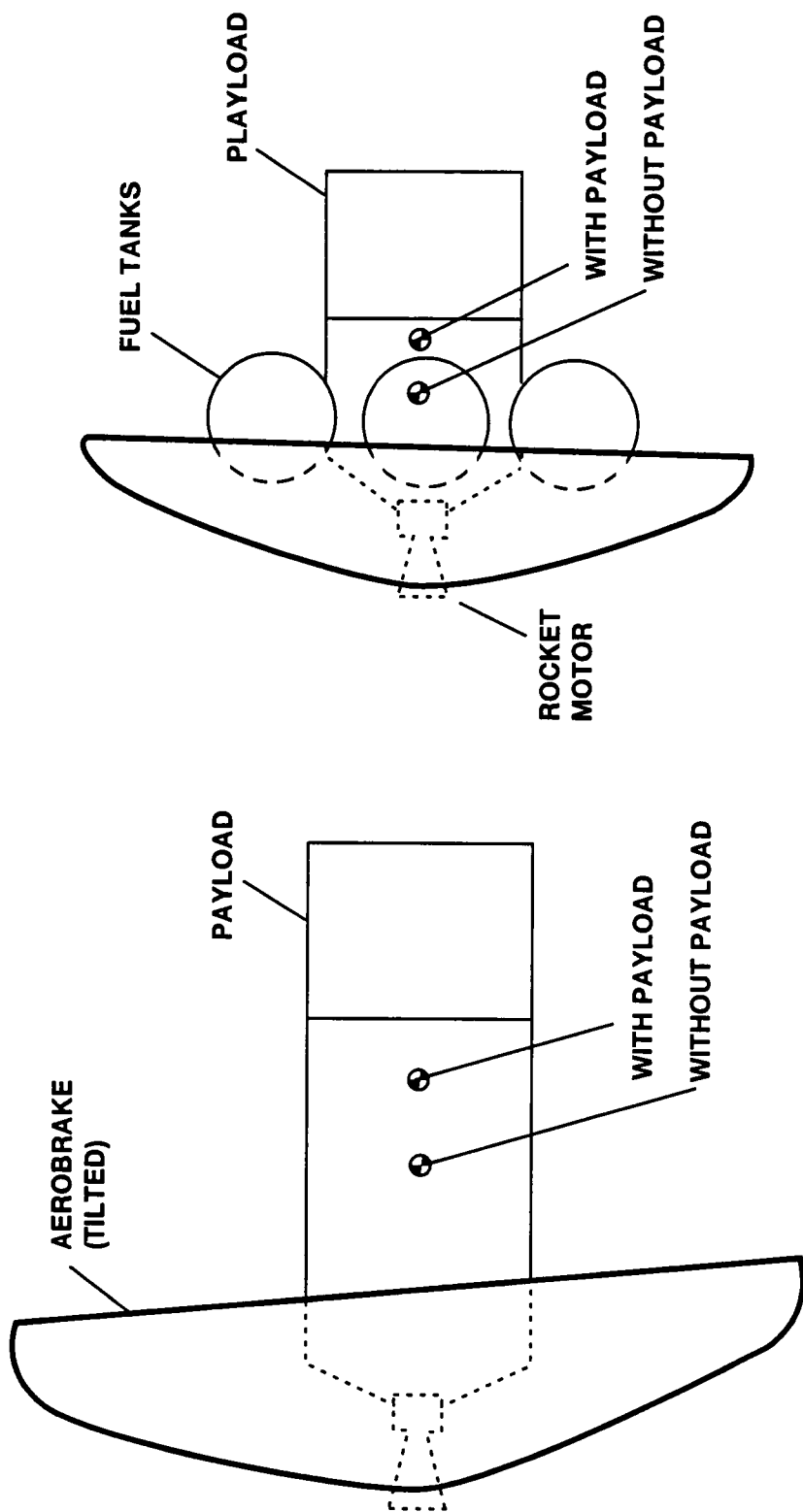


Figure 3. Axisymmetric ASTV Concepts

FOREBODY COORDINATES (INCHES)

	X	Y		X	Y
1	0.12500	0.00000	36	0.45425	1.25416
2	0.12701	0.03725	37	0.46704	1.28916
3	0.12913	0.07449	38	0.47983	1.32416
4	0.13147	0.11171	39	0.49259	1.35917
5	0.13413	0.14889	40	0.50529	1.39420
6	0.13724	0.18602	41	0.51801	1.42923
7	0.14088	0.22309	42	0.53077	1.46424
8	0.14519	0.26009	43	0.54347	1.49927
9	0.15025	0.29699	44	0.55602	1.53436
10	0.15609	0.33380	45	0.56872	1.56940
11	0.16268	0.37049	46	0.58228	1.60409
12	0.16998	0.40705	47	0.59665	1.63848
13	0.17796	0.44345	48	0.61120	1.67280
14	0.18657	0.47971	49	0.62713	1.70647
15	0.19579	0.51582	50	0.64452	1.73943
16	0.20554	0.55178	51	0.66122	1.77275
17	0.21576	0.58762	52	0.67861	1.80569
18	0.22639	0.62334	53	0.69766	1.83773
19	0.23776	0.65883	54	0.71781	1.86909
20	0.25015	0.69396	55	0.73968	1.89955
21	0.26310	0.72891	56	0.76258	1.92871
22	0.27590	0.76391	57	0.78770	1.95627
23	0.28856	0.79895	58	0.81758	1.97847
24	0.30132	0.83397	59	0.85118	1.99452
25	0.31407	0.86898	60	0.88787	1.99977
26	0.32681	0.90400	61	0.92398	1.99027
27	0.33954	0.93902	62	0.95500	1.97000
28	0.35228	0.97404			
29	0.36503	1.00905			
30	0.37780	1.04406			
31	0.39060	1.07908			
32	0.40338	1.11406			
33	0.41611	1.14908			
34	0.42879	1.18412			
35	0.44150	1.21915			

$$\frac{r_n}{R} = 1.000$$

$$\frac{r_c}{R} = 0.125$$

$$\frac{r_{cyl}}{R} = 0.300$$

$$\frac{r_f}{R} = 0.047$$

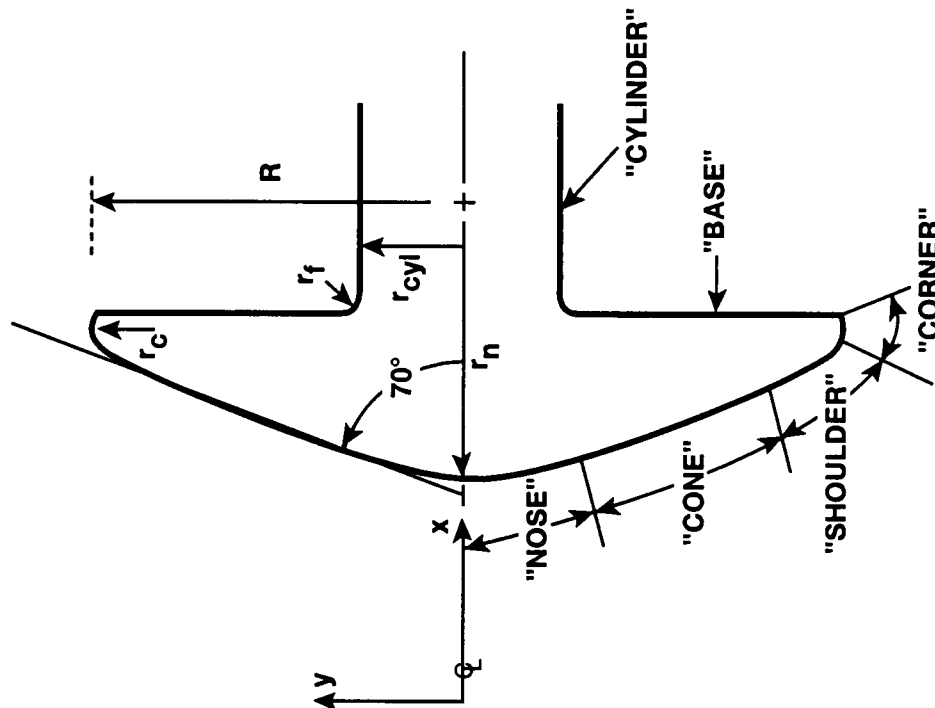


Figure 4. ASTV Configuration for the Present Study

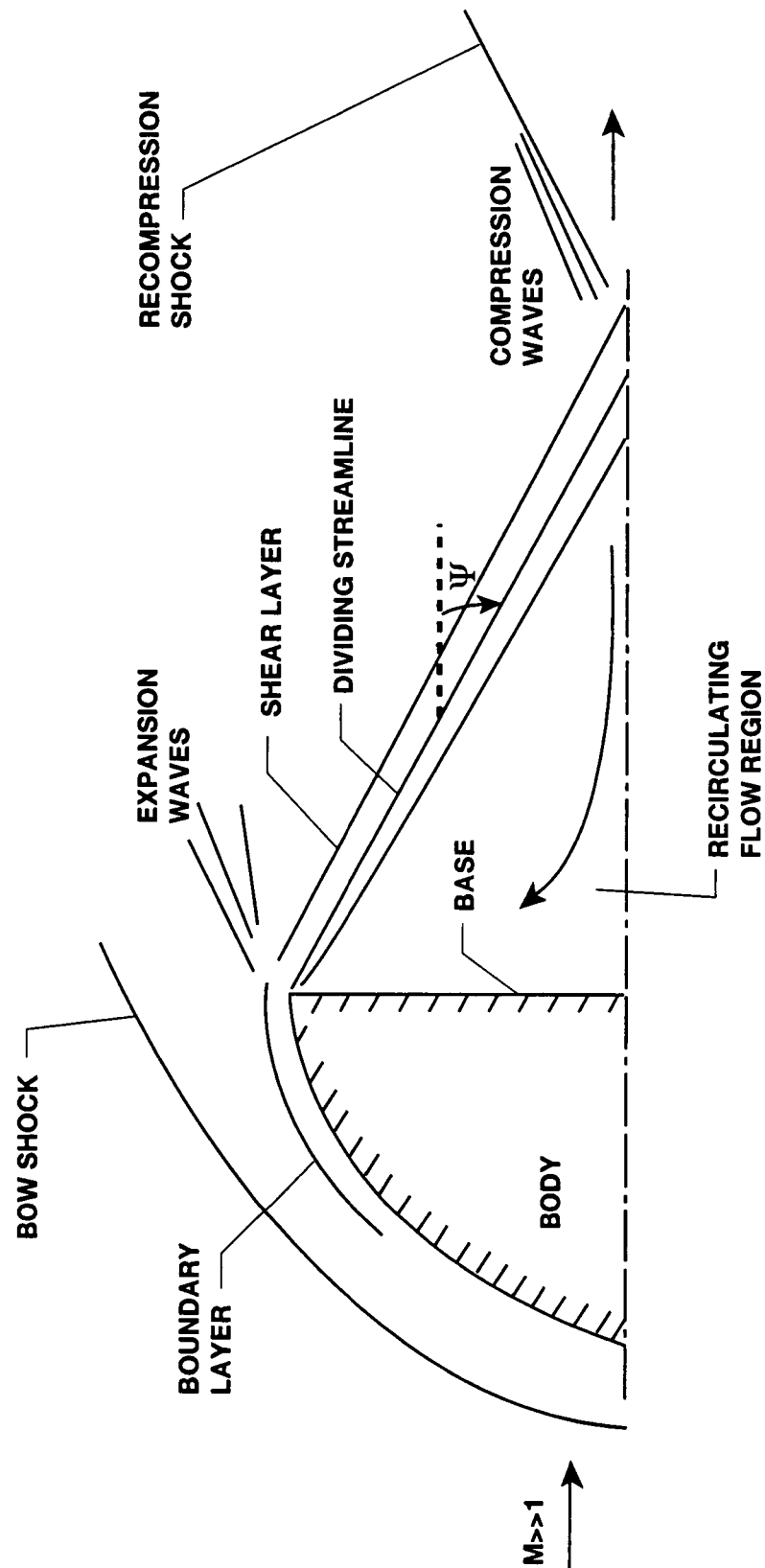


Figure 5. Simplified Model of Near-Wake Flow of Hypersonic Blunt Body

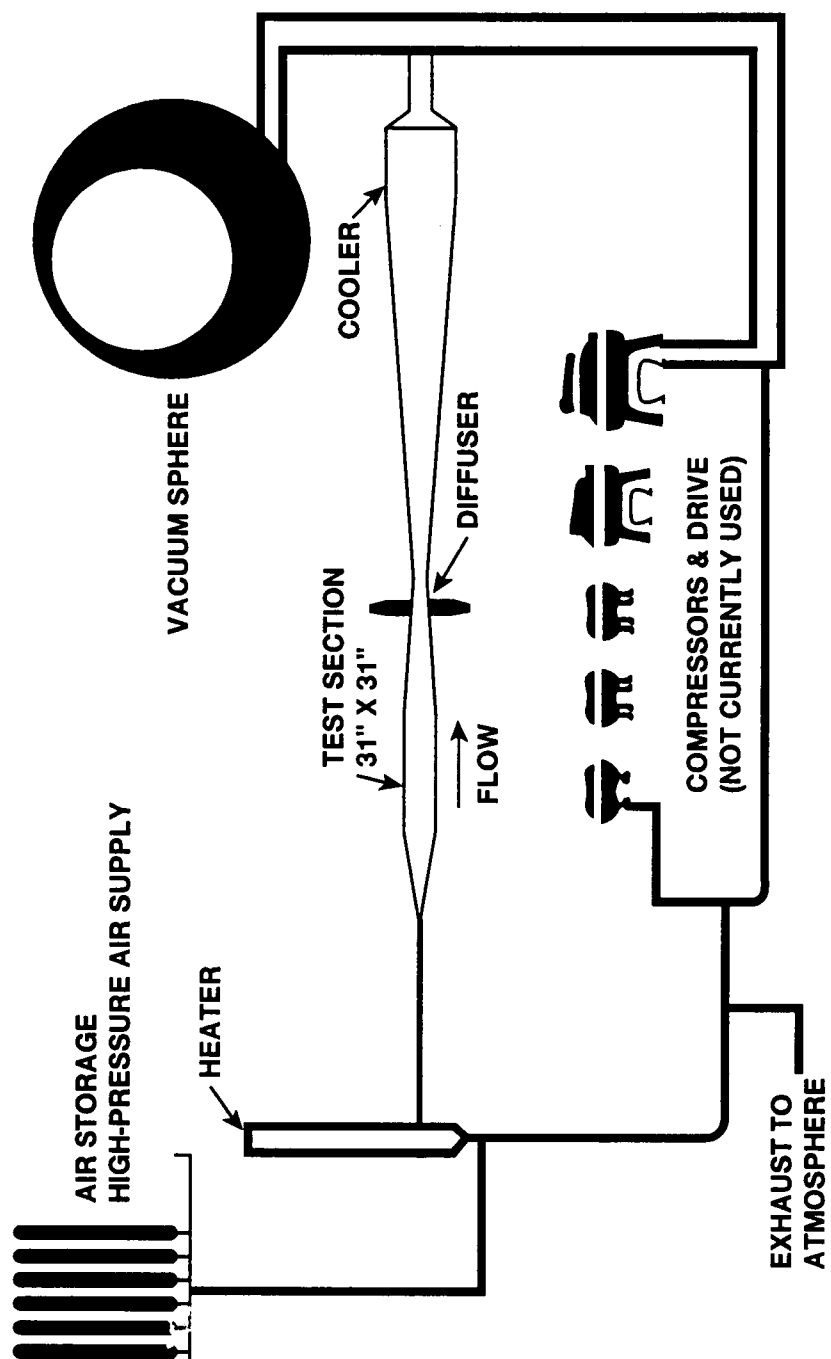


Figure 6. Diagram of the NASA Langley 31-Inch Mach 10 Tunnel

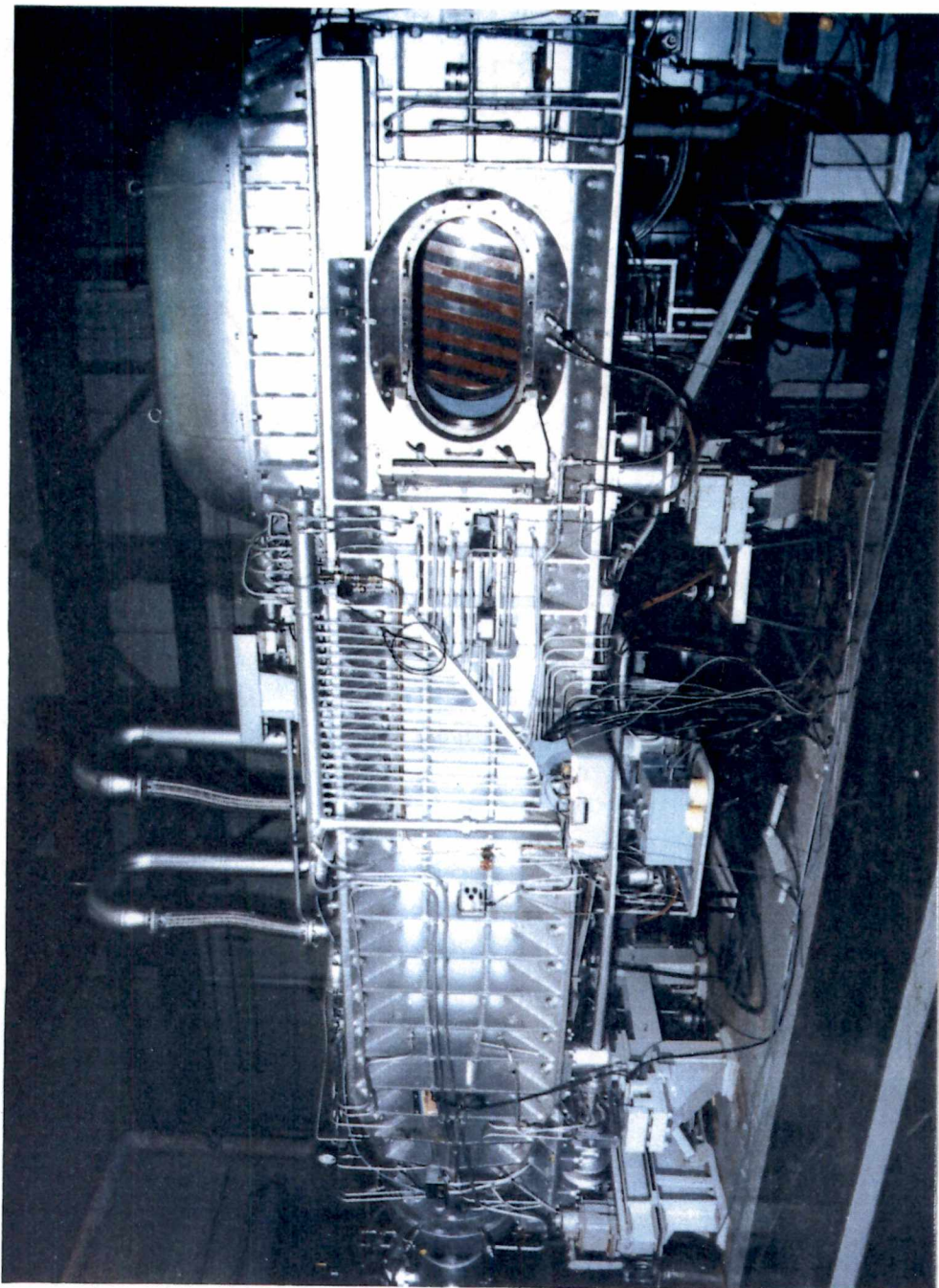
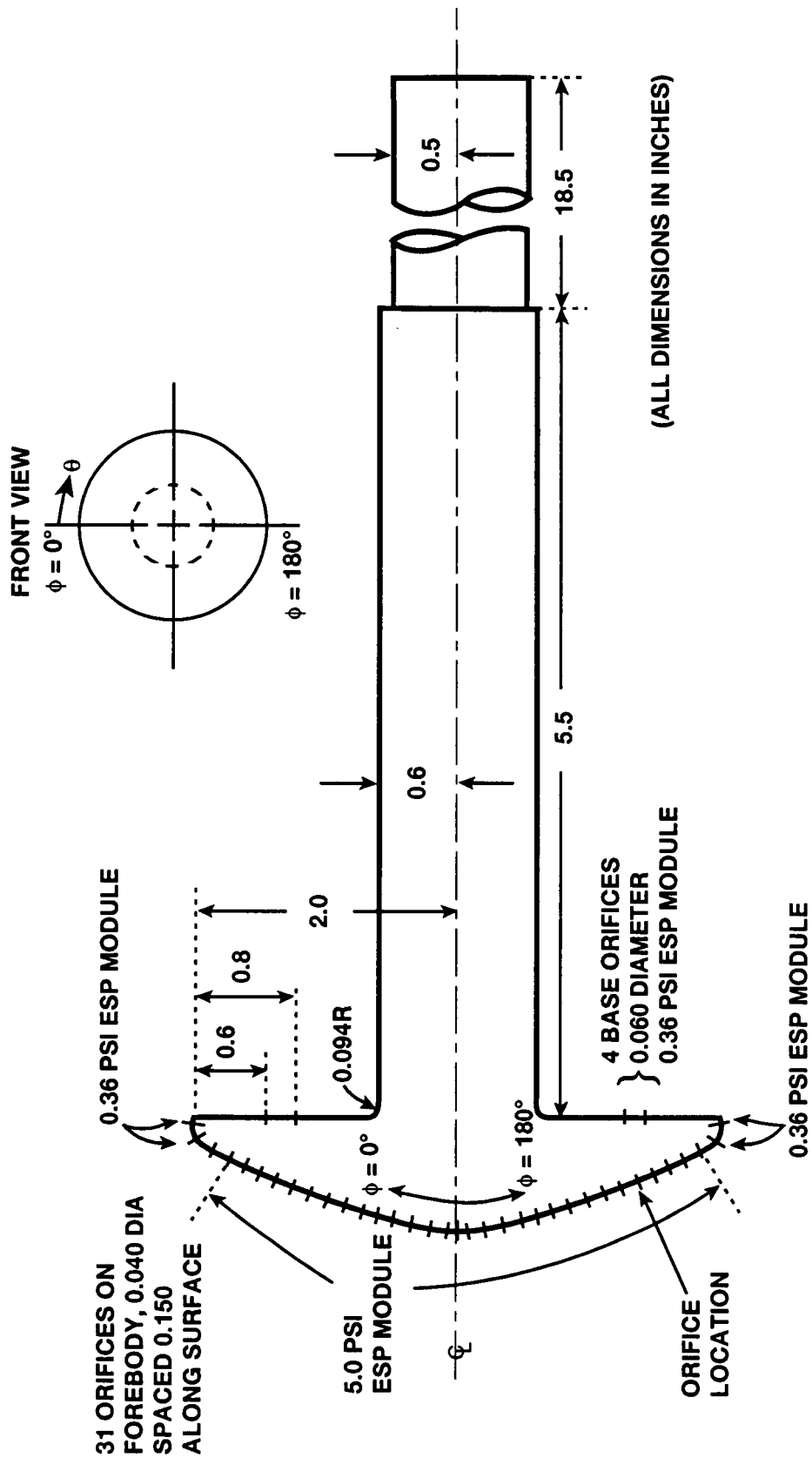
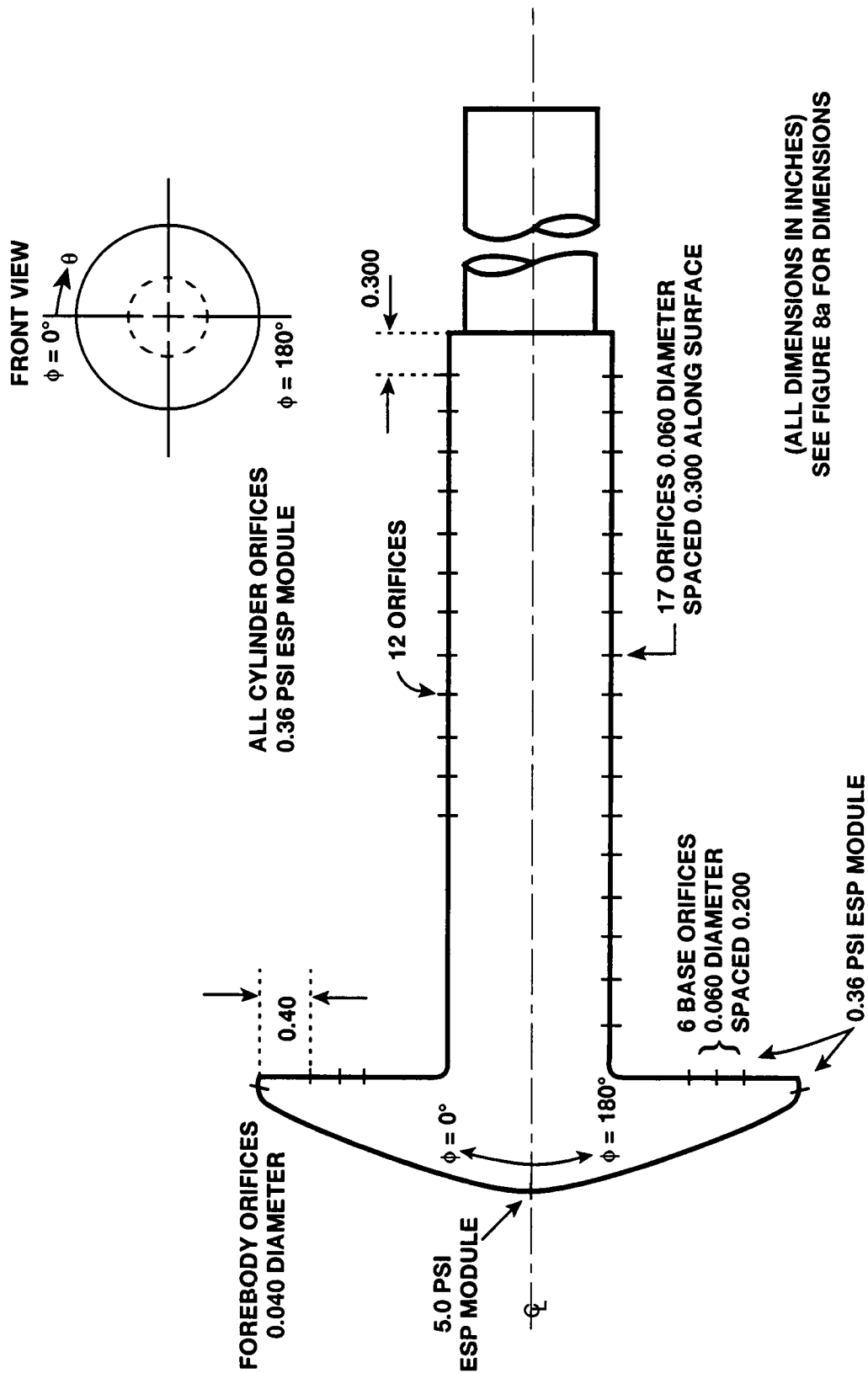


Figure 7. Photograph of the NASA Langley 31-Inch Mach 10 Tunnel



a) Forebody Model

Figure 8. Pressure Model Details



b) Cylinder Model

Figure 8. Concluded

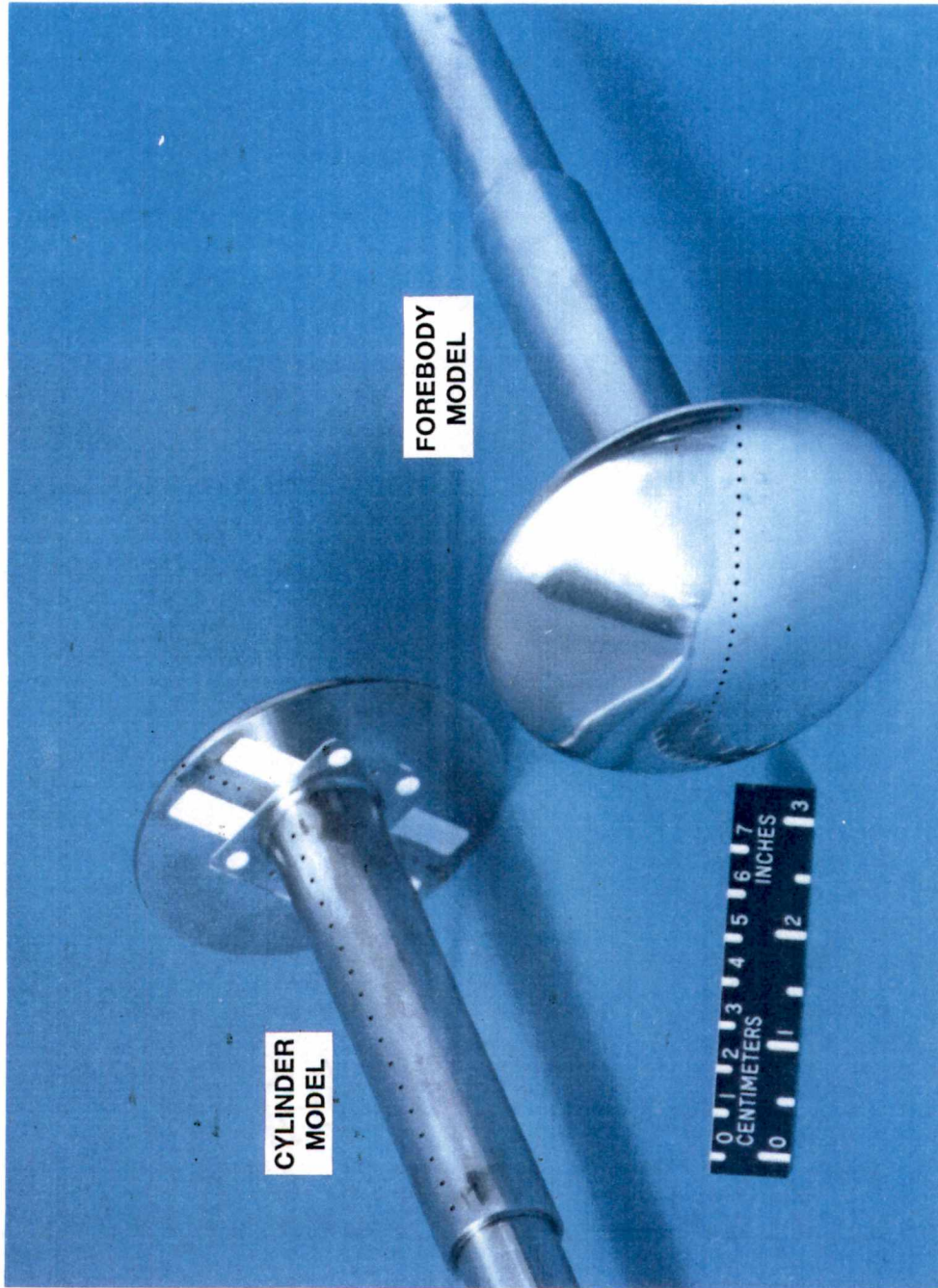


Figure 9. Photograph of Forebody and Cylinder Pressure Models

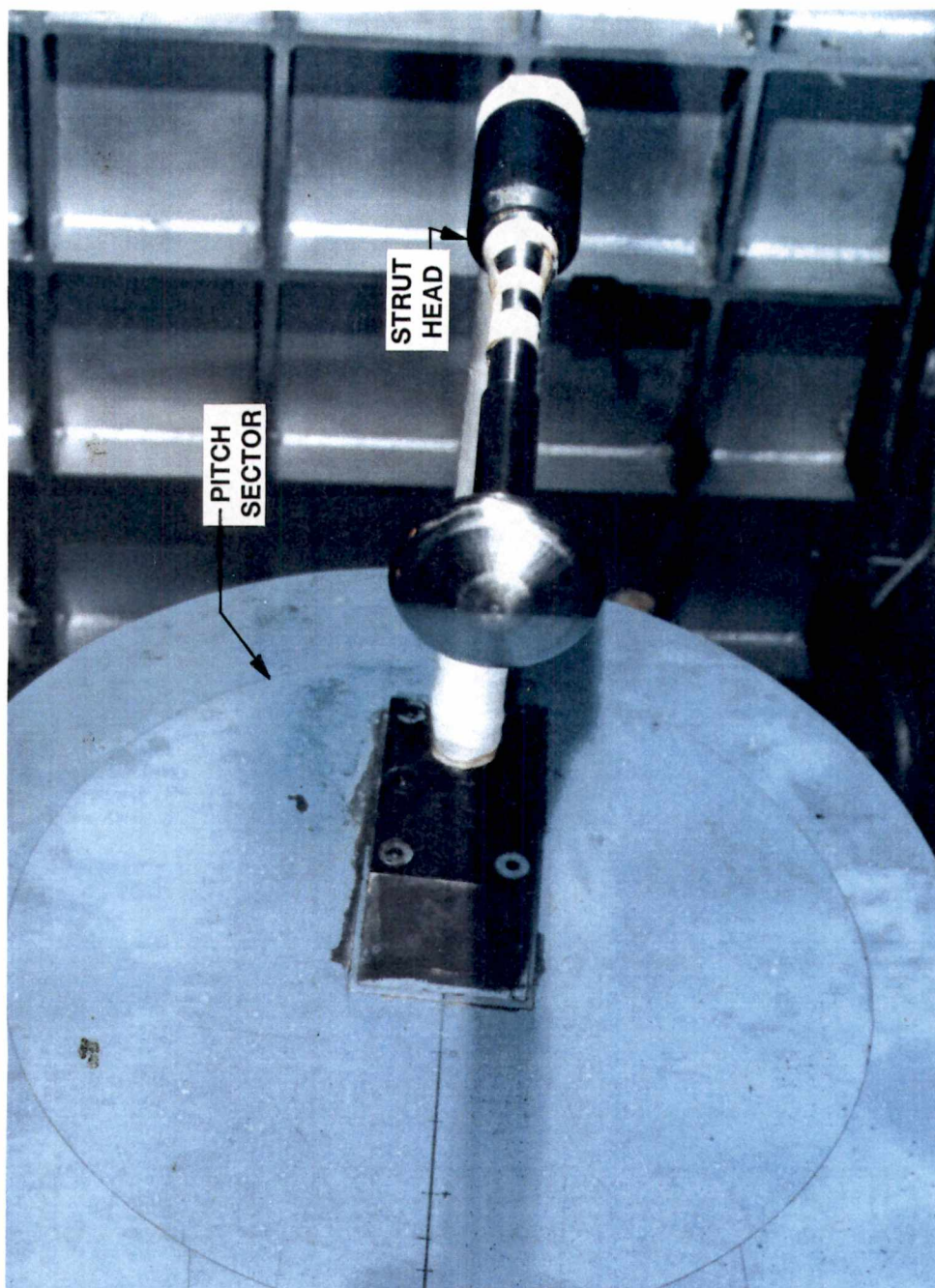


Figure 10. Photograph of Cylinder Pressure Model Installed in Wind Tunnel

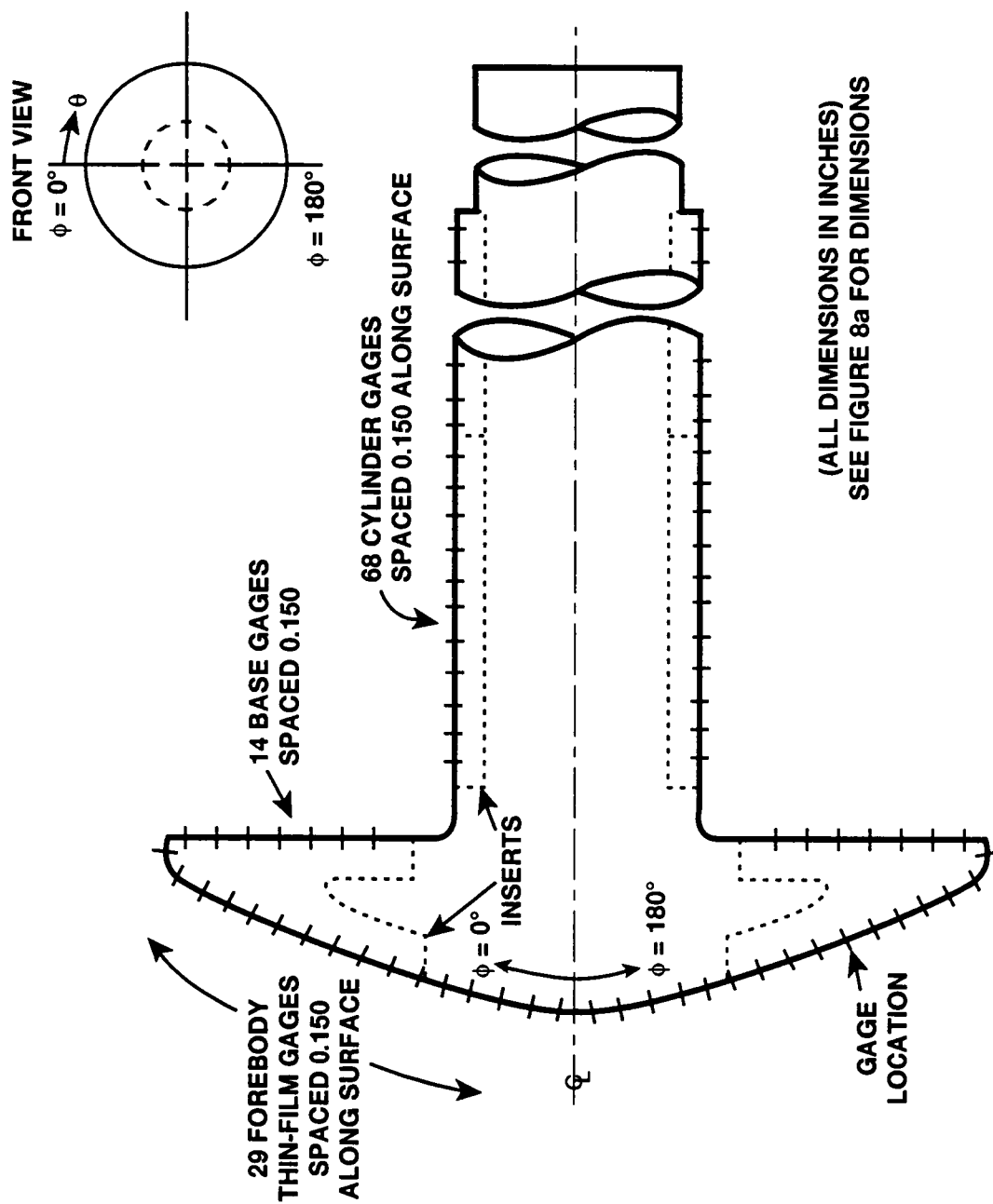
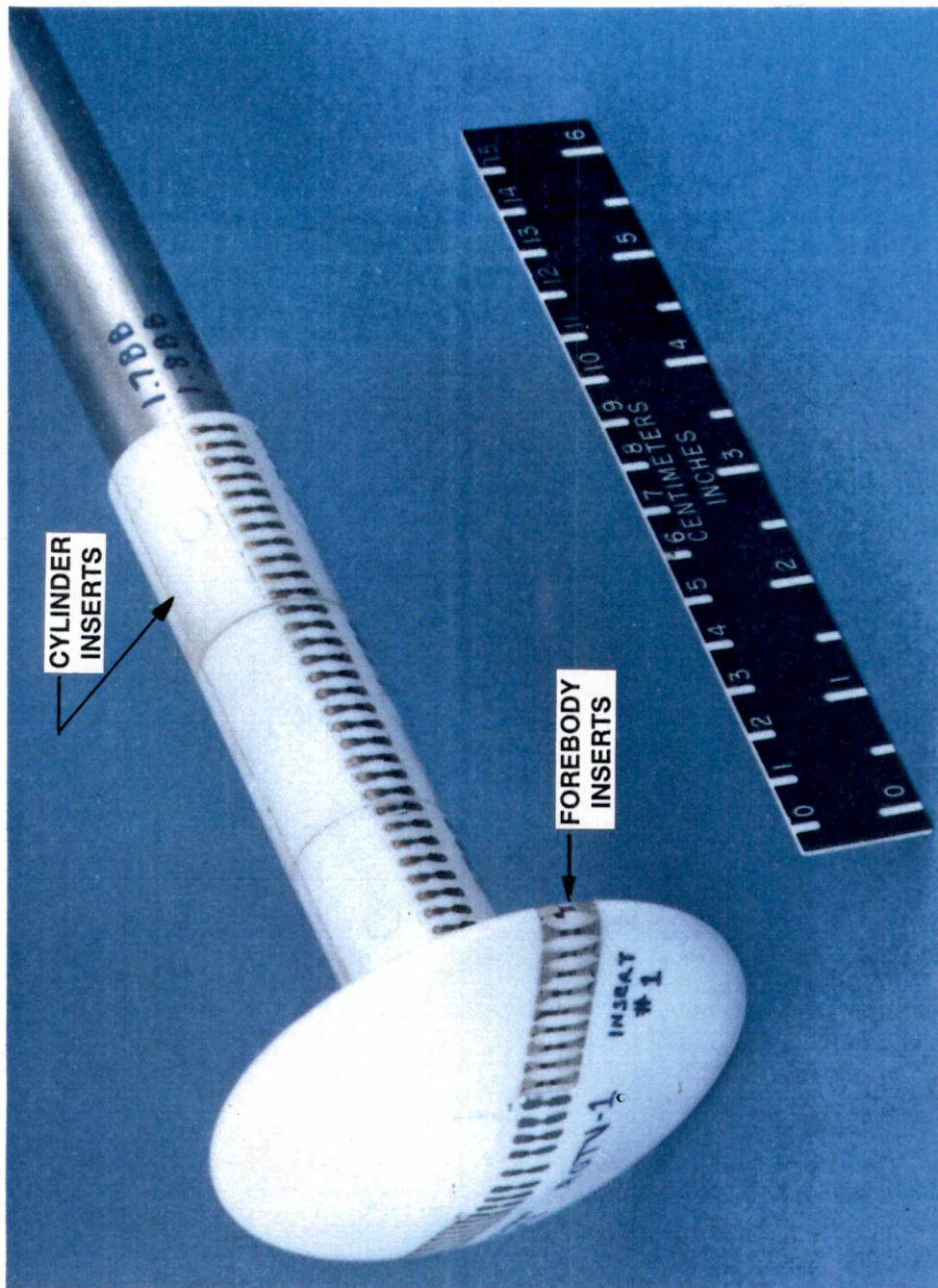
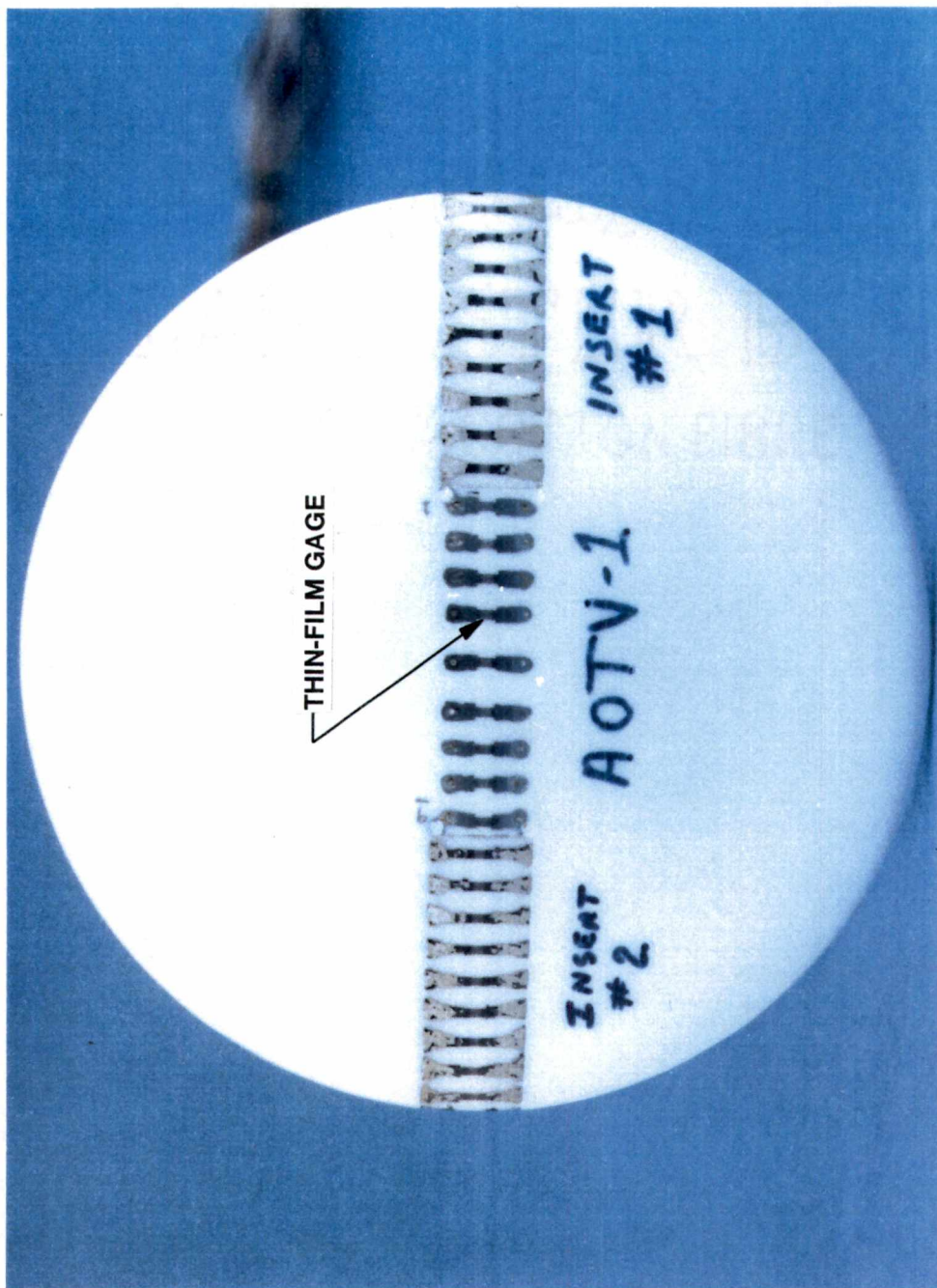


Figure 11. Heat Transfer Model Details



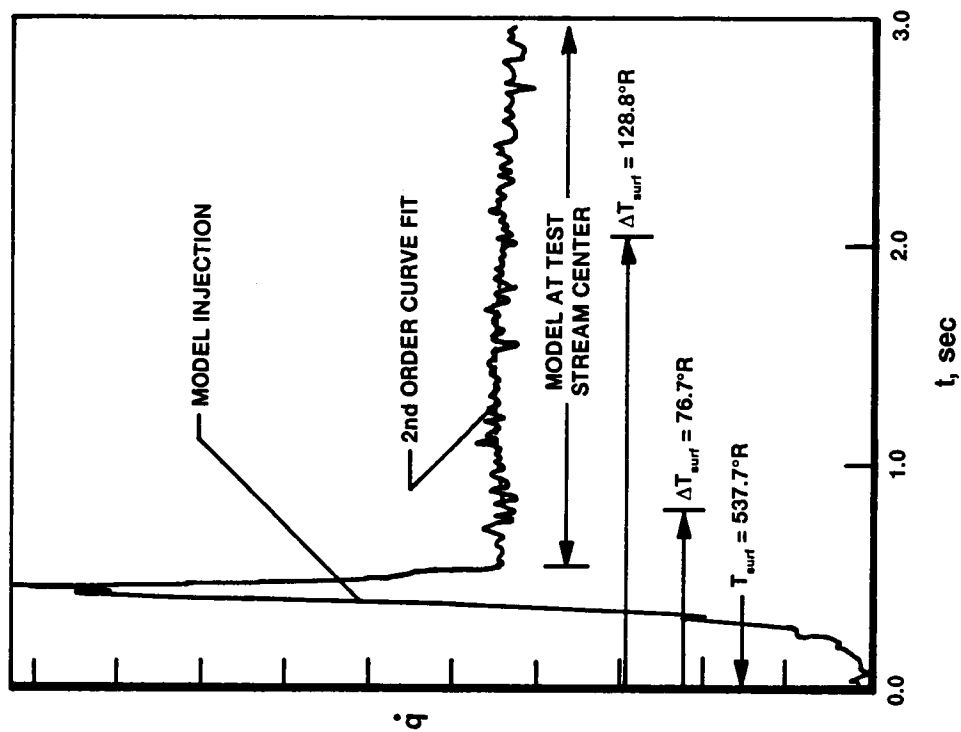
a) Thin-Film ASTV Model

Figure 12. Photograph of Heat Transfer Model

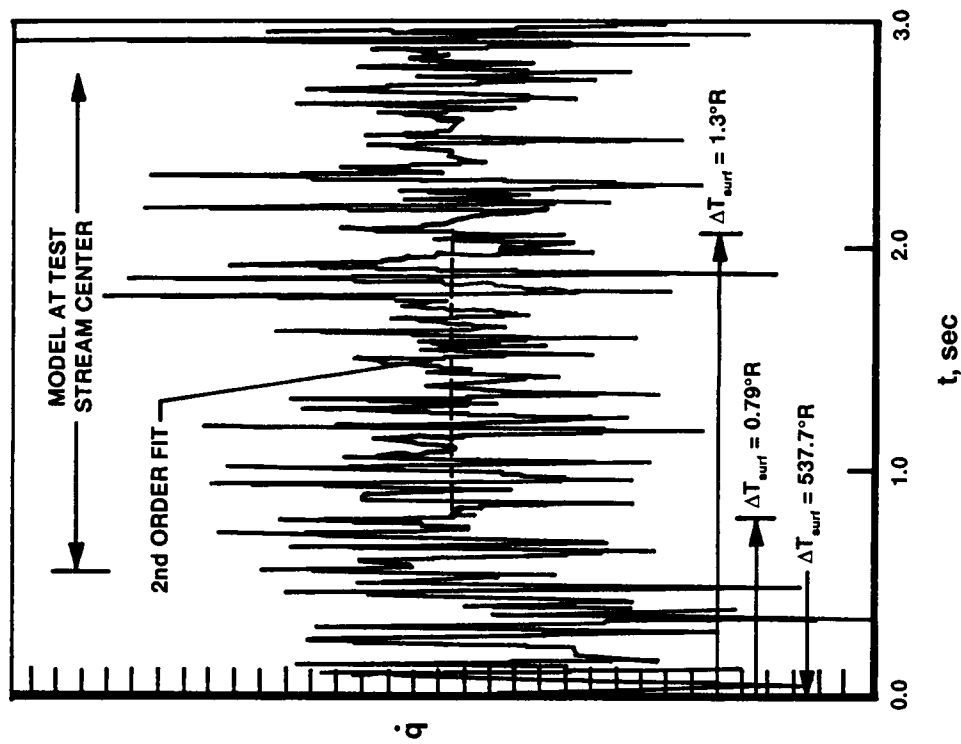


b) Close-up of Forebody and Thin-Film Gages

Figure 12. Concluded

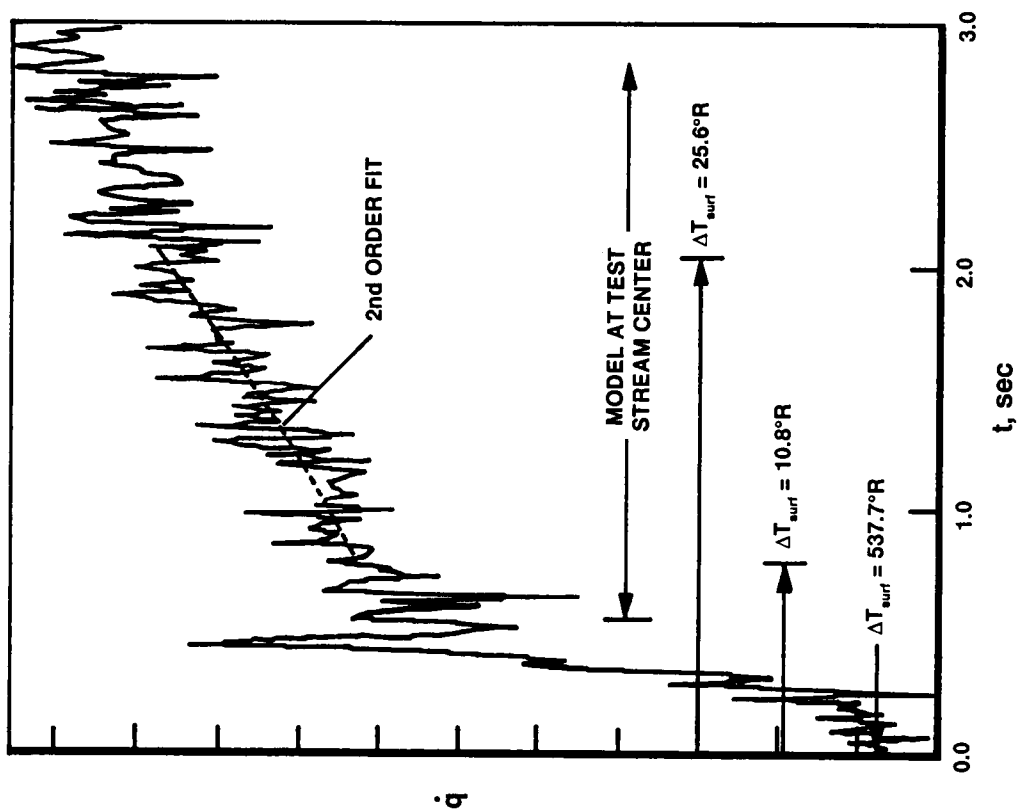


a) Example of High Signal-to-Noise Ratio (Stagnation Point)



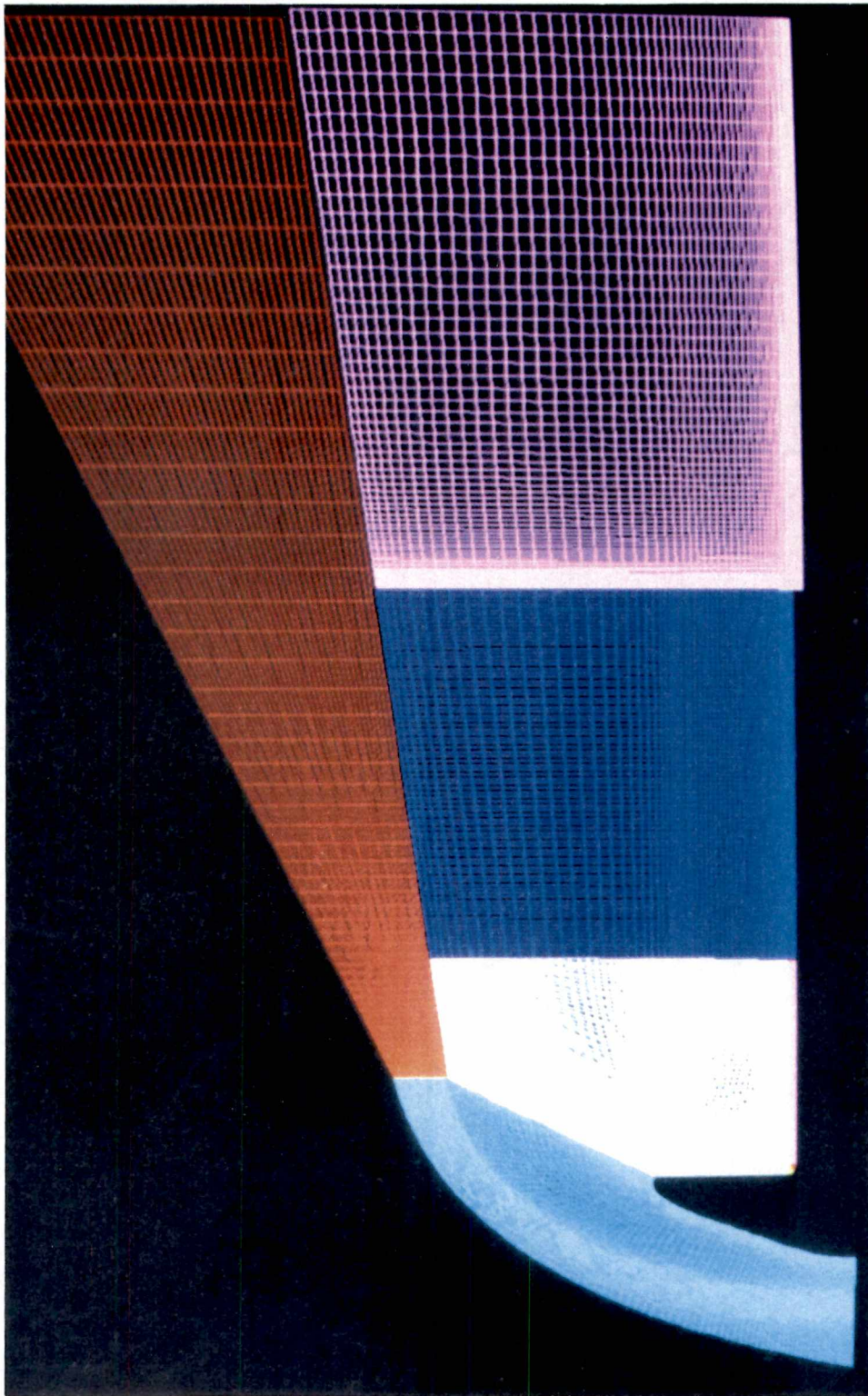
b) Example of Low Signal-to-Noise Ratio (Base Region)

Figure 13. Heating Rate Time Histories Indicating Range of Signal-to-Noise Ratio (Test 281, Run 4)



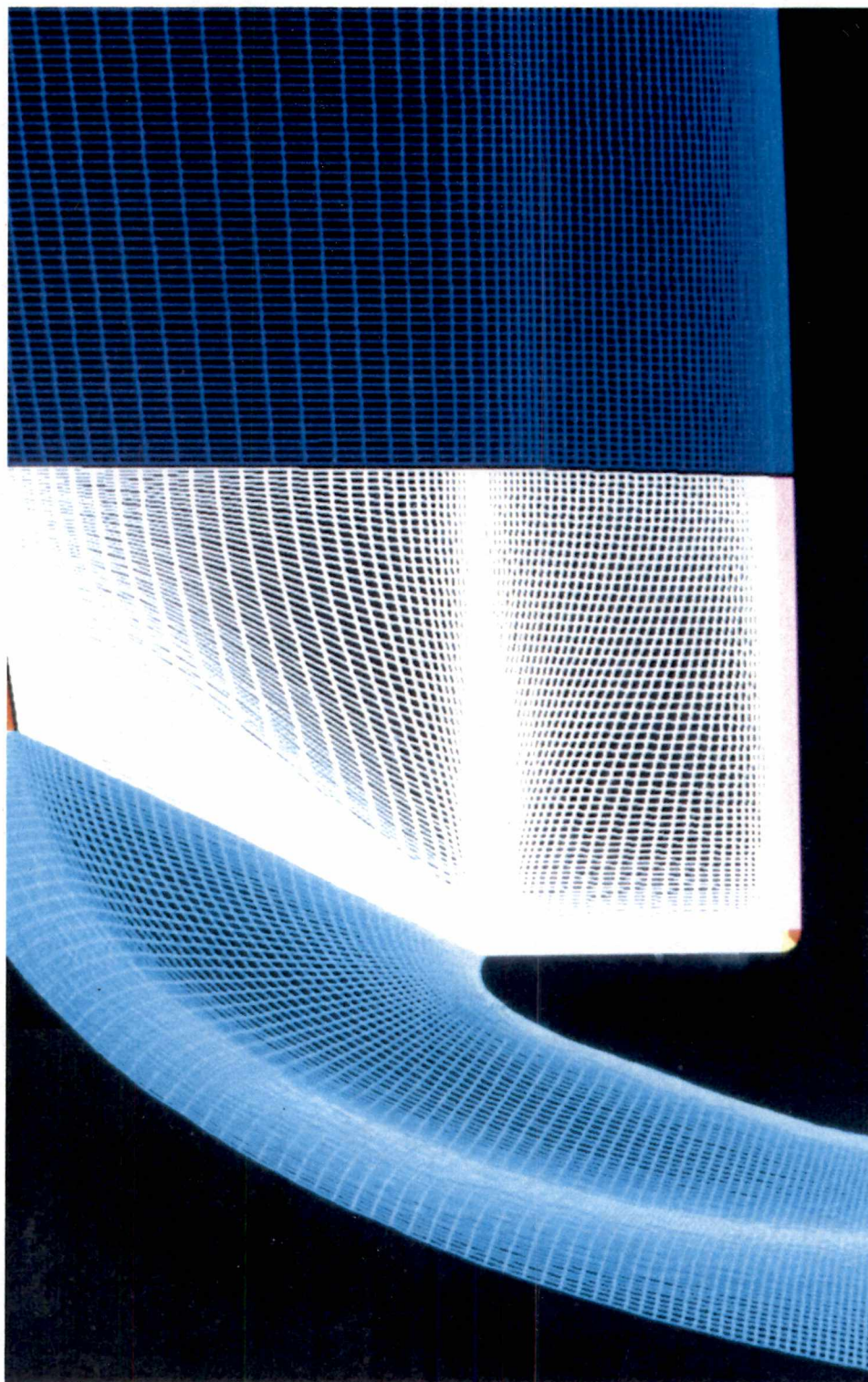
c) Example of Medium Signal-to-Noise Ratio and Effects of Conduction (Forebody Corner)

Figure 13. Concluded



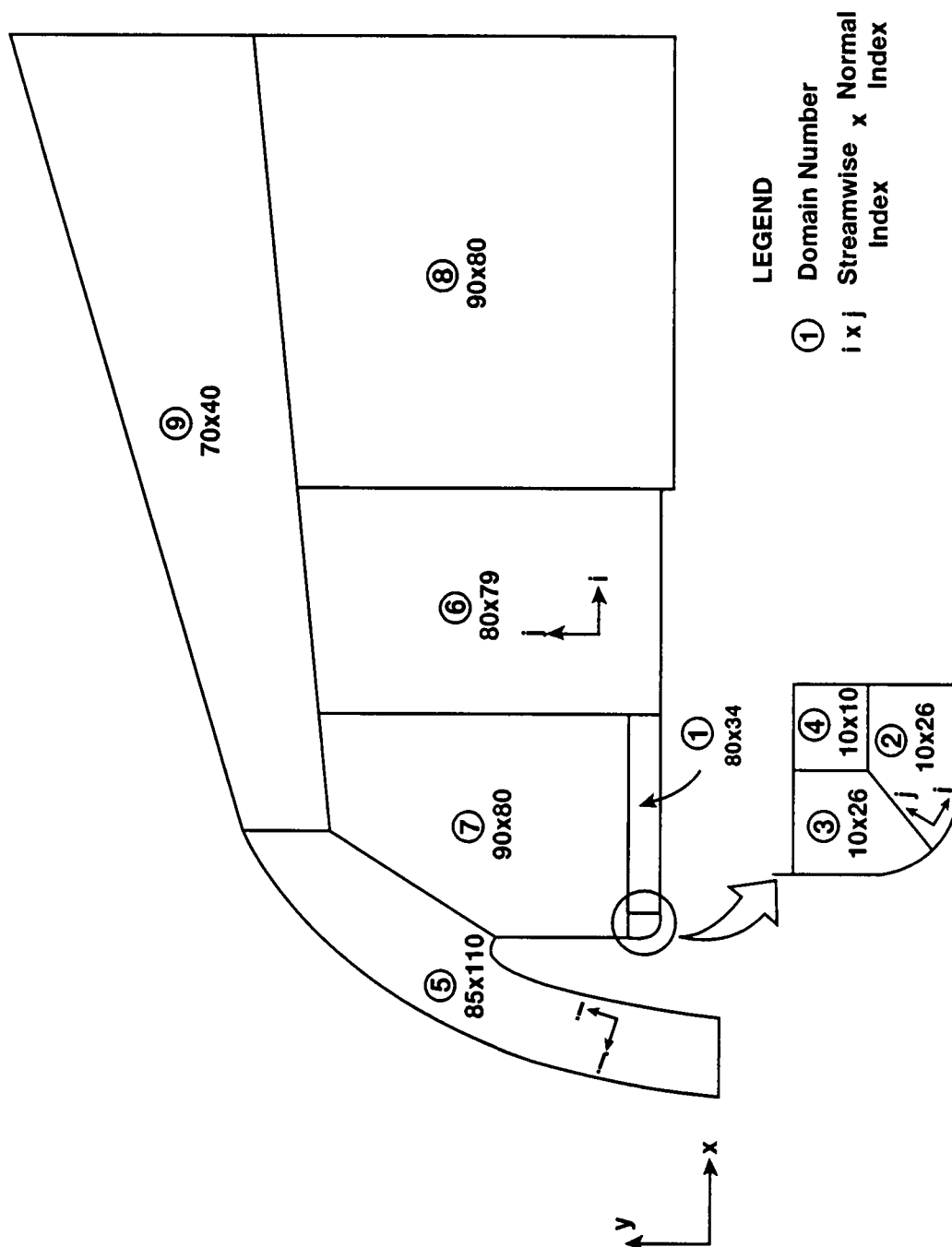
a) View of Entire Grid, Showing 9 Separate Domains

Figure 14. TUFF Grid Structure



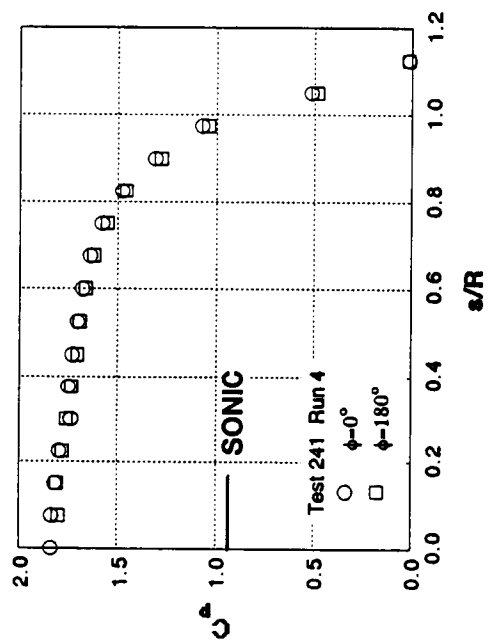
b) Enlarged View of Forebody and Near-Wake Domains

Figure 14. Continued

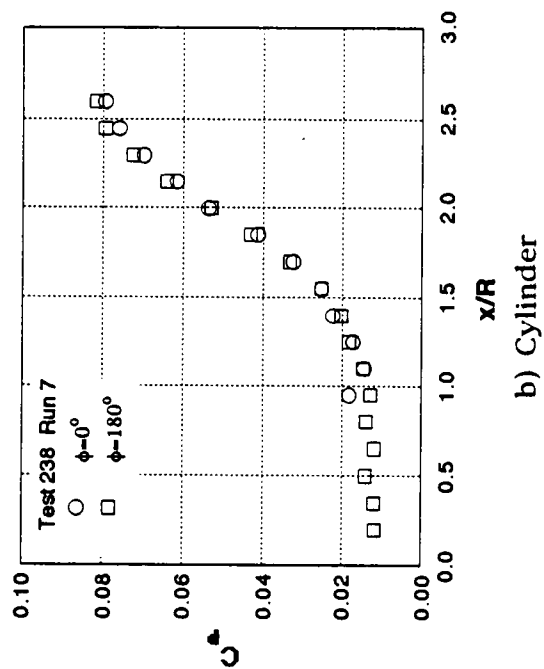


c) Domain Numbering and Size

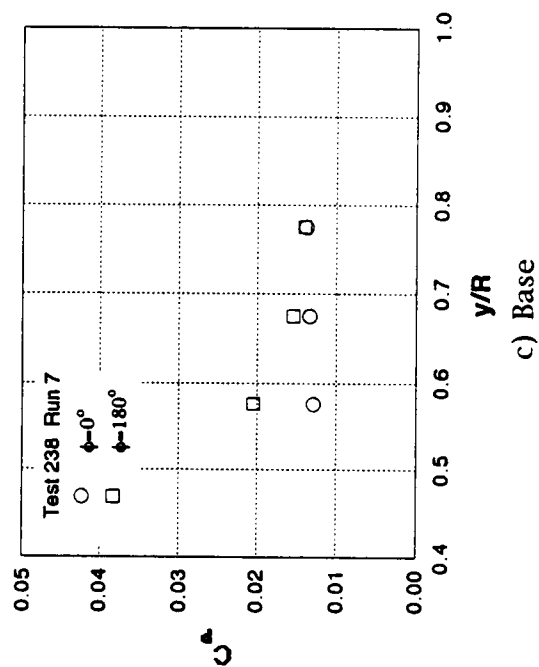
Figure 14. Concluded



a) Forebody



b) Cylinder



c) Base

Figure 15. Experimental Pressure Distributions at $\theta = 0^\circ$

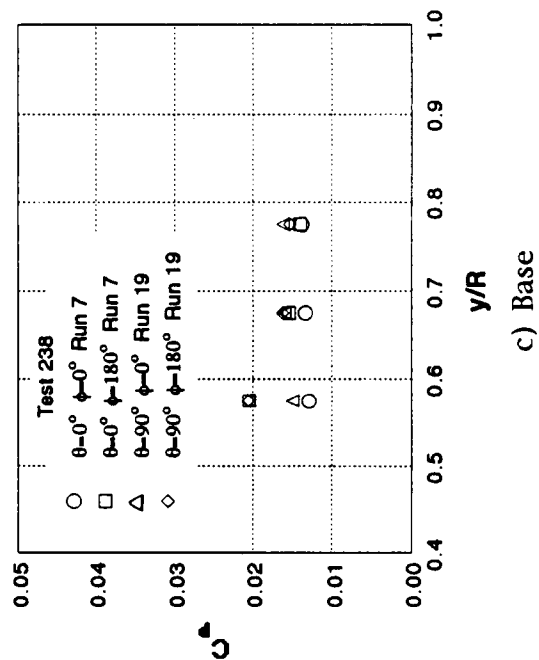
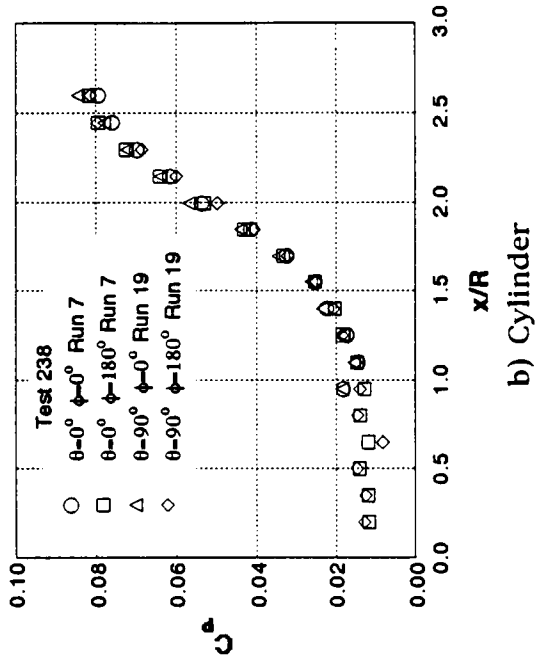
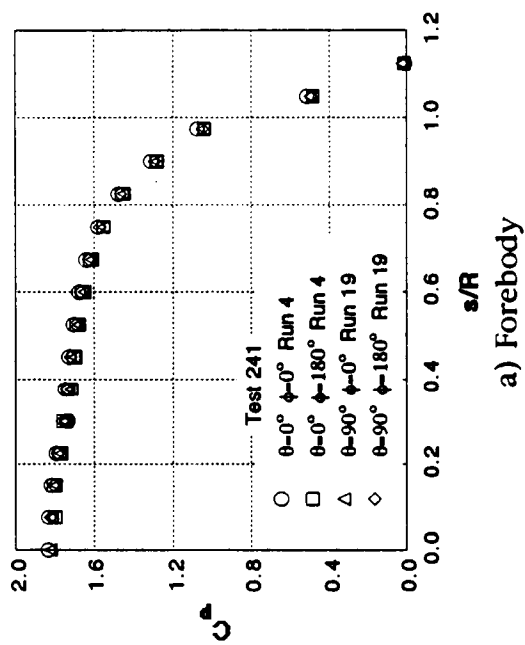
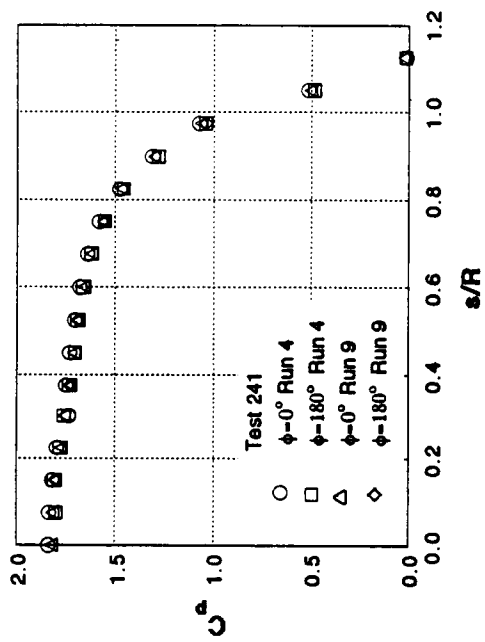
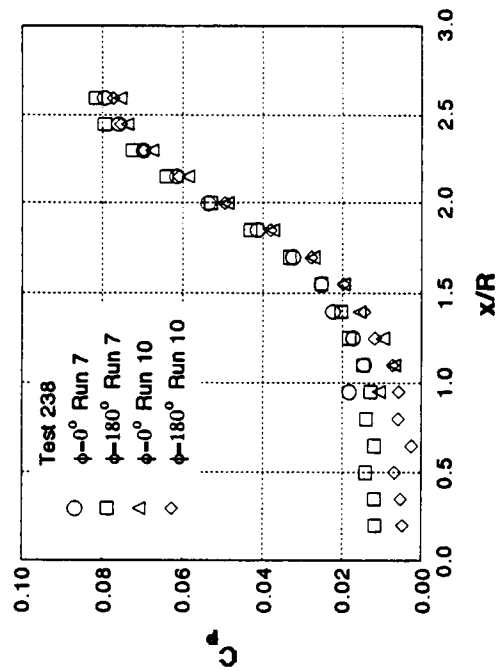


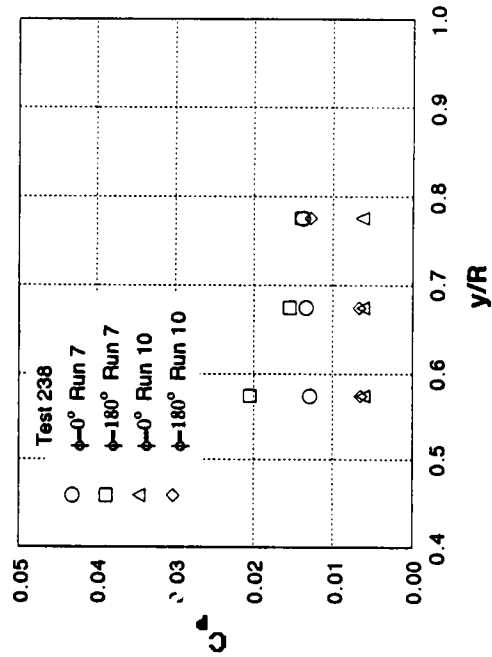
Figure 16. Experimental Pressure Distributions at $\theta = 0^\circ$ and $\theta = 90^\circ$



a) Forebody

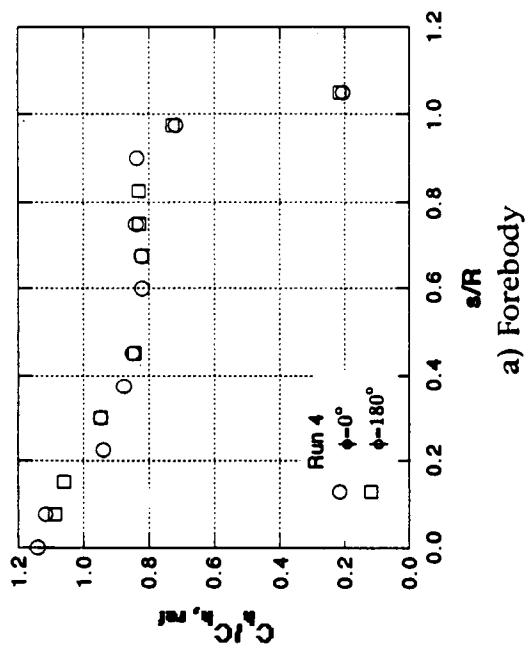


b) Cylinder

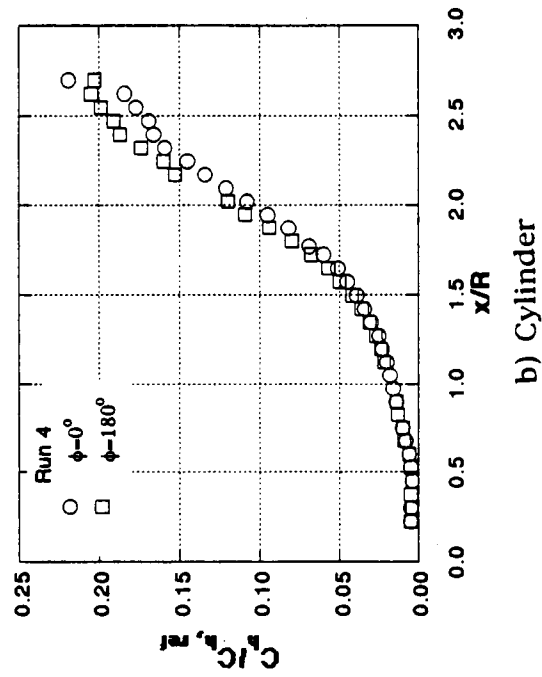


c) Base

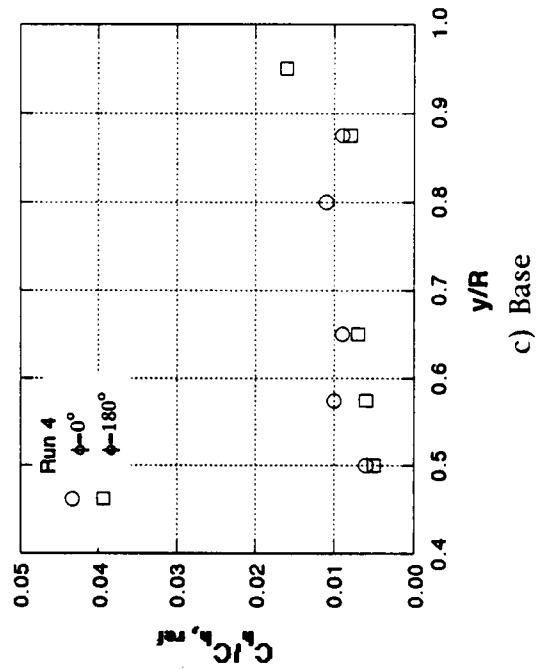
Figure 17. Experimental Pressure Distributions - Data Repeatability at $\theta = 0^\circ$



a) Forebody

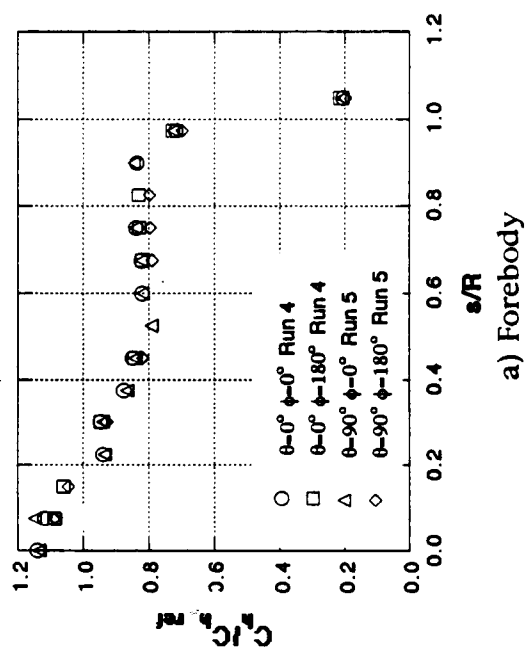


b) Cylinder

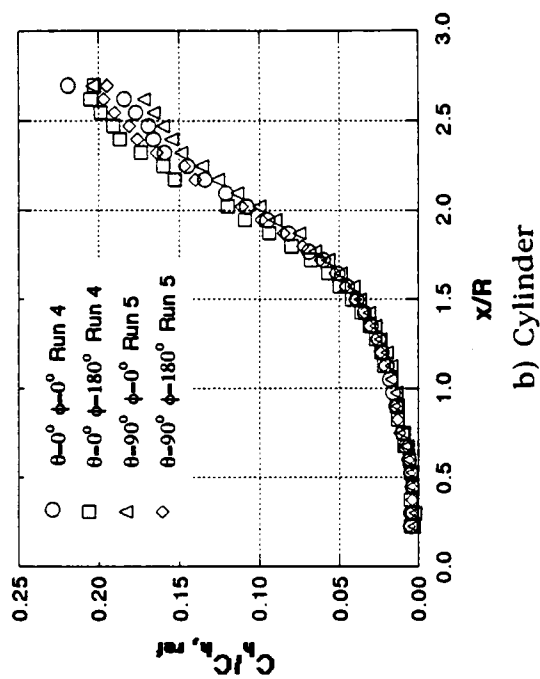


c) Base

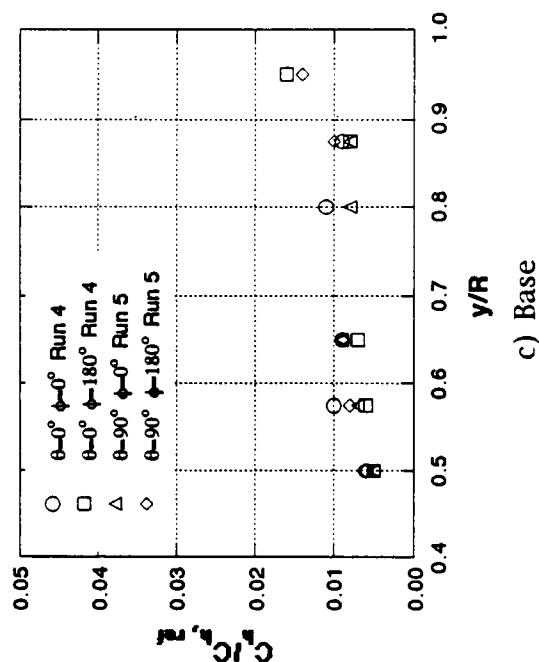
Figure 18. Experimental Heat Transfer Distributions at $\theta = 0^\circ$



a) Forebody



b) Cylinder



c) Base

Figure 19. Experimental Heat Transfer Distributions at $\theta = 0^\circ$ and $\theta = 90^\circ$

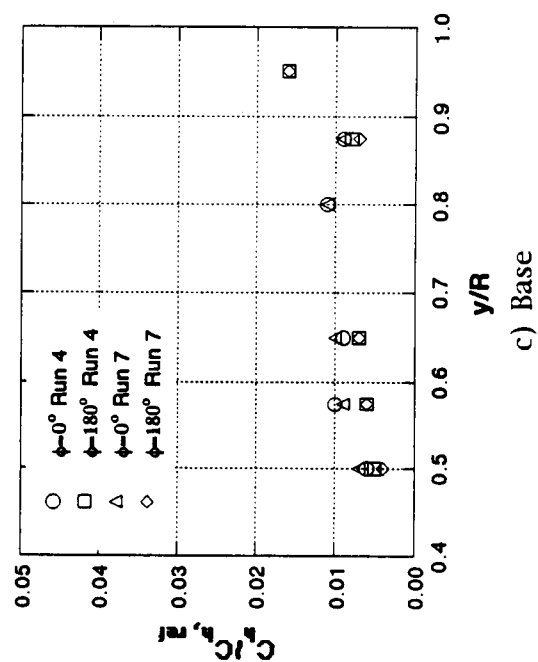
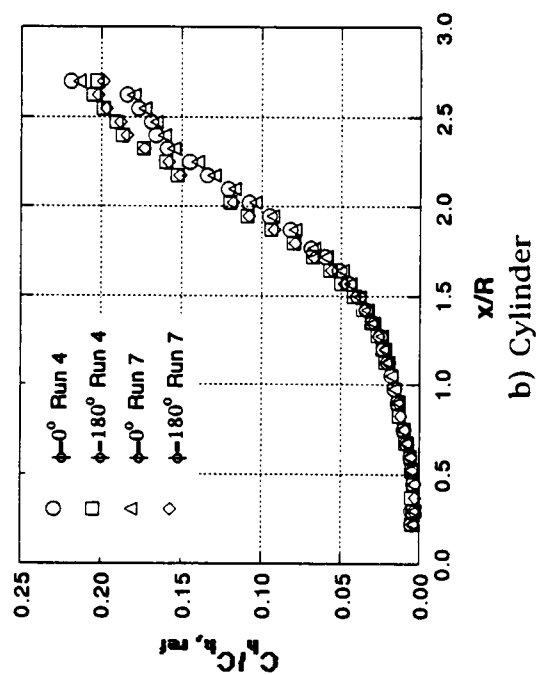
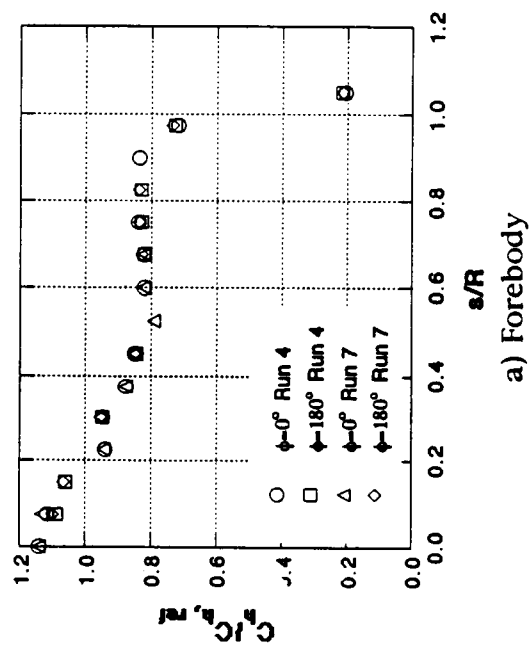


Figure 20. Experimental Heat Transfer Distributions - Data Repeatability at $\theta = 0^\circ$

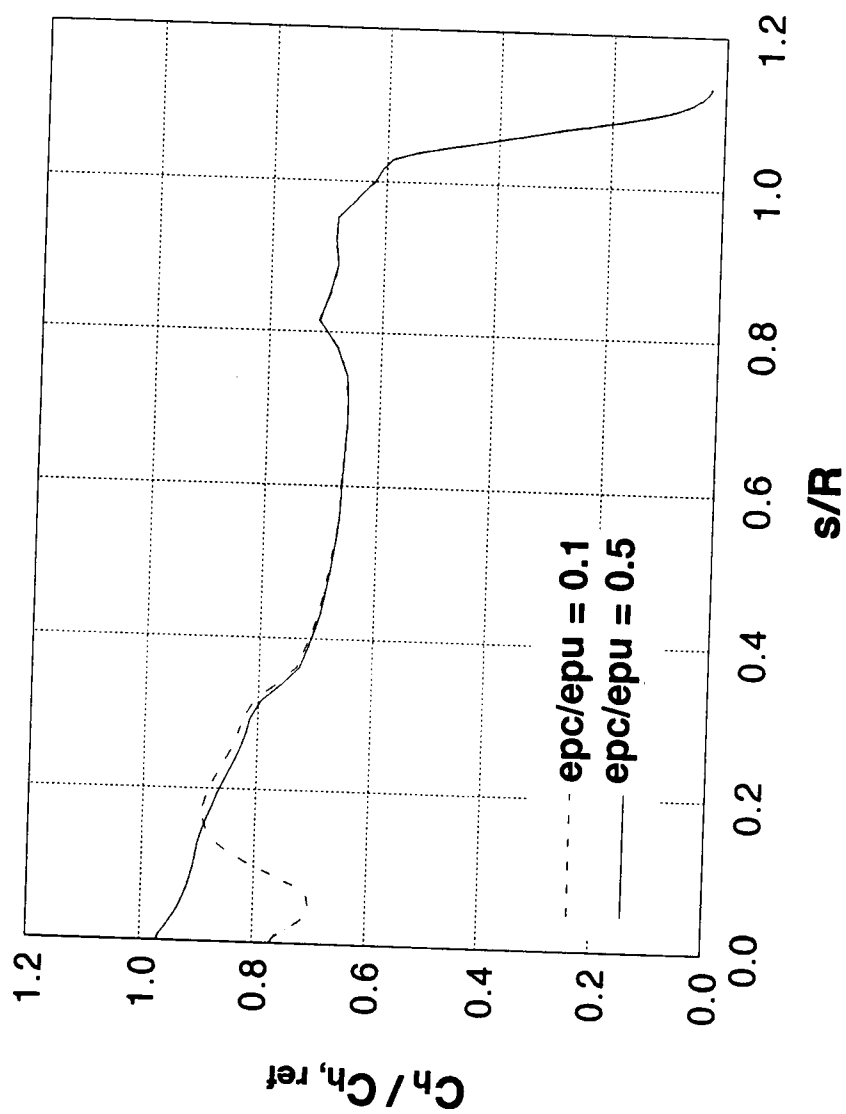
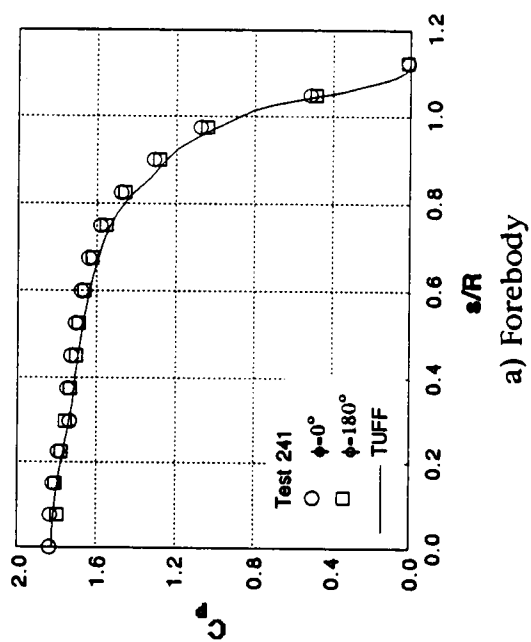
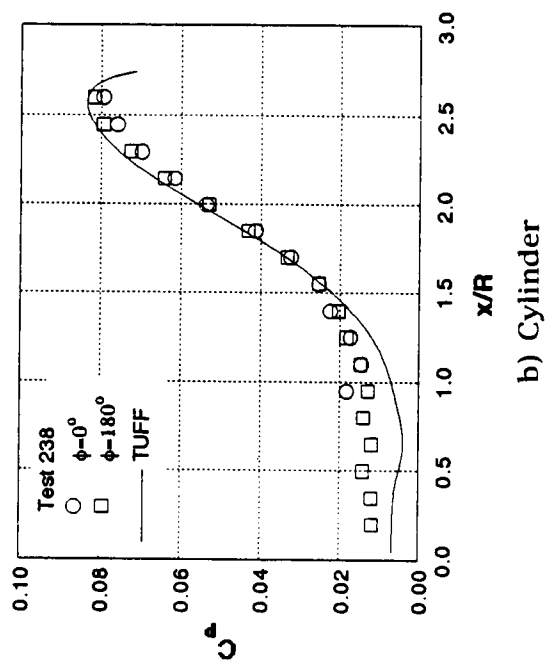


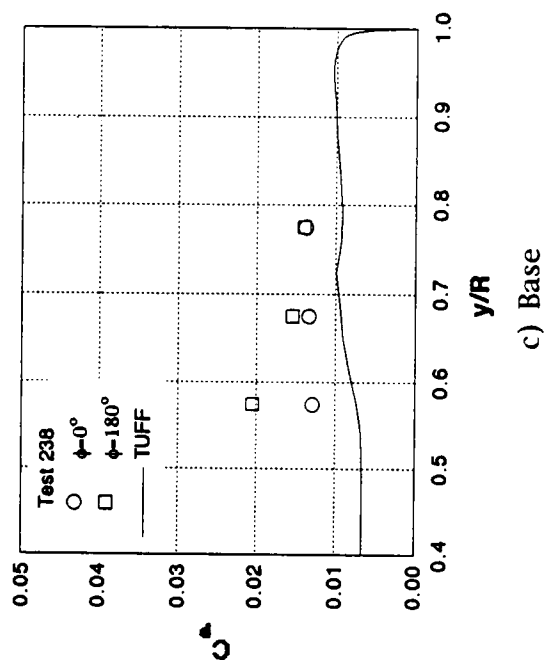
Figure 21. Computed Heat Transfer Anomaly on Forebody



a) Forebody



b) Cylinder



c) Base

Figure 22. Comparison of Experimental and Computed Pressure Distributions at $\theta = 0^\circ$

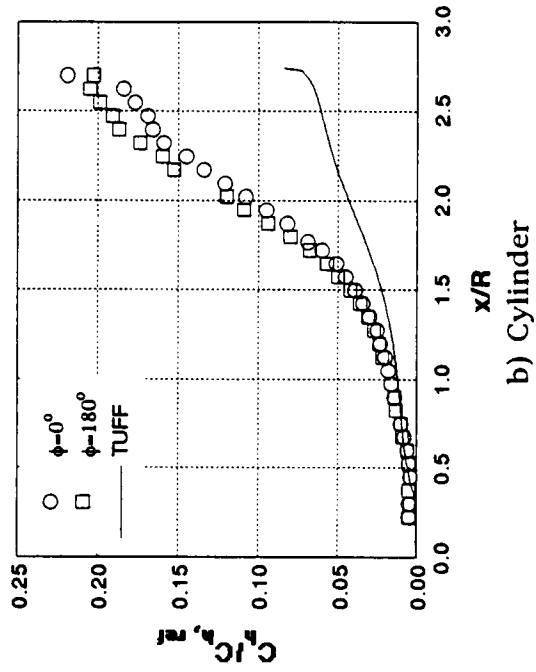
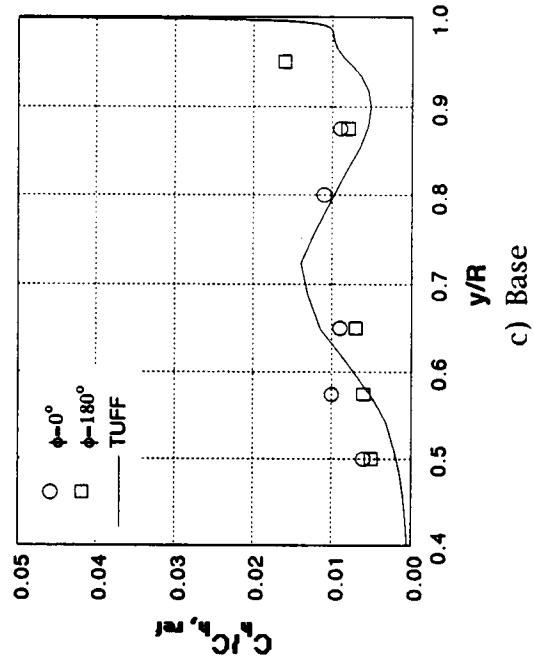
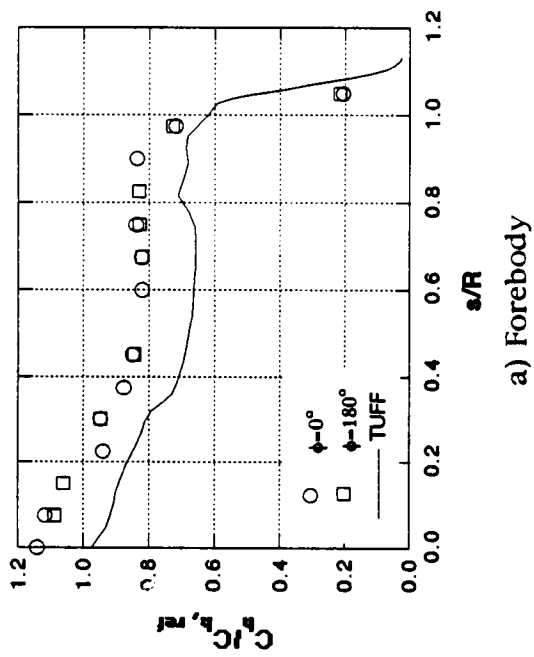


Figure 23. Comparison of Experimental and Computed Heat Transfer Distributions at $\theta = 0^\circ$

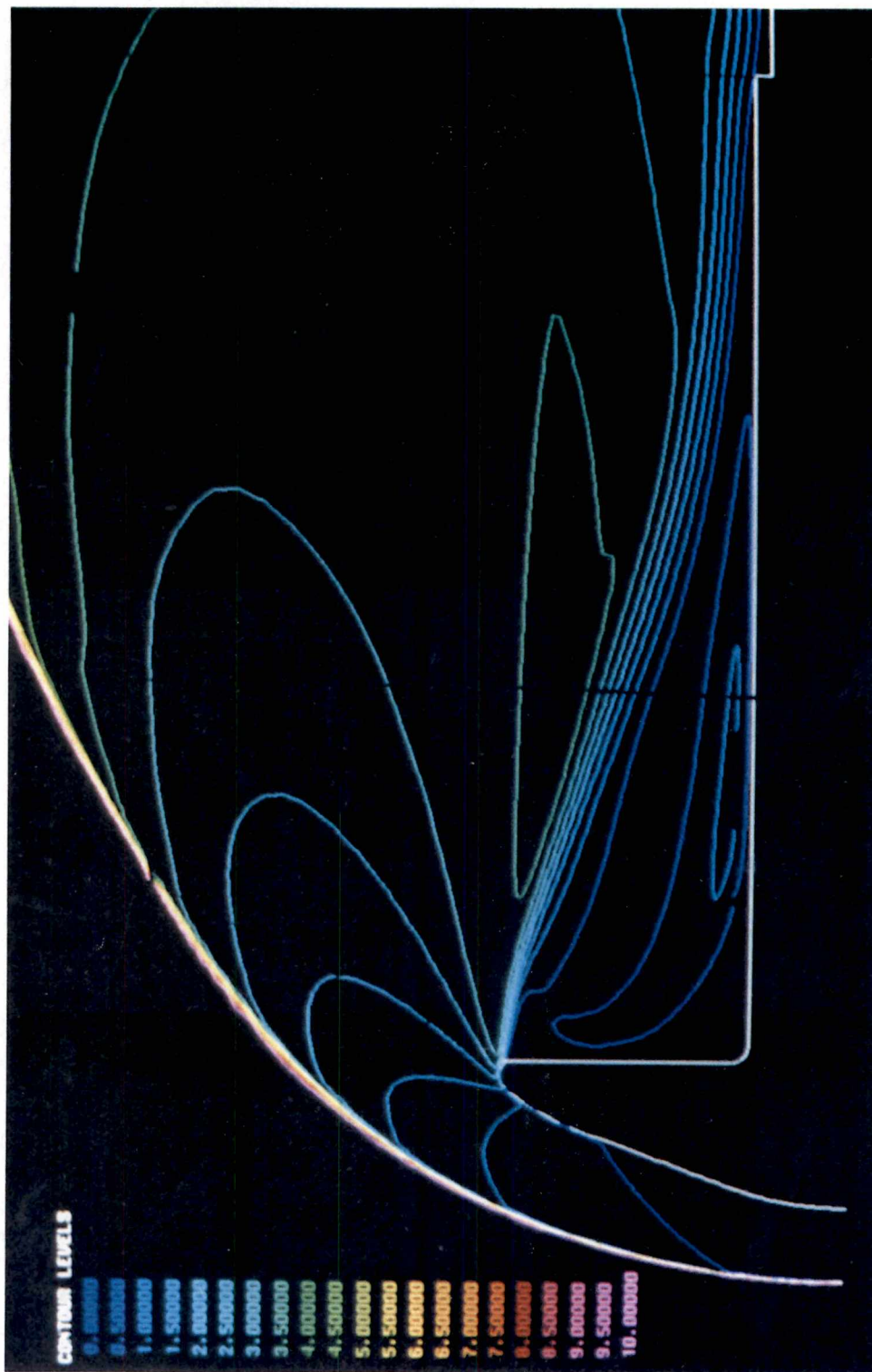
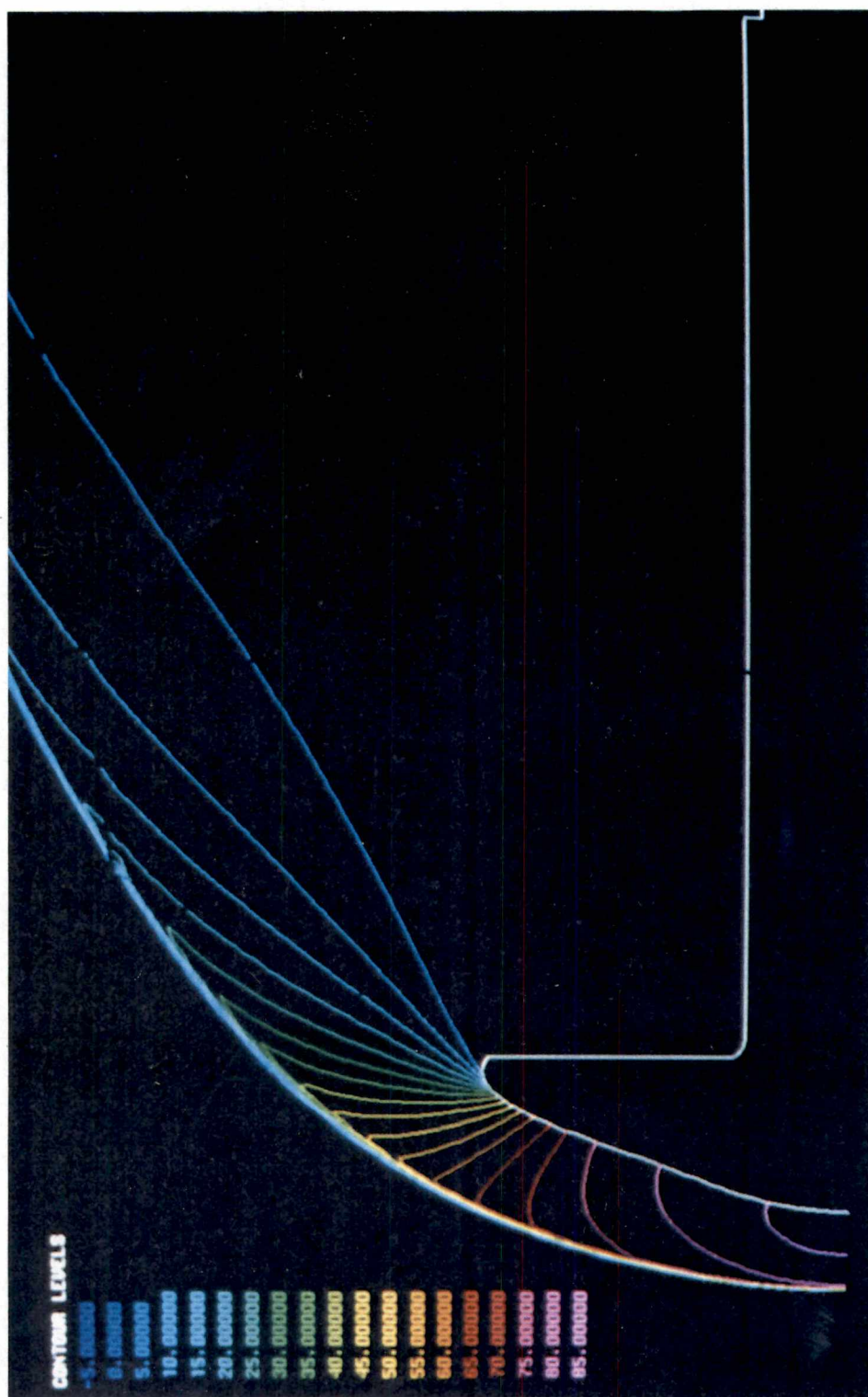
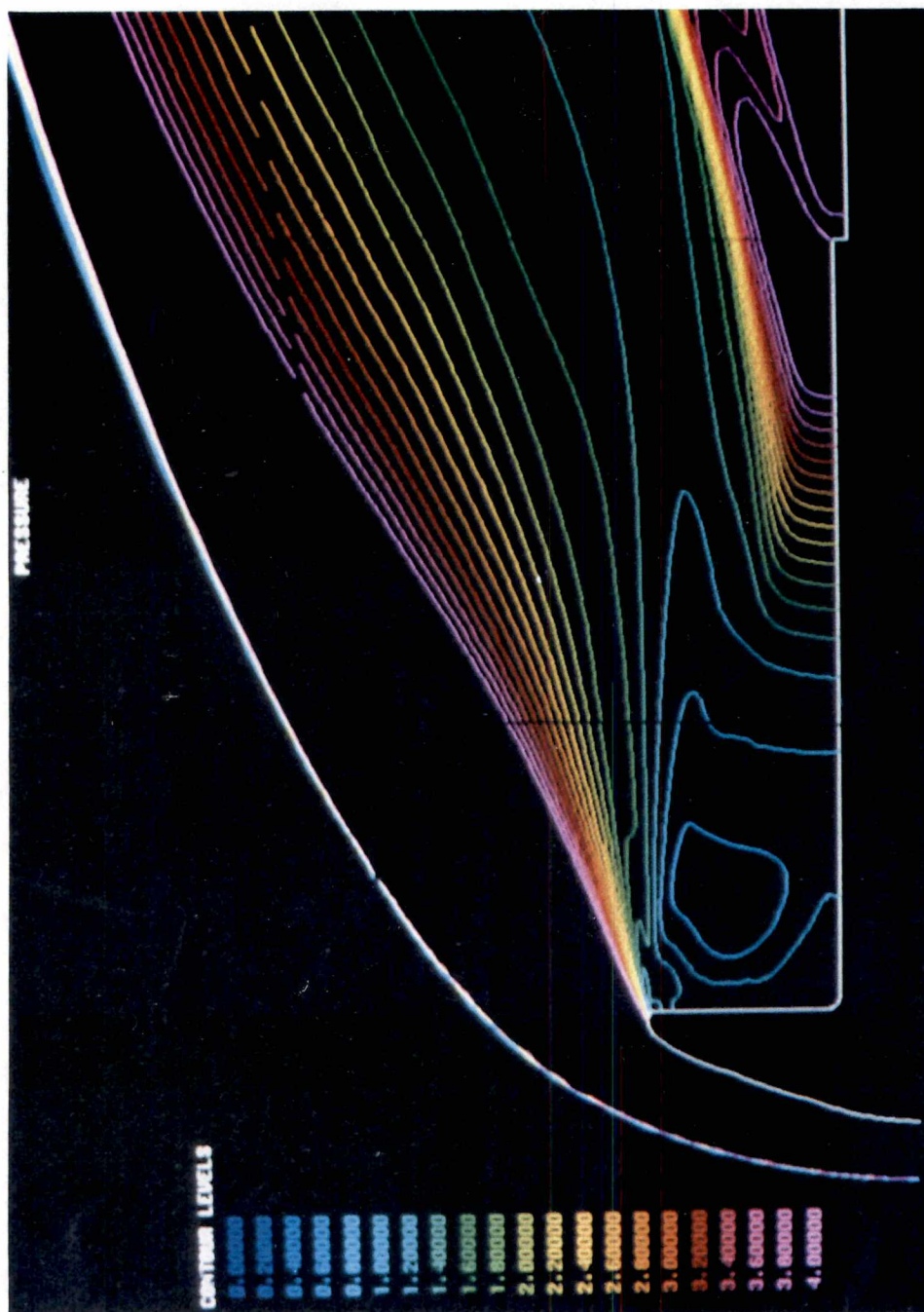


Figure 24. Computed Mach Contours for the ASTV Flowfield



a) Forebody

Figure 25. Computed Static Pressure Contours for the ASTV Flowfield

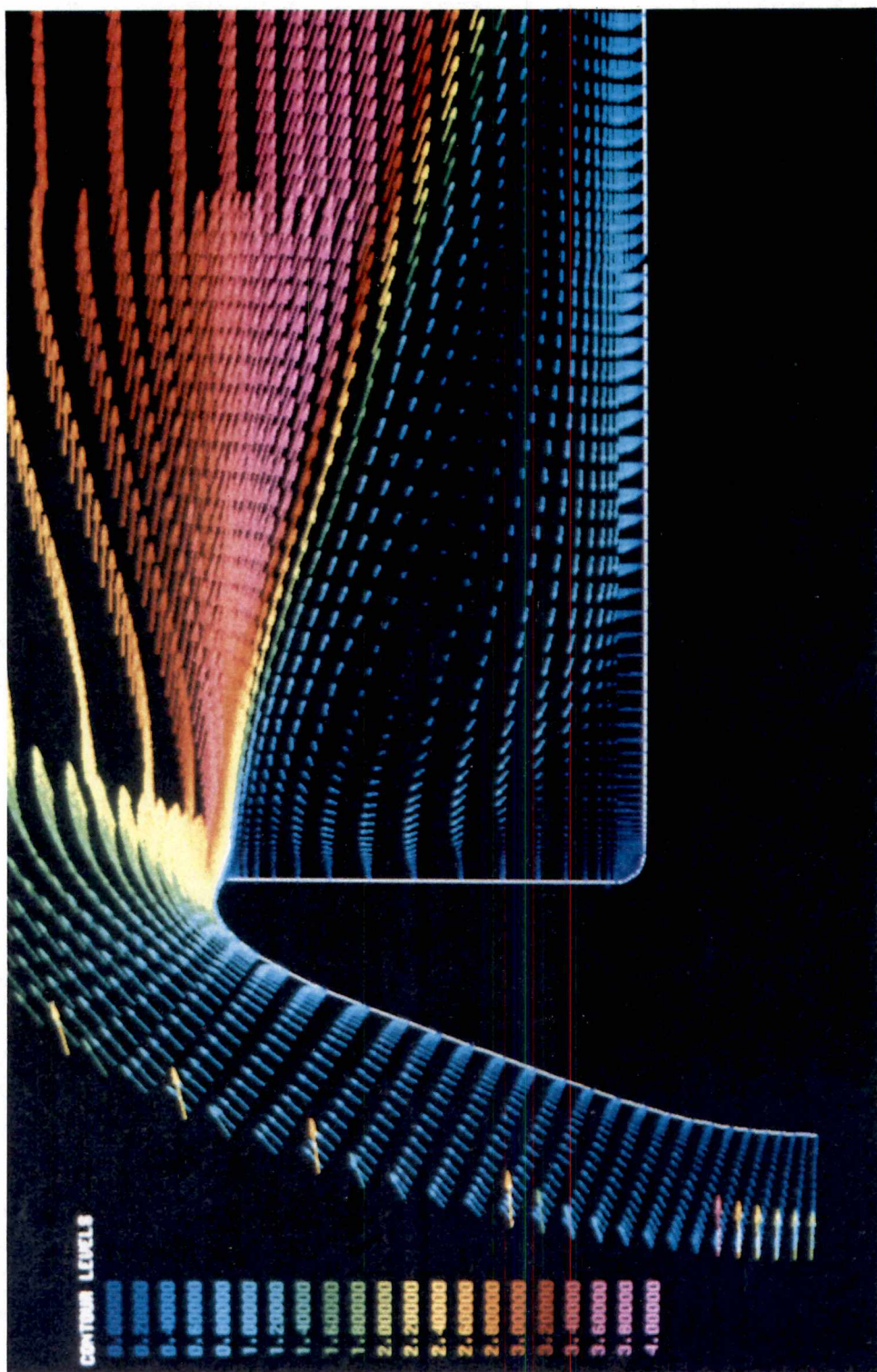


b) Near-Wake

Figure 25. Concluded

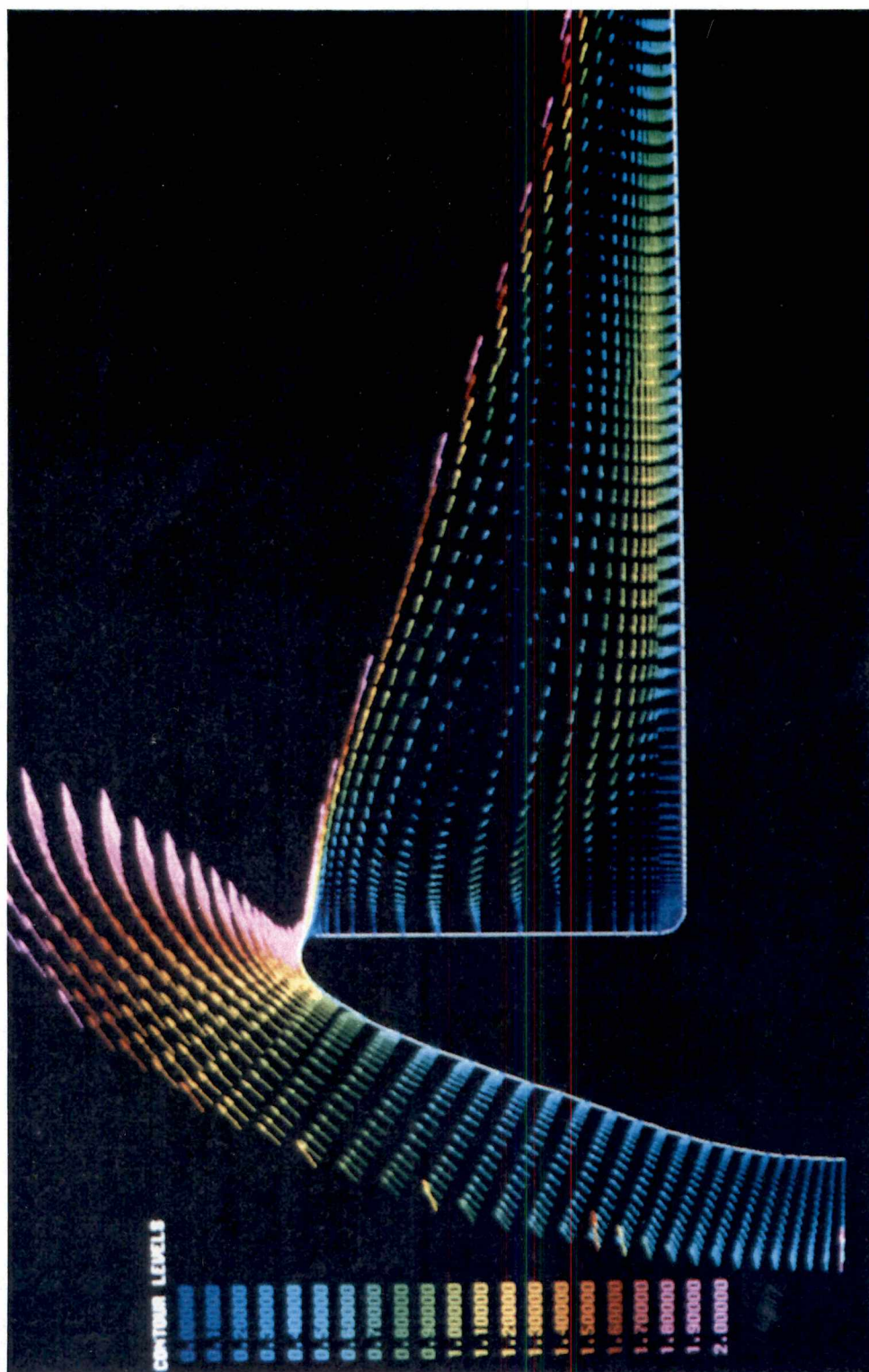


Figure 26. Computed Sonic Line for the ASTV Flowfield



a) Forebody and Near-Wake

Figure 27. Computed Velocity Vectors Colored by Mach Number for the ASTV Flowfield



b) Detail of Base Region Recirculation

Figure 27. Concluded

APPENDIX B

Uncertainty Analysis for Pressure Measurements

APPENDIX B

Uncertainty Analysis for Pressure Measurements

Manufacturer specifications for the ESP measurement system state that the overall system uncertainty is 0.07% of the full-scale reading. This corresponds to a maximum uncertainty of 0.00025 psia for the 0.36 psi module and 0.0035 psia for the 5.0 psi module. Other errors may have been introduced from the ESP calibration (tubing effects as discussed earlier), settling time, and in the calculation of the pressure coefficient. The overall uncertainty can be considered to consist of two parts: the precision, which is the variation between repeated measurements, and the bias, which is the constant or systematic error. The manufacturers specification only states the precision of the measured pressure and does not account for known bias. Uncertainty may be centered about the measurement and is defined as (Abernethy and Thompson, 1973):

$$U = \pm (B + t_{95}S)$$

where B is the bias limit, S is the precision index, and t_{95} is the 95th percentile point for the two-tailed Students "t" distribution. If a large number of samples are considered in the determination of the precision, as was the case for the present work, t_{95} can be set to 2. In the present tests there were two known sources of bias: a bias in the calibration due to tubing effects (line losses, leaks, or outgassing) and a bias due to settling time. It was difficult to quantify the bias in the calibration due to tubing effects since a detailed investigation

was not conducted. The bias due to settling time has already been quantified as within 1% of the measured pressure. Lacking further information, a total bias equal to 1% of the pressure coefficient was assumed as a worst case for the 0.36 psi module. The total bias was assumed to be negligible for the 5.0 psi module. The precision of the flow quantities used in the calculation of the pressure coefficients was assumed to be 1% of the parameter value. This assumption was based on the estimated uncertainty ($\pm 1\%$) of the Pitot pressure from the facility calibration which is used to calculate flow properties.

The precision of the calculated pressure coefficient was then determined from the Taylor's series expansion method (Abernethy and Thompson, 1973), where:

$$S_{C_p} = \sqrt{\left(\frac{\partial C_p}{\partial q_\infty} S_{q_\infty}\right)^2 + \left(\frac{\partial C_p}{\partial p_\infty} S_{p_\infty}\right)^2 + \left(\frac{\partial C_p}{\partial p} S_p\right)^2}$$

$$S_{q_\infty} = 0.01q_\infty$$

$$S_{p_\infty} = 0.01p_\infty$$

$$S_p = 0.00025 \text{ psia (0.36 psi ESP module)}$$

$$S_p = 0.00350 \text{ psia (5.0 psi ESP module)}$$

and $C_p = \frac{p - p_\infty}{q_\infty}$. The lowest measured pressure on the 0.36 psi module resulted in the maximum uncertainty for the tests. This lowest pressure was nominally 0.01 psia ($C_p=0.01$) and was located on the cylinder near to the forebody base. The estimated uncertainty in terms of the pressure coefficient for this measurement was ± 0.0007 or $\pm 7\%$ of the value. The lowest pressure on the 5.0 psi module was 0.684 psia ($C_p=1.06$) and occurred on the forebody shoulder. The estimated uncertainty in terms of the pressure coefficient for this

measurement was determined to be ± 0.013 or $\pm 1.2\%$ of the value. The table below lists the estimated uncertainty in terms of both the measured pressure and pressure coefficient for the range of values for the tests.

C_p	p, psia	Model Region	S_{C_p}	U in C_p	U in $\%C_p$
0.01	0.010	Cylinder/Base	0.0006	± 0.0007	$\pm 7.0\%$
0.08	0.055	Cylinder	0.0009	± 0.0017	$\pm 2.1\%$
0.50	0.336	Forebody Shoulder	0.0052	± 0.0102	$\pm 2.0\%$
1.06	0.684	Forebody Shoulder	0.0130	± 0.0130	$\pm 1.2\%$
1.83	1.170	Forebody Nose	0.0200	± 0.0200	$\pm 1.1\%$

VITA

Thomas P. Dye was born in Orlando, Florida on May 26, 1965. He attended elementary schools in Lancaster, California and Gaithersburg, Maryland. He graduated from Seneca Valley High School in June, 1983 and attended Montgomery College in Germantown, Maryland for one year, studying engineering. He transferred to the Georgia Institute of Technology to study Aerospace Engineering in August, 1984. In August, 1987 he graduated from Georgia Tech with High Honors and received a regular commission as a Second Lieutenant in the United States Air Force. His initial assignment was as a Research Engineer at the NASA Langley Research Center in Hampton, Virginia where he performed research in hypersonics. He then was transferred to the Arnold Engineering Development Center (AEDC) in August, 1990 where he served as an Aerospace Systems Test Manager in the Propulsion Wind Tunnel and von Karman Gas Dynamics Facility. In January, 1990 he enrolled at the University of Tennessee Space Institute as a part time student. There he received a Master of Science Degree in Aerospace Engineering in December, 1993. Captain Dye currently serves as Manager of Cooperative Programs in the AEDC Office for International Affairs.