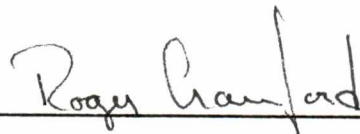


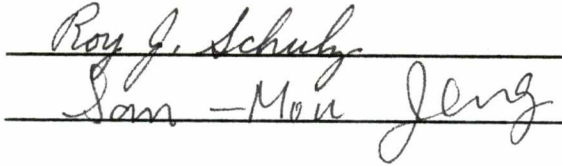
To the Graduate Council:

I am submitting herewith a thesis written by Royce Dean Purvis entitled "A PARC Code Approach for an ASTF Exhaust Header/Test Cell Bypass Flow Analysis." 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 Mechanical Engineering.

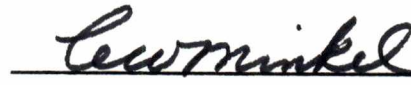


Roger Crawford, Major Professor

We have read this thesis
and recommend its acceptance:



Accepted for the Council



Vice Provost
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 acknowledgement of 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 Royce Dean Purvis

Date October 27, 1989

A PARC CODE APPROACH FOR AN ASTF
EXHAUST HEADER/TEST CELL BYPASS FLOW ANALYSIS

A Thesis
Presented for the
Master of Science
Degree
The University of Tennessee, Knoxville

Royce Dean Purvis

December 1989

ACKNOWLEDGEMENTS

I hereby thank Doctor Roger Crawford, who served as my major professor for his assistance in the preparation of this thesis. I also thank Doctor Roy Schulz and Doctor San-Mou Jeng for serving as my committee members. Kyle Cooper, Doctor A. Vakili, Tim Swafford, Ron Polce, John Leming, Gary Knox, Jim Rector, Scott Henninger, Lynn Moran, and Glen Lazalier were instrumental in helping my understanding of the problem and I give them special thanks. Finally, I give thanks to Shirley Miller for the excellent typing and printing of this thesis and to my wife Molly for her unwavering support.

Further reproduction of the work reported herein is authorized to satisfy the needs of the U.S. Government.

ABSTRACT

The object of this study was to evaluate the effect of a test cell bypass in operation upon the flow field of a jet engine test facility's exhaust header. A simplified analysis of the mixing of the flow was performed and a flow field simulation computer program, the Propulsion Ames Research Center (PARC) Code, is presented for engineers to use to obtain a more complete analysis of the flow field. A suggested method for setting up the PARC computer program input files is provided which the AEDC engineer could use with the Cray computer to analyze the mixing characteristics of the exhaust header during bypass operations. The information could then be used to optimize exhausters selection to prevent excessive cold air ingestion by the compressors or to determine a solution to warm up the cold flow.

TABLE OF CONTENTS

CHAPTER		PAGE
I.	INTRODUCTION	1
	Statement of Problem	1
	Background	1
II.	FLOW DEFINITION	22
	Physical Configuration	22
	Flow Conditions	25
	Assumptions	25
	Station 1	25
	Station *1	26
	Station 2	27
	Station 3	28
III.	SLOPE OF THE JET SPREAD	34
IV.	PROPULSION AMES RESEARCH CENTER (PARC) CODE	41
	Requirement for PARC	41
	Introduction to PARC	41
	Input/Output Overview	43
	Grid and Initial Condition File	
	Construction	45
	Parameter File NAMELISTs Construction	52
V.	RECOMMENDATIONS AND SUMMARY	64
	Recommendations for Further Study	64
	Additional Detail and Fine Tuning	65
	Configuration Changes	67
	PARC Summary	68
	BIBLIOGRAPHY	69
	APPENDIXES	72
	APPENDIX A. Calculated Test Cell Bypass Flow	
	Conditions	73
	APPENDIX B. Two Stream Mixing Analysis	89
	APPENDIX C. Nondimensional Flow Parameter	
	Derivation	96
	VITA	101

LIST OF FIGURES

FIGURE	PAGE
1. Aerial of ASTF (Before Test Cell Bypass Modification)	2
2. Facility Site Plan View	3
3. Upper Left Hand Corner (ULHC) Flight Condition	6
4. Typical Refrigeration Turbine Arrangement	7
5. Flow Path From TC1 to C1	9
6. Heat Transfer Model Results	11
7. Test Cell Bypass Configuration #1	13
8. Test Cell Bypass Configuration #2	14
9. Test Cell Bypass Configuration #3	15
10. Test Cell Bypass Configuration #4	16
11. Results of C1 Cooldown with Bypass	17
12. Bypass Tie-in to the Exhaust Header	18
13. Physical Dimensions of the Duct (Not to Scale)	23
14. Radius and Flow Station Nomenclature	24
15. Nondimensionalized Jet Spread Parameters	35
16. Velocity Profiles at Two Cross Sections of the Exhaust Header	40

FIGURE	PAGE
17. The Two Input Data Files Required to Execute the PARC Codes	44
18. Grid and Initial Condition File	46
19. Simplified Illustration of Grid Generated by Do Loops 10 and 20	49
20. Flow Regions for Initial Condition Arrays	51
21. Parameter File NAMELISTS	53
22. NAMELIST PRTSEG Data Block Format	58
23. NAMELIST BOUNDS Construction	60
24. Suggested Flow Regions for Additional Print Segments	66

NOMENCLATURE

A	-	Area
a	-	Speed of Sound
c	-	Coefficient of the Angular Growth of Jet Thickness
c_o	-	Jet Constant
C_p	-	Constant Pressure Specific Heat
D	-	Diameter
E	-	Total Energy per Unit Volume
g_c	-	Gravity Constant
e	-	Internal Energy per Unit Mass
f	-	Force of Friction
Gr	-	Grashof Number
h	-	Enthalpy
J	-	BTU Conversion Factor, 778.2 ft lbf/BTU
k	-	Coefficient of Thermal Conductivity
L, l	-	Length
M	-	Mach Number
m	-	Ratio of Wake Flow Velocity to Primary Flow Velocity
Nu	-	Nusselt Number
$\dot{m}$	-	Massflow Rate
Pr	-	Prandtl Number
P	-	Absolute Pressure

q	-	Heat Transfer
r, R	-	Radius
Re	-	Reynolds Number
R	-	Specific Gas Constant
T	-	Temperature
U	-	Axial Velocity
V	-	Volume
v	-	Radial Velocity
x	-	Axis, Axial Distance
x'	-	Adjusted Axial Distance
Δ	-	Difference
ϵ	-	Roughness Factor
γ	-	Specific Heat Ratio
ξ	-	Collective Variable
μ	-	Coefficient of Viscosity
λ	-	Secondary to Primary Massflow Ratio
π	-	Pi, 3.1416
ρ	-	Density
β	-	Given Radius to Freejet Radius Ratio
β_k	-	Mixing Chamber/Freejet Radius Ratio
U	-	Overall Heat Transfer Coefficient

SUBSCRIPTS

1	-	Bypass/Primary Flow
2	-	Secondary/Header Flow

3	-	Mixed Header Flow
B	-	Back
d	-	Dimensional Variable
E	-	Exit Plane
f	-	Freejet
H	-	Wake Flow
m	-	Axial
mc	-	Mixing Chamber
o	-	Total
ref	-	Reference
*	-	Critical Properties

I. INTRODUCTION

Statement of Problem

A test cell bypass modification that was made to a jet engine test facility to correct an operational limitation may have introduced a problem for the facility's exhaust system. During bypass operation, subzero temperature air ejecting into the exhaust header may not warm up sufficiently before entering the exhaust compressors. Excessively cold, unmixed air injected by an exhauster may damage it and/or cause a shut down and interrupt any engine test in progress. To evaluate the flow field in the exhaust header, a simplified analysis of the mixing of the flow was performed and a flow field simulation computer program approach, the PARC code, is presented for engineers to use to obtain a more complete analysis of the flow field.

Background

The Aeropropulsion Systems Test Facility (ASTF) is the newest and largest jet engine test facility at Arnold Engineering Development Center (AEDC), Arnold Air Force Base, Tennessee (Figures 1 and 2). ASTF was designed (1972 - 1977) and built (1977 - 1984) to simulate the necessary flight conditions for testing large, up to 75,000 lb thrust class, turbine engines to determine their performance and

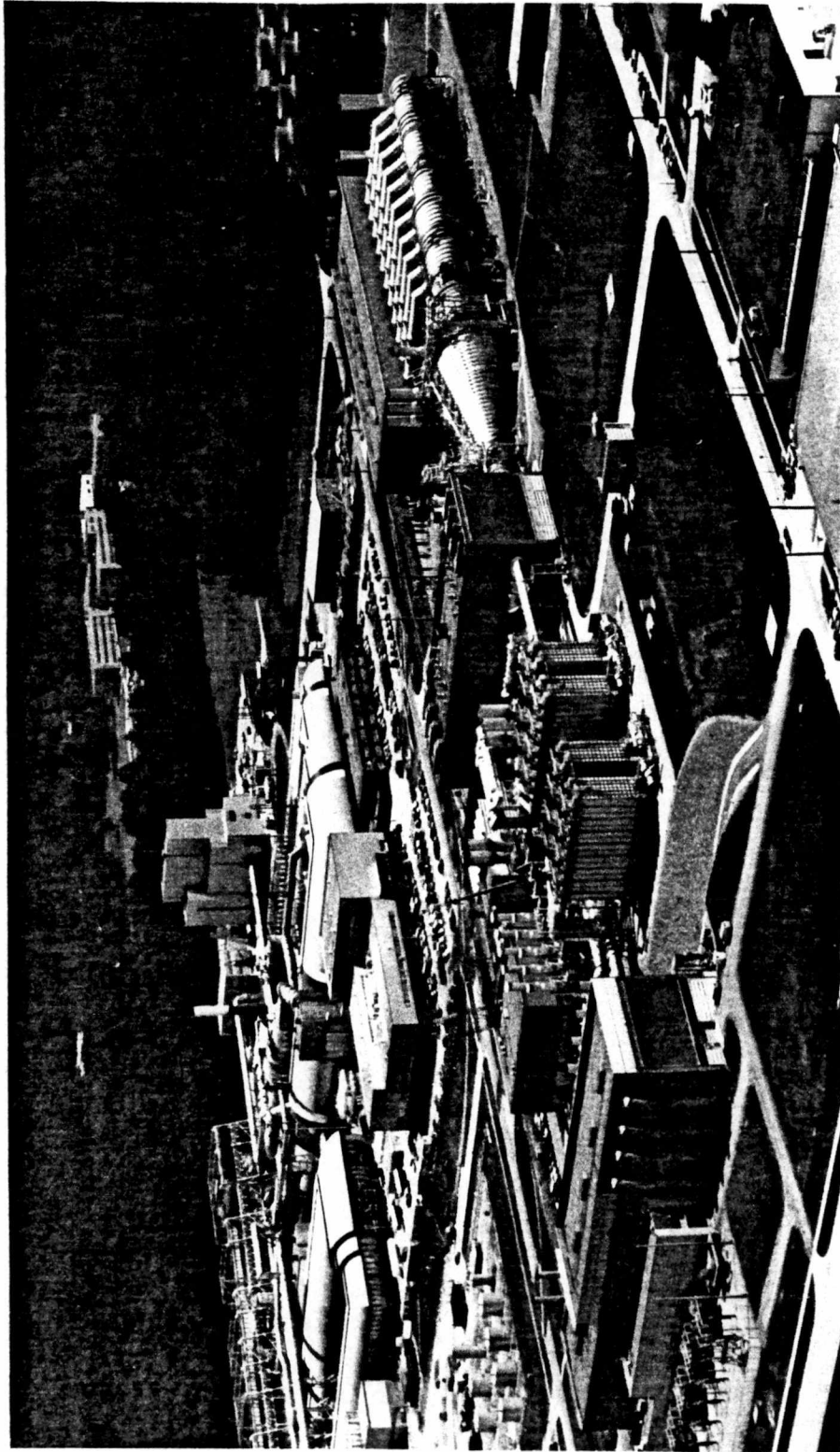


Figure 1. Aerial of ASTF (Before Test Cell Bypass Modification)

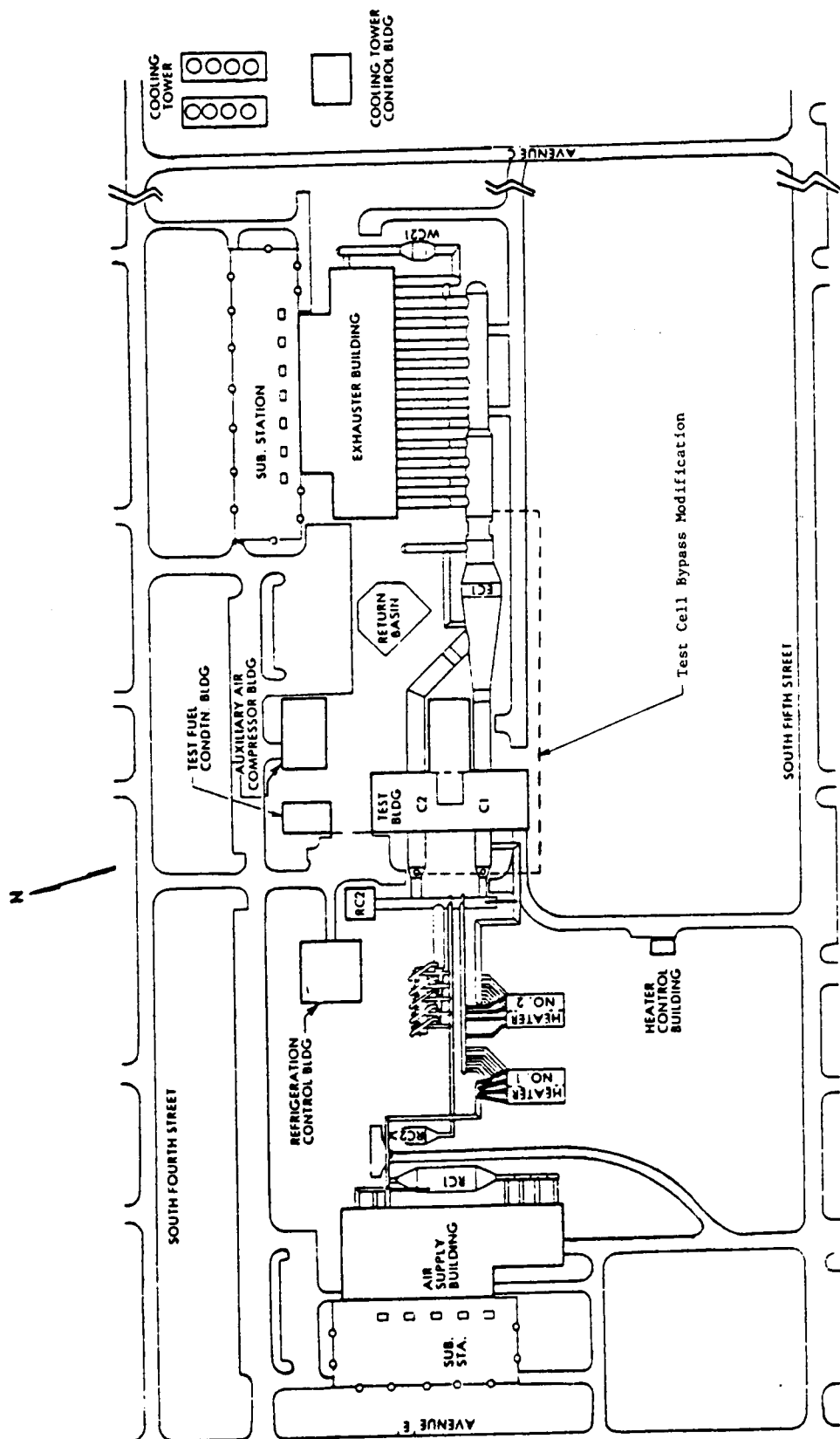


Figure 2. Facility Site Plan View

operational characteristics throughout the engine's flight envelope.

To reduce the cost of transportation on a ton/mile basis, aircraft have historically grown larger and, consequently, so have their engines. In the 1960s and '70s, subsonic high-bypass turbine engines had been increasing in thrust levels (i.e. C5 Galaxy's TF39 engine at 41,000 lb) and it was projected that continued increases in engine size and thrust would necessitate an ASTF type facility to provide additional massflow capability to test such engines. The largest test cell (J2) in existence at the Engine Test Facility (ETF) at AEDC during the early 70's has an engine thrust capacity up to 60,000 lb, but was restricted in massflow capability to about 600 lbm/sec. To increase supplied massflow and to improve overall testing capability led to the creation and construction of ASTF.

Interestingly, since the \$625 million facility came on line in October 1985, the requirement for testing large engines has yet to materialize. Thus, ASTF currently tests F100 thrust-class (20,000 to 30,000 lb thrust) engines with plant equipment originally sized for large massflow engines. The large size of the air supply compressors and exhausters makes it operationally inefficient in matching airflow to the requirements of relatively small massflow, F100 class engines. A single compressor provides several times the capacity of massflow needed, except in cases of

freejet configurations [1].¹ However, the high data acquisition rates, three-dimensional thrust stands, and increased pressure/temperature transient capabilities are some of the reasons ASTF is used to test the smaller engines.

Currently, the trend for F100 class engine test programs has been for extensive testing in the upper left hand corner (ULHC) of their flight envelope, that is, for high altitude, low Mach number simulation. Looking at Figure 3, the ULHC flight condition is at high altitude and low Mach number. This flight regime requires a facility to provide very low temperatures and pressures to the engine to simulate the ULHC environment. Engine manufacturers and test facility users request these cold conditions to provide a severe test of an engine's blowout and re-light performance at altitude.

In ASTF, the refrigeration units that supply the low temperatures are the refrigeration turbines or turbine chillers (Figure 4)[1]. During operation, a turbine absorbs power from the duct airstream by use of an airbrake, thus expanding the air and cooling its temperature from -6°F to -108°F at a rated flow of 288 lbm/s. Again, because ASTF was originally designed for large engines, the operational massflow range for one of these turbines is 100 to 300 lbm/s.

¹Numbers in brackets correspond to similarly numbered references in the Bibliography.

F100 OPERATING ENVELOPE

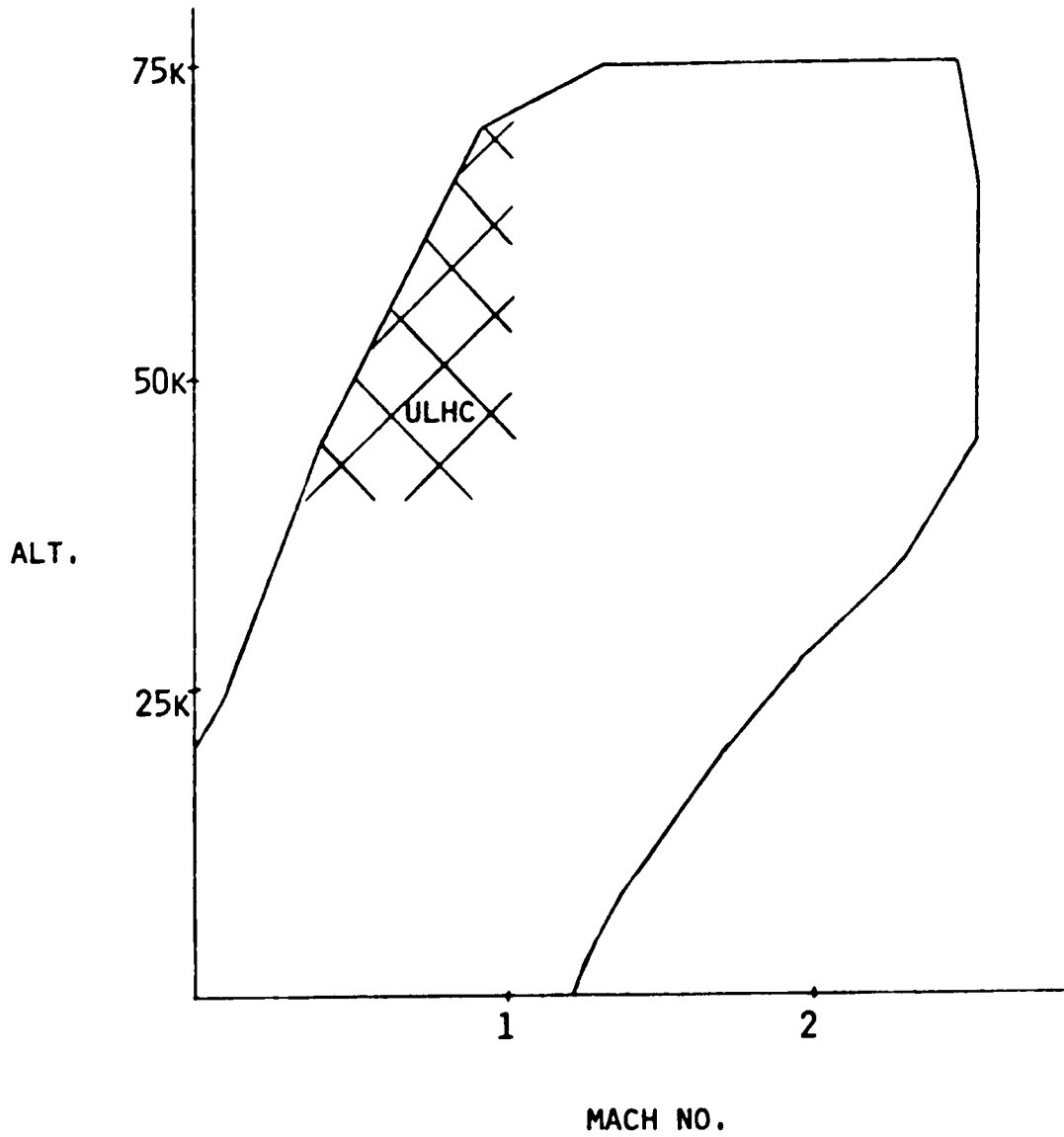


Figure 3. Upper Left Hand Corner (ULHC) Flight Condition

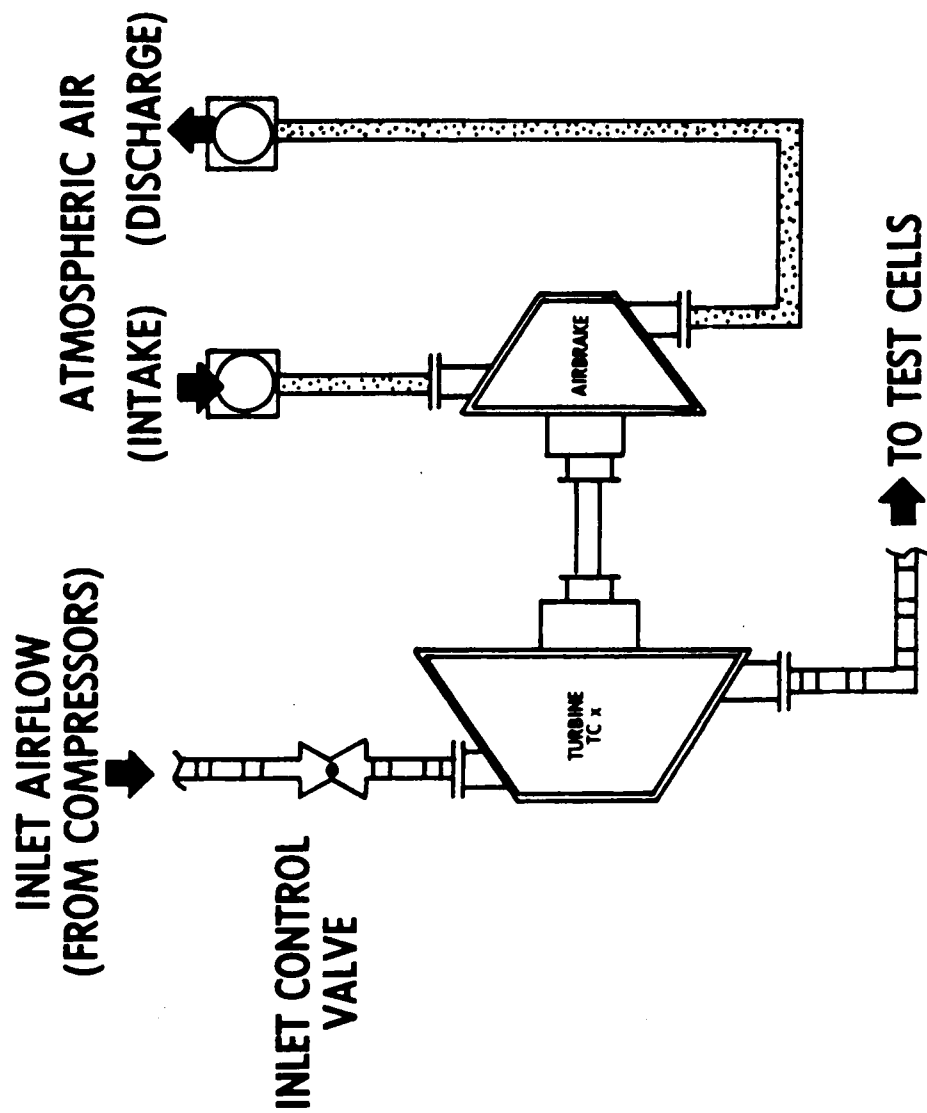


Figure 4. Typical Refrigeration Turbine Arrangement

This is a mismatch for the facility, since the F100 engines operate in the neighborhood of 50 lbm/s during ULHC testing. Achieving the low pressure requirement to simulate ULHC is no problem for ASTF but supplying the low temperatures (below 0°F) to the engine is a problem at low massflows, because of the thermal capacity of the facility for heat transfer (as a thermal source), and because of the cooling turbines.

Specifically, the two problem areas are the low turbine efficiency for massflows under 100 lbm/s and the relatively high duct heat transfer to the cold air in transit to the engine. The large mass of the steel ducting acts like a heat source to the air flow, warming it up significantly by the time it gets from the turbine chiller to the test cell. Figure 5 illustrates the flow path from turbine chiller number one (TC1) to test cell C1. The flow exits TC1, goes down the cold leg (Leg 1), enters the mixing header and makes a 90° turn, travels down the mixing header and makes another 90° turn into the C1 inlet plenum and enters the test cell. While this cold flow from leg one enters the mixing header, there is also warm flow from the hot leg (Leg 3) entering the mixing header. The mixing of these two flows is how the test cell inlet temperature is controlled and both flows can be throttled to bring the temperature up or down.

The mixing header has been identified as a major heat source that causes uncontrolled warm up of the cold process

air [2] because the header represents 300,000 lb of steel ducting. For example, in 1987, a test in C1 of a jet engine in the 30,000 lb thrust class at ULHC conditions revealed that the minimum maintainable engine inlet temperature was about -15°F at 60 lbm/s massflow [3].

Figure 6 is a heat transfer model plot [4] that illustrates the combined effect of the turbine chiller inefficiency and duct heat transfer during low massflow operations. For 50 lbm/s and under, it is evident that it takes several hours for the air flow in the test cell being supplied to an engine to reach the desired temperature condition. These hours represent unproductive test time and thus make for expensive testing, because at typical ULHC conditions ASTF consumes approximately 100 megawatts per test hour. This staggering electrical cost and the cost of the salary of approximately 50 operating personnel called for a plant modification that would increase the massflow through the plant in order to decrease the time necessary to achieve cold temperatures in the test cell (30 minutes vs. 4 hours plus). Increasing the massflow allows the turbine chillers to operate at design efficiencies and reduces the heat transfer problem between the process air flow and the duct wall because the heat transfer rate per unit massflow will have been significantly reduced.

The installation of a test cell bypass line or duct was selected as the best engineering solution [2]

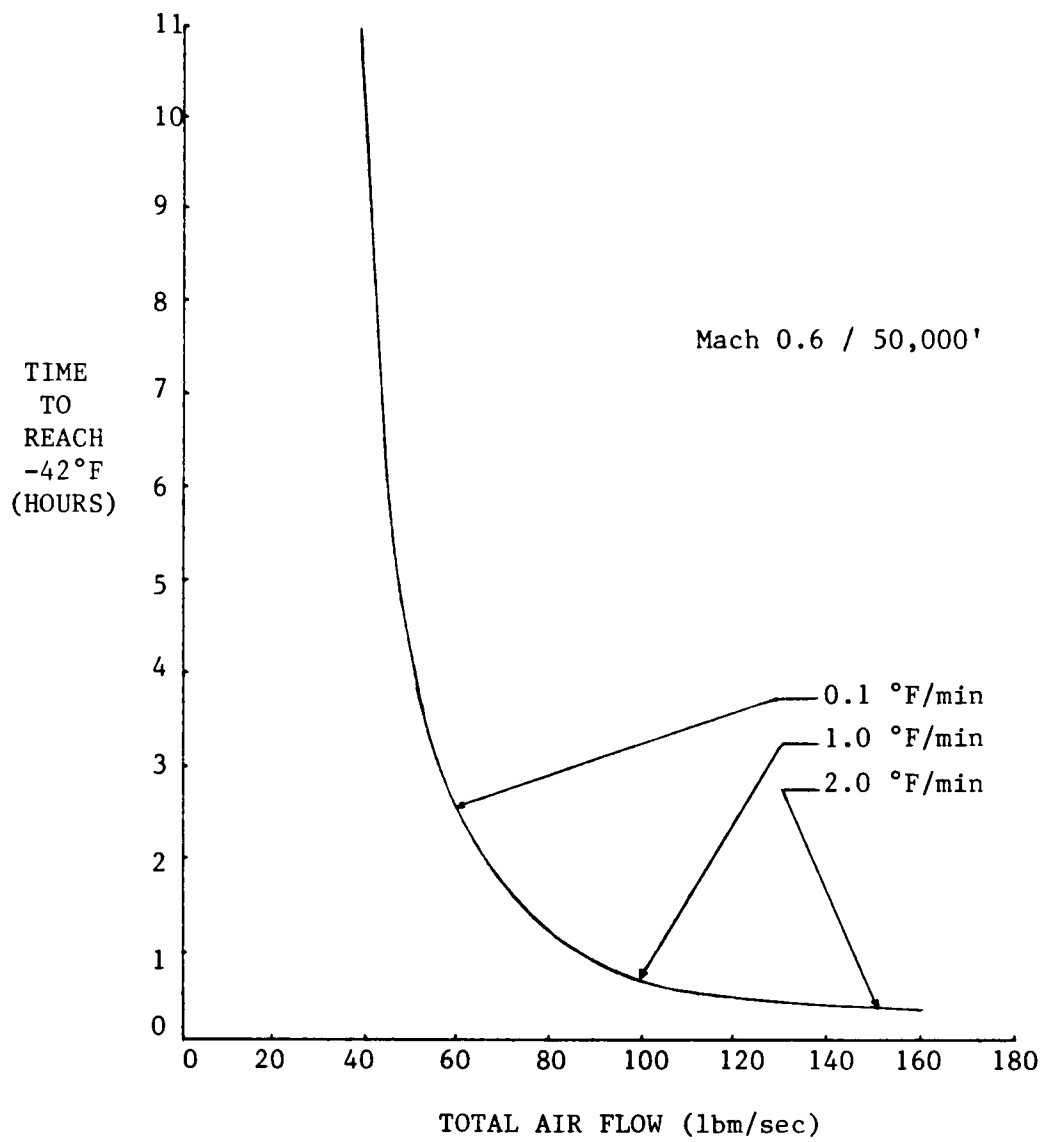


Figure 6. Heat Transfer Model Results

since it would permit operation of the plant at nearly optimum conditions while providing low test article/cell/engine massflows during ULHC testing. Figures 7 through 10 show the various configurations of the bypass that were considered during preliminary design (included here for historical reference). Figure 10 was the configuration chosen as the best engineering approach due to test schedule impacts, construction supportability, operational tradeoffs, and other factors.

During typical ULHC operation, air flows through the test cell at about 50 lbm/s and through the bypass at 1 1/2 to 2 times that of the test cell for a total mass flow through the plant of about 150 lbm/s. From Figure 6 this shows that the cooldown time of the test cell should be less than an hour to achieve the design condition of -42°F . This compares favorably with recent checkouts of the facility using the bypass. In March 1988, with a test cell flow of less than 50 lbm/s and a bypass flow of 100 to 120 lbm/s, a test cell inlet temperature (T2, Figure 11) of -44°F was achieved in about 50 minutes [5].

So, preliminary results from the several facility checkouts indicate that the bypass has accomplished its objective of reaching below -40°F temperatures in less than an hour for test cell massflows of about 50 lbm/s. However, the bypass may have introduced an operational problem for the facility, relative to the exhaust plant. In Figure 12,

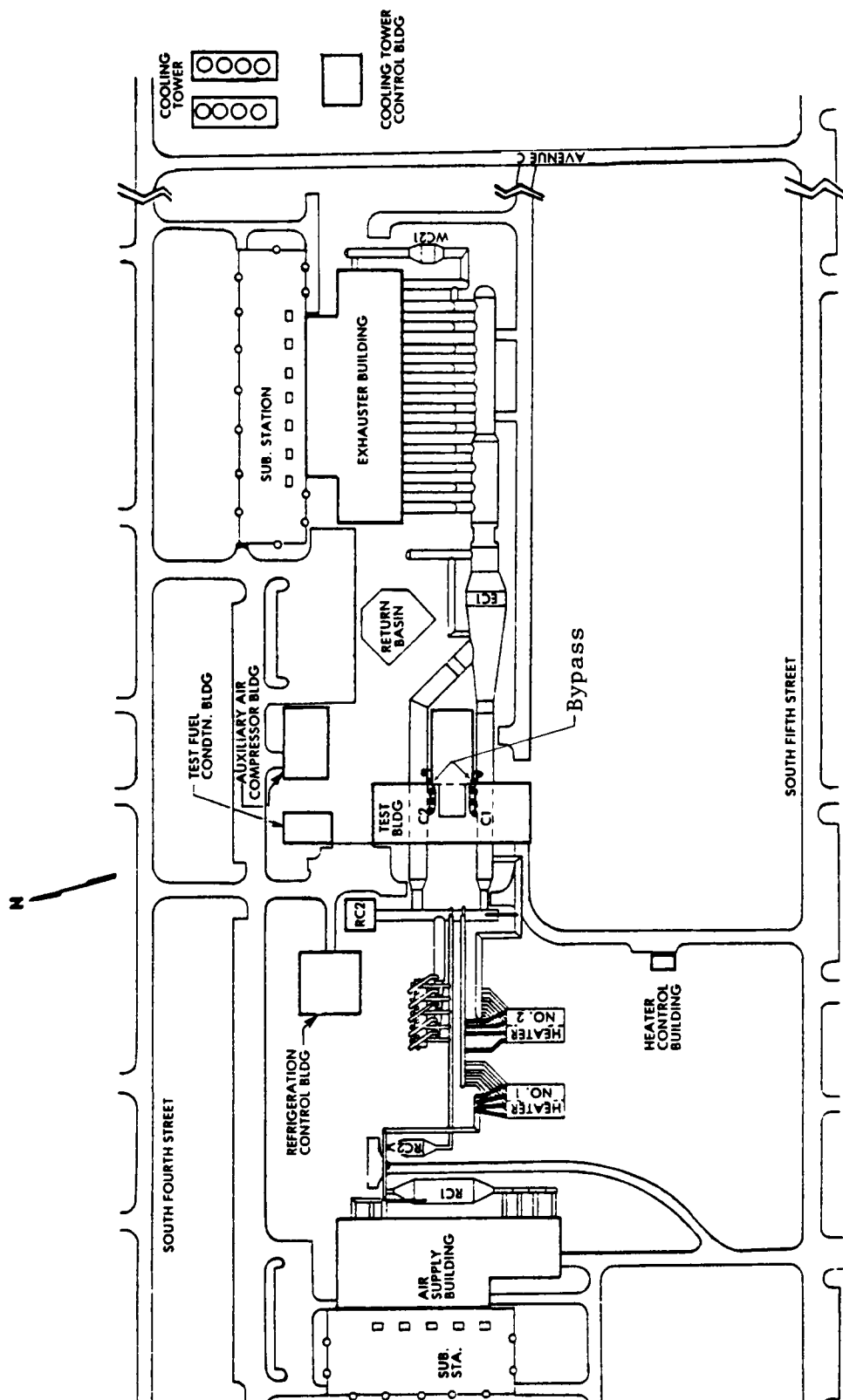


Figure 7. Test Cell Bypass Configuration #1

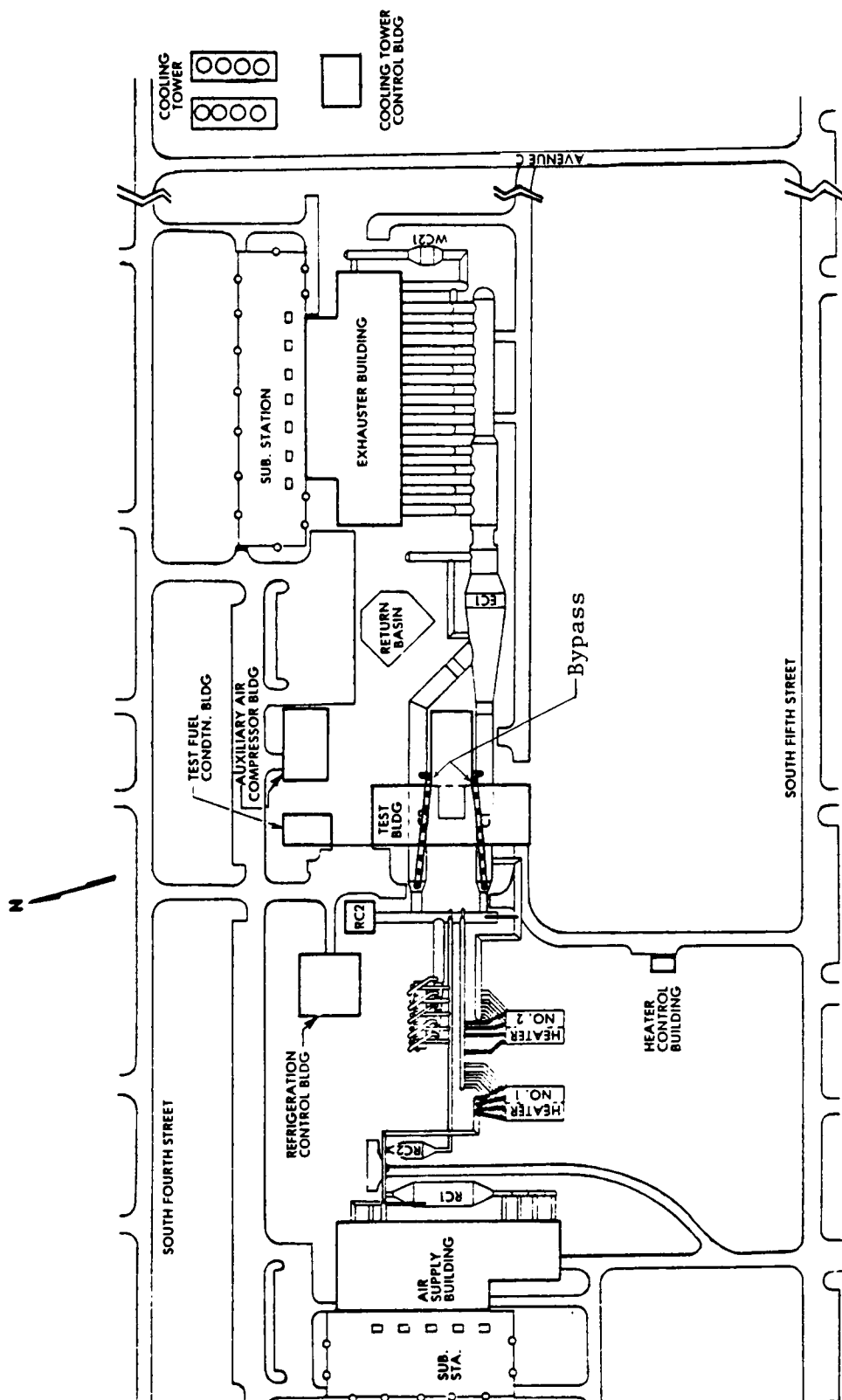


Figure 8. Test Cell Bypass Configuration #2

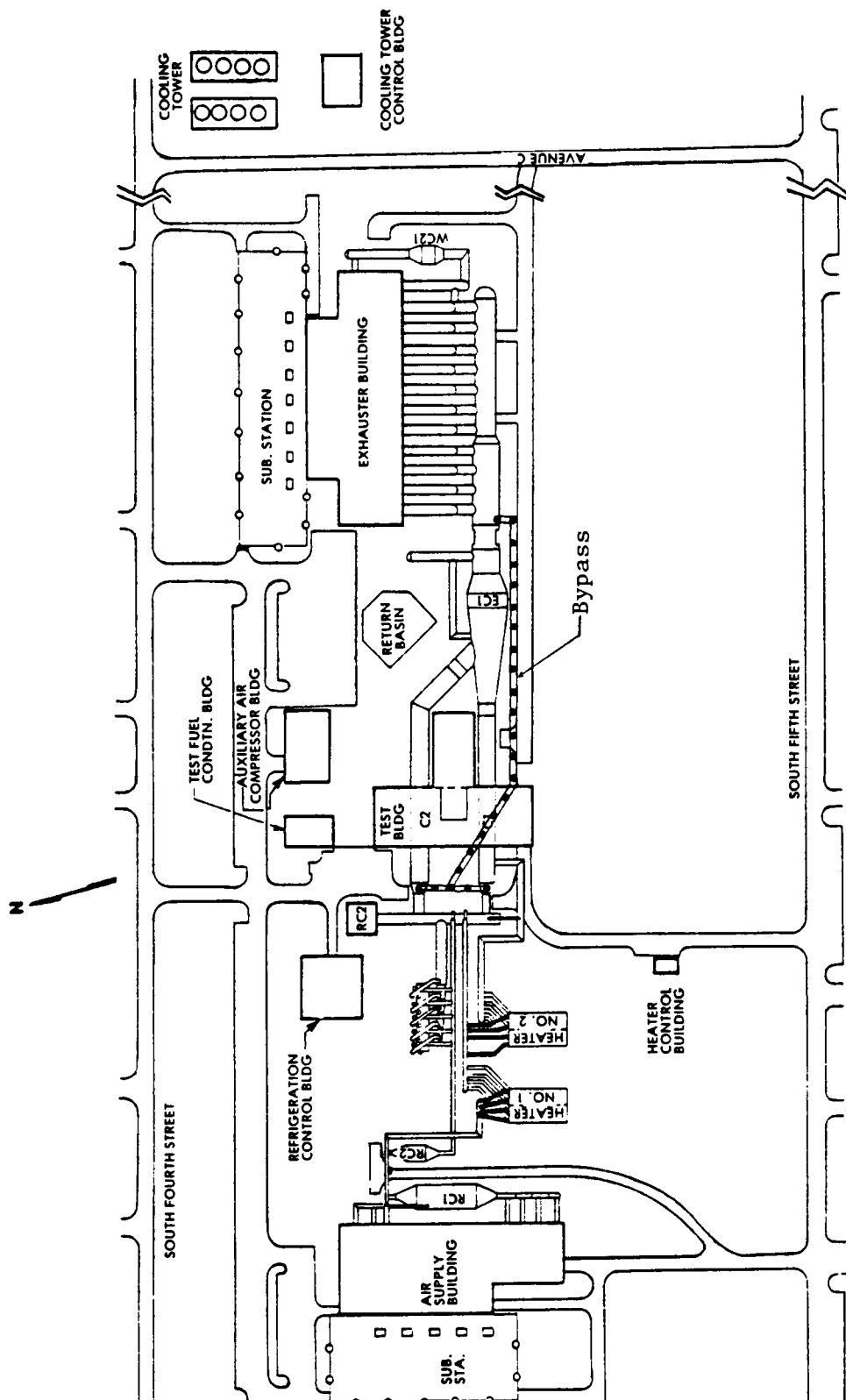


Figure 9. Test Cell Bypass Configuration #3

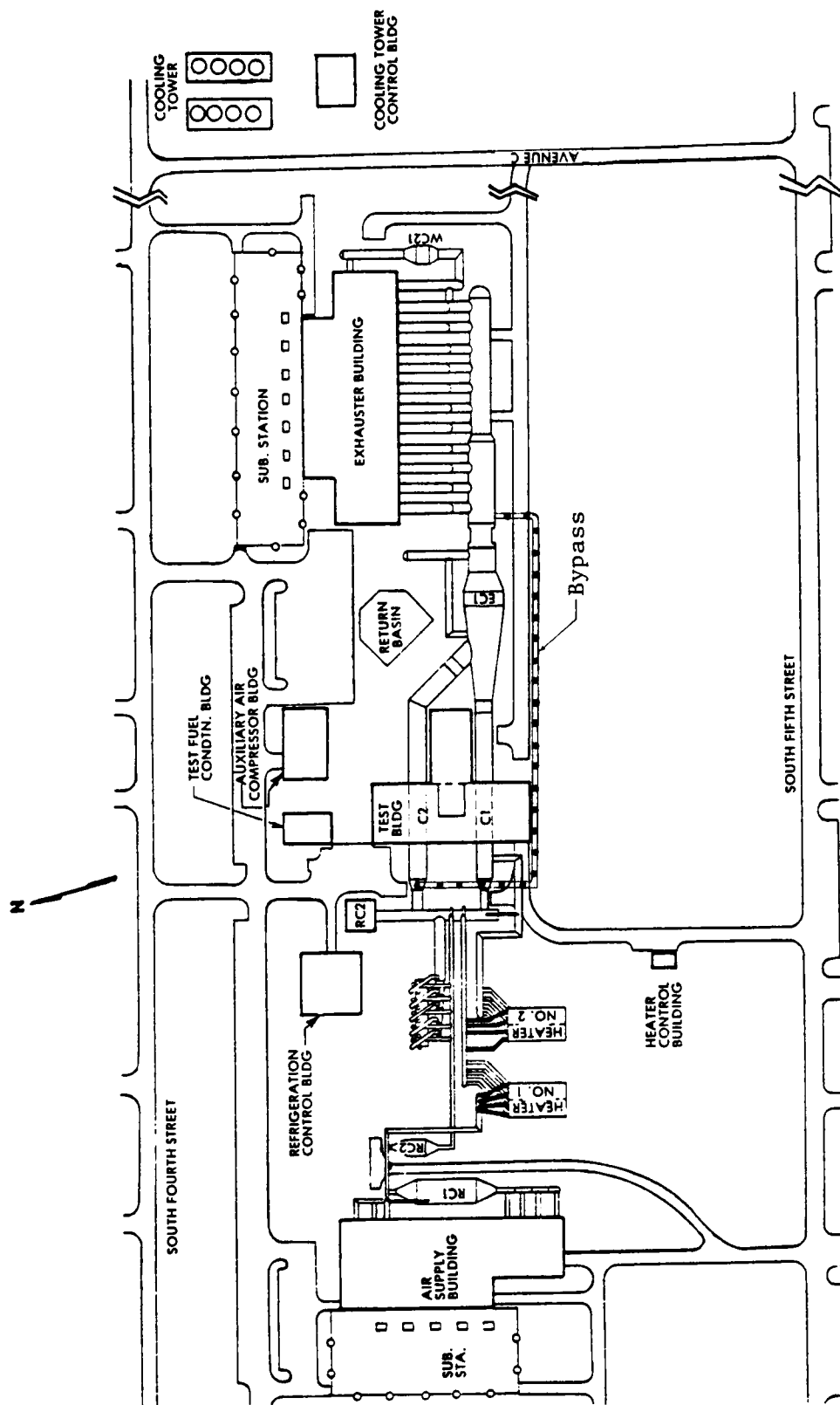


Figure 10. Test Cell Bypass Configuration #4

C1 Cooldown - Low Flow

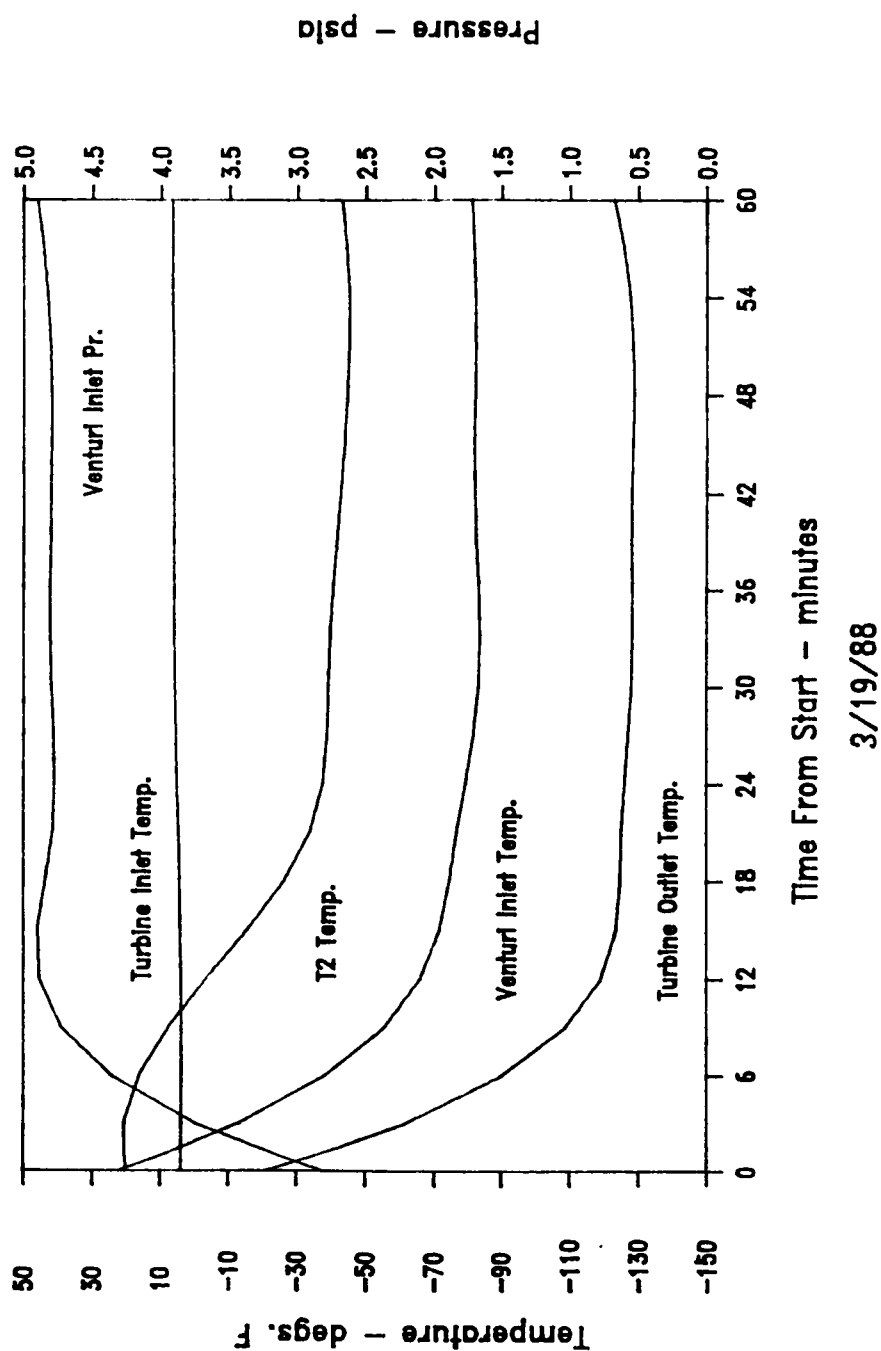


Figure 11. Results of C1 Cooldown with Bypass

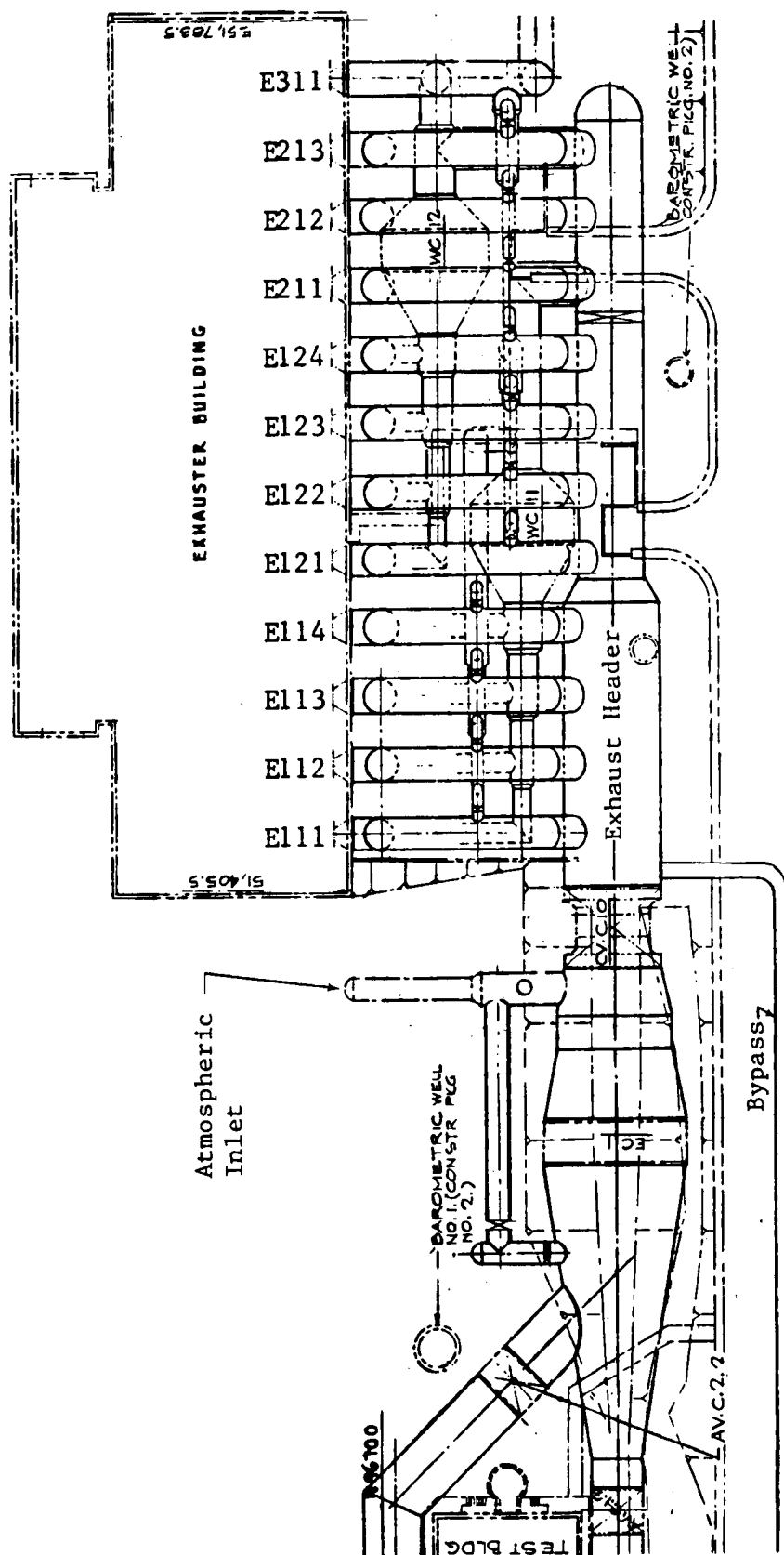


Figure 12. Bypass Tie-in to the Exhaust Header

the exhaust plant is located down stream of (or to the right of) valve CV.C.10. This valve controls the pressure in either test cell by throttling the exhaust gas flow to the exhaust compressors. The 12 exhaust gas compressors or exhausters, numbered E111-114, E121-124, E211-213, and E311, pumpdown the test cell for altitude simulation and also pump out the engine exhaust gases from the test cell. These exhausters have a requirement that their inlet air temperature must be 35⁰F or higher, because at lower temperatures the compressor inlet Foreign Object Damage (FOD) screen may ice over, causing an excessive pressure differential and collapse.

Looking at Figure 12, one can see where the bypass enters the exhaust header just downstream of valve CV.C.10. During ULHC operations, air will be entering the exhaust header from two sources: from the 5 1/2' diameter bypass and from the test cell via the 45' diameter CV.C.10 valve. The concern is how well the cold flow out of the bypass at -80⁰F will mix with the warm flow from the test cell at about 100⁰F before it gets to the exhauster(s). There is some capability to warm the flow mixture in the exhaust header by opening the upstream atmospheric inlet (Figure 12); however, this is not a dependable method since at times a test program will want ULHC testing done in the winter when the outside temperature is below freezing. It does not

help to inbleed freezing air to warm the exhaust header mixture.

The focus of this thesis is the evaluation of the mixing of the cold (primary stream) and warm (secondary) streams in the ASTF exhaust header ducting. Three methods are discussed for the evaluation and two were applied. The first is a simple, one-dimensional flow analysis of the fully-mixed conditions based on specification of the header operating pressure and the inlet flow conditions to the exhaust header from the test cell and the bypass. The second is an estimation of the fluid mechanics of the mixing using a simplified jet-into-parallel-flow model. The third method covered, but not used, is a discussion of how the real process would be modeled using a full, elliptic, Navier-Stokes program called the PARC code. Significant details of the code are presented so that design engineers could complete the study.

The PARC code, a state-of-the-art software, is presented along with a suggested method for setting up the computer program input files to solve for the flow field properties. With this, it is envisioned that the operating engineer at ASTF could use these input files along with the facility Cray computer to provide pre-test analysis of the mixing characteristics of the exhaust header during bypass operations. Engineers could then optimize exhauster

selection to minimize cold air ingestion by the compressors,
or could determine a solution to warm up the cold flow to
the exhausters using other means such as inbleed air flows.

II. FLOW DEFINITION

This section describes the physical dimensions of the ASTF exhaust header duct and the approximate characteristics of the flow inside. The assumptions and equations of flow are presented along with the rough order of magnitude analysis of the flow.

Physical Configuration

Figures 13 and 14 illustrate the duct layout with Figure 13 being the most convenient reference because it has the important dimensions listed. The exhaust header is basically comprised of three sections: a 45' diameter section about 123' long, a 116' long 32' diameter section upstream of the AV.E.4 valve, and a 109' long 32' diameter section downstream of the valve. The first four low stage exhausters have 13' diameter inlet ducts (shown as arcs in Figure 13) feeding off of the 45' diameter section, the second four low stage exhausters feed off of the 116' long section, and the high stage exhausters feed off of the 109' long section with valve AV.E.4 being used to configure the low and high stage exhausters in series operation. The 5 1/2' diameter bypass enters the header from the side and dumps cold flow approximately 100' upstream of the 12' long converging section. The bypass nozzle is assumed to be exactly axisymmetric with the header centerline.

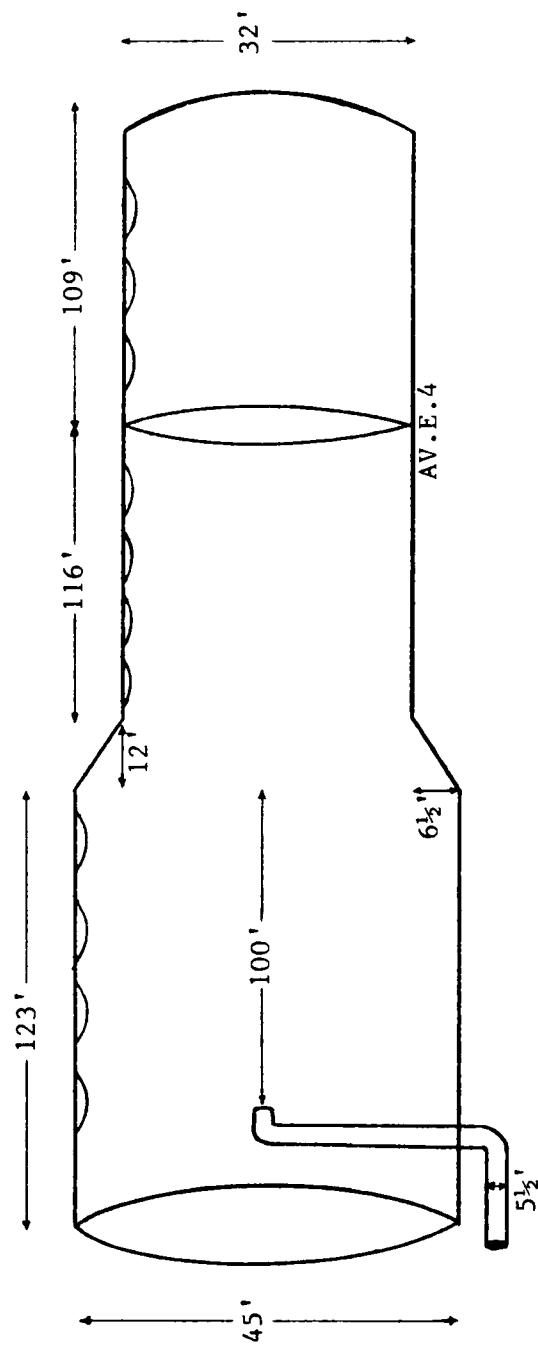


Figure 13. Physical Dimensions of the Duct (Not to Scale)

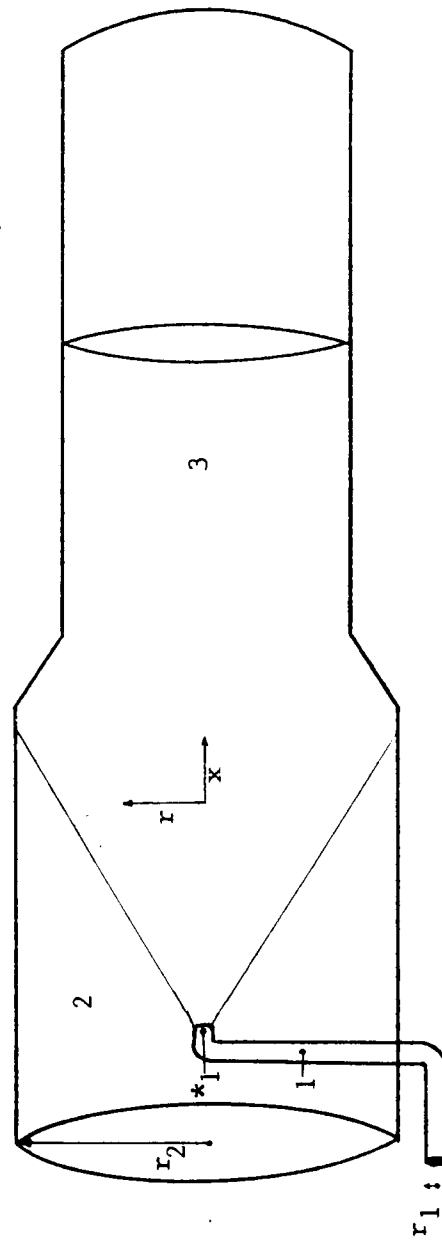


Figure 14. Radius and Flow Station Nomenclature

Figure 14 illustrates the radius and station nomenclature that will be discussed. Number subscripts refer to the station associated with the particular flow parameter. The following section will present the approximate flow conditions at each station.

Flow Conditions

Reference Figure 14 for flow station locations.

Assumptions. In order to take advantage of the numerous graphs and tables available in most compressible flow texts, the following assumptions are made:

1. Adiabatic, no wall friction, one-dimensional compressible flow
2. Steady state, ideal gas
3. Constant specific heat: $\gamma = 1.4$, and gas constant: R

These assumptions are used in the calculation of approximate flow field conditions for the flow stations in Figure 14. This idealized analysis is intended as a ball park baseline reference to gauge PARC application results.

Station 1. The massflow of the air inside the bypass is given as approximately 100 lbm/s when the exhaust header pressure is 1.5 lbf/in² [11] and the inlet conditions to the bypass are:

$$P_o = 5.5 \text{ lbf/in}^2$$

$$T_o = -80^\circ\text{F} = 380^\circ\text{R}$$

Station *1. The exit conditions of the bypass, as obtained by Fanno Flow analysis (Appendix A)[12] and assuming bypass exit static pressure equal header static pressure, are characterized by:

$$P_{*1} = 1.5 \text{ lbf/in}^2$$

$$T_{*1} = 381 \text{ }^{\circ}\text{R}$$

$$U_{*1} = 395 \text{ ft/s}$$

$$M_{*1} = 0.41$$

$$P_{o_{*1}} = 1.7 \text{ lbf/in}^2$$

$$T_{o_{*1}} = 394 \text{ }^{\circ}\text{R}$$

Other:

$$A_{*1} = \pi (2.75 \text{ ft})^2$$

$$= 23.76 \text{ ft}^2$$

$$\dot{m}_{*1} = 100 \text{ lbm/s}$$

$$\rho_{*1} = \frac{P_{*1}}{RT_{*1}}$$

$$= \frac{(1.5 \text{ lbf/in}^2)(144 \text{ in}^2/\text{ft}^2)}{(53.3 \text{ ft lbf/lbm}^{\circ}\text{R})(381^{\circ}\text{R})}$$

$$= 0.01064 \text{ lbm/ft}^3$$

Station 2. Pressure, temperature, and massflow values for this section, the secondary flow, were provided by AEDC engineers [11]:

$$P_2 = 1.5 \text{ lbf/in}^2$$

$$T_2 = 100 \text{ }^\circ\text{F} = 560 \text{ }^\circ\text{R}$$

$$\dot{m}_2 = 100 \text{ lbm/s}$$

$$A_2 = \pi \left[(22.5 \text{ ft})^2 - (2.75 \text{ ft})^2 \right]$$

$$= 1566.68 \text{ ft}^2$$

$$\rho_2 = \frac{(144 \text{ in}^2/\text{ft}^2)(1.5 \text{ lbf/in}^2)}{(53.3 \text{ ft lbf/lbm}^\circ\text{R})(560^\circ\text{R})}$$

$$= 0.00724 \text{ lbm/ft}^3$$

$$U_2 = \frac{100 \text{ lbm/s}}{(0.00724 \text{ lbm/ft}^3)(1566.68 \text{ ft}^2)}$$

$$= 8.82 \text{ ft/s}$$

$$a_2 = \sqrt{\gamma R T_2 (32.2 \text{ ft lbm/s}^2 \text{ lbf})}$$

$$= 1160.26 \text{ ft/s}$$

$$M_2 = \frac{8.82 \text{ ft/s}}{1160.26 \text{ ft/s}}$$

$$= 0.0076$$

$$P_{o_2} = (1.5 \text{ lbf/in}^2) \left(1 + 0.2 (0.0076)^2 \right)^{3.5}$$

$$= 1.50006 \text{ lbf/in}^2$$

$$T_{o_2} = (560 \text{ } ^\circ\text{R}) \left(1 + 0.2 (0.0076)^2 \right)$$

$$= 560.0065 \text{ } ^\circ\text{R}$$

Station 3. This station is located where the bypass/primary flow has mixed with the header/secondary flow such that the jet radius extends to the wall of the header. The 1-D continuity equation is:

$$\dot{m}_3 = \dot{m}_{*1} + \dot{m}_2$$

$$= 100 \text{ lbm/s} + 100 \text{ lbm/s}$$

$$= 200 \text{ lbm/s}$$

For this two stream mixing problem, the following equations [10] (based on the conservation equations for nearly constant pressure mixing, see Appendix B for derivation) are used to determine Station 3 flow parameters:

$$U_3 = \frac{-\xi_4 \xi_3}{2(\xi_1 - \xi_4 \xi_2)} + \sqrt{\frac{\xi_4^2 \xi_3^2}{4(\xi_1 - \xi_4 \xi_2)^2} + \frac{\xi_1 \xi_5}{\xi_1 - \xi_4 \xi_2}}$$

$$P_3 = \xi_3 - \xi_2 U_3$$

$$T_s = \frac{P U}{\zeta_1}$$

$$\lambda = \frac{m_2}{m_{*1}} = 1$$

where

$$\zeta_1 = \frac{m_{*1} (1+\lambda) R}{A_s}$$

$$= \frac{(100)(1+1)(53.3)}{\pi(22.5)^2} \frac{\text{lbf}}{\text{ft s}^\circ R}$$

$$= 6.7026 \frac{\text{lbf}}{\text{ft s}^\circ R}$$

$$\zeta_2 = \frac{m_{*1} (1+\lambda)}{g_c A_s}$$

$$= \frac{(100)(1+1)}{(32.2)\pi(22.5)^2} \frac{\text{lbf s}}{\text{ft}^3}$$

$$= 0.003905 \frac{\text{lb f s}}{\text{ft}^3}$$

$$\xi_3 = P_{*1} + \frac{\dot{m}_{*1} (U_{*1} + \lambda U_2)}{g_c A_3}$$

$$= (1.5)(144) + \frac{(100)(395 + (1)(8.82))}{(32.2) \pi (22.5)^2} \frac{\text{lb f}}{\text{ft}^2}$$

$$= 216.771 \frac{\text{lb f}}{\text{ft}^2}$$

$$\xi_4 = 2C_{p3} J g_c$$

$$= 2 \left[\left(\frac{\dot{m}_{*1}}{\dot{m}_{*1} + \dot{m}_2} \right) C_{p*1} + \left(\frac{\dot{m}_2}{\dot{m}_{*1} + \dot{m}_2} \right) C_{p2} \right] J g_c$$

$$= 2 \left[\left(\frac{100}{200} \right) (0.35) + \left(\frac{100}{200} \right) (0.52) \right] (778.2)(32.2) \frac{\text{ft}^2}{\text{s}^2 \text{ } ^\circ \text{R}}$$

$$= 21800.5 \frac{\text{ft}^2}{\text{s}^2 \text{ } ^\circ \text{R}}$$

$$\xi_5 = \frac{2C_{p*1} J g_c}{1+\lambda} \left[T_{*1} + \lambda \frac{C_{p2}}{C_{p*1}} T_2 \right] + \frac{U_{*1}^2 + \lambda U_2^2}{1+\lambda}$$

$$\begin{aligned}
&= \frac{2(0.35)(778.2)(32.2)}{1+1} \left[381 + (1) \left(\frac{0.52}{0.35} \right) (560) \right] \frac{\text{ft}^2}{\text{s}^2} \\
&+ \frac{\left[(395)^2 + (1)(8.82)^2 \right]}{1+1} \frac{\text{ft}^2}{\text{s}^2} \\
&= 1.0716 \times 10^7 \frac{\text{ft}^2}{\text{s}^2}
\end{aligned}$$

Computing coefficients:

$$\xi_4 \xi_8 = (21800.5)(216.771) \frac{\text{lbf}}{\text{s}^2 \text{ } ^\circ \text{R}}$$

$$= 4.7257 \times 10^6 \frac{\text{lbf}}{\text{s}^2 \text{ } ^\circ \text{R}}$$

$$\xi_4 \xi_2 = (21800.5)(0.003905) \frac{\text{lbf}}{\text{ft s } ^\circ \text{R}}$$

$$= 85.131 \frac{\text{lbf}}{\text{ft s } ^\circ \text{R}}$$

$$\xi_8 \xi_1 = (1.0716 \times 10^7)(6.7026) \frac{\text{ft lbf}}{\text{s}^3 \text{ } ^\circ \text{R}}$$

$$= 7.1828 \times 10^7 \frac{\text{ft lbf}}{\text{s}^3 \text{ } ^\circ\text{R}}$$

Substitute and compute Station 3 velocity:

$$U_3 = \frac{-4.7257 \times 10^6}{2(6.7026 - 85.131)} \pm$$

$$\sqrt{\frac{(4.7257 \times 10^6)^2}{4(6.7026 - 85.131)^2} + \frac{7.1828 \times 10^7}{6.7026 - 85.131}} \quad \frac{\text{ft}}{\text{s}}$$

$$= 30.127 \times 10^3 \pm \sqrt{9.0767 \times 10^9 - 915.84 \times 10^3} \quad \text{ft/s}$$

$$= 30.127 \times 10^3 \pm 30.112 \times 10^3 \text{ ft/s}$$

The highest solution would be nonphysical; thus,

$$U_3 = 30.127 \times 10^3 - 30.112 \times 10^3 \text{ ft/s}$$

$$= 15.203 \text{ ft/s}$$

Compute pressure and temperature:

$$P_3 = 216.771 - (0.003905)(15.203) \text{ lbf/ft}^2$$

$$= 216.712 \text{ lbf/ft}^2$$

$$= 1.505 \text{ lbf/in}^2$$

$$T_s = \frac{(216.771)(15.203)}{6.7026} \text{ } ^\circ\text{R}$$

$$= 491.69 \text{ } ^\circ\text{R}$$

$$\rho_s = \frac{\dot{m}_s}{U_s A_s}$$

$$= \frac{100 \text{ lbm/s}}{(15.203 \text{ ft/s})\pi(22.5 \text{ ft})^2}$$

$$= 0.00414 \text{ lbm/ft}^3$$

$$a_s = 49.03 \sqrt{491.7} \text{ ft/s}$$

$$= 1087.20 \text{ ft/s}$$

$$M_s = \frac{15.203 \text{ ft/s}}{1087.20 \text{ ft/s}}$$

$$= 0.0140$$

$$P_{o_s} = (1.5 \text{ lbf/in}^2)(1 + 0.2(0.0140)^2)^{3.5}$$

$$= 1.5002 \text{ lbf/in}^2$$

$$T_{o_s} = (491.7^\circ\text{R})(1 + 0.2(0.0140)^2)$$

$$= 491.711 \text{ } ^\circ\text{R}$$

III. SLOPE OF THE JET SPREAD

For an approximate slope of the constant pressure boundary, Abramovich's [8] jet spread data for mixing chambers is used. One of the graphs in this reference, Figure 13.12, contains nondimensionalized curves of the radius of the jet versus its axial location in the mixing chamber for given values of the ratio of the wake flow velocity to the primary flow velocity ($m=U_H/U_1$). For convenient reference, a simplified version of Abramovich's Figure 13.12 is shown in Figure 15.

For this case:

$$m = \frac{U_H}{U_{*1}} = \frac{\sqrt{2 \frac{P_o - P_g}{\rho_2}}}{U_{*1}}$$

$$= \frac{\sqrt{2 \frac{P_2 + 1/2 \rho_2 U_2^2 - P_g}{\rho_2}}}{U_{*1}}$$

$$m = \frac{\sqrt{2 \frac{1.5 + \frac{1}{2} (0.00724)(8.82)^2 - 1.505}{0.00724}}}{395 \text{ ft/s}} \text{ ft/s}$$

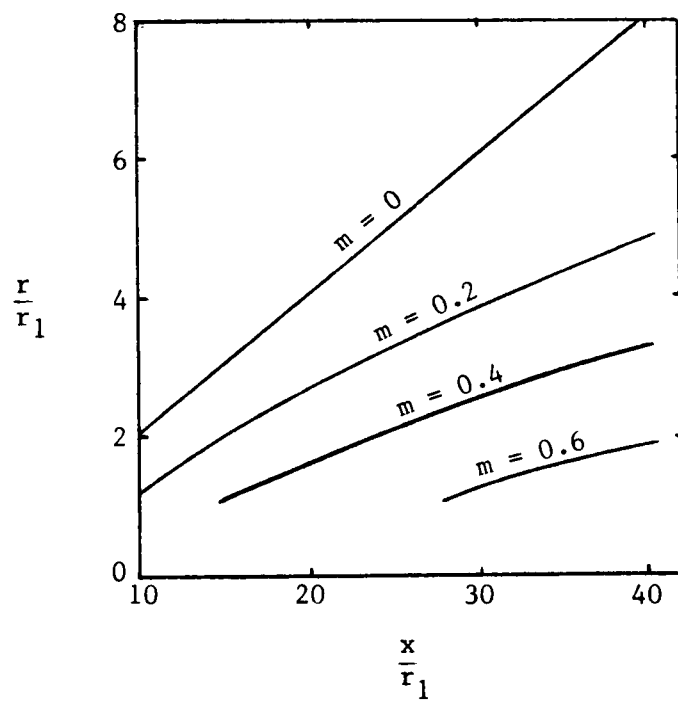


Figure 15. Nondimensionalized Jet Spread Parameters

$$m = \frac{8.74 \text{ ft/s}}{395 \text{ ft/s}}$$

$$= 0.022$$

This value of m and the graph abscissa (x/r_1) value are both needed in order to determine what the ordinate (r/r_1) value from the graph should be for this particular application. With this ordinate value one then determines the jet radius (for a given axial distance) and consequently the slope of the jet.

The abscissa of this graph is the ratio of the axial distance to the nozzle radius (x/r_1) . Figure 13.12 [8] is constructed for an angular jet growth coefficient of $c=0.2$. In order to use the figure for $c \neq 0.2$, the parameter x/r_1 must be adjusted/calculated from the relation [8]:

$$\frac{x'}{r_1} = \frac{x}{r_1} \frac{c}{0.2} \quad (\text{III.a})$$

where x is the axial distance (100 ft for this case), x' is the adjusted axial distance and c is defined by:

$$c = \frac{c_0}{2} \left(1 + \frac{\rho_2}{\rho_{*1}} \right)$$

The empirical value of c_0 is 0.22 for turbulent gas streams [8]. Thus, for this case:

$$\begin{aligned} c &= \frac{0.22}{2} \left[1 + \frac{0.00724 \text{ lbm/ft}^3}{0.01064 \text{ lbm/ft}^3} \right] \\ &= 0.1848 \end{aligned}$$

and

$$\frac{x'}{r_1} = \left(\frac{100 \text{ ft}}{2.75 \text{ ft}} \right) \left(\frac{0.1848}{0.2} \right)$$
$$= 33.61$$

From the graph for $m = 0.022$ and $x/r_1 = 33.61$:

$$\frac{r}{r_1} = 4.5$$

$$r(x=100 \text{ ft}) = 4.5 r_1$$
$$= (4.5)(2.75 \text{ ft})$$
$$= 12.4 \text{ ft}$$

So, at a distance 100 ft from the nozzle, the graph indicates the jet radius is 12.4 ft, or that the jet is 10.1 ft from the header wall. Near the wall, there will be some uncertainty associated with the angle of the jet spread. With reference to the x'/r_1 value, the higher ρ_2 is, the wider the spread of the jet is. However, the m value has a greater influence on how widely the jet spreads. For m , the smaller the secondary velocity (U_2) is, then the wider the jet will spread.

From Abramovich, velocity profiles are calculated for two arbitrary cross sections of the header (@ $x = 115$ and 228 ft) to further aid in visualizing the flow field and for user reference when gauging PARC code results.

Abramovich's basic formula for the velocity field is:

$$\Delta U = \Delta U_m \left(1 - \beta^{1.5}\right)^2$$

or

$$U - U_H = \left(U_m - U_H\right) \left(1 - \beta^{1.5}\right)^2$$

where U is the absolute velocity, U_m the axial velocity, and U_H the wake flow velocity. The dimensionless parameter β is the ratio of a given radius, r , to the freejet radius, r_f , at the given cross section.

The velocity profiles are constructed for U versus r at $x = 115$ ft and 228 ft. Abramovich's auxiliary graphs are used for determining the values of the various parameters that go into the velocity field equation. From equation III.a, for $x = 115$ ft and 228 ft:

$$\frac{x'}{r} = 38.64, 76.61$$

From his Figure 13.12, for these x'/r values and for $m = 0.022$:

$$\frac{r_f}{r_1} = 5.0, 9.9$$

The ratio r_f/r_1 is equal to $r_{mc}/r_1\beta_k$; thus, β_k is obtained. The dimensionless parameter β_k is the ratio of the mixing chamber radius, r_{mc} , to the freejet radius r_f . For this case:

$$\beta_k = \frac{r_{mc}}{r_1} \frac{r_1}{r_f}$$

$$= 1.16, 0.59$$

From Abramovich's Figure 13.8 for these values of β_k :

$$\frac{\Delta U_m}{\Delta U_s} = 5.1, 1.8$$

where

$$\Delta U_s = U_s - U_H$$

$$= 15.203 \text{ ft/s} - 8.740 \text{ ft/s}$$

$$= 6.463 \text{ ft/s}$$

and thus

$$\Delta U_m = 39.96 \text{ ft/s}, 11.63 \text{ ft/s}$$

The other variable, β , in his velocity field equation is defined by:

$$\beta = \frac{r}{r_f}$$

$$= \frac{r}{r_{mc}} \beta_k$$

Figure 16 shows a plot of U versus r/r_{mc} at $x = 115 \text{ ft}$ and 228 ft , where

$$U = \Delta U_m \left[1 - \left(\frac{r}{r_{mc}} \beta_k \right)^{4.5} \right]^2 + U_H$$

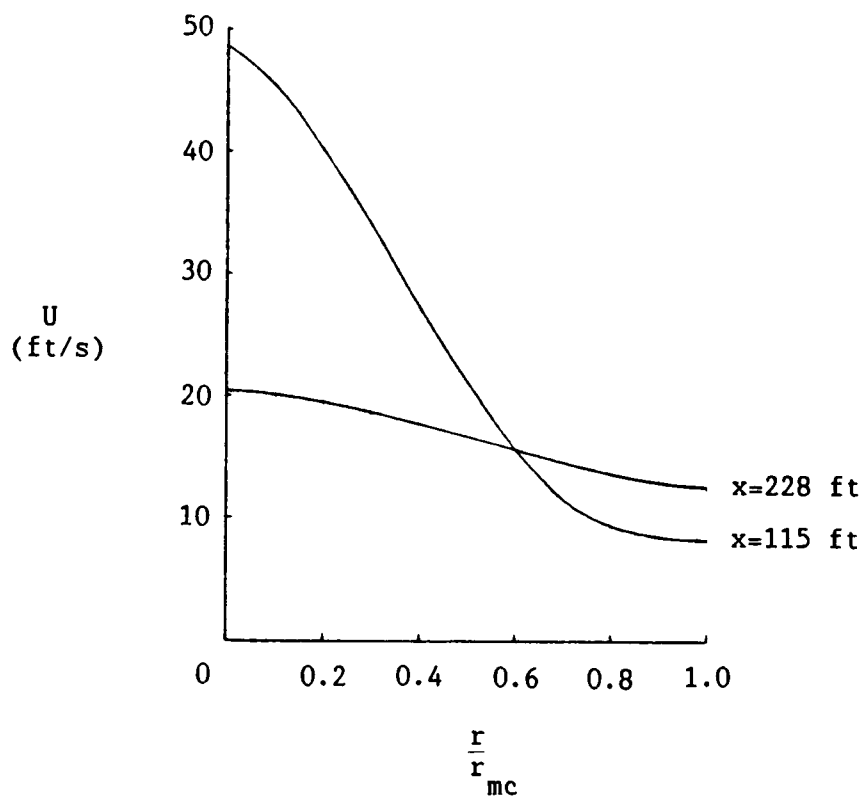


Figure 16. Velocity Profiles at Two Cross Sections of the Exhaust Header

IV. PROPULSION AMES RESEARCH CENTER (PARC) CODE

Requirement for PARC

Flow analysis calculations in the previous chapters give a general idea of the characteristics of the flow field in the header during bypass operation. A more detailed analysis can be achieved with the use of the PARC Code, a general purpose computational tool to simulate the flow field. With PARC, selected portions of the flow field may be examined or changes to the flow field configuration could be made for the evaluation of untried operational conditions.

Introduction to PARC

G.K. Cooper of AEDC/Sverdrup authored an AEDC Technical Report [9] that explains the theory and usage of the PARC code. This code is a flow-field simulation computer program based on the Navier-Stokes equations for which several variants are presented including 2-D, axisymmetric, and 3-D. Also included are specializations in inviscid, laminar, and turbulent flows. This code was developed to satisfy the requirement for a general purpose computational fluid dynamics (CFD) tool for analyzing problems involving complex flows. It has been proven capable of treating many flow problems that could not be adequately dealt with by the use

of other, more specialized CFD codes. An implicit scheme is used that solves a set of nondimensionalized equations produced by central-differencing the Navier-Stokes equations on a computational grid. The difference equations form a set of equations in matrix format. Simultaneous solution of these matrix equations is carried out by Gaussian elimination.

The code requires the user to input a file containing an ordered set of nodal points that represent or model the geometric boundaries of the flow problem, and the distribution of flow-field points between the boundaries. This file contains the initial and boundary condition variables (pressure, temperature, etc.), and the flow field grid. Numerous output options are available to control the amount of printed and plotted data.

Cooper's report provides background and technical details necessary to use the code and is comprised of three major sections: a brief introduction, theory of the equations used in the formulation of the code, and the code usage or program execution section. Major topics in the equation theory section are the Navier-Stokes equations, thermodynamic properties, algorithm technique for solving the equations, boundary conditions, and the turbulence model. The section is good reference for one interested in the composition of the code. The program execution section is a "mini" user's guide and discusses the input and output

options available to assist the user in setting up his particular flow field configuration accurately for analysis by PARC. There are also appendices with 2-D and 3-D examples and error message explanations.

Input/Output Overview

There are two input files that the user must create for execution of the PARC code (Figure 17). The first and larger of the two is known as the restart file or grid and initial condition file. One of the sections of this file is the grid which the user must create. The grid is a set of boundary and flow-field nodal points representing the geometric configuration of the flow problem. The other section of this file is the user's initial conditions for the flow field to be simulated and is used on the very first run of PARC as a starting point. Subsequently, this file section is created by runs of the PARC programs.

The second user-assembled file is the execution and parameter input file (parameter file for short). It too is composed of two sections, one of which contains user account numbers and file location nomenclature. The second section is the NAMELISTS parameter inputs. These NAMELISTS provide the user with numerous options to select in order to specify the problem as closely as possible and to control program execution and desired output format. If desirable, the user can change selected options for "what if" experiments on the

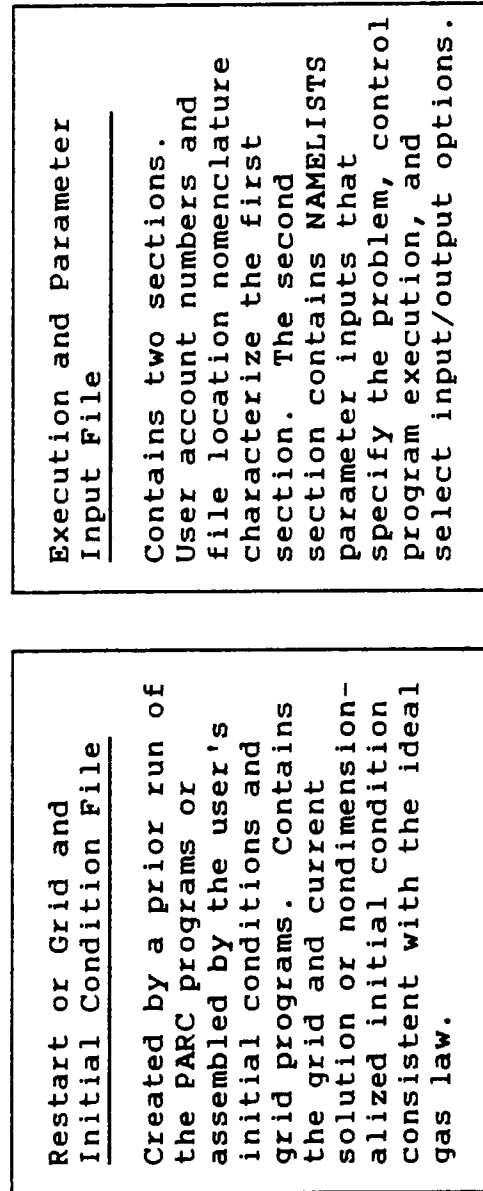


Figure 17. The Two Input Data Files Required to Execute the PARC Codes

flow field. Some of the options are axisymmetric, 2-D, 3-D, inviscid, thin-layer, fully viscous, laminar, turbulent, and various boundary conditions.

The output is specified in the NAMELISTS section from the various options available. The print frequency of the convergence history can be selected along with options for printing and/or plotting the flux balances (conservation of mass, momentum, and energy). An option is available for reducing the amount of printed output by providing information on regions of interest within the flow field. With this option, varying degrees of fine or coarse detail can be selected. The user should keep in mind the tradeoff between the need for reams of detailed output and the cost of computer time. Finally, a record of the run parameters used is automatically a part of every printed output.

Grid and Initial Condition File Construction. The listing of the Grid and Initial Condition File for the ASTF exhaust/header bypass is shown in Figure 18 and a line by line explanation of the listing is now presented. The first section generates the grid and the last section sets the initial conditions.

<u>Line No.</u>	<u>Function</u>
1.	Names computer program "Grid".
2.	Sets number of grid lines for the J (axial) and K (radial) indices.

```

1.  PROGRAM GRID
2.  PARAMETER (JD=229, KD=91)
3.  DIMENSION RH(JD,KD),RU(JD,KD),RV(JD,KD), E(JD,KD)
4.  DIMENSION X(JD,KD),R(JD,KD)
5.  DATA G/1.4/,NC1/0/
6.  GM1=G-1.
7.  DELX=228./228
8.  C FORM THE X GRID
9.  DO 10 J=1,JD
10. DO 10 K=1,KD
11. IF(J.EQ.1) THEN
12. X(J,K)=0.
13. ELSE
14. X(J,K)=X(J-1,K)+DELX
15. ENDIF
16. 10 CONTINUE
17. C FORM THE R GRID
18. DO 20 J=1, JD
19. IF(X(J,KD).LE.100.) R2=22.5
20. IF(X(J,KD).GT.112.) R2=16.
21. IF((X(J,KD).GT.100.).AND.(X(J,KD).LE.112.)) THEN
22. R2=22.5-(X(J,KD)-100.)*0.5167
23. ENDIF
24. DELR=R2/(KD-1)
25. R(J,1)=0.0
26. DO 20 K=2,KD
27. R(J,K)=R(J,K-1)+DELR
28. 20 CONTINUE
29. C FORM THE ARRAYS OF NON-DIMENSIONAL CONSERVATION VARIABLES
30. C FORM THE (1,1)-(1,12) ARRAY A
31. UMACH=0.41
32. FACT=(1.+2*UMACH**2)
33. PBAR=FACT**(-3.5)/G
34. DO 100 K=1,12
35. RH(1,K)=FACT**(-2.5)
36. RU(1,K)=RH(1,K)*UMACH*SQRT(1./FACT)
37. RV(1,K)=0.
38. E(1,K)=PBAR/GM1+.5*(RU(1,K)**2)/RH(1,K)
39. 100 CONTINUE
40. C FORM THE (1,13) - (1,91) ARRAY B
41. UMACH = 0.00724
42. FACT=(1.+2*UMACH**2)
43. PBAR=FACT**(-3.5)/G
44. DO 200 K=13,KD
45. RH(1,K)=FACT**(-2.5)
46. RU(1,K)=RH(1,K)*UMACH*SQRT(1./FACT)
47. RV(1,K)=0.
48. E(1,K)=PBAR/GM1+.5*(RU(1,K)**2)/RH(1,K)
49. 200 CONTINUE

```

Figure 18. Grid and Initial Condition File

```

50. C FORM THE (2,1) - (101,51) ARRAY C
51.      M=2
52.      N=11
53.      L=15
54.      LL=16
55.      DO 300 I=1,10
56.      DO 400 J=M,N
57.      UMACH=0.41
58.      FACT=(1.+2*UMACH**2)
59.      PBAR=FACT**(-3.5)/G
60.      DO 500 K=1,L
61.      RH(J,K)=FACT**(-2.5)
62.      RU(J,K)=RH(J,K)*UMACH*SQRT(1./FACT)
63.      RV(J,K)=0.
64.      E(J,K)=PBAR/GM1+.5*(RU(J,K)**2)/RH(J,K)
65.      500 CONTINUE
66. C FORM THE (2,2) - (101,91) ARRAY D
67.      UMACH = 0.00724
68.      FACT=(1.+2*UMACH**2)
69.      PBAR = FACT**(-3.5)/G
70.      DO 600 K=LL,KD
71.      RH(J,K)=FACT**(-2.5)
72.      RU(J,K)=RH(J,K)*UMACH*SQRT(1./FACT)
73.      RV(J,K)=0.
74.      E(J,K)=PBAR/GM1+.5*(RU(J,K)**2)/RH(J,K)
75.      600 CONTINUE
76.      400 CONTINUE
77.      M=M+10
78.      N=N+10
79.      L=L+4
80.      LL=LL+4
81.      300 CONTINUE
82. C FORM THE (102,1) - (229, 91) ARRAY E
83.      UMACH=0.00414
84.      FACT=(1.+2*UMACH**2)
85.      PBAR=FACT**(-3.5)/G
86.      DO 700 J=102,JD
87.      DO 800 K=1,KD
88.      RH(J,K)=FACT**(-2.5)
89.      RU(J,K)=RH(J,K)*UMACH*SQRT(1./FACT)
90.      RV(J,K)=0.
91.      E(J,K)=PBAR/GM1+.5*(RU(J,K)**2)/RH(J,K)
92.      800 CONTINUE
93.      700 CONTINUE
94.      NC1=0
95.      WRITE(20)NC1,G
96.      WRITE(20)X,R
97.      WRITE(20)RH,RU,RV,E
98.      STOP
99.      END

```

Figure 18. (Continued)

3. Dimensions arrays for the nondimensional density, x-momentum (axial), R-momentum (radial), and total energy per unit volume.
4. Dimensions arrays for the x and R coordinates of the grid.
5. Sets ratio of specific heats (γ) equal to 1.4 and iteration count to zero.
6. Sets variable denoting $\gamma - 1$.
7. Scales distance between J-grid lines is set to one foot.
- 8.-28. Performs do loops 10 and 20 to form the grid shown in Figure 19. This figure shows a simplified form of the grid for clarity. The actual grid would contain 229 J-lines and 91 K-lines.

The first part of the header has a 22 1/2 ft radius so the scale distance between the K-grid lines is 0.25 ft. The second part of the header has a 16 ft radius and the scale distance between its K-grid lines is 0.178 ft. The user would need to keep this in mind while reviewing any plot outputs.
22. Sets the slope of the convergent portion of the header to 0.5167.

Since the computer analysis will be done for an axisymmetric flow, the constructed grid represents an axial cross section of the duct with the K=1 grid line representing the axis of symmetry.

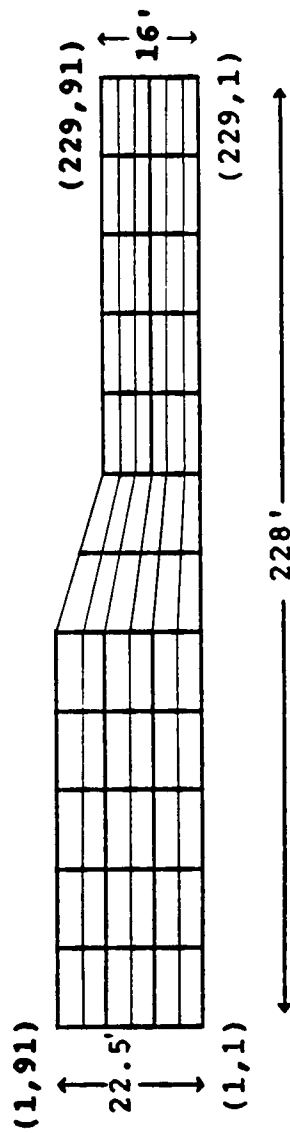


Figure 19. Simplified Illustration of Grid Generated by Do Loops 10 and 20

The next section sets the initial conditions and is somewhat more complex; hence, its length. It divides the flow grid into five regions for setting the initial conditions A,B,C,D, and E (Figure 20). Cooper's [9] scheme from the example problems was adopted for setting the initial conditions with the use of local Mach number and nondimensionalized flow parameters.

- 30. - 39. Sets initial conditions for region A (exit face of the bypass).
- 31. Sets local Mach number equal to 0.41.
- 32. Sets nondimensional temperature (see Appendix C for derivation of this and other nondimensional equations).
- 33. Sets nondimensional pressure.
- 34. - 39. Performs the do loop that sets values for nondimensional density, axial momentum, radial momentum, and total energy.
- 40. - 49. Sets region B initial conditions for warm and slow secondary flow. Nondimensional parameters are calculated.
- 50. - 81. Treats regions C and D such that the jet spread discussed previously in Chapter III is partially accounted for. The J and K indices increment in a stair-step fashion to approximate the slope of the jet spread. The Mach number for region C is set to 0.41. Region D flow conditions are set the same as region

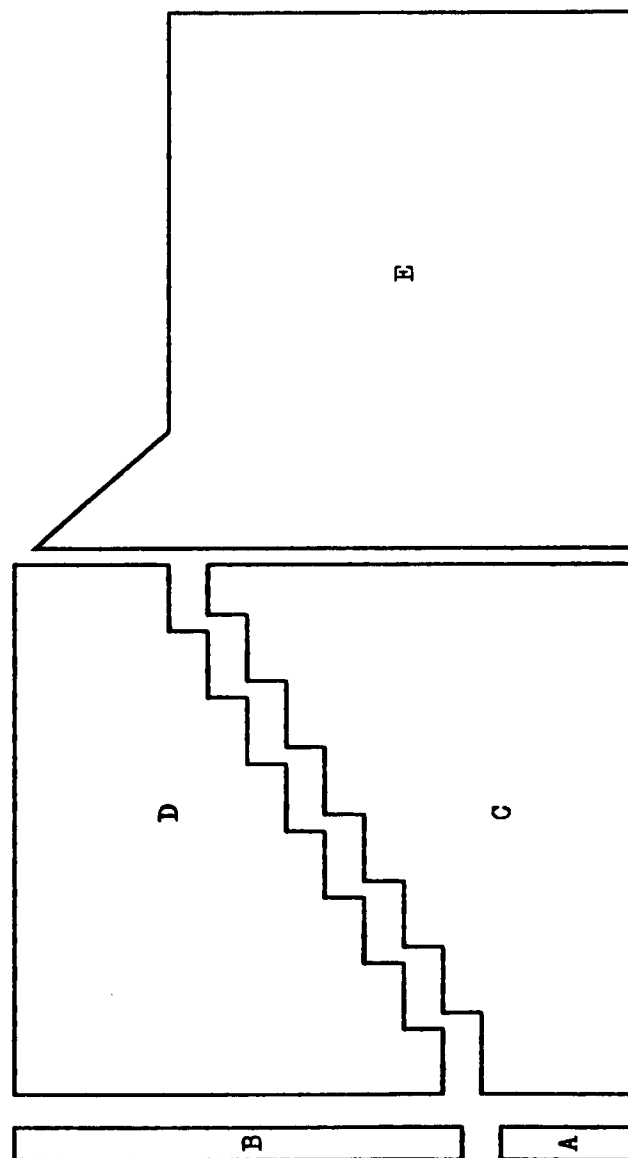


Figure 20. Flow Regions for Initial Condition Arrays

B's. PARC will take the user's initial conditions and adjust accordingly to account for shocks, etc. PARC can proceed from arbitrary initial conditions and only a factor of two improvement in convergence rate can be expected between good and bad initial condition input. A ballpark (rather than arbitrary) estimate of the expected steady-state conditions has been attempted (as detailed in previous chapters) for this thesis in order to avoid convergence problems.

82. - 93. Sets region E initial conditions to represent the mixture of the bypass/primary and header/secondary flows (as discussed in Chapter II).

94. - 99. Writes output and sets end of file statements.

Parameter File NAMELISTs Construction. The first section of the parameter file will not be discussed further other than that it contains user account numbers and file location nomenclature peculiar to the user's organization. Cooper provides a couple of samples used by Sverdrup Technology for the Cray computer.

The listing of the second section, the parameter file NAMELISTs, is shown in Figure 21 and a line by line (as necessary) explanation is now presented. The listing consists of three NAMELISTs: INPUTS, PRTSEG, and BOUNDS. It is recommended that the NAMELIST BOUNDS portion be reviewed first by the reader in order to better understand the other two NAMELISTs.

```

1.//DD*
2.$INPUTS
3.  NMAX=2500,                JMAX=229,   KMAX=91,   NP=2500,
4.  PREF=1.50006,            TREFR=560.0065,
5.  NPSEG=1,                  IFXPRT=0,
6.  DIS2=0.25,                RE=661440.0,
7.  DTCAP=20.0,
8.  NSPRT=10,                  IAXISY=1,          STOPL2=1.E-10,
9.  LAMIN(1)=0,                LAMIN(2)=1,
10.$END
11.$PRTSEG
12. JKLPI(1,1,1)=1,229,10,JKLPI(1,2,1)=1,91,10,IPORD(1)=2,
13.$END
14.$BOUNDS
15.  NJSEG=3,
16.    JLINE(1)=1,
17.    JKLOW(1)=2,            JKHIGH(1)=12,        JTYPE(1)=-10,
18.    JSIGN(1)=1,
19.    JLINE(2)=1,
20.    JKLOW(2)=13,          JKHIGH(2)=90,        JTYPE(2)=-10,
21.    JSIGN(2)=1,
22.    JLINE(3)=229,
23.    JKLOW(3)=2,          JKHIGH(3)=90,        JTYPE(3)=0,
24.    JSIGN(3)=-1,        PRESSJ(3)=0.71650,  TEMPJ(3)=0.878,
25.  NKSEG=2,
26.    KLINE(1)=1,
27.    KJLOW(1)=1,          KJHIGH(1)=229,      KTYPE(1)=51,
28.    KSIGN(1)=1,
29.    KLINE(2)=91,
30.    KJLOW(2)=1,          KJHIGH(2)=229,      KTYPE(2)=50,
31.    KSIGN(2)=-1,
32.$END
33.//DD*
34./EOJ
35.END OF DATA

```

Figure 21. Parameter File NAMELISTs

<u>Line No.</u>	<u>Function</u>
2. - 10.	NAMELIST INPUTS. Selects options to specify the problem to be solved.
3.	Sets maximum number of iterations, NMAX, to be performed during PARC execution. Typical iteration range is from 2,000 to 20,000. Initially, value is set low until program is running without error messages. Sets J and K maximums, JMAX and KMAX, for the grid. Sets flow-field print frequency, NP, to 2500. This option specifies the frequency that the flow field is printed.
4.	Sets reference pressure and temperature. Station 2 is chosen as reference and its total pressure (1.50006 lbf/in ²) and total temperature (560.0065°R) are the nondimensionalizing denominators for the five segments discussed in NAMELIST BOUNDS. This reference temperature and pressure satisfies the suggested criteria that the reference flow parameters are realized somewhere in the flow and are consistent with the ideal gas law.
5.	Number of print segments, NPSEG, is one. The printed output for the segment is specified in NAMELIST PRTSEG. Flux balance history parameter, IFXPRT, is zero. A value of zero excludes the flux balances (mass, momentum, and energy) from the printed convergence history. A value of one causes them to be calculated

and printed. Suggest that flux balances not be outputted until program is running without error messages.

6. Sets level of artificial viscosity (DIS2). Cooper recommends that it be started at a maximum value of 0.25 and later can be reduced as low as is consistent with stability and accuracy. Since the PARC code uses central differencing, artificial viscosity is required to improve the robustness of the calculations. It improves stability and accuracy where strong shocks and pressure gradients are concerned. The PARC code requires a reference Reynold's number:

$$\begin{aligned}
 &= \frac{\rho_{\text{ref}} a_{\text{ref}} x_{\text{ref}}}{\mu_{\text{ref}}} \\
 &= \frac{(0.00724 \text{ lbm/ft}^3)(1160.26 \text{ ft/s})(1\text{ft})}{(1.27 \times 10^{-5} \text{ lbm/ft s})} \\
 &= 6.6144 \times 10^5
 \end{aligned}$$

Reference length $x_{\text{ref}} = 1 \text{ ft}$ is the unit length for the dimensions of the flow field.

7. DTCAP is a parameter that sets maximum time-step size for the run. It controls the convergence rate and is set to a high value of 20.0. This value needs to be reduced on subsequent runs so that the value of "DT,"

printed as part of the convergence history, matches it throughout the run.

8. NSPRT is a parameter that sets the frequency at which the convergence history parameters are printed. It is set to be calculated and printed every tenth iteration. Thus, the number of lines of convergence data printed will be $(NMAX=2500)/(NSPRT=10)=250$. Axisymmetric form of the Navier-Stokes equations is selected by setting IAXISY equal to one (zero for 2-D). STOPL2 is the convergence parameter value that PARC execution will terminate at. Set at 1×10^{-10} per Cooper's recommendation.
9. To specify turbulent flow, LAMIN for J-index is set to zero and LAMIN for K-index is set to one. The value for INVISC option was not specified, so by default it sets to fully viscous. Also not specified is the turbulent Prandtl number and its default value is 0.9.
11. - 13. NAMELIST PRTSEG. Specifies the flow-field segments as called for by NPSEG.
12. JKLPI arrays specify the index coordinate limits and print increments for each segment called for by NPSEG (one segment in this case). IPORD array determines the order in which the J and K indices are used in printing the segment.

NAMELIST PRTSEG provides the user with the capability to study selected portions of the flow field in detail using

one or more segments. For this case, only one segment was specified (NPSEG=1) for a rough look at the whole flow field in order to keep the program simple. The user should insure the program is running smoothly before attempting more complex segments. The printout for a segment will consist of blocks of data (Figure 22) whose format is determined by the way the JKLPI and IPORD arrays are set up.

The generic definition of the JKLPI array is: (1-D printout, J or K index, segment no.) = J or K coordinate limits, and print increment/step size is from coordinate limits N1 to N2. JKLPI (1,1,1)=1,229,10 means the J-index (1) limits are from J=1 to 229 and starting at J=1 data will be calculated and printed every tenth J-line. There will be a total of 22 increments because J=229 does not fall on a tenth increment and won't be printed. JKLPI (1,2,1)=1,91,10 means the K-index (2) limits are from K=1 to 91 and starting at K=1 data will be calculated and printed at every tenth K-line for a total of nine increments. The data to be calculated and printed in both cases is P, T, M, P_o , U, V, X, and R in dimensional form.

With the setting of IPORD(1)=2, the blocks of data will be printed for the K-index (2) first with the J-index (1) held constant (vice versa for IPORD(2)=1) as shown in Figure 22. This presents the data in a "marching" fashion so the user can study the flow-field values as they progress down the duct.

J = 1									
K	P	T	M	P _o	U	V	X	R	
1									
11									
21									
31									
41									
51									
61									
71									
81									
91									

Figure 22. NAMELIST PRTSEG Data Block Format

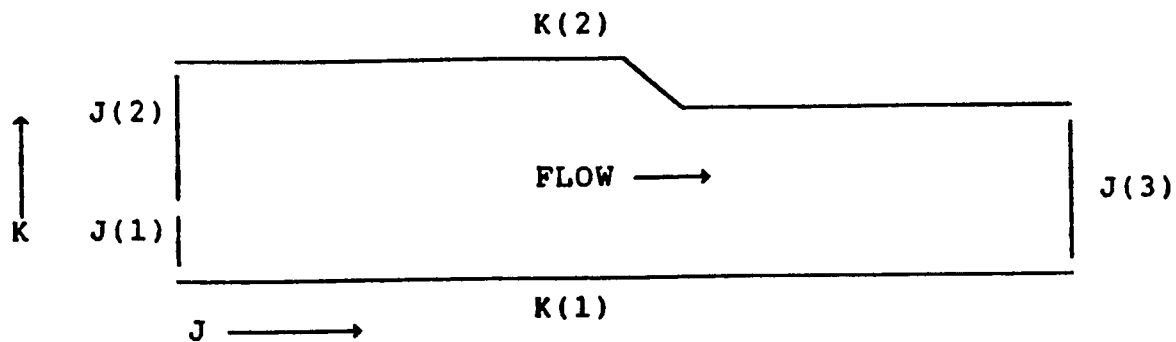
The printed output for this problem will consist of three portions. The first and smallest will be a summary listing of the inputs as specified in the three NAMELISTs. The second will be 250 lines (NMAX/NSPRT) of convergence history. The third and last will be 23 blocks of data, each twelve lines long, describing the flow field. After studying this basic output, the user can further tailor the input and/or output options to meet specific requirements.

14. - 32. NAMELIST BOUNDS. The flow boundaries are split into segments and the appropriate boundary-condition information is determined for each segment. For this problem there are five segments and Figure 23 summarizes at a glance the construction of BOUNDS. Notice that segments do not share points and that surface normals in the direction of the index vector equal one (minus one in the opposite direction).

15. There are three J-constant segments.

16. The first J-constant segment, J(1), has a J=1 index. the radial half of the bypass exit face. Boundary condition for J(1) is a type -10 which is a fixed

17. J(1) segment size is from K=2 to K=12 and represents the radial half of the bypass exit face. Boundary condition for J(1) is a type -10 which is a fixed condition. Dimensionless pressure and temperature are not specified since they are calculated from the Mach number which is fixed ($M_{*1} = 0.41$).



J Constant Segments							
Seg. No.	J-Index	K-Index		Sign of Normal	Boundary Conditions		
		Min.	Max.		Type	Pres.	Temp.
J(1)	1	2	12	1	-10	-	-
J(2)	1	13	90	1	-10	-	-
J(3)	229	2	90	-1	0	0.71650	0.878
K Constant Segments							
Seg. No.	K-Index	J-Index		Sign of Normal	Boundary Conditions		
		Min.	Max.		Type	Pres.	Temp.
K(1)	1	1	229	1	51	-	-
K(2)	91	1	229	-1	50	-	-

Boundary Condition Type	
-10	Fixed inlet conditions. P & T are calculated from fixed Mach no.
0	Free boundary. P & T are initially assigned but allowed to float.
51	Axis of symmetry. P & T not required.
50	Slip wall. P & T not required.

Figure 23. NAMELIST BOUNDS Construction

18. Sign of normal for J(1) is 1.
19. J-constant segment J(2) has a J=1 index.
20. J(2) segment size is from K=13 to K=90 and represents the radial half of the inlet secondary flow. Boundary-condition for J(2) is also a type -10 which is a fixed condition. Again, dimensionless pressure and temperature are not specified since they are calculated from the Mach number which is fixed ($M_2 = 0.0076$).
21. Sign of normal for J(2) is 1.
22. J-constant segment J(3) has a J=229 index.
23. J(3) segment size is from K=2 to K=90 and represents the radial half of the 16 ft duct. J(3)'s position corresponds to valve AV.E.4 in Figure 13, page 23. For this case, the valve is idealized as fully open with no flow impedance. Boundary condition type 0 is a free boundary where nondimensional pressure and temperature are initially assigned but allowed to float.
24. Sign of normal for J(3) is -1. Nondimensionalized J(3) pressure is $(1.5047 \text{ lbf/in}^2)/(1.50006 \text{ lbf/in}^2)(\gamma)$ or 0.71650. Nondimensional J(3) temperature is $(491.711^\circ\text{R})/(560.0065^\circ\text{R})$ or 0.878. To calculate nondimensional pressure, the Station 3 pressure given by $P_3 = 1.505 \text{ lbf/in}^2$ must be adjusted for a 16 ft radius duct rather than 22 1/2 ft. From continuity:

$$\begin{aligned}
U'_3 &= \frac{A_3}{A'_3} U_3 \\
&= \frac{(22.5 \text{ ft})^2}{(16 \text{ ft})^2} (15.203 \text{ ft/s}) \\
&= 30.06 \text{ ft/s}
\end{aligned}$$

Assuming density is roughly unchanged, the adjusted pressure is:

$$\begin{aligned}
P'_3 &= P_3 + \frac{1}{2} \frac{\rho_3 U_3^2}{g_c} - \frac{1}{2} \frac{\rho_3 U'^2_3}{g_c} \\
&= 1.505 \frac{\text{lbf}}{\text{in}^2} \\
&\quad + \frac{1}{2} \frac{(0.00414 \text{ lbm/ft}^3) \left[(15.203 \text{ ft/s})^2 - (30.06 \text{ ft/s})^2 \right]}{\left[32.2 \frac{\text{ft lbm}}{\text{s}^2 \text{ lbf}} \right] \left[144 \frac{\text{in}^2}{\text{ft}^2} \right]} \\
&= 1.5047 \text{ lbf/in}^2
\end{aligned}$$

25. There are two K-constant segments.
26. K(1) segment has a K=1 index.
27. K(1) segment size is from J=1 to J=229 and represents the axis of symmetry which is a boundary-condition type 51. Nondimensionalized pressure and temperature are not required for this boundary condition.
28. Sign of normal for K(1) is 1.
29. K(2) segment has a K=91 index.

30. $K(2)$ segment size is from $J=1$ to $J=229$ and represents the header wall. Boundary-condition type 50 is a slip wall and does not require specification of a nondimensionalized pressure or temperature.
31. Sign of normal for $K(2)$ is -1.

V. RECOMMENDATIONS AND SUMMARY

Recommendations for Further Study

ASTF is a relatively new facility, having been completed in 1984, and operational procedures are still in development. Its capacity and potential are still being explored and, in the process, problems have appeared. The propulsion and wind tunnel testing business is very schedule driven because there are many users waiting with their test articles. AEDC engineers are under pressure to analyze any test-limiting problems thoroughly and to present the solution in an expeditious manner. As a result, many computational tools and methods have been developed that are state of the art for examining flow fields, for example.

PARC is one state-of-the-art tool and its application to the bypass/header flow configuration has been presented here in as much detail as necessary to minimize learning-curve time so the user can proceed with flow field computations. An AEDC engineer with a need to analyze the flow field in this portion of the ASTF facility can use this thesis in conjunction with the PARC users manual. With the input files of this thesis and user account numbers in parameter file format, the engineer can proceed to the Cray computer and obtain detailed flow analysis.

Additional Detail and Fine Tuning. Once the program is running smoothly, the first adjustment the user should make is to specify one or more additional print segments to "zoom in" on selected portions of the flow field. Figure 24 shows two regions of flow suggested for additional study. One region is just downstream of the bypass exit and thus would be an area to study because of the expansion and mixing effects. Also, studying the influence of the converging duct on the flow field would be interesting. To study these two regions the user would need to specify two print segments in addition to the original segment to get the printout of detailed data. Only the radial halves of the region need to be specified for axisymmetric flow (as shown in Figure 24). For these two additional print segments modify line 5 of NAMELIST to:

5. NPSEG = 3, IFXPRT = 0,

Insert two more lines after NAMELIST line 12:

13. JKLPI(1,1,1)1,11,1, JKLPI(1,2,1)1,21,1, IPORD(1)=2,

14. JKLPI(1,1,1)96,118,1, JKLPI(1,2,1)1,91,5, IPORD(1)=2,

The mod to line 5 tells PARC there are three segments specified in NAMELIST PRTSEG. Line 13 specifies that a segment output data for the first ten feet $((11-1)*1$ ft between J-lines) downstream of the bypass exit for a radial distance of five feet $((21-1)*0.25$ ft between K-lines). The increment is one for J and K so there will be eleven blocks of data (J=1 to 11) for this segment with each block

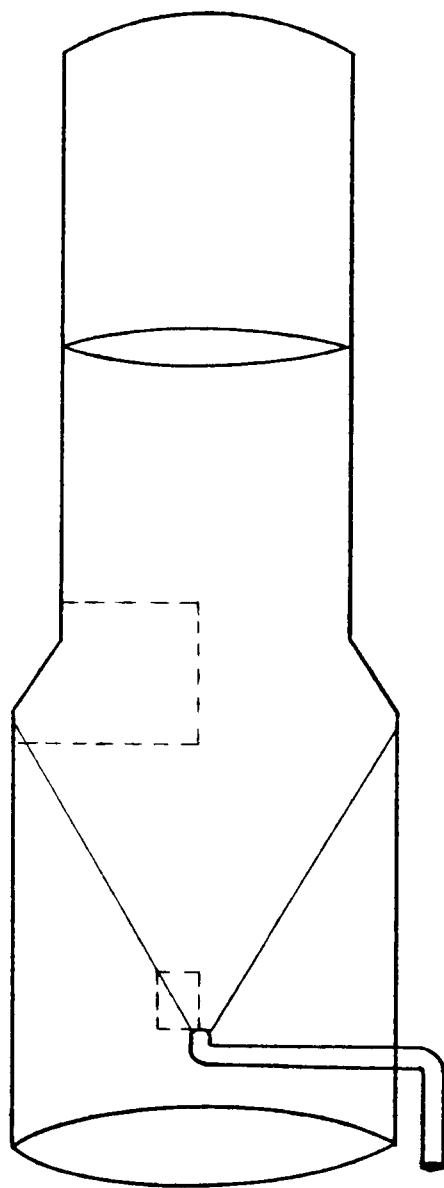


Figure 24. Suggested Flow Regions for Additional Print Segments

containing twenty-one lines of data ($K=21$). Line 14 specifies a segment from the duct centerline to the wall, five feet either side of the twelve-foot converging ducting (reference Figure 24). A J-increment of one will output twenty-three blocks of data ($118-96+1$) for this segment with each block containing nineteen lines of data ($K=1$ to 91, increment of 5). This should be sufficient to show the user the shock effects for the first region and the converging duct effect on the second region. The output segments can be further tailored to reduce or increase data requirements for any specific region.

Fine-tuning adjustments can be made to some of the input parameters to study the effects upon the flow field or the convergence rate. To name a few, the turbulent Prandtl number can be adjusted from its default value of 0.9 down to 0.7 and/or the artificial viscosity could be reduced as low as is consistent with stability and accuracy. To adjust convergence, the maximum number of iterations (NMAX) could be increased or the convergence parameter (STOPL2) can be raised or lowered as required. Plot options are also available for the convergence parameter and/or for the IFXPLT flux balances (mass, momentum, and energy).

Configuration Changes. While there are quite a few options for additional detail and fine tuning, there are almost an endless number of ways to configure the ducting arrangement. Grid files can be constructed that

represent the ducting with one or more exhausters operating, a modification of the bypass exit, the AV.E.4 configuration valve in a closed or open position, the inclusion of the remaining 109' header section, etc. After achieving proficiency in the construction of various axisymmetric/2-D grid files of the header/bypass, the user could ultimately progress to 3-D. While more difficult, 3-D would more closely approximate actual flow fields. An accurate, 3-D model would be the optimum tool to use to analyze the effect upon the header flow when varying exhauster selection during bypass operation. The scope of any such effort would undoubtedly be proportional to the requirement for the data.

PARC Summary

PARC helps to satisfy the requirement for a general purpose CFD tool that is applicable to engine test facility needs. It has proven capable of solving many problems with its flexibility of application and state of the art capability. It can also contribute to improved validation of the ASTF math model which will in turn provide more accurate estimation of duct heat transfer. Accurate heat transfer estimation is important for the determination of facility performance. The powerful combination of PARC and the Cray computer is recommended as the analysis tool for the study of bypass/header flow fields.

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Polce, R.L. A Systems Description of the Aeropropulsion Systems Test Facility (ASTF). Master's thesis, The University of Tennessee, Knoxville, UTSI, 1982.
2. Deibner, G.W. Heat Transfer in the Aeropropulsion Test Facility Mixing Header. Master's thesis, The University of Tennessee, Knoxville, UTSI, 1987.
3. Lazalier, G.R., Henninger, D.S., and Moran, A.H. Results of Test Cell Bypass Tests. Sverdrup memorandum to AEDC/DOPT, 1987.
4. Moran, A.H. ASTF Mathematical Model Initial Baseline. Arnold Engineering Development Center, 1986.
5. Leming, John H. Cooldown Data from ASTF C1 Bypass Checkout #3. Arnold Engineering Development Center, 1988.
6. Saad, Michel A. Compressible Fluid Flow. Englewood Cliffs, New Jersey: Prentice-Hall, 1985.
7. Van Wylen, Gordon J. and Sonntag, Richard. Fundamentals of Classical Thermodynamics. New York: John Wiley & Sons, 1978.
8. Abramovich, G.N. The Theory of Turbulent Jets. Cambridge: The Massachusetts Institute of Technology, 1963.
9. Cooper, G.K. "The PARC Code: Theory and Usage," AEDC-TR-87-24, October 1987.
10. Schulz, R.J. Gas Turbine Theory Notes. The University of Tennessee Space Institute, October 1989.
11. Leming, John H., Knox, G.D., and Rector J. Private communications on ASTF Test Cell Bypass Operations. Arnold Engineering Development Center, Spring 1989.
12. Crawford, R., Schulz, R.J., and Vakili, A.D. Private Communications on ASTF Test Cell Bypass Flow Analysis. The University of Tennessee Space Institute, October 1989.

13. Incropera, F.P., and Dewitt, D.P. Fundamentals of Heat Transfer. New York: John Wiley & Sons, 1981.
14. Shapiro, A.H. The Dynamics and Thermodynamics of Compressible Fluid Flow. New York: The Ronald Press, 1953.

APPENDIXES

APPENDIX A

CALCULATED TEST CELL BYPASS FLOW CONDITIONS

CALCULATED TEST CELL BYPASS FLOW CONDITIONS

The technique used to estimate the flow conditions of the bypass flow, as it leaves the bypass header, was to first assume that the flow was choked at the exit at a static pressure equal to the exhaust header pressure.

Choked Flow Assumption

The choked bypass exit conditions that would be known are:

$$M_{*1} = 1.0$$

$$\dot{m}_{*1} = 100 \text{ lbm/s}$$

$$P_{*1} = 1.5 \text{ lbf/in}^2$$

$$A_{*1} = \pi(2.75 \text{ ft})^2$$

$$= 23.76 \text{ ft}^2$$

From these known conditions, the remaining unknown condition would be the gas static temperature. The temperature is calculated from the mass-conservation principle as follows:

$$\dot{m}_{*1} = \rho_{*1} U_{*1} A_{*1} = \frac{P_{*1}}{RT_{*1}} M_{*1} \sqrt{\gamma g_c RT_{*1}} A_{*1}$$

$$\dot{m}_{*1} = P_{*1} M_{*1} A_{*1} \sqrt{\frac{\gamma g_c}{RT_{*1}}}$$

$$\dot{m}_{*1}^2 = P_{*1}^2 M_{*1}^2 A_{*1}^2 \frac{\gamma g_c}{RT_{*1}}$$

$$T_{*1} = \frac{P_{*1}^2 M_{*1}^2 A_{*1}^2 \gamma g_c}{\dot{m}_{*1}^2 R}$$

$$T_{*1} = \frac{\left[(1.5)(144)(1.0)(23.76) \right]^2 (1.4)(32.2)}{(100)^2 (53.36)} \text{ } ^\circ\text{R}$$

$$= 2222.9 \text{ } ^\circ\text{R}$$

This temperature indicates that the end of the duct cannot be choked, as assumed. Therefore, the next analysis of the bypass flow was to compute the flow as a one-dimensional flow with wall friction and/or heat transfer in order to estimate outflow conditions. The first step taken was to compute heat transfer effects on the flow.

Heat Transfer Calculations

Assume fully developed flow in the bypass duct with constant natural convection heat transfer conditions on the

outer wall, which is exposed to atmospheric conditions.

Assume:

- nominal bypass duct length of 500 ft
- bypass wall thickness of 1/4 inch
- wall material of carbon steel
- ambient temperature of 70°F (530 °R)
- ambient pressure of 1 atm.

I. External Free Convection Surface Coefficient

$$Nu_D = CRa_D^n = \frac{\bar{h}D}{k}$$

$$Ra_D = Gr_D Pr = \frac{g\beta |T_s - T_\infty| D^3}{\nu \alpha}$$

$$n = \frac{1}{4} \text{ (laminar), } \frac{1}{3} \text{ (turbulent)}$$

All properties based on film temperature, T_f , where

$$T_f = \frac{1}{2} (T_s + T_\infty)$$

The correlation is:

$$Nu_D = \left\{ 0.60 + 0.387 Ra_D^{1/4} / \left[1 + \left(\frac{0.559}{Pr} \right)^{9/16} \right]^{8/27} \right\}^2$$

for $10^{-5} \leq Ra_D \leq 10^{12}$; equation 9.34 from Fundamentals of Heat Transfer (FHT)[13]. An alternate form is given in Table 9.2 of FHT.

First Compute Ra_D :

$$1. \quad \text{Compute } T_f = \frac{1}{2} (380 + 530) = 455 \text{ } ^\circ\text{R}$$

$$= 252.78 \text{ } ^\circ\text{K} \approx 253^\circ\text{K}$$

$$2. \quad \beta = - \left. \frac{1}{\rho} \frac{\delta \rho}{\delta T} \right|_P = \frac{1}{T} \text{ for an ideal gas}$$

$$\beta = 3.956 \times 10^{-3} \text{ } ^\circ\text{K}^{-1}$$

$$3. \quad \text{Assume } T_s = 380^\circ\text{R} = 211.1 \text{ } ^\circ\text{K}$$

$$T_\infty = 530^\circ\text{R} = 294.4 \text{ } ^\circ\text{K}$$

$$|T_s - T_\infty| = |211.1 - 294.4| = 83.3 \text{ } ^\circ\text{K}$$

$$4. \quad \nu @ 253^\circ\text{K} = 11.44 \times 10^{-6} \text{ m}^2/\text{s}$$

$$\alpha @ 253^\circ\text{K} = 15.9 \times 10^{-6} \text{ m}^2/\text{s}$$

$$D^3 = \left(\frac{5.5}{3.2808} \right)^3 = 4.711 \text{ m}^3$$

Thus,

$$Ra_D = \frac{(9.8 \text{ m/s}^2)(3.956 \times 10^{-3} \text{ }^\circ\text{K}^{-1})(83.3^\circ\text{K})(4.711 \text{ m}^3)}{(11.44 \times 10^{-6} \text{ m}^2/\text{s})(15.9 \times 10^{-6} \text{ m}^2/\text{s})}$$

$$= 8.364 \times 10^{10}$$

5. $Pr = 0.720$

Compute $\overline{Nu}_D$,

$$\overline{Nu}_D = \left\{ 0.60 + \frac{(0.387)(8.364 \times 10^{10})^{1/4}}{\left[1 + \left(\frac{0.559}{0.720} \right)^{9/16} \right]^{8/27}} \right\}^2$$

$$= 478.3 \text{ (Eqn. 9.34)}$$

Alternatively, from Table 9.2 of FHT, for $Ra_D = 8.364 \times 10^{10}$,

$$\overline{Nu}_D = 0.125 (8.364 \times 10^{10})^{0.333}$$

$$= 542$$

Therefore, using an average value for $\overline{Nu}_D$,

$$\overline{Nu}_D = \frac{1}{2} (478.3 + 542)$$

$$= 510$$

Calculate the surface coefficient as

$$\begin{aligned}\bar{h} &= \overline{Nu}_D \frac{k}{D} \\ &= \frac{(510) (22.3 \times 10^{-3} \text{ W/m } ^\circ\text{K})}{1.676 \text{ m}} \\ &= 6.78 \text{ W/m}^2 \text{ } ^\circ\text{K}\end{aligned}$$

II. Pipe Thermal Conductivity

Assume that the pipe wall is at the film temperature. Thus, $T_v = 253 \text{ } ^\circ\text{K}$, and the thermal conductivity for steel is

$$k_s = 41 \text{ W/m}^\circ\text{K}$$

for standard (carbon - Mn - Si) structural carbon steel.

III. Internal Forced-Convection Surface Coefficient

$$\overline{Nu}_D = F(Re_D, Pr) = \frac{\bar{h} D}{k}$$

First, compute flow Reynolds number.

$$Re_D = \frac{\rho U D}{\mu}$$

Assume air at $380 \text{ } ^\circ\text{R}$ for a start:

$$\rho = \frac{P}{R T}$$

$$= \frac{(5.5)(144)}{(53.36)(380)} \frac{\text{lbm}}{\text{ft}^3}$$

$$= 3.91 \times 10^{-2} \frac{\text{lbm}}{\text{ft}^3}$$

$$= \left[3.91 \times 10^{-2} \frac{\text{lbm}}{\text{ft}^3} \right] \left[3.2808 \frac{\text{ft}}{\text{m}} \right]^3 \left[\frac{1}{2.2046} \frac{\text{kg}}{\text{lbm}} \right]$$

$$= 0.626 \text{ kg/m}^3$$

$$\mu = 132.5 \times 10^{-7} \frac{\text{N s}}{\text{m}^2}$$

$$U = ?$$

To compute U, assume 100 lbm/s through the 5.5 ft diameter bypass at the inlet flow conditions.

$$\dot{m} = \rho U A$$

$$= \sqrt{\frac{\gamma g c}{R}} \left(1 + \frac{\gamma-1}{2} M^2 \right)^{-\frac{\gamma+1}{2(\gamma-1)}} \frac{P_{0, \text{AM}}}{\sqrt{T_0}}$$

Given:

$$\dot{m} = 100 \text{ lbm/s}$$

$$P_0 = 5.5 \text{ lbf/in}^2$$

$$T_0 = 380 \text{ } ^\circ\text{R}$$

$$A = \frac{\pi}{4} (5.5 \text{ ft})^2 = 23.76 \text{ ft}^2$$

substitute and solve for M,

$$100 = \sqrt{\frac{(1.4)(32.2)}{53.36}} (1 + 0.2M^2)^{-3.0} \frac{(5.5)(144)(M)(23.76)}{\sqrt{380}}$$

$$0.1128 = M(1 + 0.2M^2)^{-3}$$

$$M = 0.1137$$

Hence,

$$U = Ma$$

$$= 0.1137 \sqrt{(1.4)(32.2)(53.36)(380)}$$

$$= 108.7 \text{ ft/s}$$

$$= 33.12 \text{ m/s}$$

The Reynolds number is

$$Re_D = \frac{\rho U D}{\mu}$$

$$= \frac{(0.626 \text{ kg/m}^3)(33.12 \text{ m/s})(1.676 \text{ m})}{132.5 \times 10^{-7} \frac{\text{Ns}}{\text{m}^2}}$$

$$= 2.547 \times 10^6$$

Flow is turbulent. Thus, use fully-developed flow Nusselt number correlation:

$$\overline{Nu}_D = 0.023 Re_D^{0.8} Pr^n$$

This correlation is Equation 8.58 from FHT[13]. Where,

$$n = 0.3 \text{ for cooling, } T_s < T_m$$

$$Pr = 0.74 \text{ at } 380^\circ R \text{ (approx. } 200^\circ K)$$

Hence,

$$\begin{aligned}\overline{Nu}_D &= 0.023 \left(2.547 \times 10^4 \right)^{0.8} \left(0.74 \right)^{0.3} \\ &= 2801.2\end{aligned}$$

The surface coefficient, $\bar{h}$, is

$$\begin{aligned}\bar{h} &= \overline{Nu}_D \frac{k}{D} \\ &= \frac{(2801) (18.1 \times 10^{-3} \text{ W/m } ^\circ K)}{1.676 \text{ m}} \\ &= 30.25 \text{ W/m}^2 \text{ } ^\circ K\end{aligned}$$

IV. Formulate the Overall Heat Transfer Coefficient

For heat transfer in a cylindrical system, through resistances in series, the overall heat transfer coefficient is:

$$U = \frac{1}{\frac{1}{h_1} + \frac{r_1}{k_A} \ln \left(\frac{r_2}{r_1} \right) + \frac{r_1}{k_B} \ln \left(\frac{r_3}{r_2} \right) + \frac{r_1}{k_C} \ln \left(\frac{r_4}{r_3} \right) + \frac{r_1}{r_4} \frac{1}{h_4}}$$

from Equation 3.30 in FHT, for a cylinder with three wall thicknesses, and internal and external convection heat transfer. Since the bypass duct has only one thickness of conducting material, the equation reduces to

$$U = \frac{1}{\frac{1}{h_1} + \frac{r_1}{k_s} \ln \left(\frac{r_2}{r_1} \right) + \frac{r_1}{r_2} \frac{1}{h_2}}$$

where

$h_1 = \bar{h}$ for internal convection heat transfer

$h_2 = \bar{h}$ for external free convection heat transfer

k_s = steel thermal conductivity

r_1, r_2 = inner and outer duct radii.

The heat transfer equation, Equation 3.29 in FHT, is

$$q = UA_1(T_{\infty,1} - T_{\infty,2})$$

where

$$A_1 = 2\pi r_1 L$$

$$= 2\pi(0.8316)(500/3.2808)\text{m}^2$$

$$= 796.3 \text{ m}^2$$

$T_{\infty,1}$ = duct air temperature

$T_{\infty,2}$ = outside air temperature

Since $T_{\infty,2} > T_{\infty,1}$, the heat flow is into the air in the duct.

Compute U

$$\begin{aligned} U &= \left[\frac{1}{30.25} + \frac{0.8316}{41} \ln(1.0076) + \frac{0.8316}{0.838} \frac{1}{6.78} \right]^{-1} \frac{\text{W}}{\text{m}^2 \text{ } ^\circ\text{K}} \\ &= 5.569 \frac{\text{W}}{\text{m}^2 \text{ } ^\circ\text{K}} \end{aligned}$$

Thus, the heat flow into the pipe is

$$\begin{aligned} q &= (796.3)(5.569)(211.1 - 294.4) \text{ W} \\ &= 3.69598 \times 10^5 \text{ W} \\ &= 350.32 \text{ BTU/s} \end{aligned}$$

For the given massflow rate of 100 lbm/s, this corresponds to a temperature rise computed from

$$\dot{m}_1 C_{p_1} (T_{\text{out}} - T_{\text{in}}) = q$$

$$T_{\text{out}} = T_{\text{in}} + \frac{q}{\dot{m}_1 C_{p_1}}$$

$$= 380 \text{ } ^\circ\text{R} + \frac{350.32}{(100)(0.2)} \text{ } ^\circ\text{R}$$

$$= 394.6 \text{ }^{\circ}\text{R}$$

Thus, only a 15 $^{\circ}\text{R}$ temperature rise is predicted to occur on a still day (free convection dominating the heat transfer rates). Therefore, the bypass flow may be treated as nearly adiabatic. Hence, to first order, a Rayleigh flow is not a valid model for the bypass flow; instead, a friction-based Fanno flow will be applied to estimate flow variation along the bypass pipe.

Fanno Flow Analysis of the Bypass Flow

The heat transfer analysis suggests that, at least on a still day where free convection heat transfer on the outside of the duct determines the heat transfer rate to the air flow, the flow stays nearly adiabatic. Thus, the flow analysis relevant to this problem is the so-called Fanno Flow: adiabatic flow with friction in a constant area duct. The parameter $4 f l/D$ is the controlling parameter for Fanno Flow. For the carbon steel duct assumed, the parameter is computed as follows:

$$f = \frac{\tau_w}{\frac{1}{2} \rho U^2}$$

For

$$\text{Re}_D = 2.547 \times 10^6$$

$$\epsilon/D = 0.0004 \text{ (as typ.)}$$

$$f = 0.004$$

Hence,

$$\frac{4fL}{D} = \frac{4(0.004)(500)}{5.5}$$

$$= 1.4545$$

For an inlet Mach number to the bypass of 0.1137,

$$\left. \frac{4fL}{D} \right|_{in} - \frac{4fL}{D} = \left. \frac{4fL}{D} \right|_{out}$$

$$56.238 - 1.4545 = 54.78$$

This says that the inlet flow has barely accelerated.

Computing the bypass outlet Mach number by interpolation from Table B.4 in The Dynamics and Thermodynamics of Compressible Fluid Flow [14]:

$$M_{*1} = 0.1155$$

Thus, the acceleration of the flow is negligible. From

$$P_{o*1} = 4.937 \text{ lbf/in}^2$$

$$T_{o*1} = 394 \text{ }^\circ\text{R}$$

$$M_{*1} = 0.1155$$

compute massflow:

$$\dot{m}_{*1} = \sqrt{\frac{\gamma g c}{R}} \left[1 + \frac{\gamma-1}{2} M_{*1}^2 \right]^{\frac{-(\gamma+1)}{2(\gamma-1)}} \frac{P_{o*1} A_{*1} M_{*1}}{\sqrt{T_{o*1}}}$$

$$= \sqrt{\frac{(1.4)(32.2)}{53.36}} \left[1 + 0.2(0.1155)^2 \right]^{-3}$$

$$\left[\frac{(4.94)(144)(23.76)(0.1155)}{\sqrt{394}} \right] \frac{\text{lbm}}{\text{s}}$$

$$= 89.64 \text{ lbm/s}$$

This is fairly close to the known flow of 100 lbm/s. Thus, the Fanno analysis predicts that the flow in the bypass pipe will be subsonic and that frictional pressure losses are relatively small. Hence, the pressure losses in the bypass pipe will come from a combination of inlet (contraction) losses, valve (throttle) losses, and elbow losses.

Exit Flow Conditions

The heat transfer and frictional flow analyses of the flow in the bypass pipe indicate that neither heat transfer effects nor friction effects result in a choked outflow condition. Since heat transfer is so relatively small, it is assumed that the outflow conditions are determined by mass conservation subject to the following constraints:

$$P_{*1} = 1.5 \text{ lbf/in}^2 \quad \text{(static pressure of outflow in equilibrium with the header pressure)}$$

$$T_{o*1} = 380 \text{ }^\circ\text{R} \quad \text{(total temperature remains constant)}$$

Given a massflow of 100 lbm/s under these conditions,
solving the mass-conservation equation for the Mach number
of the outflow gives:

$$M_{*1} = 0.41$$

$$U_{*1} = 395 \text{ ft/s}$$

$$P_{o*1} = 1.7 \text{ lbf/in}^2$$

$$T_{o*1} = 394 \text{ }^{\circ}\text{R}$$

APPENDIX B
TWO STREAM MIXING ANALYSIS

TWO STREAM MIXING ANALYSIS

This analysis refers to the mixing of two, parallel, one-dimensional streams with the following additional, simplifying assumptions:

1. No wall friction
2. No wall heat transfer
3. Steady state flow
4. Constant duct area; $A_3 = A_{*1} + A_2$

The analysis is based on the conservation of mass, momentum, and energy; and is accomplished by using the primary variables of pressure, temperature, and velocity.

Conservation of mass (continuity):

$$\dot{m}_2 + \dot{m}_{*1} = \dot{m}_3$$

Define

$$\lambda = \dot{m}_2 / \dot{m}_{*1}$$

Thus,

$$\dot{m}_{*1} (\lambda + 1) = \rho_3 U_3 A_3$$

$$\dot{m}_{*1} (\lambda + 1) = \frac{P_3}{R_s T_3} U_3 A_3$$

$$\frac{P_3 U_3 A_3}{R_3 T_3} = \dot{m}_{*1} (1 + \lambda)$$

or

$$P_3 U_3 - \frac{\dot{m}_{*1} (1 + \lambda) R_3 T_3}{A_3} = 0$$

Define

$$\zeta_1 = \frac{\dot{m}_{*1} (1 + \lambda) R_3}{A_3}$$

Then,

$$P_3 U_3 - \zeta_1 T_3 = 0 \quad (B.1)$$

Conservation of momentum:

$$F_{net} = 0 = P_3 A_3 + \dot{m}_3 U_3 - \left[P_{*1} A_{*1} + \dot{m}_{*1} U_{*1} \right] - \left[P_2 A_2 + \dot{m}_2 U_2 \right]$$

$$P_3 A_3 + \dot{m}_3 U_3 = P_{*1} A_{*1} + P_2 A_2 + \dot{m}_{*1} U_{*1} + \dot{m}_2 U_2$$

$$P_3 A_3 + \dot{m}_3 (1 + \lambda) U_3 = P_{*1} A_{*1} + P_2 A_2 + \dot{m}_{*1} \left[U_{*1} + \lambda U_2 \right]$$

Assuming $P_{*1} = P_2$ (equal static pressures in the initial plane of mixing), then

$$P_{*1} A_{*1} + P_2 A_2 = P_{*1} A_3$$

$$P_g A_g + \dot{m}_{*1} (1 + \lambda) U_g = P_{*1} A_g + \dot{m}_{*1} (U_{*1} + \lambda U_2)$$

$$P_g + \frac{\dot{m}_{*1} (1 + \lambda) U_g}{A_g} = P_{*1} + \frac{\dot{m}_{*1} (U_{*1} + \lambda U_2)}{A_g}$$

Define

$$\xi_2 = \frac{\dot{m}_{*1} (1 + \lambda)}{A_g}$$

$$\xi_g = P_{*1} + \frac{\dot{m}_{*1} (U_{*1} + \lambda U_2)}{A_g}$$

Then,

$$P_g + \xi_2 U_g = \xi_g \quad (\text{B.2})$$

Conservation of energy:

$$\dot{m}_g \left(h_g + \frac{1}{2} U_g^2 \right) = \dot{m}_{*1} \left(h_{*1} + \frac{1}{2} U_{*1}^2 \right) + \dot{m}_2 \left(h_2 + \frac{1}{2} U_2^2 \right)$$

$$\begin{aligned} \dot{m}_{*1} (1 + \lambda) \left(C_{p_g} T_g + \frac{1}{2} U_g^2 \right) &= \dot{m}_{*1} \left(C_{p_{*1}} T_{*1} + \frac{1}{2} U_{*1}^2 \right) \\ &+ \lambda \dot{m}_{*1} \left(C_{p_2} T_2 + \frac{1}{2} U_2^2 \right) \end{aligned}$$

Reduce and rearrange,

$$U_3^2 + 2C_{P_3} T_3 = \frac{\left[C_{P_{*1}} T_{*1} + \lambda C_{P_2} T_2 \right]}{1 + \lambda} + \frac{\left[\frac{1}{2} U_{*1}^2 + \frac{1}{2} \lambda U_2^2 \right]}{1 + \lambda}$$

Define

$$\lambda^* = \frac{\lambda C_{P_2}}{C_{P_{*1}}}$$

Thus,

$$U_3^2 + 2C_{P_3} T_3 = \frac{C_{P_{*1}} \left[T_{*1} + \lambda^* T_2 \right]}{1 + \lambda} + \frac{\left[\frac{1}{2} U_{*1}^2 + \frac{1}{2} \lambda U_2^2 \right]}{1 + \lambda}$$

Define

$$\xi_4 = 2 C_{P_3}$$

$$\xi_5 = \frac{C_{P_{*1}} \left[T_{*1} + \lambda^* T_2 \right]}{1 + \lambda} + \frac{\left[\frac{1}{2} U_{*1}^2 + \frac{1}{2} \lambda U_2^2 \right]}{1 + \lambda}$$

Thus,

$$U_3^2 + \xi_4 T_3 = \xi_5 \quad (B.3)$$

Now solve the three algebraic, conservation equations; B.1, B.2, and B.3; for P_3 , T_3 , and U_3 . From B.2,

$$P_3 = \xi_3 - \xi_2 U_3$$

Substitute into B.1:

$$\left(\xi_3 - \xi_2 U_3 \right) U_3 - \xi_1 T_3 = 0$$

Then,

$$T_3 = \left(\xi_3 - \xi_2 U_3 \right) \frac{U_3}{\xi_1}$$

Substitute for T_3 into B.3 and solve for U_3 :

$$U_3^2 + \xi_4 \left(\xi_3 - \xi_2 U_3 \right) \frac{U_3}{\xi_1} = \xi_5$$

$$U_3^2 \left(1 - \frac{\xi_4 \xi_2}{\xi_1} \right) - \frac{\xi_4 \xi_3}{\xi_1} U_3 - \xi_5 = 0$$

$$U_3 = \frac{-\xi_4 \xi_3}{2 \left(\xi_1 - \xi_4 \xi_2 \right)} \pm \sqrt{\frac{\xi_4^2 \xi_3^2}{4 \left(\xi_1 - \xi_4 \xi_2 \right)^2} + \frac{\xi_5 \xi_1}{\xi_1 - \xi_4 \xi_2}}$$

If either stream 2 or stream *1 or both are supersonic, then two possible solutions are given by the above equation: one solution "+" refers to supersonic fully-mixed flow, the

other solution "-" refers to subsonic fully-mixed flow. For the ASTF test cell bypass/exhaust header problem, both streams 2 and *1 are subsonic; so, the physically real solution is the subsonic solution.

APPENDIX C

NONDIMENSIONAL FLOW PARAMETER DERIVATION

NONDIMENSIONAL FLOW PARAMETER DERIVATION

$$\text{FACT} = (1. + .2 * \text{UMACH} ** 2)$$

$$= (1 + 0.2 M^2)$$

$$= \left(1 + \frac{\gamma - 1}{2} M^2 \right)$$

$$= \frac{T_o}{T}$$

$$\text{PBAR} = \text{FACT} ** (-3.5) / G$$

$$= \frac{\text{FACT}^{-3.5}}{\gamma}$$

$$= \frac{\left(1 + \frac{\gamma - 1}{2} M^2 \right)^{-3.5}}{\gamma}$$

$$= \frac{1}{\gamma} \frac{1}{\left(1 + \frac{\gamma - 1}{2} M^2 \right)^{3.5}}$$

$$= \frac{1}{\gamma} \frac{1}{\left(1 + \frac{\gamma - 1}{2} M^2 \right)^{\gamma / (\gamma - 1)}}$$

$$= \frac{1}{\gamma} \frac{1}{P_o / P}$$

$$= \frac{P}{\gamma P_o}$$

$$RH = FACT^{**}(-2.5)$$

$$= FACT^{-2.5}$$

$$= \left(1 + \frac{\gamma-1}{2} M^2\right)^{-2.5}$$

$$= \frac{1}{\left(1 + \frac{\gamma-1}{2} M^2\right)^{2.5}}$$

$$= \frac{1}{\left(1 + \frac{\gamma-1}{2} M^2\right)^{1/(\gamma-1)}}$$

$$= \frac{1}{\rho_0/\rho}$$

$$= \frac{\rho}{\gamma \rho_0}$$

$$RU = RH * UMACH * SQRT(1./FACT)$$

$$= \frac{\rho}{\rho_0} M \sqrt{\frac{1}{1 + \frac{\gamma-1}{2} M^2}}$$

$$= \frac{\rho}{\rho_0} M \sqrt{\frac{T}{T_0}}$$

$$= \frac{\rho}{\rho_o} M \sqrt{\frac{T}{T_{ref}}}$$

$$= \frac{\rho}{\rho_o} \frac{M \sqrt{\gamma RT}}{\sqrt{\gamma RT_{ref}}}$$

$$= \frac{\rho}{\rho_o} \frac{Ma}{a_{ref}}$$

$$= \frac{\rho}{\rho_o} \frac{U}{a_{ref}}$$

$$E = \frac{E_d}{\gamma P_{ref}} = \frac{PBAR}{\gamma-1} + \frac{1}{2} \frac{RU^2}{RH}$$

$$= \frac{PBAR}{\gamma-1} + \frac{1}{2} \frac{RU^2}{RH}$$

$$= \left(\frac{1}{\gamma-1} \right) \left(\frac{P}{\gamma P_o} \right) + \frac{1}{2} \frac{\left(\frac{\rho}{\rho_o} \frac{u}{a_{ref}} \right)^2}{\rho/\rho_o}$$

$$= \left(\frac{1}{\gamma-1} \right) \left(\frac{1}{\gamma} \right) \left(\frac{P/R}{P_o/R} \right) + \frac{1}{2} \frac{\rho}{\rho_o} \left(\frac{u}{a_{ref}} \right)^2$$

$$= \left(\frac{1}{\gamma-1} \right) \left(\frac{1}{\gamma} \right) \left(\frac{\rho/T}{\rho_o/T_o} \right) + \frac{1}{2} \frac{\rho}{\rho_o} \left(\frac{u^2}{\gamma RT_{ref}} \right)$$

dimensionalized ($\gamma P = \rho T$):

$$E_d = \left(\frac{\rho T}{\gamma - 1} \right) \left(\frac{1}{\gamma} \right) + \frac{1}{2} \rho u^2$$

$$= \rho \left(\frac{T}{\gamma - 1} \frac{1}{\gamma} + \frac{1}{2} u^2 \right)$$

$$= \rho \left(e + \frac{1}{2} u^2 \right)$$

VITA

Royce Dean Purvis was born in El Paso, Texas on February 4, 1960. He attended elementary and high school in Odessa, Texas. In 1980 he transferred from Odessa College to the University of Texas at Austin and received a Bachelor of Science Degree in Mechanical Engineering in May of 1983.

In February of 1984 he entered United States Air Force Officer Training School and after graduation was stationed at Arnold Engineering Development Center in Tennessee. During this period he worked in the ASTF Program Office, Engine Test Facility Operations Division, and attended The University of Tennessee Space Institute on a part time basis. In December of 1989 he was awarded the Master of Science Degree in Mechanical Engineering.

He is married to the former Molly Elise Forrest of Odessa, Texas.