

DTIC

ELECTE

AUG 12 1993

CR 93.003

NCEL

C

June 1993

Contract Report

An Investigation Conducted by
Adaptive Research Corporation
Huntsville, AL

AD-A267 833



STRUCTURED FINITE VOLUME MODELING OF U.S. NAVY AIRCRAFT ENGINE TEST CELLS

TASK 2: TURBOPROP ENGINE -

FINAL REPORT - VOLUME 1

Abstract This report presents results of the numerical simulation of a U.S. Naval turboprop test cell facility. The ultimate purpose of this simulation was to provide the Navy with a numerical model to be used for the evaluation of the aerothermal performance of test cells. This simulation was performed using the structured finite volume (SFV) computer code. A description of the physical model, mathematical details, boundary conditions, and results of the study are presented and covered in Volume 1.

Volume 2, Code Documentation and Listings, provides a copy of the input files developed for the modeling of turboprop test cells.

93-18644



NAVAL CIVIL ENGINEERING LABORATORY PORT HUENEME CALIFORNIA 93043-4328

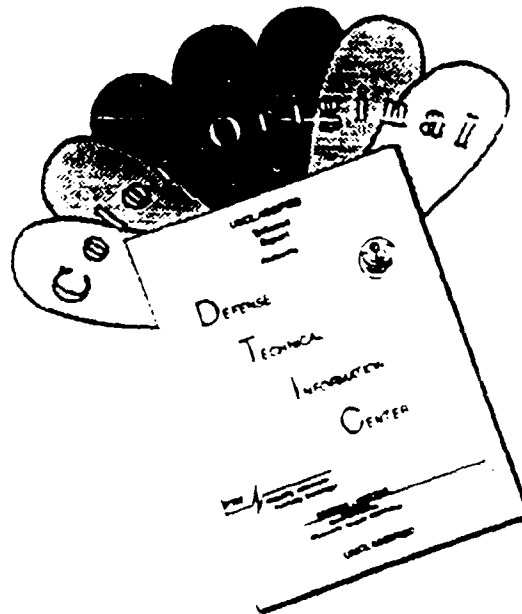
Approved for public release; distribution is unlimited.

METRIC CONVERSION FACTORS

Approximate Conversions to Metric Measures			
Symbol	When You Know	Multiply by	To Find
in ft yd mi	inches	2.5	centimeters
	feet	30	centimeters
	yards	0.9	meters
	miles	1.6	kilometers
in ² ft ² yd ² mi ²	square inches	6.5	square centimeters
	square feet	0.09	square meters
	square yards	0.8	square meters
	square miles	2.6	square kilometers
	acres	0.4	hectares
MASS (weight)			
oz	ounces	28	grams
lb	pounds	0.45	kilograms
	short tons (2,000 lb)	0.9	tonnes
tsp Tbsp fl oz c pt qt gal ft ³ yd ³	teaspoons	5	milliliters
	tablespoons	15	milliliters
	fluid ounces	30	milliliters
	cups	0.24	liters
	pints	0.47	liters
	quarts	0.95	liters
	gallons	3.8	liters
	cubic feet	0.03	cubic meters
	cubic yards	0.76	cubic meters
TEMPERATURE (exact)			
°F	Fahrenheit temperature	5/9 (after subtracting 32)	Celsius temperature
Approximate Conversions from Metric Measures			
Symbol	When You Know	Multiply by	To Find
mm cm m km	millimeters	0.04	inches
	centimeters	0.4	inches
	meters	3.3	feet
	kilometers	1.1	yards
cm ² m ² km ² ha	square centimeters	0.16	square inches
	square meters	1.2	square yards
	square kilometers	0.4	square miles
	hectares (10,000 m ²)	2.5	acres
MASS (weight)			
g	grams	0.035	ounces
kg	kilograms	2.2	pounds
t	tonnes (1,000 kg)	1.1	short tons
ml l l l m ³ m ³	milliliters	0.03	fluid ounces
	liters	2.1	pints
	liters	1.06	quarts
	liters	0.26	gallons
	cubic meters	35	cubic feet
	cubic meters	1.3	cubic yards
TEMPERATURE (exact)			
°C	Celsius temperature	9/5 (then add 32)	Fahrenheit temperature

* 1 in = 2.54 (exactly). For other exact conversions and more detailed tables, see NBS Misc. Publ. 286, Units of Weights and Measures, Price \$2.25, SD Catalog No. C-13 10 286.

DISCLAIMER NOTICE



THIS DOCUMENT IS BEST QUALITY AVAILABLE. THE COPY FURNISHED TO DTIC CONTAINED A SIGNIFICANT NUMBER OF COLOR PAGES WHICH DO NOT REPRODUCE LEGIBLY ON BLACK AND WHITE MICROFICHE.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-018	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE June 1993		3. REPORT TYPE AND DATES COVERED Final; December 1990 - September 1992
4. TITLE AND SUBTITLE STRUCTURED FINITE VOLUME MODELING OF U.S. NAVY AIRCRAFT ENGINE TEST CELLS TASK 2: TURBOPROP ENGINE - FINAL REPORT - VOLUME 1			5. FUNDING NUMBERS PR - 0604215N C - N47408-91-C-1228 DN - 661008	
6. AUTHOR(S) P. L. Daley and W. A. Mahaffey				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Adaptive Research Corporation 4960 Corporate Drive, Suite 100-A Huntsville, AL 35805			8. PERFORMING ORGANIZATION REPORT NUMBER CR 93.003	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) Commander / Naval Civil Engineering Laboratory Naval Air Systems Command 560 Laboratory Drive Code 09Y Facilities Systems Division/L53 Washington, DC 20362-5101 Port Hueneme, CA 93043-4328			10. SPONSORING/MONITORING AGENCY REPORT NUMBER	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) This report presents results of the numerical simulation of a U.S. Naval turboprop test cell facility. The ultimate purpose of this simulation was to provide the Navy with a numerical model to be used for the evaluation of the aerothermal performance of test cells. This simulation was performed using the structured finite volume (SFV) computer code. A description of the physical model, mathematical details, boundary conditions, and results of the study are presented and covered in Volume 1. Volume 2, Code Documentation and Listings, provides a copy of the input files developed for the modeling of turboprop test cells.				
14. SUBJECT TERMS Computational fluid dynamics, test cells, aviation test facilities			15. NUMBER OF PAGES 59	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UL	

TABLE OF CONTENTS

Section No.		Page No.
1.	INTRODUCTION	1
1.1	Purpose of the Report	1
1.2	Background	1
1.3	Objective of the Study	1
1.4	Outline of the Report	2
2.	DESCRIPTION OF THE SIMULATED PROCESS	3
2.1	Geometry	3
3.	MATHEMATICAL FORMULATION	5
3.1	The Structured Finite Volume (SFV) Mathematical Formulation	5
3.1.1	The Governing Equations	5
3.1.2	General Form of the Governing Equations	7
3.1.3	General Form of the Finite-Domain Equations	8
3.1.4	Auxiliary Relations	9
4.	NUMERICAL DETAILS AND PHYSICAL MODELS	10
4.1	Introduction	10
4.2	Grid Description	10
4.3	Properties of the Fluid	10
4.4	Boundary Condition	11
4.4.1	Engine	11
4.4.2	Propeller	12

DTIC QUALITY INSPECTED 3

Accession For	
NTIS	CRA&I ☒
DTIC	TAB ☐
Unannounced ☐	
Justification	
By	
Distribution /	
Availability Codes	
Dist	Avail and/or Special
A-1	

4.4.2.1	Thrust	12
4.4.2.2	Swirl	14
4.4.2.3	Turbulence	15
4.4.3	Inlet	15
4.4.4	Outlet	16
4.4.5	Wall Boundaries	16
4.4.6	Heat Transfer through Augmenter Tube	17
5.1	DISCUSSION OF RESULTS	18
5.1	Velocity	18
5.2	Temperature	18
5.3	Other Quantities	19
6.	CONCLUSION	20
7.	REFERENCES	21
	FIGURES	22
	APPENDIX A	
	APPENDIX B	

LIST OF FIGURES

Page No.

1.	Computational Grid	22
2.	Outline of Features (ISO View)	23
3.	Outline of Features (Side View)	24
4.	Outline of Features (Top View)	25
5.	Velocity Vectors	26
6.	Velocity Vectors Prop Region	27
7.	Velocity Vectors Down Stream of Prop	28
8.	Velocity Vectors Exit of Engine	29
9.	Velocity Vectors Chimney Section	30
10.	Velocity Vectors Front of Test Cell	31
11.	Temperature	32
12.	Temperature Exit of Engine	33
13.	Temperature Augmenter Tube	34
14.	Temperature Exit of Augmenter Tube	35
15.	Temperature Exit of Chimney Section	36
16.	Temperature Augmenter Lip	37
17.	Pressure	38
18.	Turbulence Kinetic Energy Exit of Engine	39
19.	Turbulence Kinetic Energy Down Stream of Prop	40

1. INTRODUCTION

1.1 Purpose of the Report

This report describes the development and applications of a computer model that simulates steady flow inside a turboprop engine test facility. This model was developed using the Structured Finite Volume (SFV) code [1,2], a general purpose computational fluid dynamics (CFD) code. The report provides details of the mathematical model and reviews the results obtained from the computations. This work was undertaken by Adaptive Research Corporation, for the Naval Facilities Engineering Command, under Contract Number N47408-91-C-1228.

1.2 Background

U. S. Navy aircraft engines completing maintenance or repair are operated in test cells where they must meet performance specifications before reinstallation. Test cells are fully enclosed, sound absorbing hangers. The principle function of these facilities is to provide repeatable, specific test conditions. In the process, the test cells must attenuate the jet and propeller noise sufficiently for personnel to carry out normal activities near the facility, cool and control the engine exhaust flow to the point that cell components are not damaged, and meet environmental standards.

The Navy has recently completed construction of the prototypes of a new generation of test cells designed for turboprop and turboprop engines. These test cells, located at the Marine Corps Air Facility, Camp Pendleton, California, and the Naval Air Station, Sigonella, Italy are scheduled to begin operating shortly.

1.3 Objective of the Study

The objective of this work is to provide a computational fluid dynamics model, employing the SFV code, of these prototype engine test cells for use in performance evaluation and for troubleshooting.

1.4 Outline of the Report

This report consists of six main sections. Following this introduction, Section 2 presents the geometric details of the turboprop test cell along with operating conditions. Next, Section 3 contains information about the mathematical basis of the numerical model. In Section 4, the numerical details of the model such as boundary conditions are presented. The results of the present study are presented in Section 5, and the conclusions drawn are given in the final section.

2. DESCRIPTION OF THE SIMULATED PROCESS

2.1 Geometry

Details of the geometry are provided in Appendix A. The engine with propeller is placed in a rectangular test bay. Air is drawn into the test bay through the plenum. This air, for the most part, then enters an orifice before contact with the propeller. The two rows of intake vanes, plenum, and orifice provide a smooth, low velocity, low angle of approach airflow to the propeller. Exhaust from the engine, along with some cooler test bay air entrained by the exhaust jet, leave the facility through the augmentor tube, discharging to a chimney and then to the atmosphere. The rest of the through flow is discharged through a set of baffles on the back side of the test bay. A set of acoustic baffles are also located at the top of the chimney.

The engine considered in this modeling effort is the T56 turboprop engine manufactured by Allison Corporation. The operating conditions for the engine are listed in Table 1. The ambient conditions outside the cell were 29.92 mm Hg ($1.01325 \times 10^5 \text{ N/m}^2$), 77 F (25 C), and a dry bicomponent atmosphere (76.83 wt per cent N_2).

TABLE 1. T56 Turboprop Engine Operating Conditions

Exhaust Flow	32.4 lb/s 14.7 kg/s
Fuel Flow	0.833 lb/s 0.375 kg/s
Average Exit Temperature	1100 F 600 C
Exhaust Composition (wt%)	0.7479 N ₂ 0.1411 O ₂ 0.8100 CO ₂ 0.3000 H ₂ O
Prop Speed	1021 RPM
Power Output	4591 SHP

3. MATHEMATICAL FORMULATION

3.1 The Structured Finite Volume (SFV) Mathematical Formulation

The SFV computer code is a general-purpose CFD code in wide use throughout government, industry, and academia. It can model two- or three-dimensional, laminar or turbulent, single- or two-phase fluid flows in arbitrary geometric flow domains. SFV solves the conservation equations for mass, momentum, and energy. The details of this solution procedure follow.

3.1.1 The Governing Equations

The flow field in a given geometry can be described by the conservation equations for mass, momentum, and energy. These equations can be expressed in the following form

$$\frac{\partial}{\partial t} + \nabla (\rho \nabla) = 0 \quad (1)$$

$$\frac{\partial(\rho \nabla)}{\partial t} + \nabla (\rho \nabla \nabla) = -\nabla p + \nabla (\mu_{\text{eff}} \nabla \nabla) \quad (2)$$

$$\frac{\partial(\rho \nabla)}{\partial t} + \nabla (\rho \nabla h) = \nabla \bar{q} + \mu_{\text{eff}} \nabla^2 + \frac{\partial p}{\partial t} + \nabla \nabla P \quad (3)$$

where

- ∇ — is the time-mean velocity;
- ρ — is the gas density;
- p — is the static pressure;
- μ_{eff} — is the "effective" viscosity;
- h — is the enthalpy;
- $\bar{q}$ — is the diffusive energy flux; and
- ϕ — is the dissipation function.

The effective viscosity is defined by the relation

$$\mu_{\text{eff}} = \mu + \mu_t ; \mu_t = C_\mu \rho \frac{k^2}{\epsilon}$$

where μ is the molecular viscosity and μ_t , the turbulent viscosity, is deduced by employing a turbulence model. SFV employs the two-equation k - ϵ turbulence model [3]. This treatment of turbulence requires the solution of two additional partial differential equations

$$\frac{\partial(\rho k)}{\partial t} + \nabla(\rho \nabla k) = \nabla \left[\frac{\mu_{\text{eff}}}{\sigma_k} \nabla k \right] + P_k - \rho \epsilon \quad (4)$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \nabla(\rho \nabla \epsilon) = \nabla \left[\frac{\mu_{\text{eff}}}{\sigma_\epsilon} \nabla \epsilon \right] + \frac{\epsilon}{k} \left[C_1 P_k - \rho C_2 \epsilon \right] \quad (5)$$

where

- k — is the turbulence kinetic energy;
- ϵ — is the dissipation rate of turbulence kinetic energy;
- $C_1, C_2, \sigma_\epsilon, \sigma_k$ — are model constants; and
- P_k — is the volumetric production rate of k , defined by

$$P_k = \mu_t \left[\frac{\partial V_i}{\partial x_j} + \frac{\partial V_j}{\partial x_i} \right] \frac{\partial V_i}{\partial x_j}$$

When single-phase three-dimensional flow is under consideration, the dependent variables that require solution are

- p — the static pressure;
- u — the horizontal-direction velocity component;
- v — the vertical-direction velocity component;
- w — the axial-direction velocity component;
- k, ϵ — turbulence quantities (described above);
- h — the static enthalpy;

C1 — the mass fraction of engine gases.

3.1.2 General Form of the Governing Equations

The governing equation for each dependent variable (φ) can be reduced to a single general form which can be succinctly represented in vector notation as

$$\frac{\partial(\rho\varphi)}{\partial t} + \nabla \cdot [\rho \nabla \varphi + \mathbf{J}_\varphi] = S_\varphi \quad (6)$$

The source term S_φ includes both sources and sinks of dependent variable (φ) plus any other terms beyond those which appear on the left-hand side of the equation.

Because of the generality of the SFV code, a standard gradient-diffusion law is provided for the diffusion flux, namely

$$\mathbf{J}_\varphi = -\Gamma_\varphi \nabla \varphi \quad (7)$$

Where diffusion of the conserved quantities does not follow the standard form of Equation 7, compensating terms are added to the source term S_φ . In that equation, Γ_φ is the exchange coefficient for the variable φ , defined as

$$\Gamma_\varphi = \frac{\mu_{\text{eff}}}{\sigma_{\varphi,\text{eff}}} \quad (8)$$

where $\sigma_{\varphi,\text{eff}}$ is the effective Prandtl/Schmidt number governing diffusive transport of the variable φ .

The exception to the general form given by Equation 6 is the continuity (mass conservation) equation. The pressure, p , has been classified as a dependent variable, but does not appear as the subject of a transport equation. Instead, the pressure is associated with the continuity equation which can be manipulated to derive a pressure-correction equation. SFV employs a variant of the SIMPLE pressure-correction procedure [4]. The transport coefficient Γ_φ and source term S_φ are provided for each dependent variable in Table 2.

TABLE 2. Transport Equations and Turbulence Model Constraints

	φ	Γ_φ	S_φ
Continuity	1	0	0
Momentum:			
x-direction	u	μ_{eff}	$-\frac{\partial p}{\partial x}$
y-direction	v	μ_{eff}	$-\frac{\partial p}{\partial y}$
z-direction	w	μ_{eff}	$-\frac{\partial p}{\partial z}$
Kinetic Energy	k	$\frac{\mu_t}{\sigma_k}$	$P_k - \rho\epsilon$
Dissipation Rate	ϵ	$\frac{\mu_t}{\sigma_\epsilon}$	$\frac{\epsilon}{k} (C_1 P_k - C_2 \rho\epsilon)$
Enthalpy	h	$(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t})$	0
Concentration	C	$(\frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t})$	0

$C_\mu = 0.09,$ $C_1 = 1.44,$ $C_2 = 1.92,$ $\sigma_k = 1.0,$ $\sigma_\epsilon = 1.314$

3.1.3 General Form of the Finite-Domain Equations

Integration of the generalized conservation equation (6) yields a finite-difference analog for each dependent variable φ , for each control cell into which the flow geometry is divided. The integration process is carried out, and the terms in the finite-domain equations are

assembled in the manner described in many standard texts on numerical analysis [5]. For each simulation performed, the SFV program will assemble and then solve a set of simultaneous coupled equations of the general form (for point P)

$$a_P \phi_P = a_E \phi_E + a_W \phi_W + a_N \phi_N + a_S \phi_S + a_H \phi_H + a_L \phi_L + S_\phi \quad (9)$$

where the subscripts E, W, N, S, H, and L represent the neighbor points in space. The coefficients, a_N and others, represent the effects of convection and diffusion. S_ϕ is the linearized source term. The central coefficient a_P is defined as

$$a_P = a_E + a_W + a_N + a_S + a_H + a_L \quad (10)$$

The set of equations (9) are assembled and solved for each dependent variable in sequence. The solution of the flow field then proceeds iteratively to steady-state.

3.1.4 Auxiliary Relations

To complete the mathematical formulation of a problem the specification of additional information is required: namely, fluid properties and boundary conditions.

SFV allows the arbitrary specification of fluid properties. The models resulting from this study will employ fluid properties typical of the conditions present in the Naval Air Station, Sigonella, Italy test cells. SFV also allows several different types of flow and thermal boundary conditions. These will be appropriately chosen so as not to sacrifice the fidelity of the models and also to allow for easy modification in the course of the parametric studies.

4. NUMERICAL DETAILS AND PHYSICAL MODELS

4.1 Introduction

This section of the report describes the various aspects of the modeling method necessary to simulate the flow and heat transfer inside the test facility. First, the computational grid employed in this simulation is described. Next, the physical properties are defined. This is followed by a discussion of the boundary conditions used in the computational model.

4.2 Grid Description

The first step in setting up a numerical model is the specification of the computational grid. For this study a three dimensional Body-Fitted Coordinate (BFC) grid was employed. The grid for this modeling effort is quite complex. In order to simplify the grid generation, a procedure has been developed to allow for relatively easy modifications of the test cell configuration.

In this method the user specifies various geometric quantities. The standard input files then create a set of data files for the EasyMesh [6] program which is a two dimensional BFC package. The user must then run this grid generation package to create a set of two dimensional grids. The standard input files are then re-executed to produce the final grid by stacking, blending, and rotation of the two dimensional grid planes.

The final computational grid for this study is shown in Figure 1. This grid has 34 cells across the test bay, 36 cells in the vertical direction, and 75 computational cells in the axial direction. The grid is clustered in areas of geometric changes, areas of expected high shearing action, and areas of heat transfer. In Figures 2 through 4 outlines of the internal components are presented.

4.3 Properties of the Fluid

The properties of the gases mixture inside the test cell were calculated based on the local conditions. The local heat capacity was computed using the gas mass fraction and the cell

temperature using JANNAF [7] polynomial expressions. The gas density was calculated using the ideal gas law (requiring local pressure, molecular weight, and temperature). The laminar kinematic viscosity was assumed to have a constant value of $1.08 \times 10^{-4} \text{ ft}^2/\text{s}$ ($1.0 \times 10^{-5} \text{ m}^2/\text{s}$). The turbulent viscosity, which tends to dominate, was calculated from the local turbulence quantities.

Heat transfer through the augments tube wall required the specification of the material thermal conductivity. Inside the room the thermal conductivity of mineral fiber with a value of 0.22 Btu/ft-s-R (0.38 W/m-K); was used. In the chimney the conductivity of steel was used. The value of steel was taken to be 26 Btu/ft-s-R (45 W/m-K).

4.4 Boundary Conditions

For this model, the boundary conditions requiring specifications were

- the engine,
- the propeller,
- inlet into test cell,
- outlets of test cell and chimney,
- wall boundaries, and
- heat transfer through the augments wall.

These boundary conditions and sources are discussed in the following sections.

4.4.1 Engine

The engine is modeled as a hollow cylindrical shape with a variable cross sectional area. Inside this region a plate of computational faces were blocked. On the intake side a mass sink is applied which corresponded to the exit mass flow minus the fuel intake.

On the exhaust side a mass source was applied to account for the engine mass flow. Other boundary conditions applied the appropriate sources for the momentum, enthalpy, turbulence, and concentrations. The exit turbulence quantities were calculated assuming a

15 per cent turbulence intensity from the following equations:

$$k = 1/2 (vI_t) \quad (11)$$

$$\epsilon = 0.1643 k^{1.5} / l_m \quad (12)$$

where

I_t — is the turbulence intensity and

l_m — is the mixing length (based on a gap distance or inlet radius)

4.4.2 Propeller

The propeller was modeled using an actuator disk model, where in all of the net results of the propeller (wash, swirl, turbulence) are modeled as source terms in the momentum and turbulence equations. The three aspects of the propeller which were modeled were thrust (axial flow generation), swirl (swirling flow generation) and turbulence (turbulence generation by the whirling blades). The actual methodology of modeling each of these three physical characteristics is now detailed.

4.4.2.1 Thrust

The thrust of the propeller was modeled as an momentum source in the axial-direction momentum equation. The actual expressions used were supplied by Hamilton Standard Corporation, the manufacturer of the propeller being modeled. The performance of a propeller is characterized by correlating the ratio of the thrust coefficient (C_T) versus the power coefficient (C_P). These coefficients are defined as:

$$C_P = P / \rho n^3 D^5 \quad (13)$$

with

P — power absorbed by propeller,

ρ — reference fluid density,

n — rotational speed of propeller, and

D — diameter of propeller:

$$C_T = T / \rho n^2 D^4 \quad (14)$$

with

T — thrust produced by the propeller.

All quantities are assumed to be in consistent units. The propeller is characterized by a plot of C_T/C_P versus C_P . This plot was also supplied by Hamilton Standard (see Appendix B). Thus, the user of the numerical model supplies shaft horse power SHP (which gives power), RPM (which gives n), and prop diameter. The code converts all of these to MKS units and evaluates a C_P . The prop characteristic curve is coded into GROUND by data statements and the code interpolates as required to generate C_T/C_P . With the evaluation of C_T the thrust is calculated from Equation 14.

The average axial induced velocity of the propeller is given as

$$W_p = \sqrt{\frac{T}{2\rho A_p}} \quad (15)$$

with

A_p — area of whirling prop ($\pi D^2/4$).

The code computes W_p as a check on prop model performance only, it is not otherwise needed. The thrust per unit-mass-of-air within the envelope of the propeller is given as:

$$T/\text{mass} = T/\rho V d = T/\rho A_p \delta_p = 2W_p^2/\delta_p \quad (16)$$

where δ_p is the thickness of the grid cells in the axial direction where the prop is located. This is how the source is actually implemented in the code for numerical consistency with the rest of the discretized equations.

4.4.2.2 Swirl

The integrated swirl generated consumes a certain percentage of engine power. This percentage is calculated by having the user specify the percentage of total power which is assumed to be wasted generating turbulences (%k). The percentage used to generate thrust (%w) is known from the thrust calculation as follows. The power used to generate thrust, P_T is given as:

$$P_T = TW_p \quad (17)$$

Then the percent of power used in thrust is $P_T/P \times 100$.

It is assumed that all engine power absorbed by the prop is used to do one of three things, namely thrust, swirl, and turbulence, and knowing %w, and %k, the percentage used for swirl (%u) is calculated. The power used by the swirl (P_u) is %u*P. It is assumed that the power used in swirl generation is distributed evenly over the face of the propeller. This power is equal to the torque used in swirl generation times the angular speed of the prop:

$$P_u = Tn \quad (18)$$

This torque is equal to the local force of the prop on the air in the direction of swirl times the distance from the center of the prop to the location under consideration.

$$T = Fr \quad (19)$$

This force is assumed to be constant in r; thus

$$P_u = Frn \quad (20)$$

or F , the magnitude of the propeller on the air locally, is given as

$$|F| = P_u / r n \quad (21)$$

The code computes local unit vectors for the prop swirl force F , then supplies $F \cdot \hat{e}_u$ as the local swirl source for the u and v momentum equations, where $\hat{e}_u$ is the local unit vector for the u and v momentum equations, available from the code.

4.4.2.3 Turbulence

The turbulence of the propeller is assumed to be proportional to local prop speed squared:

$$S = C v^2 \quad (22)$$

The power used to generate, turbulence, ($P_k = \%k * P$) is assumed to be distributed over the radius of the propeller such that the integrated power dissipated is equal to P_k . Thus

$$P_k = \int_0^{2\pi} \int_0^R C v^2 r dr d\theta \quad (23)$$

The constant C is evaluated by completing the integral and solving for C to give

$$C = \frac{64 P_k}{2\pi n^2 D^4} \quad (24)$$

and the local turbulence sources are evaluated as

$$S_k = C(rn)^2 \text{ (turbulence kinetic energy)} \quad (25)$$

and

$$S_\epsilon = S_k \frac{\epsilon}{k} \text{ (dissipation of turbulence kinetic energy).} \quad (26)$$

4.4.3 Inlet

At the inlet a fixed pressure was applied which corresponded to the ambient pressure. From this and the other boundary conditions the mass flow was computed. Since this mass flow varies during the solution process, the momentum source was continually updated according to the computed inflow. The enthalpy and concentrations were considered to be ambient. A turbulence intensity of 2 per cent was used to calculate the incoming turbulence values. A constant velocity of 3.3 ft/s (1.0 m/s) was used in the turbulence calculations.

The baffles located at the inlet and chimney were not directly modeled. These effects were modeled by calculating the momentum loss through the baffles from

$$\Delta p = 0.5 K \rho v^2$$

where

K is a loss coefficient.

Supplied with pressure drop across the baffles along with gas density and velocity an accurate determination of K is possible. Pressure data from the turboshaft test cell indicated a k-loss factor of 0.2. Because the pressure drop in the test cell was under predicted a value of 1.0 was used in this simulation.

4.4.4 Outlet

At the two 'outlets' a fixed pressure was used. If no recirculation occurs this is all that is required. However, in the chimney, the possibility of recirculation at the exit was extremely great. Momentum and energy sources were supplied if and when recirculation at the exit plane was calculated. The momentum source was based on the amount of mass being brought into the domain and the energy source was based on ambient gas conditions.

4.4.5 Wall Boundaries

At the wall boundaries the wall function approach [3] was used to account for momentum losses. This condition is not applied directly to the wall but rather at a point outside the viscous sublayer, where the logarithmic law of wall prevails. This is where turbulence is assumed to be in local equilibrium (i.e., the generation and dissipation of turbulence energy

are equal). Heat transfer was not considered except through the augments tube wall.

4.4.6 Heat Transfer through Augments Tube

To compute the heat transfer through the augments tube, a composite heat transfer treatment was used. Surface heat transfer coefficients on both surfaces (inside and outside) were calculated using the standard SFV wall friction treatment. The overall heat transfer conductance between the fluid cell adjacent to the inner wall and the one adjacent to the outer wall was evaluated using these two surface coefficients (in the form of a heat transfer resistance) and the assumed thermal resistance through the tube wall. This last resistance was evaluated using a conductivity of insulation and thickness of insulation provided by the Navy. In the chimney, where there was no insulation, the conductivity and thickness of the augments tube wall was used for the solid contribution to the overall heat transfer conduction.

5. DISCUSSION OF RESULTS

This report presents the results obtained from one computational run. At the time of this report there were no experimental data for comparison purposes.

5.1 Velocity

The velocity vectors are presented in Figures 5 through 10. A cross sectional view to the vectors are shown in Figure 5. This plot provides a good overall display of the major flow patterns within the calculation domain. Two plots of velocity around the propeller are shown in Figures 6 and 7. The first one again shows a side view. In this plot the air movement through the orifice and propeller is presented. The effect of the reduction gear is noted as the flow of air is forced around the unit. Large recirculation zones at the top and the bottom of the test cell downstream of the prop are also shown. In Figure 7 (x-view) the velocity vectors one computational cell down stream of the propeller are plotted. The highest magnitude of velocity is located at the center of the prop. This is from the assumption that a free vortex type propeller was used as noted in Section 4.4.2

Figure 8 shows the velocity vectors at the end of the engine and start of the augments tube. There is entrainment of surrounding gases into the tube at this cross section. A later temperature plot will indicate if there is any flow coming out of the augments opening. The maximum velocity of the engine is 721 ft/s (218 m/s). The velocity in the chimney section is shown in Figure 9. The augments tube is circular down the full length. The axial momentum propels the gases inside the augments tube to the aft side of the bend section. The gases do not have enough time to spread the entire cross section of the chimney. This causes a recirculation zone at the exit of the calculation.

The final vector plot (Figure 10) shows a top side view of the velocity vectors entering into the test cell. The entrance velocity is approximately 23 ft/s (7 m/s). The inlet section of the turboprop facility covers the full width and height of the test bay.

5.2 Temperature

Temperature plots are provided in Figures 11 through 15. The overall temperature field is shown in Figure 11. In this model the engine is the only heat source. The temperature around the exit of the engine is shown in Figure 12. The exit temperature of the engine is 1100 F (600 C). This temperature reduces as the engine gases mix with entrained air and

propeller wash in the augments tube as shown in Figure 12. The plot of temperature at the exit of the augments tube is given in Figure 13. At this point the temperature ranges from 602 F to 626 F (317 C to 330 C). The average temperature is approximately 615 F (324 C). The narrow range in temperature indicates a good level of mixing in the augments tube. In Figure 14 the temperature at the exit of the chimney section is plotted. A small zone of recirculation is noted by a region where the temperature is 83 F (28 C) which is close to ambient conditions of 77 F (25 C). During the iterative solution process, intermediate unconverted results indicated that a flow was exiting the entrance of the augments tube. In Figure 15 a plot of the temperature in the first cell in the augments lip is provided. It shows no recirculation of hot gases out of the lip region.

5.3 Other Quantities

In Figure 16 the pressure field located approximately 2 ft (0.6 m) from the right side looking into the cell is displayed. In this plane there is both cell depression and cell pressurization. The values range from -1.2 to 1.4 inches of water.

Figure 17 shows the turbulence kinetic energy (TKE) at the exit of the engine. The largest values occur in the shear region between the top of the engine and the top of the augments lip. In Figure 18 the TKE is plotted one cell down stream of the propeller. The largest values occur near the tip of the prop.

There was one global number calculated from this model. This was the pumping ratio of the engine. The calculated value was 1.08.

Heat transfer through the augments tube was at the rate of 1.6 BTU/s (1680 J/s). Approximately 55 per cent of this occurred in the uninsulated section of the chimney.

6. CONCLUSION

A 3-dimensional turboprop model of a Naval test cell facility has been developed. This model allows the user to investigate various parameters of interest including variations in geometry, ambient conditions, and boundary conditions. The results of this study show

- an engine pumping ratio of 1.08;
- cell depression and pressurization that range between -1.2 and 1.4" of water;
- an augmenter tube average exit temperature of 615 F; and
- recirculation at the exit of the chimney section.

There were no experimental values to compare with predicted values.

7. REFERENCES

1. "The SFV Beginner's Guide, "Adaptive Report 100, 1992.
2. "The SFV Reference Manual", Adaptive Report 200, 1992.
3. Launder, B. E. and Spalding, D. B., Mathematical Models of Turbulence, Academic Press, 1972.
4. Patankar, S. V. and Spalding, D. B., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," Int. Journal of Heat and Mass Transfer, pp 1787-1806, 1972.
5. Patankar, S. V., Numerical Flow and Heat Transfer. Hemisphere, New York, 1980.
6. Daley, P. L. and Owens, S. F., "User Manual for Grid Generation Program", CHAM Report 2008/2, November 1988.
7. Gordon, S. and McBride, B., "Computer Program for Calculation of Complex Chemical Equilibrium Compositions", NASA SP-273, 1971.

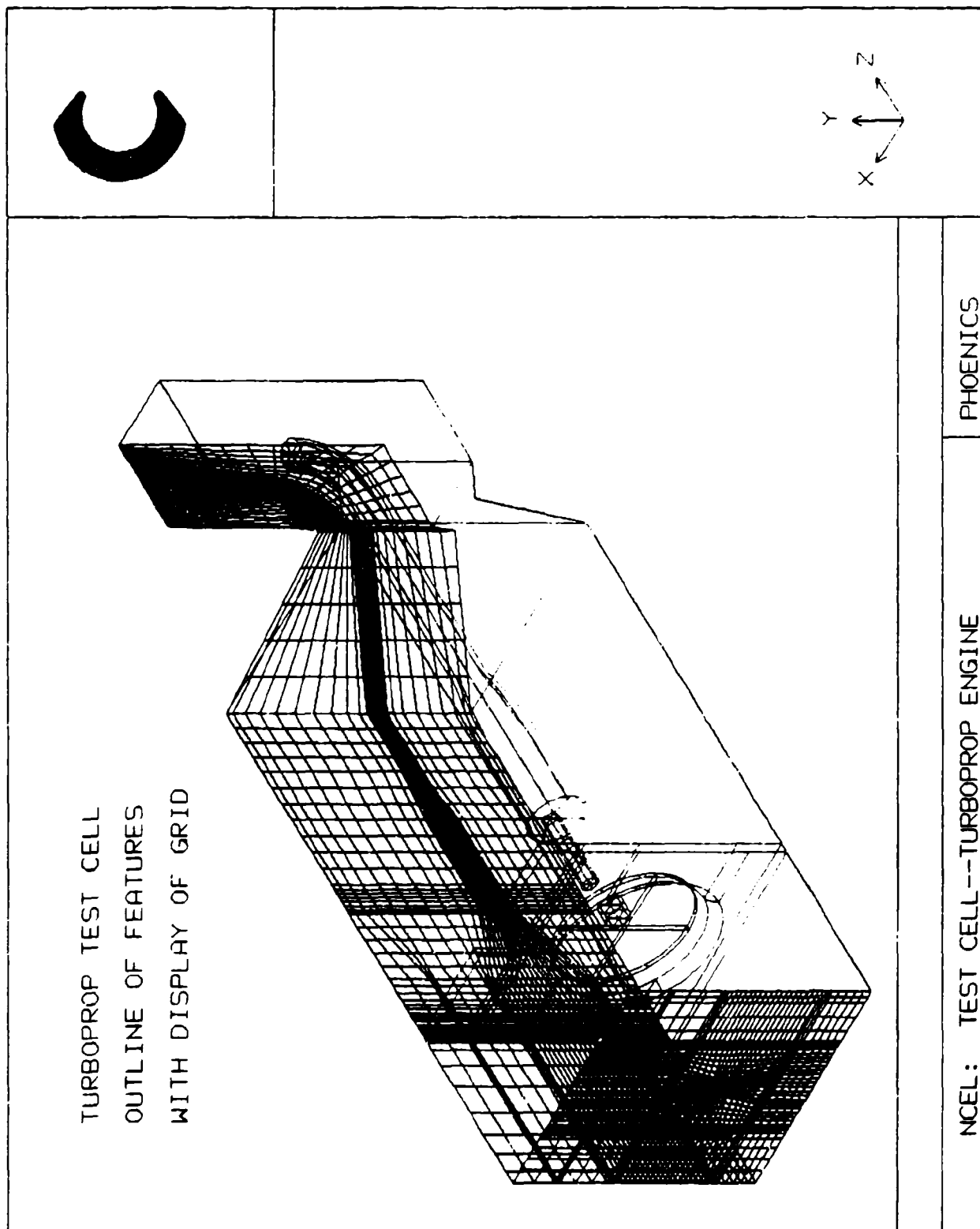


Figure 1 Computational Grid

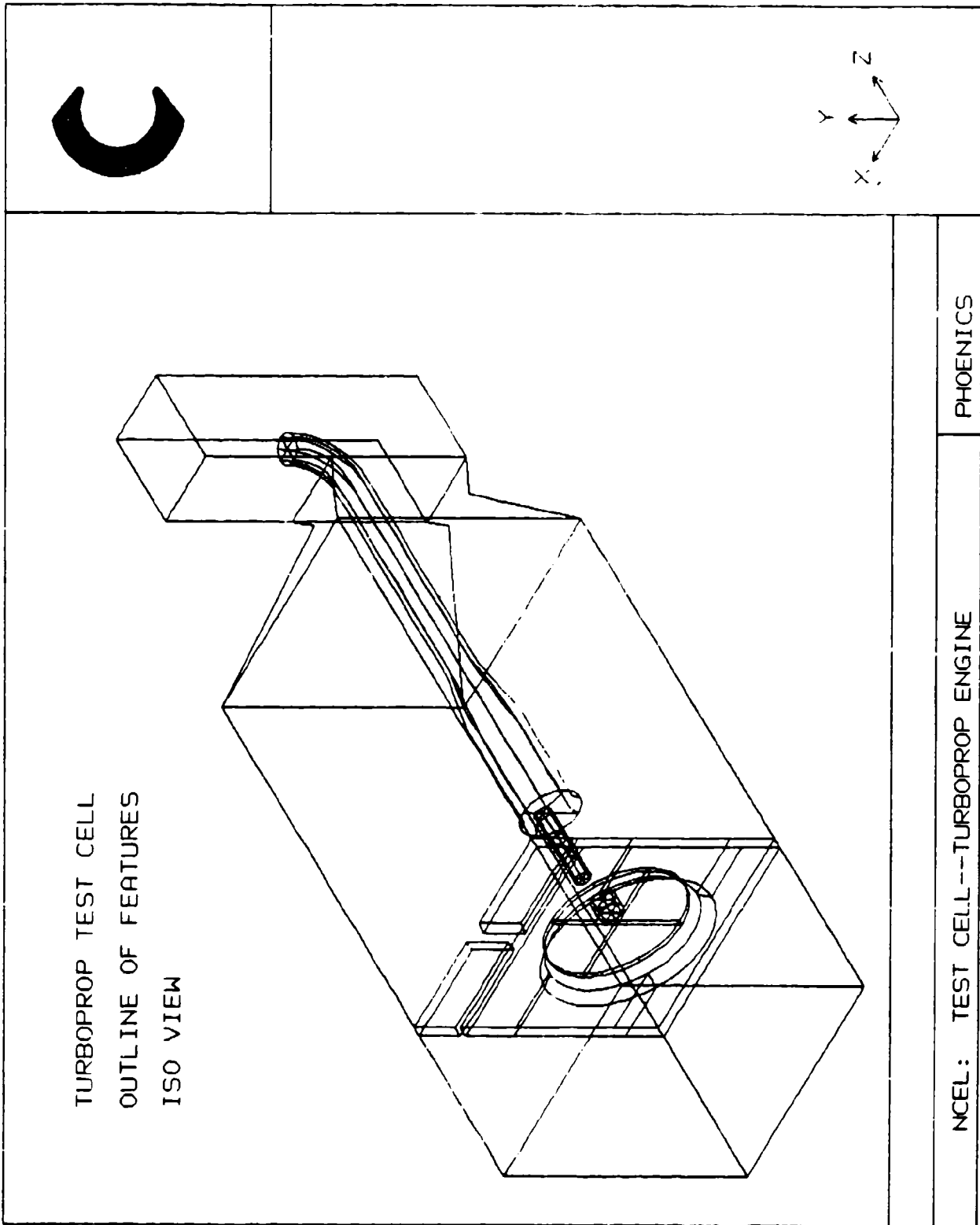
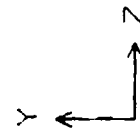
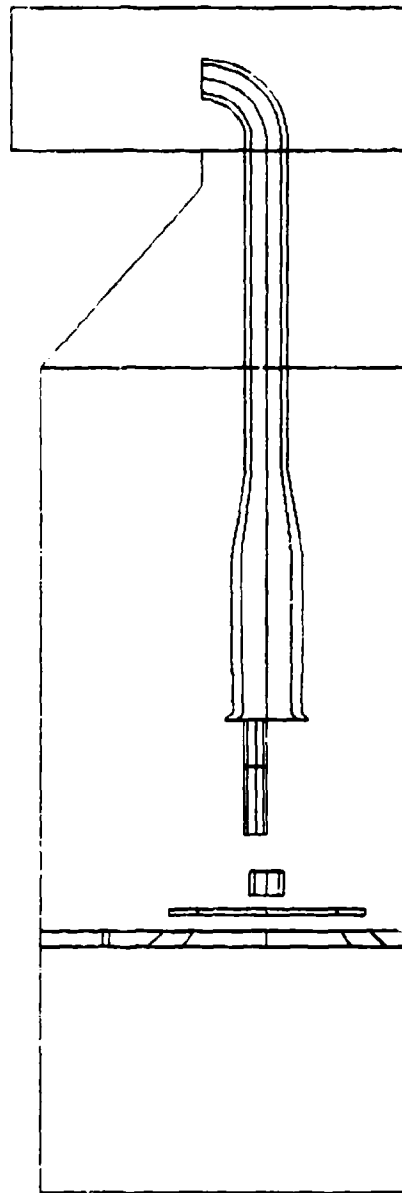


Figure 2 Outline of Features (ISO View)

TURBOPROP TEST CELL
OUTLINE OF FEATURES
SIDE VIEW



PHOENICS

NCEL: TEST CELL--TURBOPROP ENGINE

Figure 3 Outline of Features (Side View)

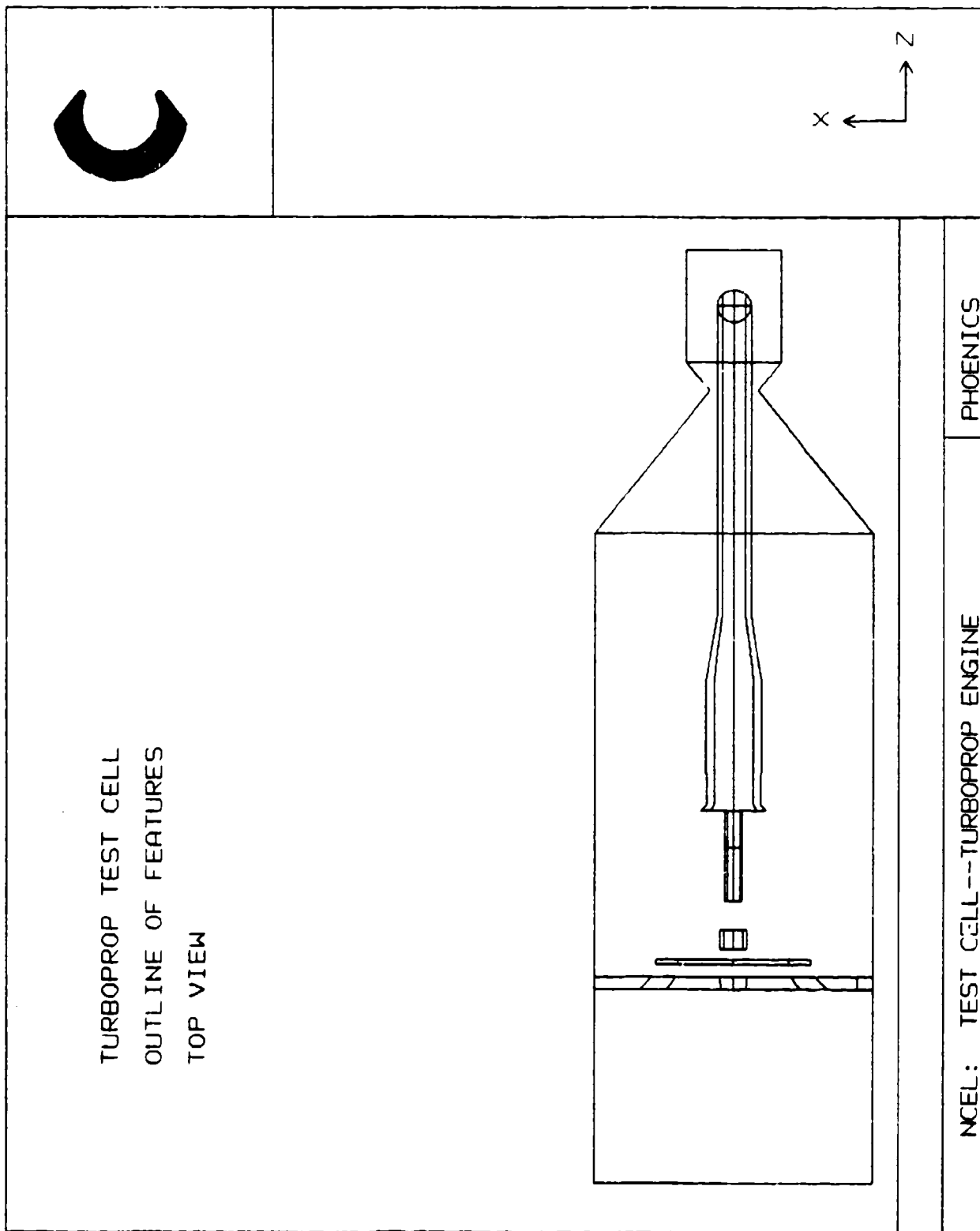


Figure 4 Outline of Features (Top View)

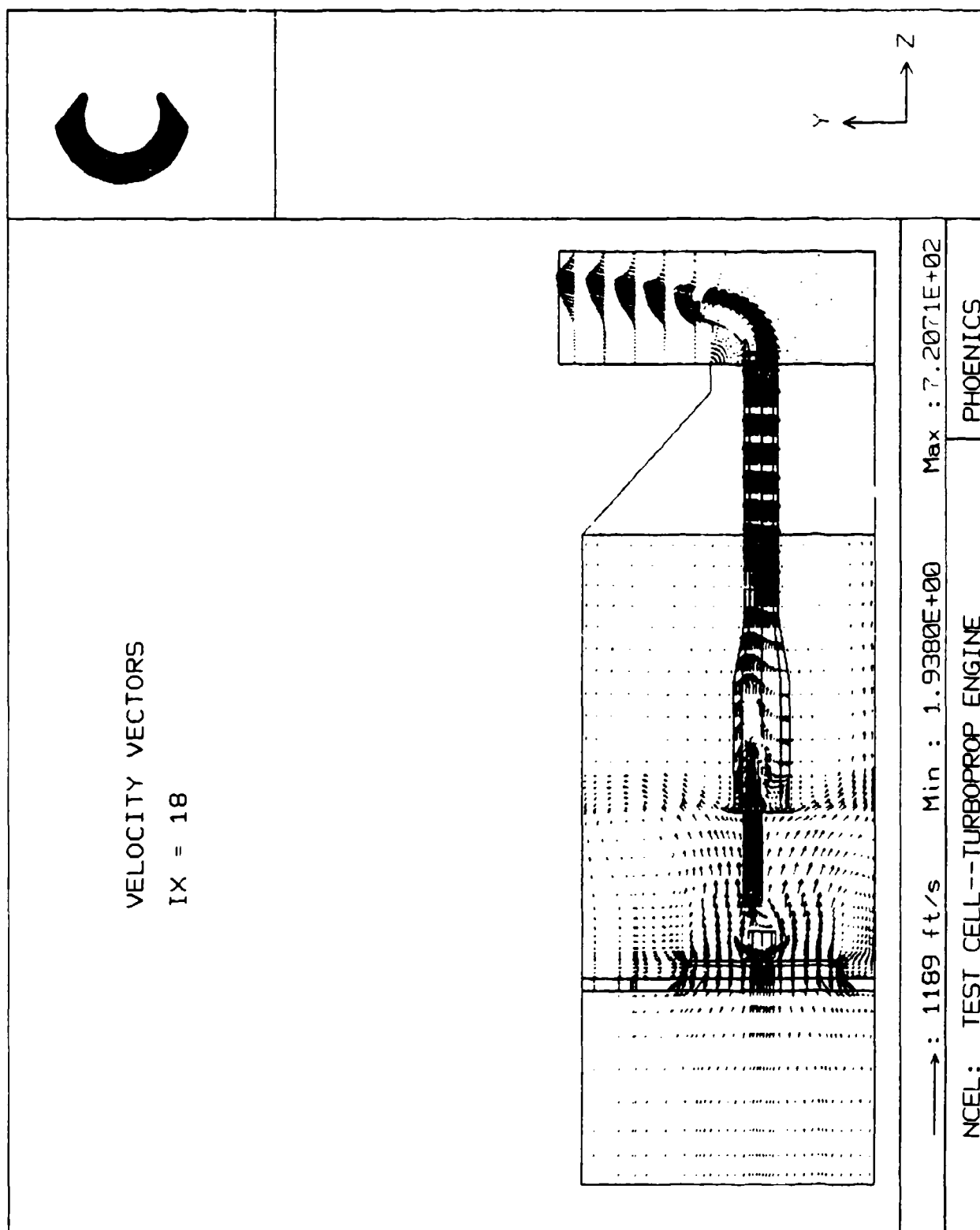


Figure 5 Velocity Vectors (Vertical Plane Through Engine)

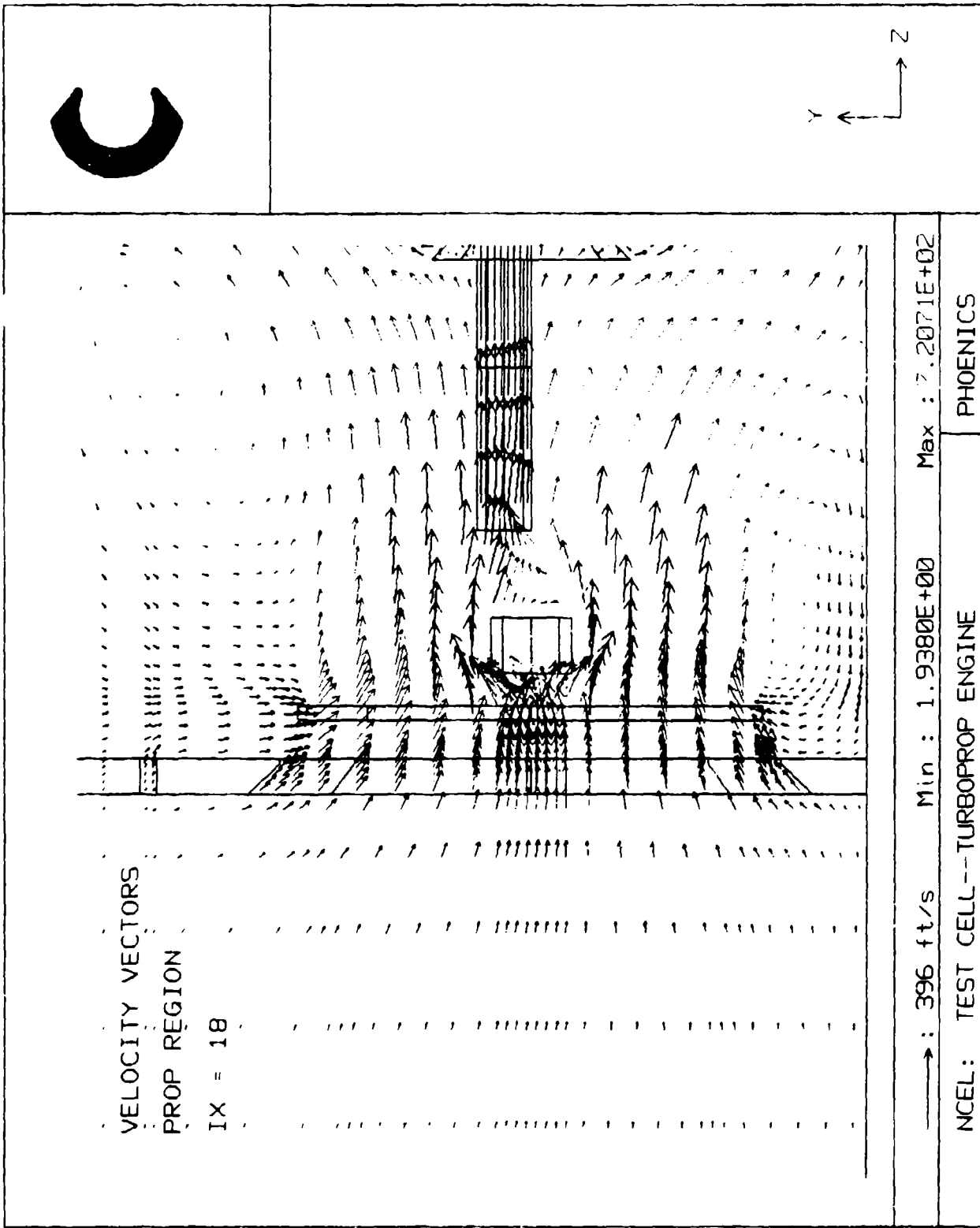


Figure 6 Velocity Vectors Prop Region (Vertical Plane)

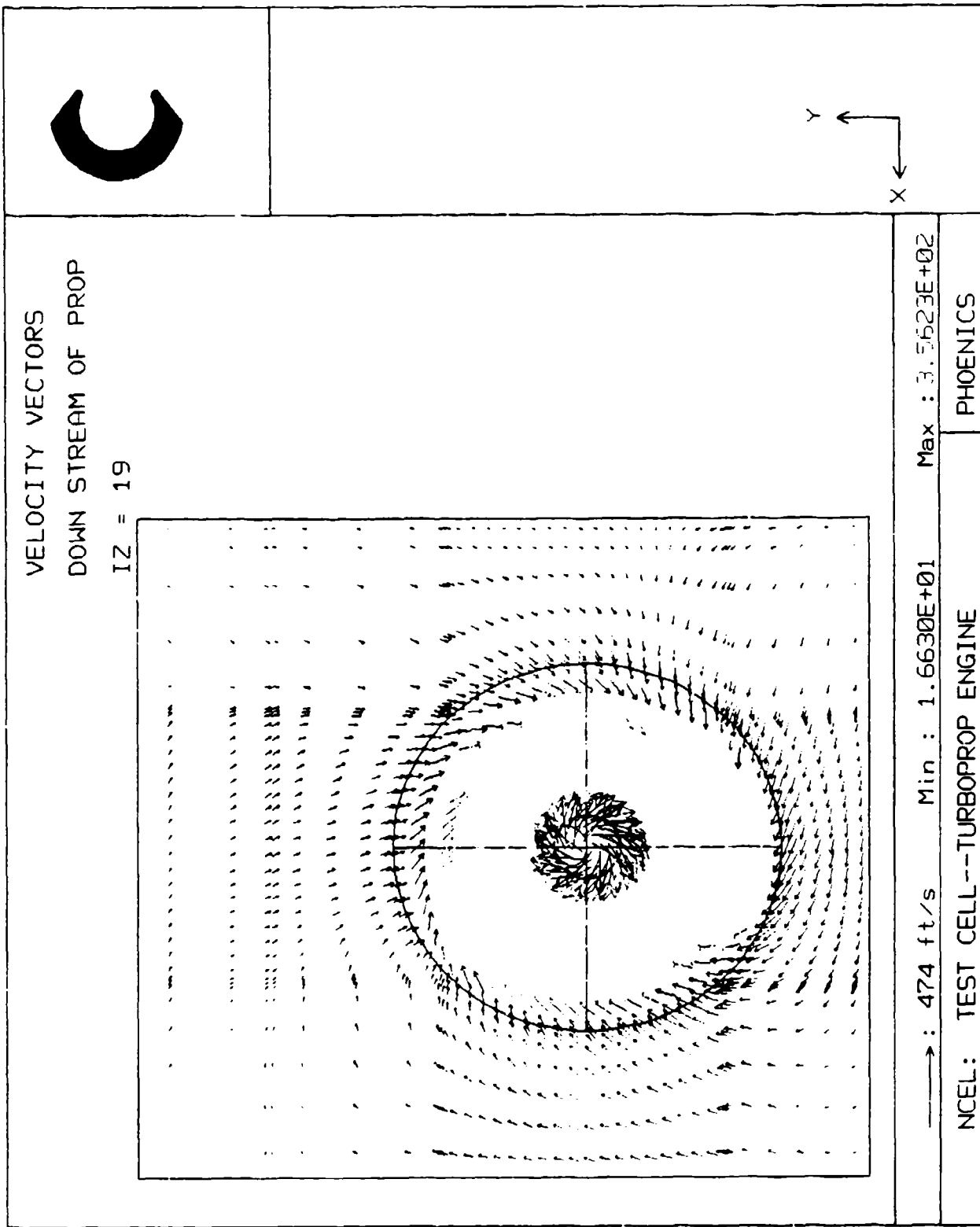


Figure 7 Velocity Vectors Down Stream of Prop (Cross Plane)

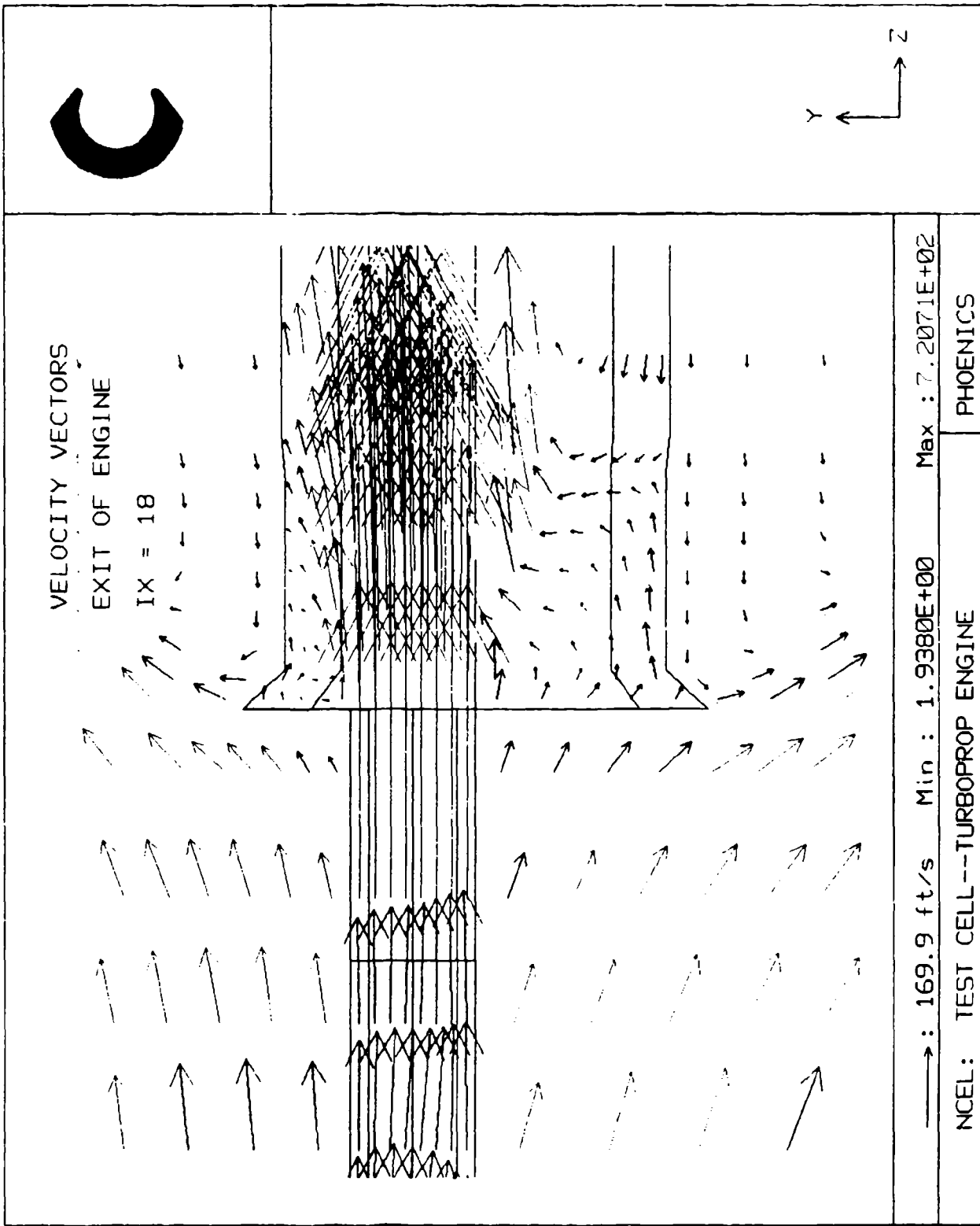


Figure 8 Velocity Vectors Exit of Engine (Vertical Plane Through Engine)

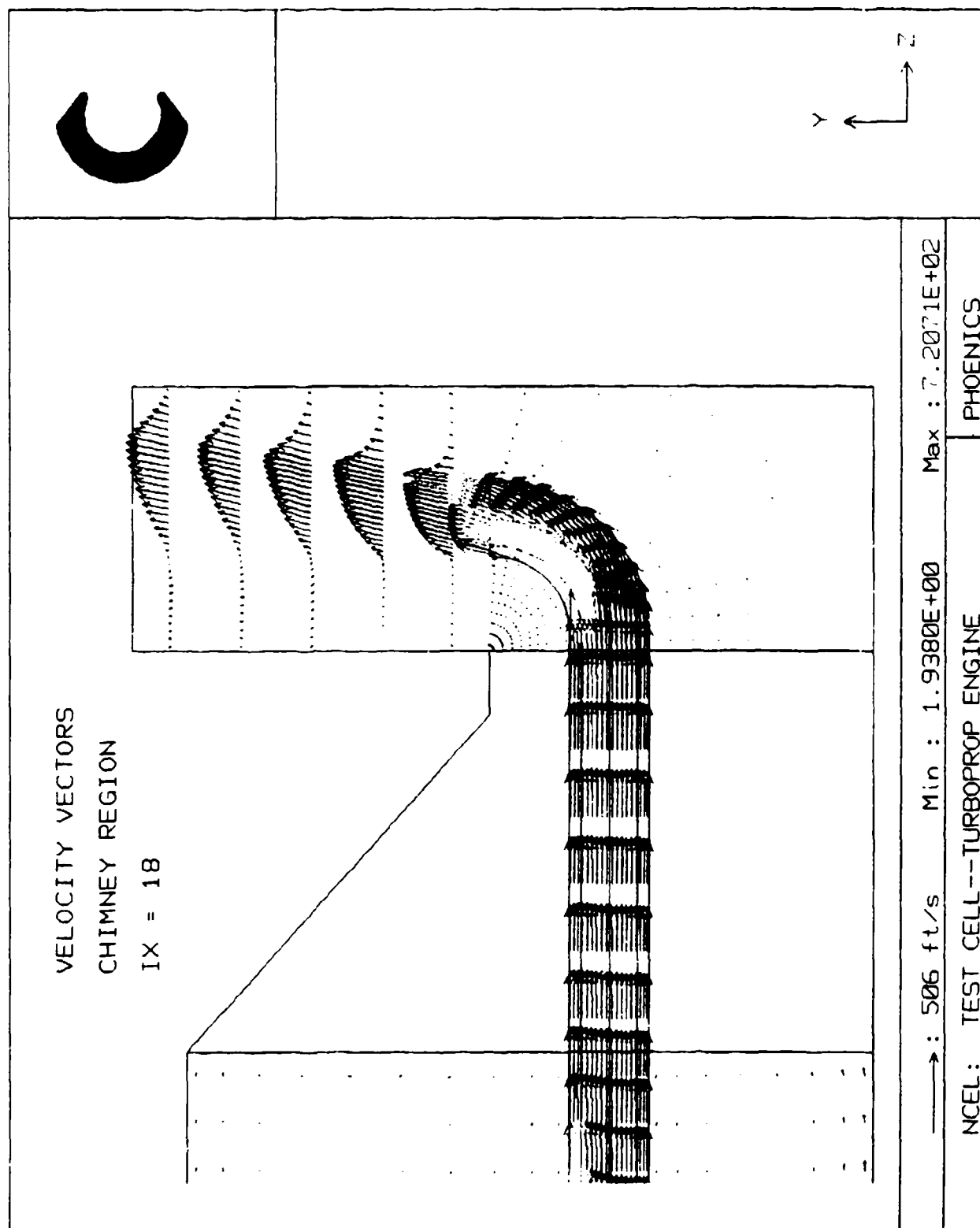


Figure 9 Velocity Vectors Chimney Section (Vertical Plane)

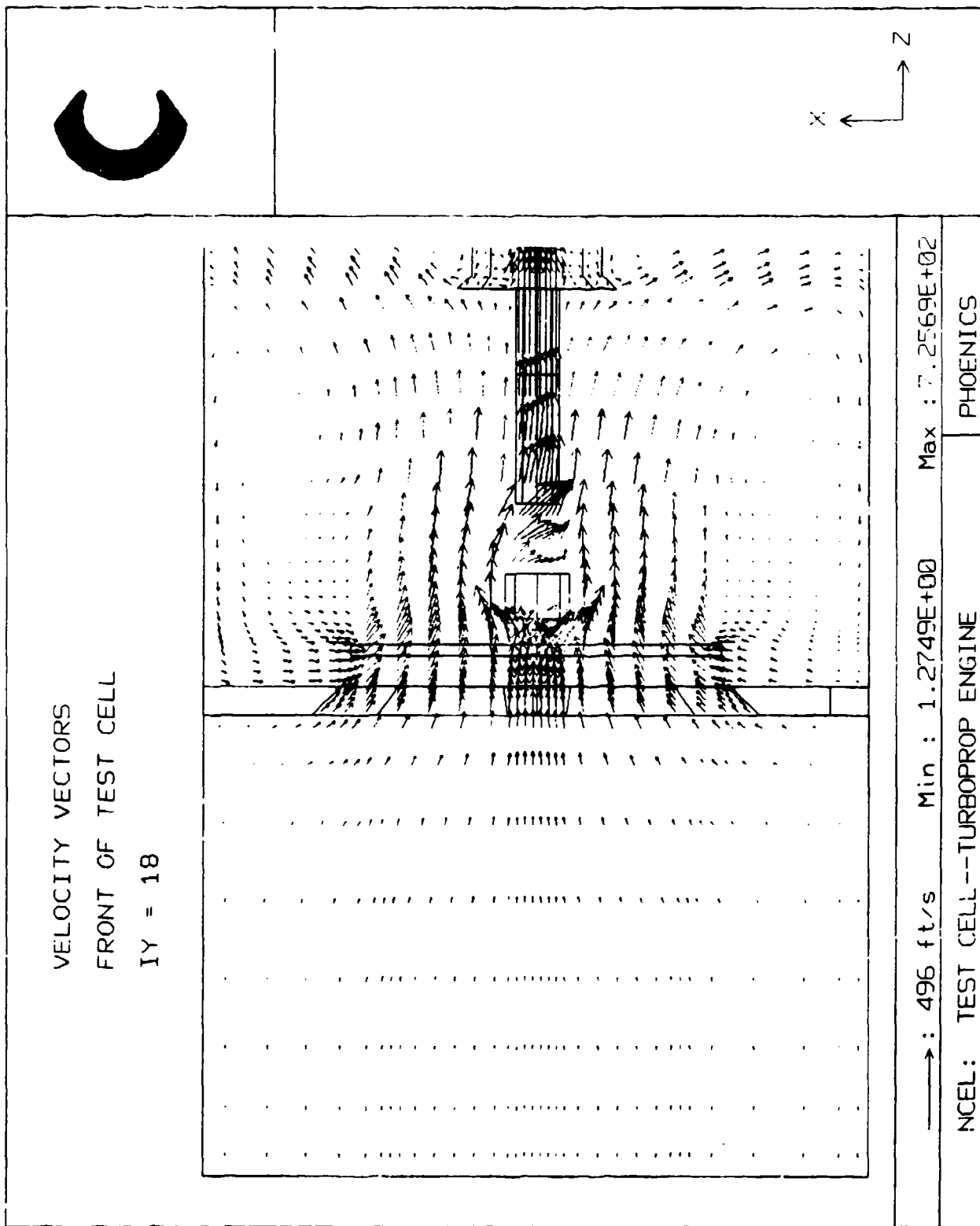
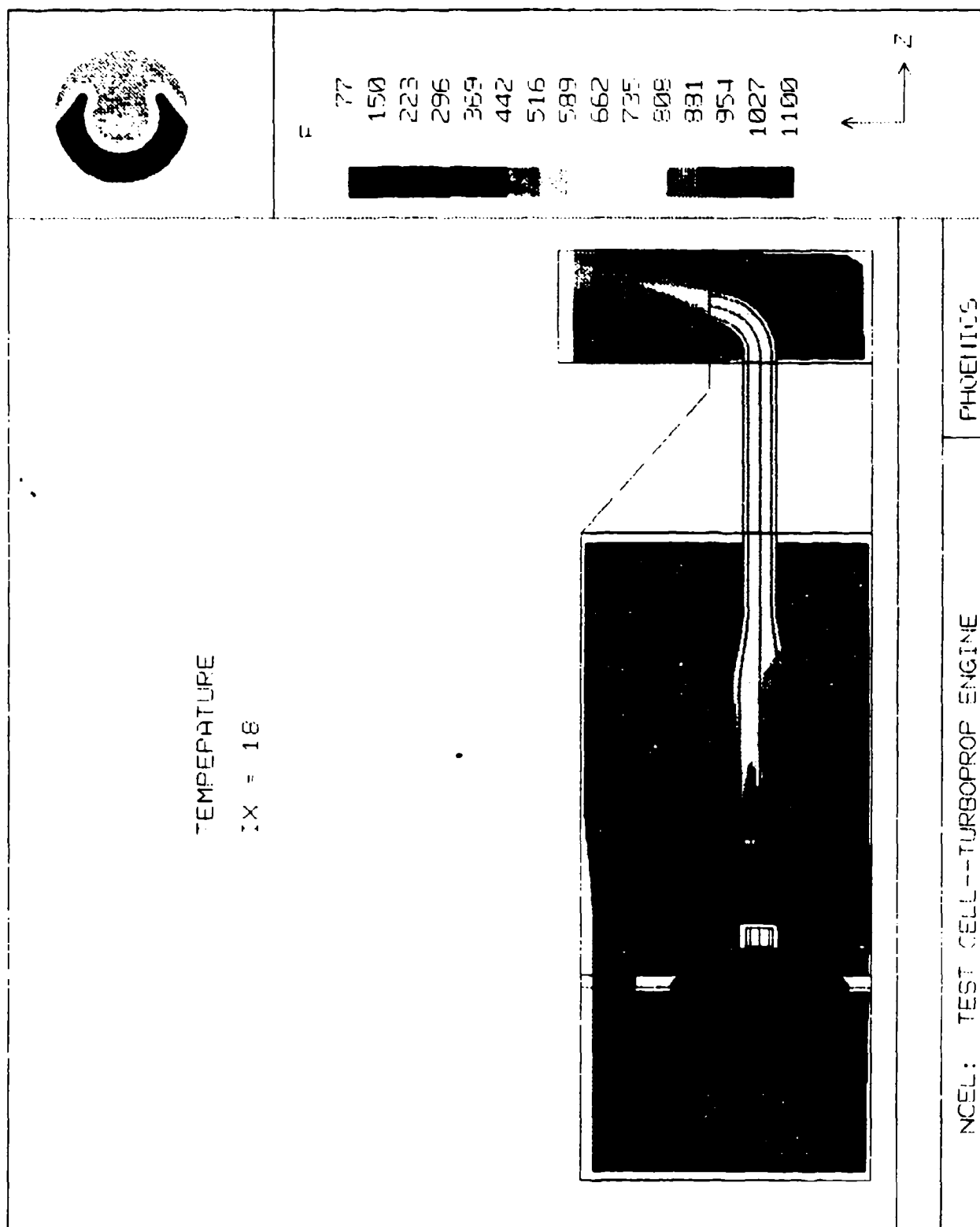
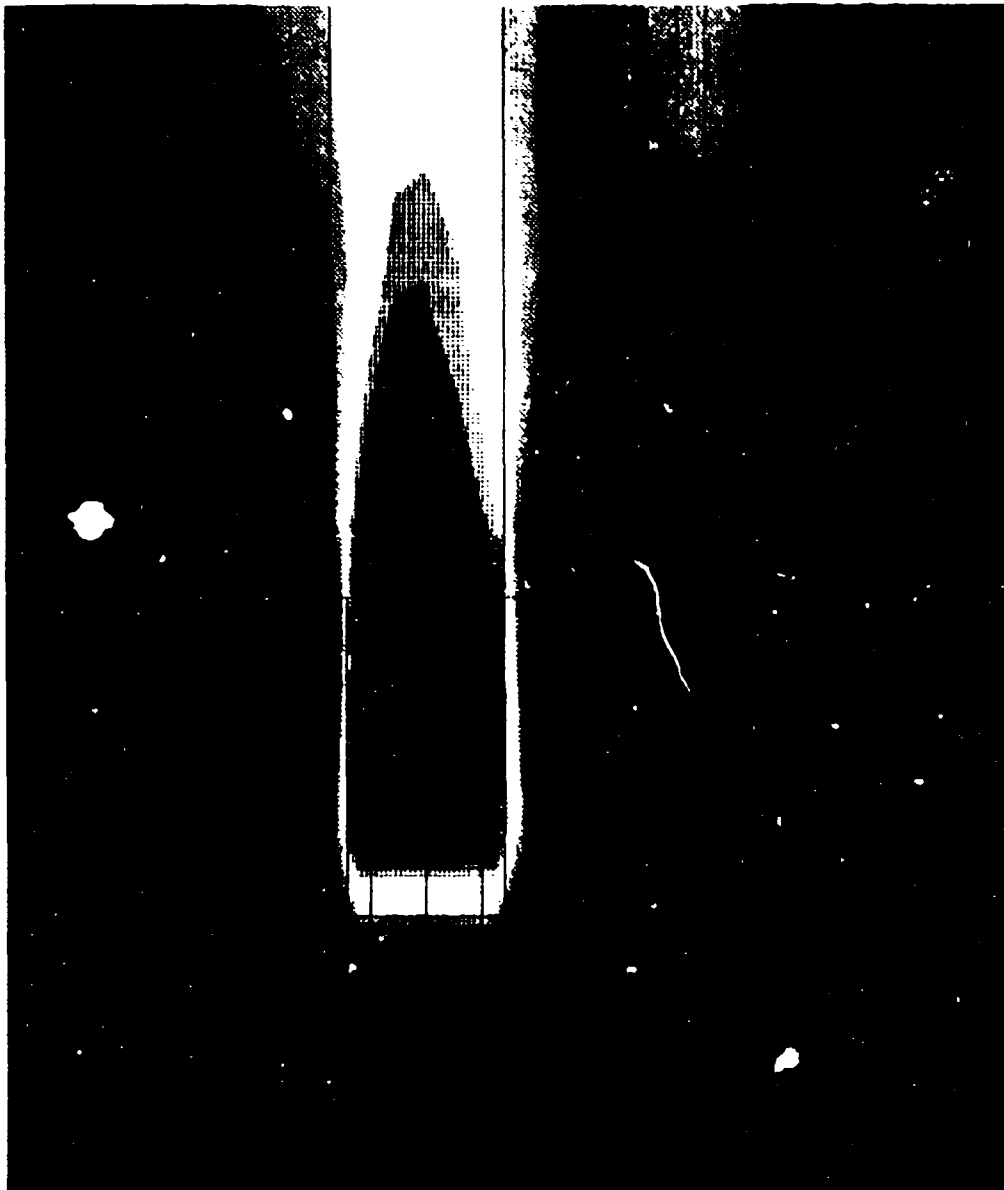


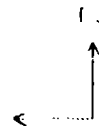
Figure 10 Velocity Vectors Front of Test Cell (Horizontal Plane Through Engine)



TEMPERATURE EXIT OF ENGINE IN = 18



77
150
222
296
359
442
516
589
662
735
808
881
954
1027
1100



NCEL: TEST CELL--TURBOPROP ENGINE PHOTOGRAPH

Figure 12 Temperature Exit of Engine (Vertical Plane Through Engine)

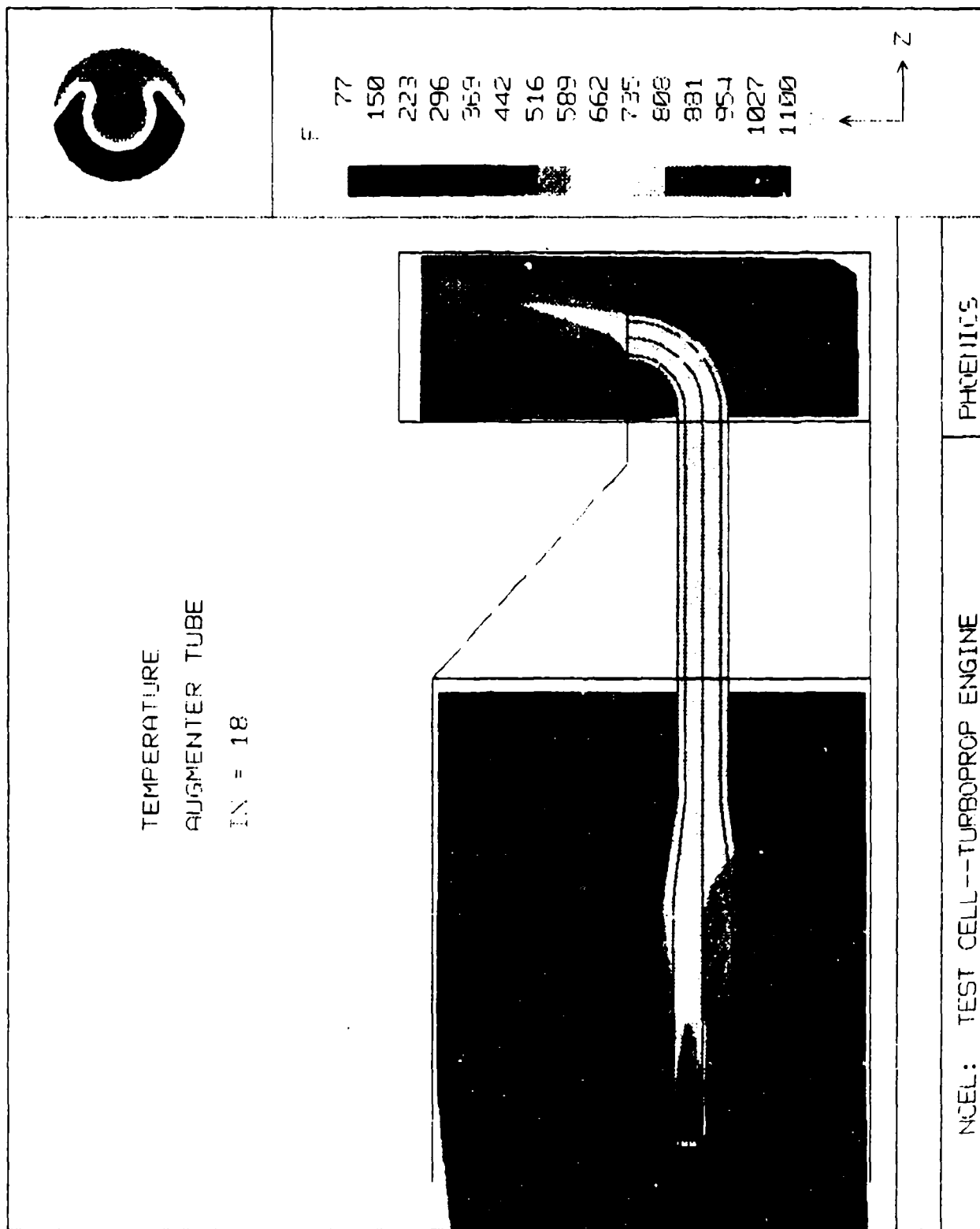


Figure 13 Temperature Augmenter Tube (Vertical Plane)

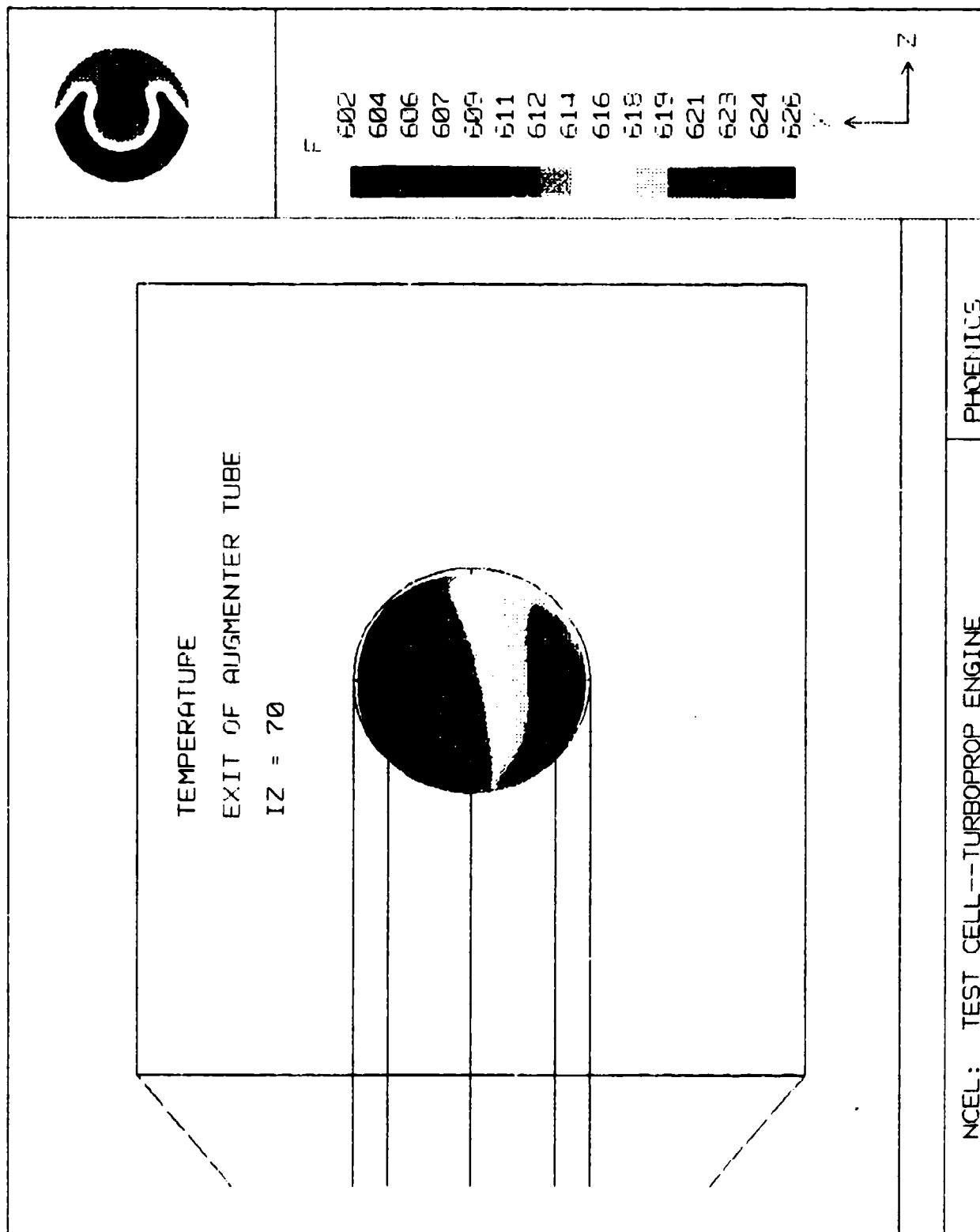


Figure 14 Temperature Exit of Augmenter Tube (Horizontal Plane)

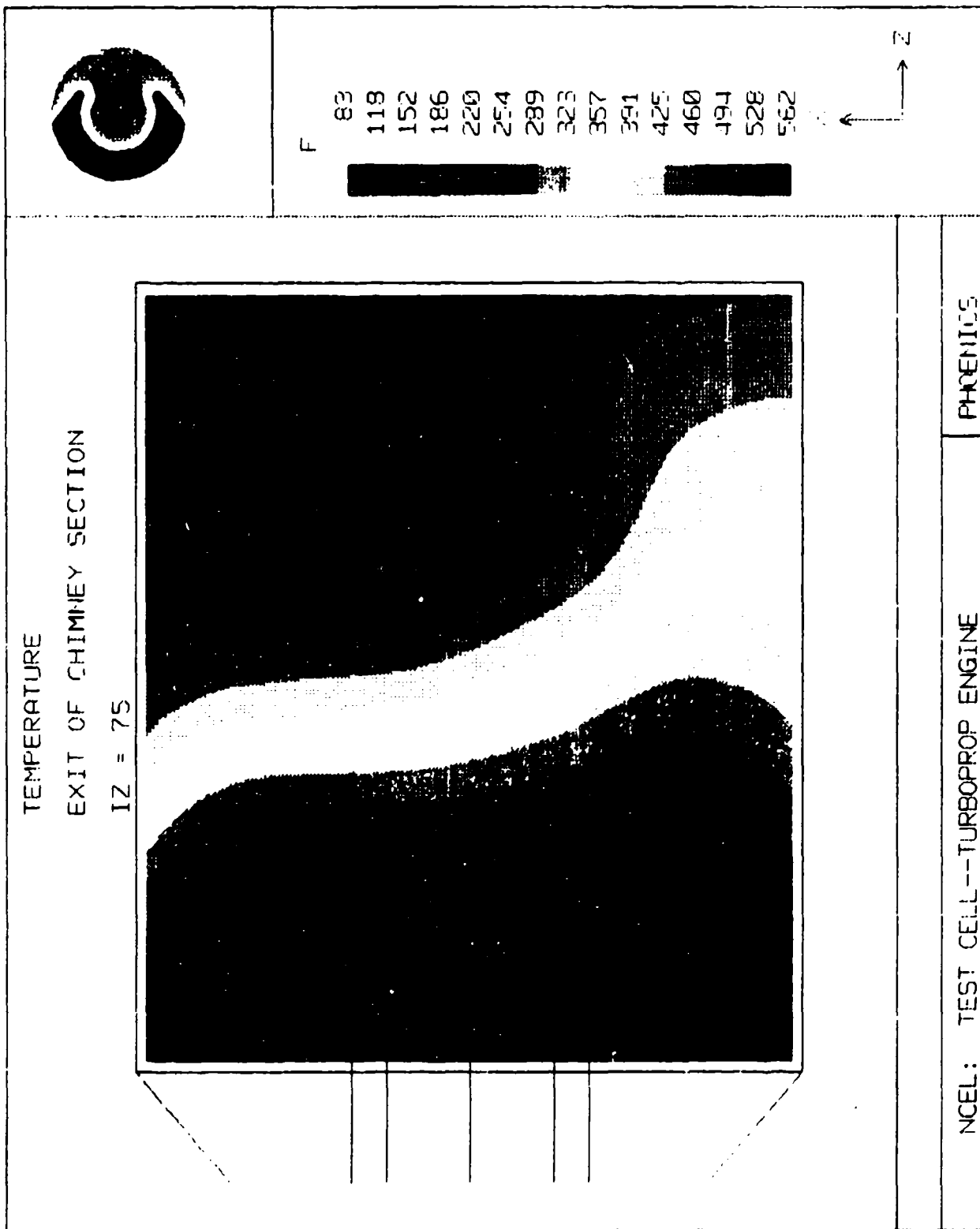


Figure 15 Temperature Exit of Chimney Section (Horizontal Plane)

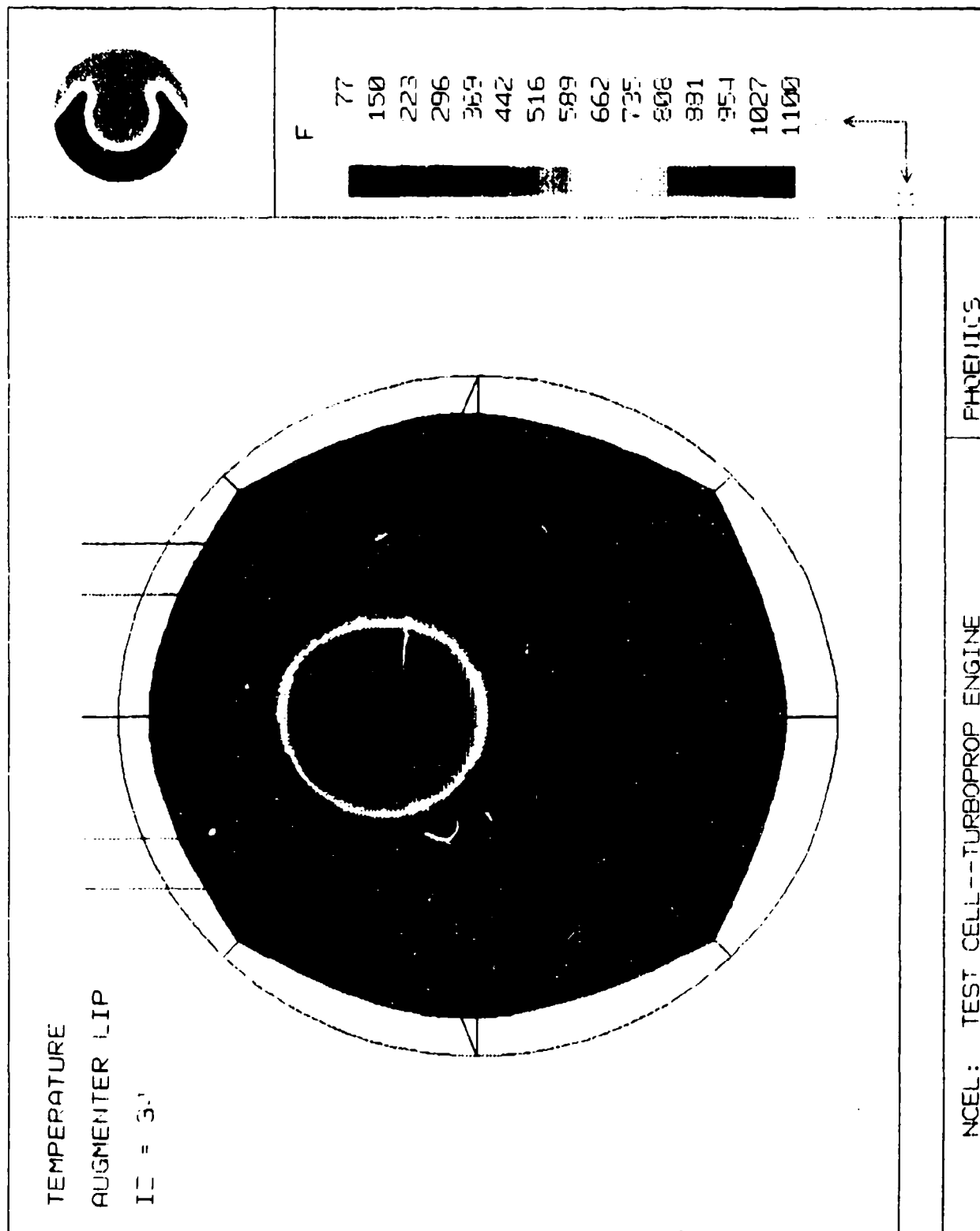


Figure 16 Temperature Augmenter Lip (Horizontal Plane)

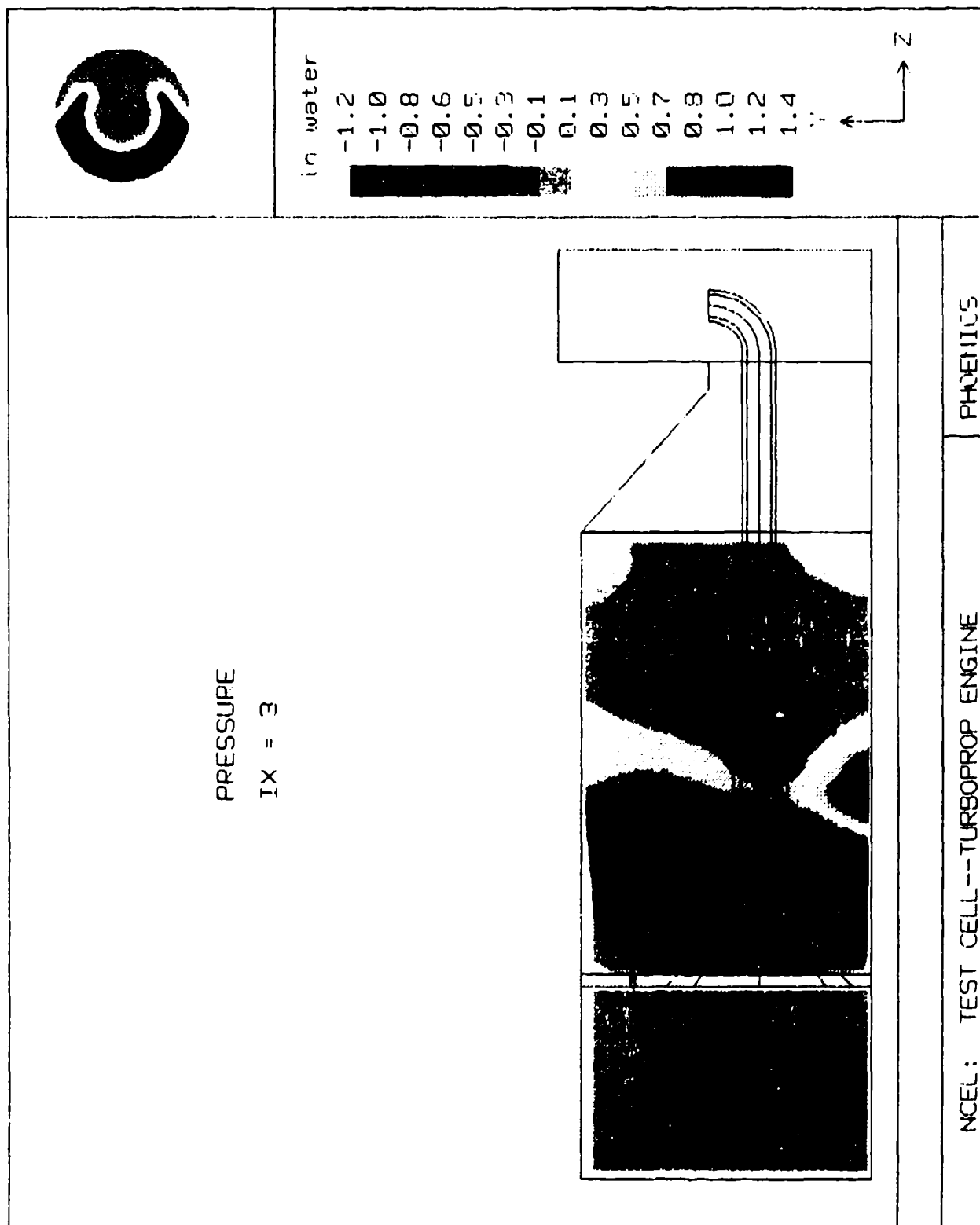


Figure 17 Pressure (Vertical Plane Near Wall)

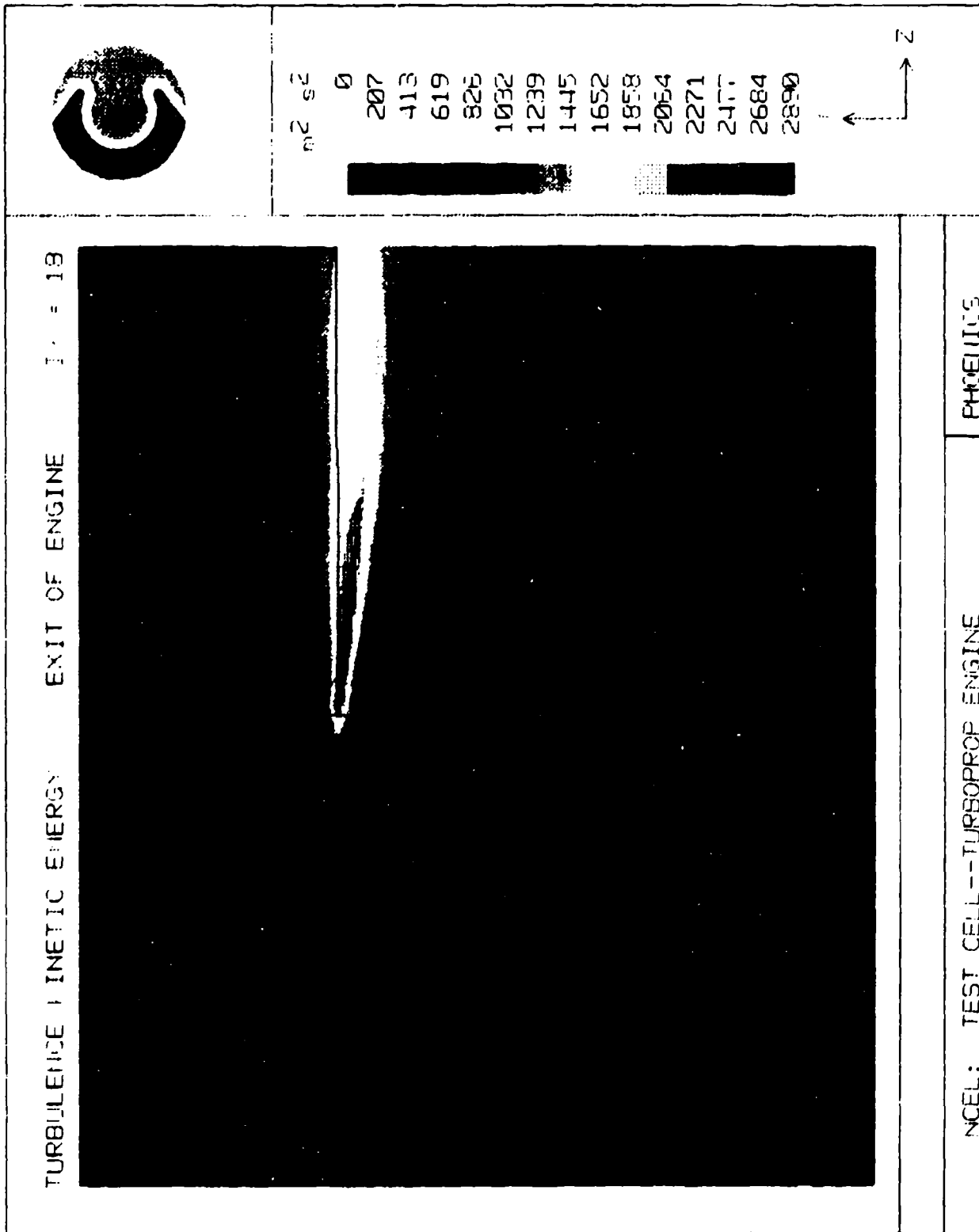


Figure 18 Turbulence Kinetic Energy Exit of Engine (Vertical Plane)

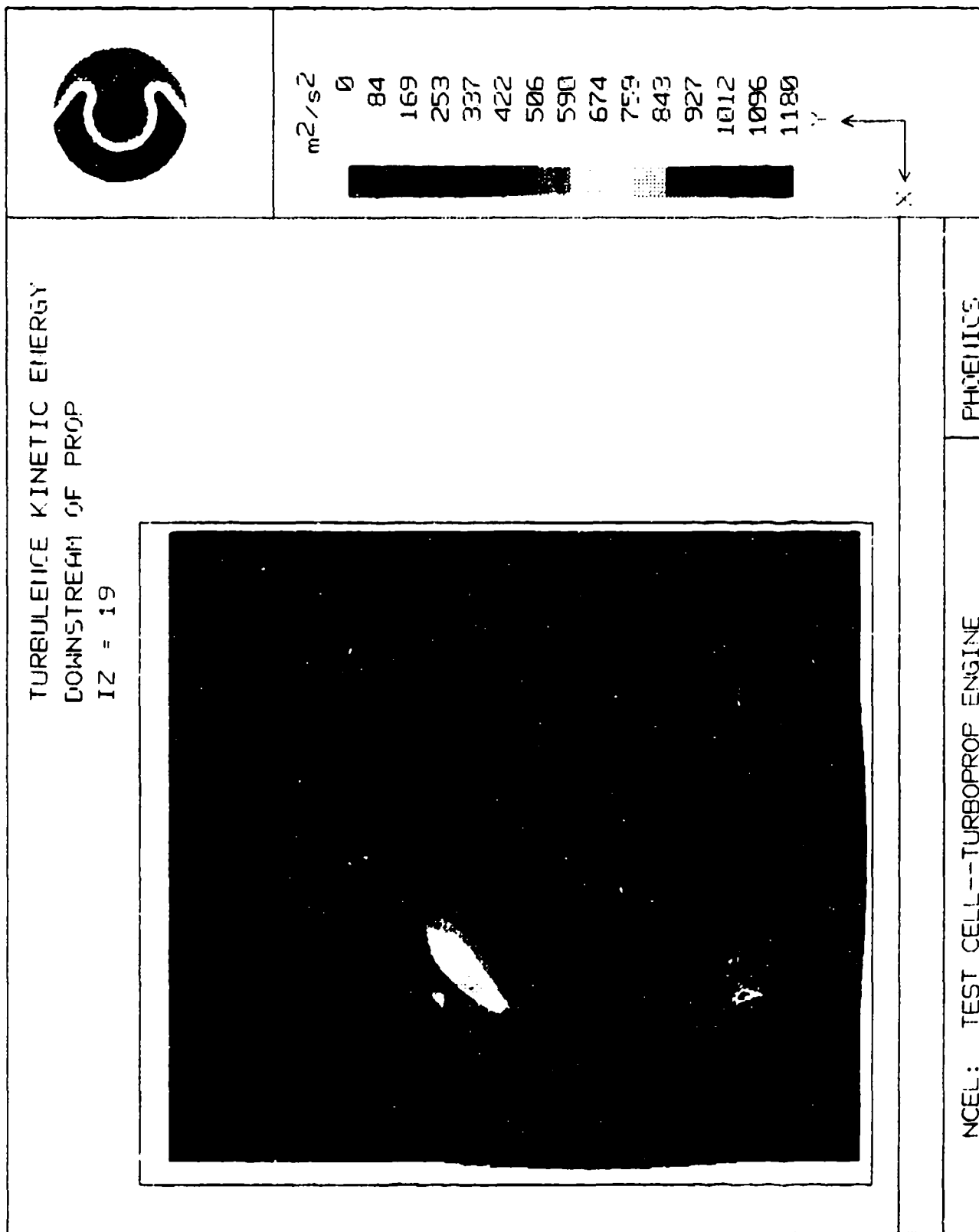
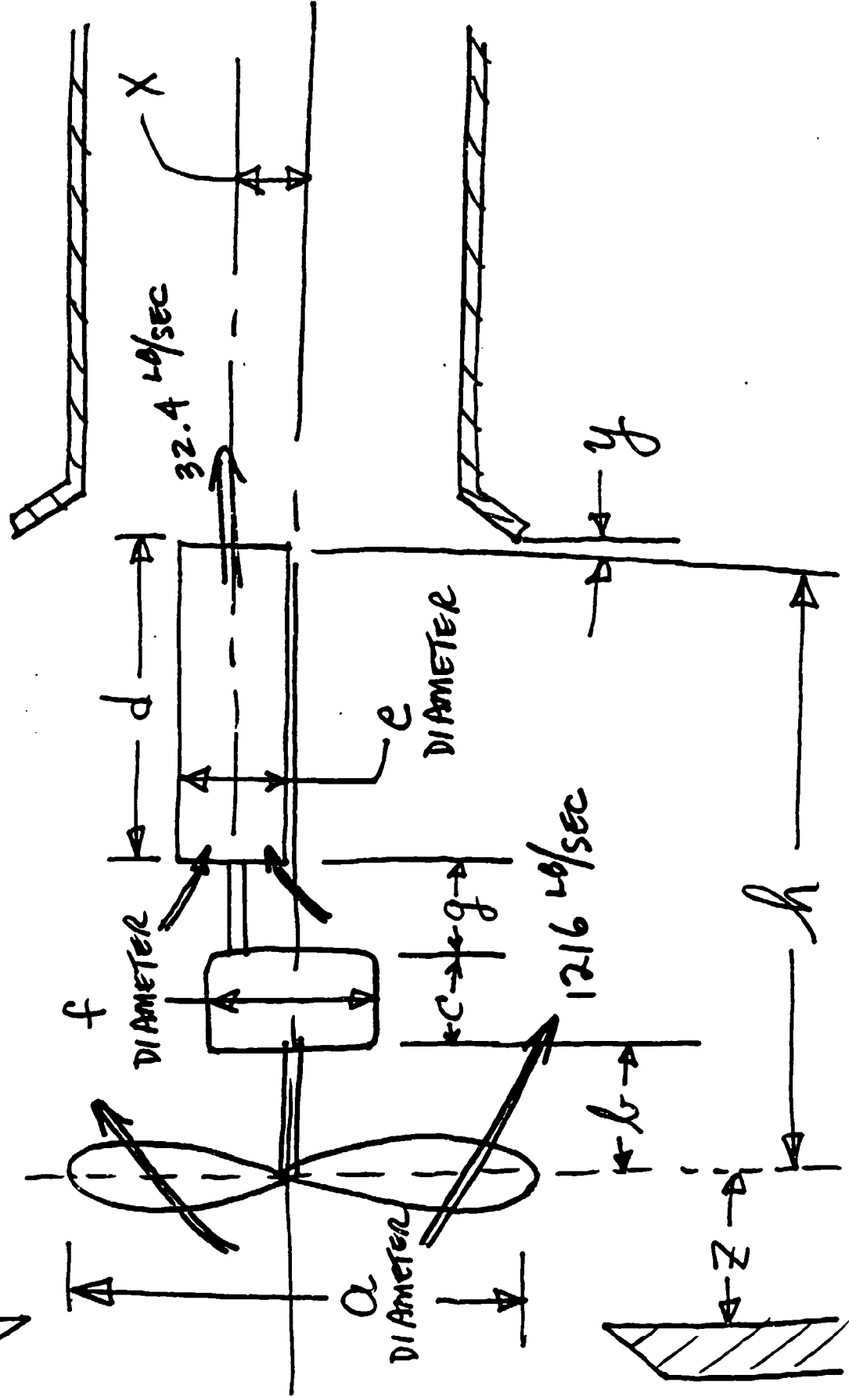


Figure 19 Turbulence Kinetic Energy Down Stream of Prop (Cross Plane)

APPENDIX A

→ + DIRECTION

T56 DIMENSIONS



T56 DIMENSIONS

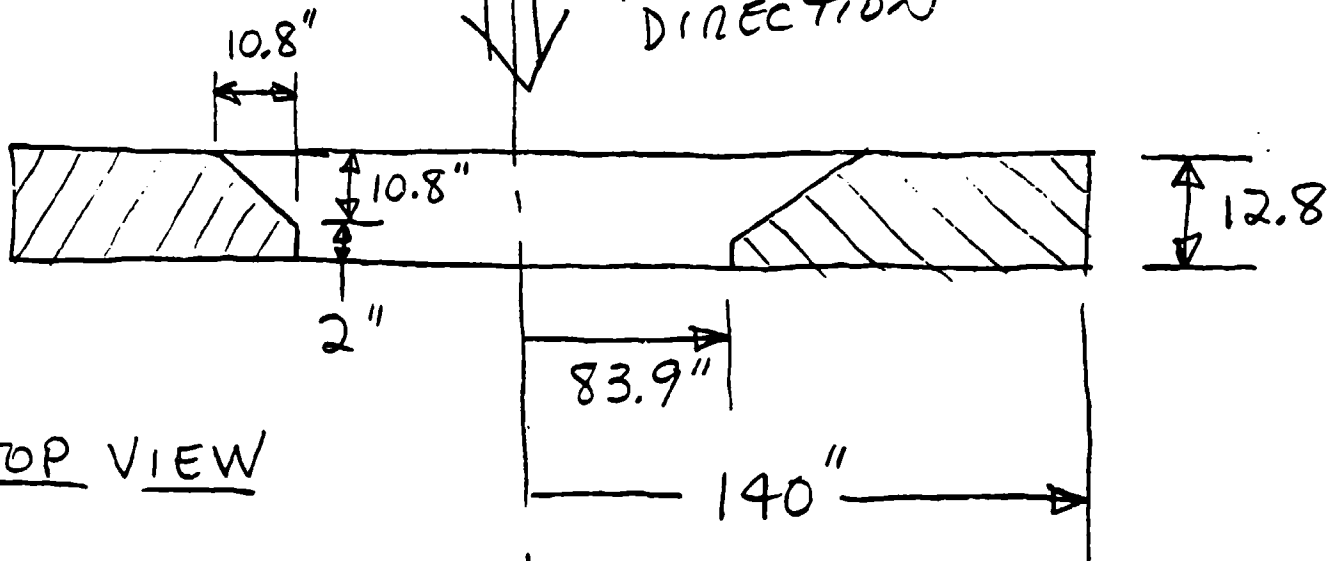
NOTATION	BASE (INCHES)	VARIATION
a ^{PROP DIA}	156	HIGHLY VARIABLE
b	14	*
c	20	*
d	96	VARIABLE
e ^{NOZZLE DIA}	18	VARIABLE
f ^{REDUCTION GEAR DIA}	27 ***	*
g	31	MAY BE IMPORTANT?
h	161	$A \leq 177" + 6"$
x	+9.	$-12" \leq X \leq +12"$
y	0.	$-16" \leq Y \leq +6" *$
z	16.	$0. \leq Z \leq 22"$

* PROBABLY WILL HAVE LITTLE EFFECT ON FLOWS?

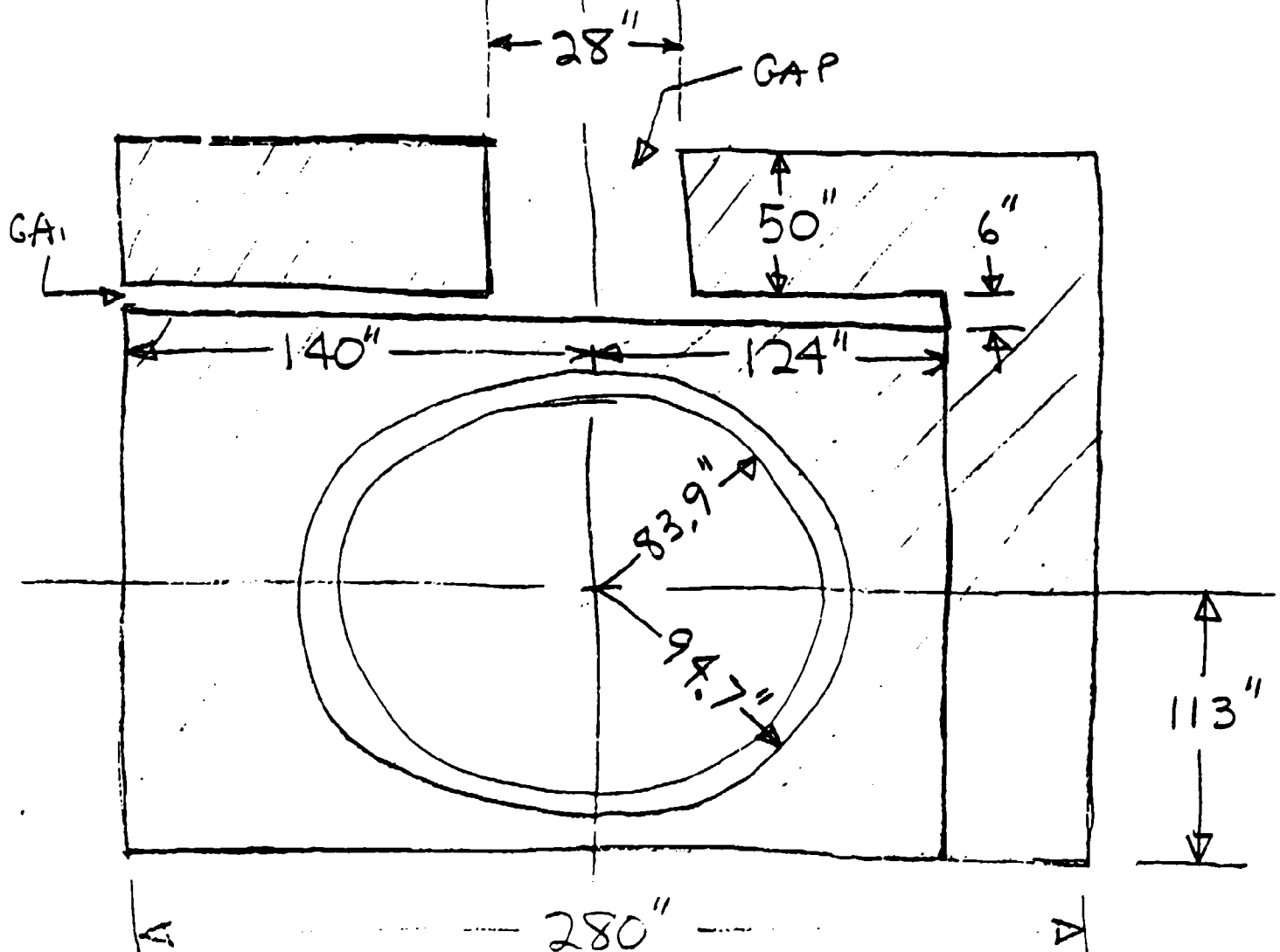
** NOT MORE THAN 6" PENETRATION OF AUG TUBE

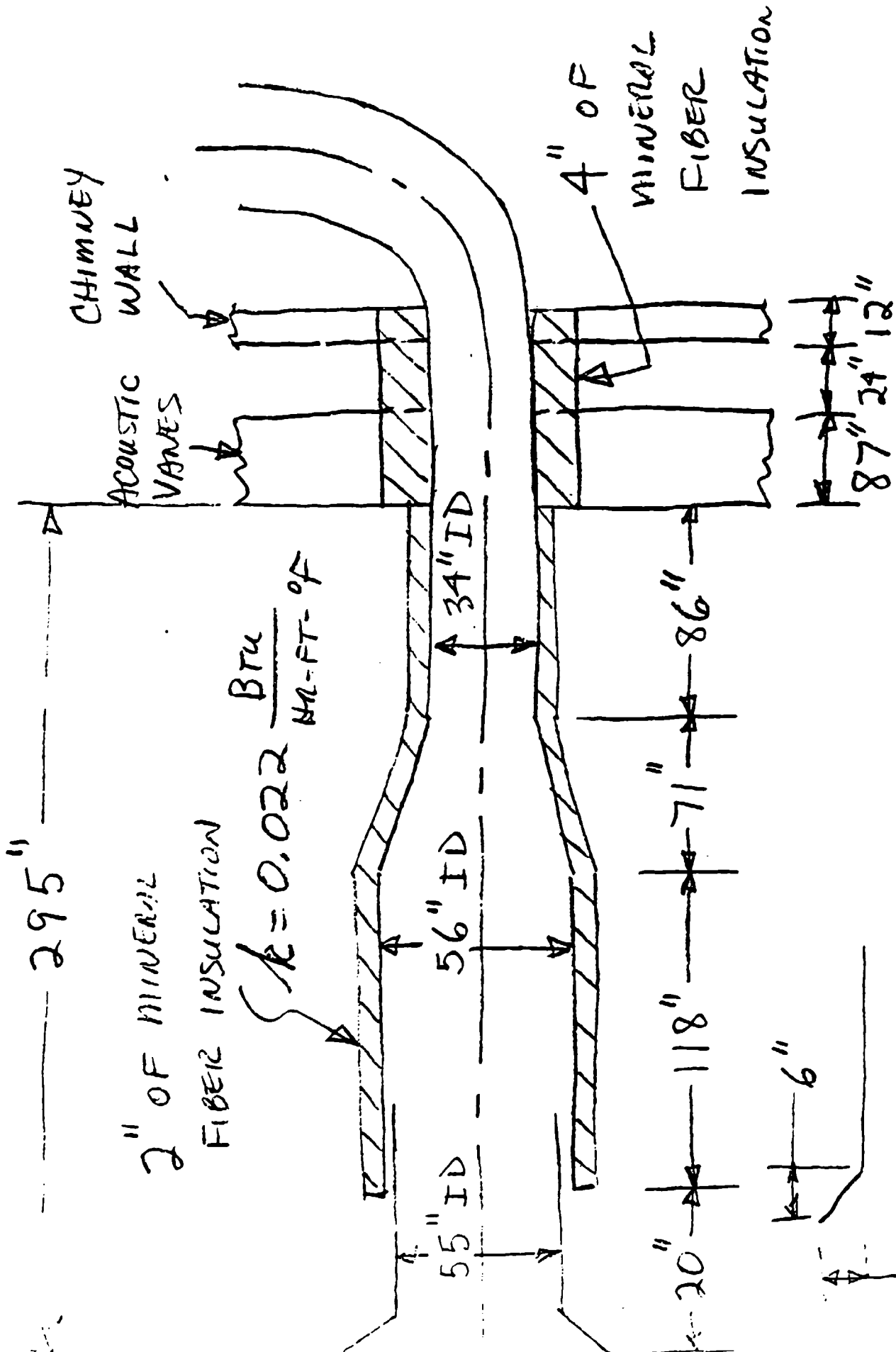
*** ASSUME REDUCTION GEAR ALWAYS SYMMETRIC WITH RESPECT TO PROPELLER AXIS OF ROTATION?

FLOW
DIRECTION

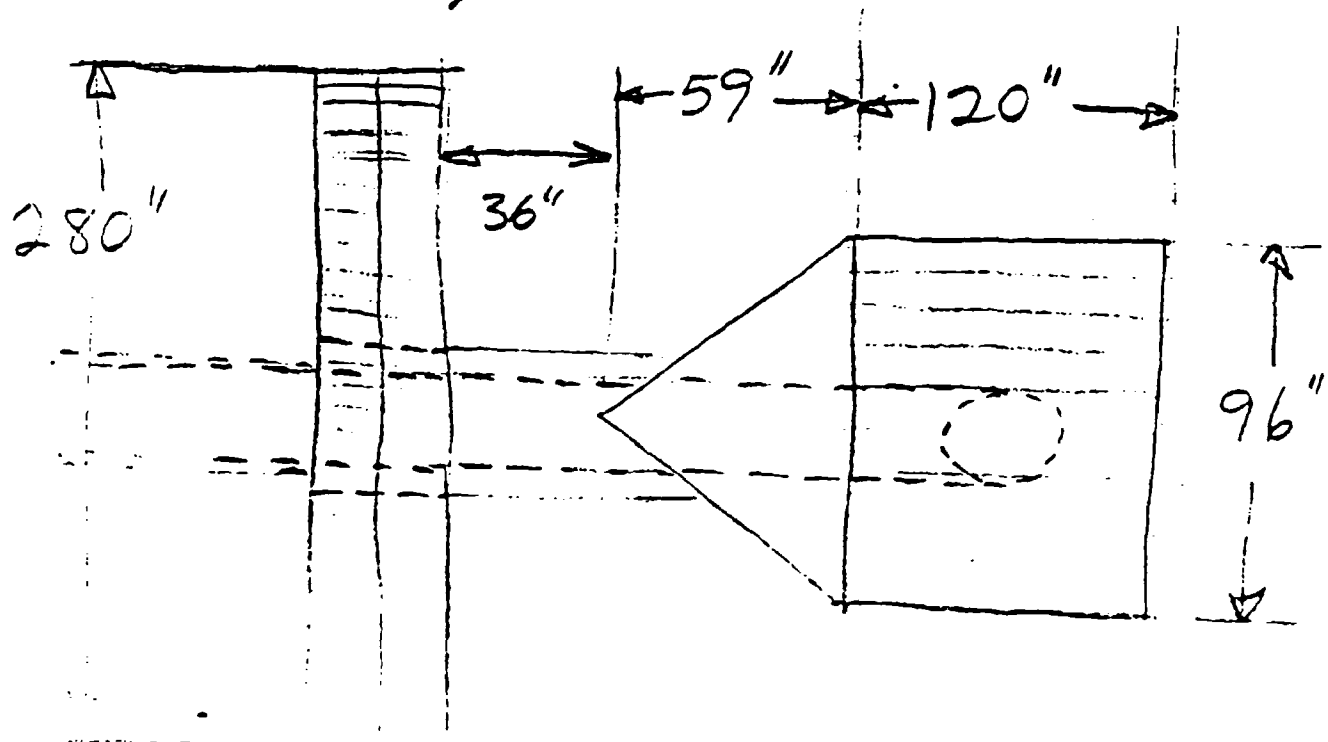
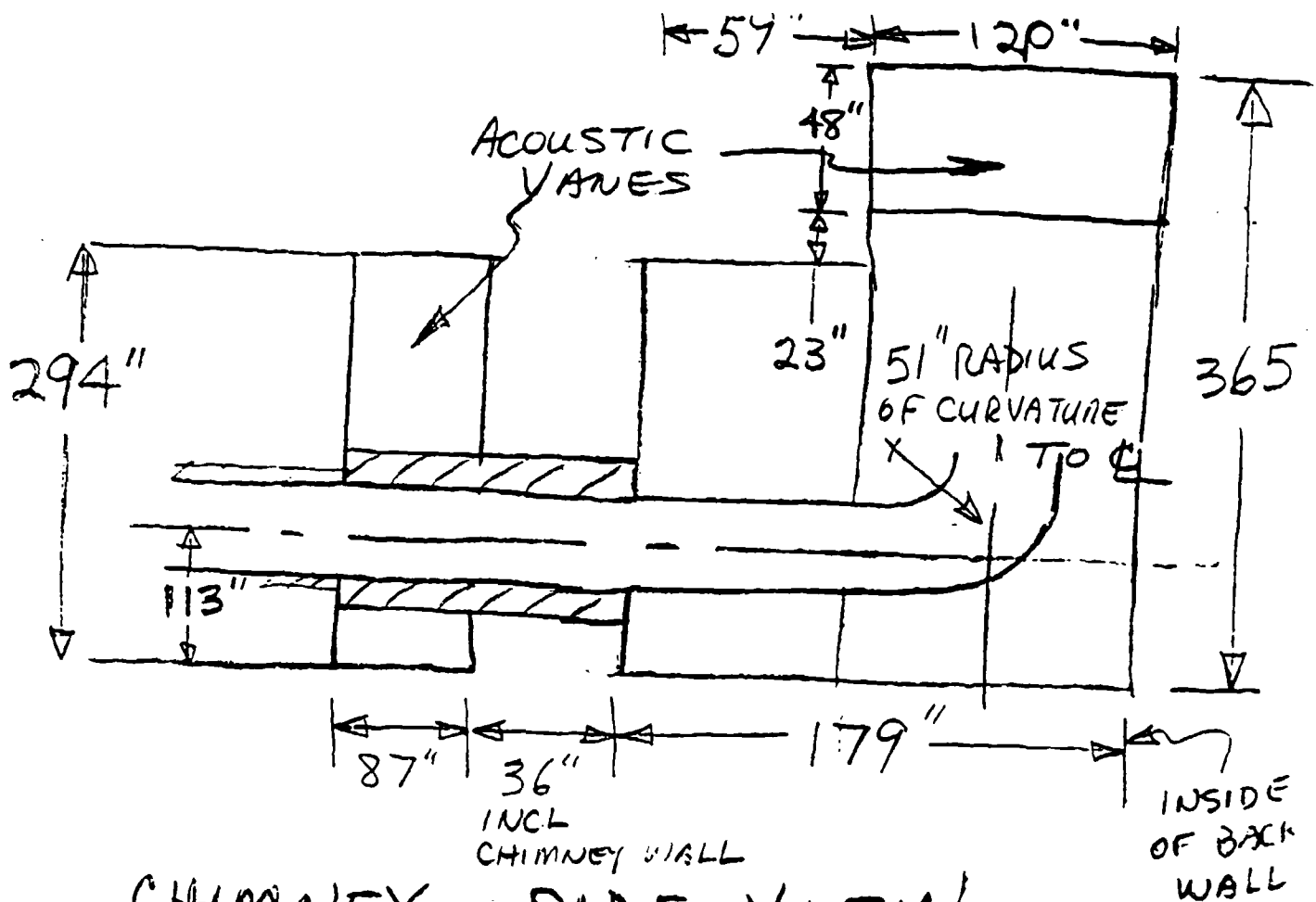


TOP VIEW



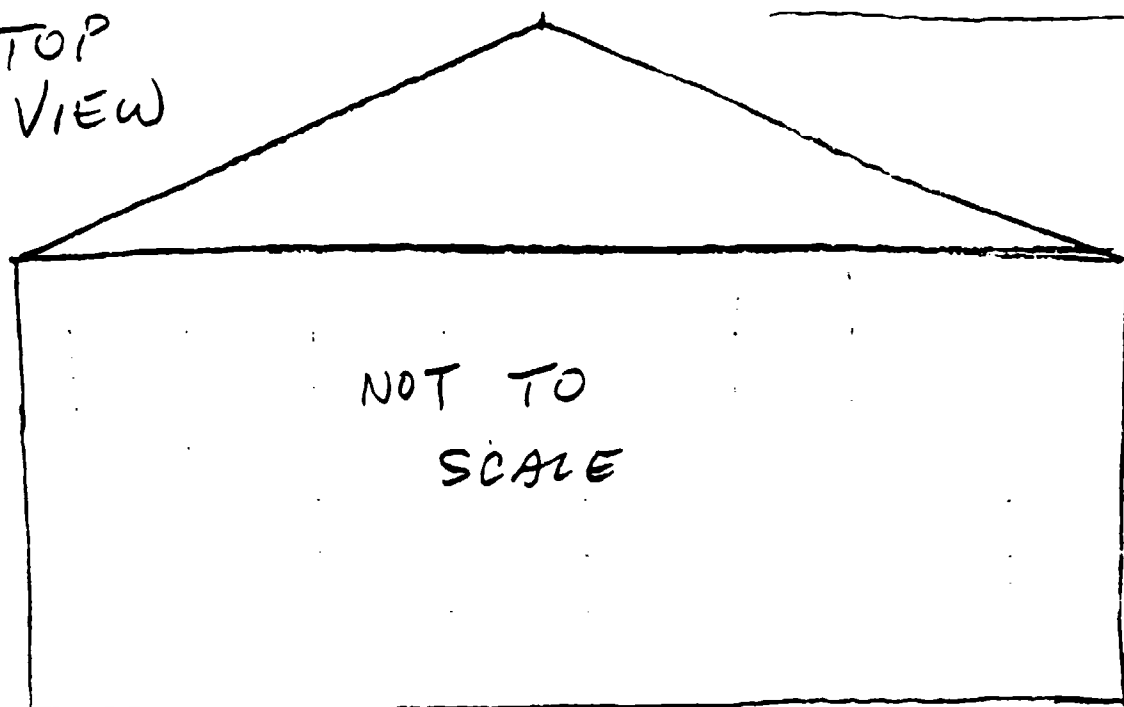


AUGMENTED TUBE



COPY AVAILABLE TO DTIC DOES NOT PERMIT FULLY LEGIBLE REPRODUCTION

TOP
VIEW



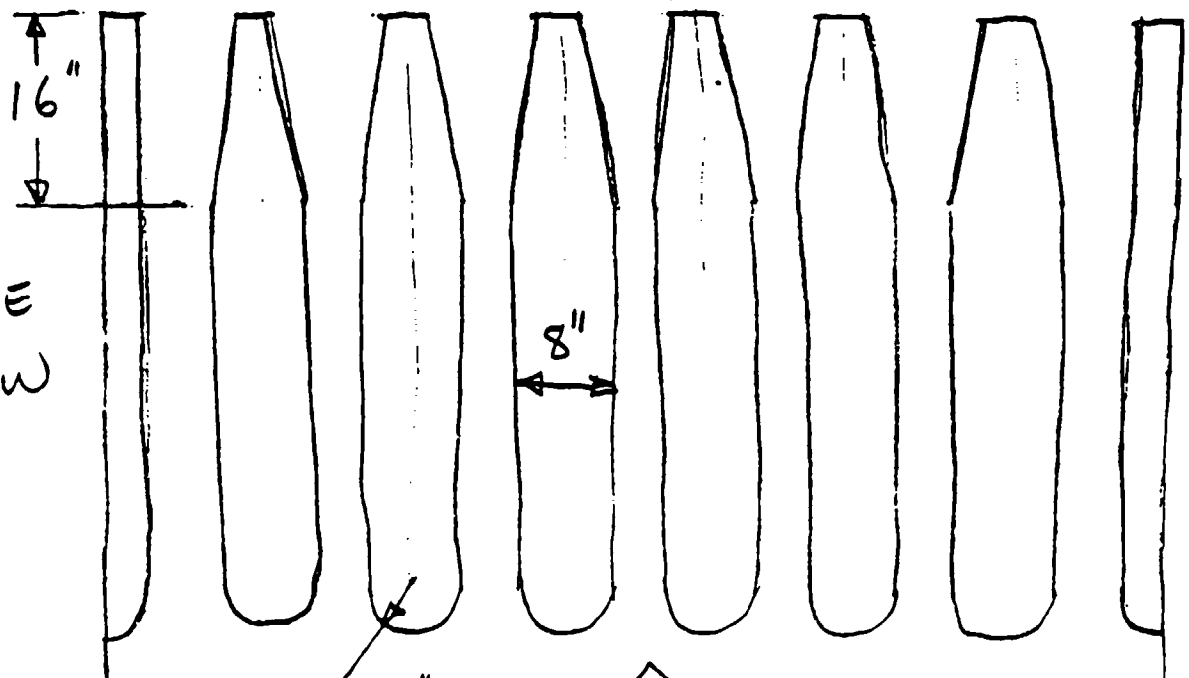
59"
120"

96"

$13\frac{5}{8}"$ $13\frac{3}{4}"$

4"

SIDE
VIEW



48"

8"

4" RADIUS

FLOW
DIRECTION

TR EXHAUST CHIMNEY

APPENDIX B

Hamilton Standard/United Technologies
Facsimile Transmission

From: Robert W. Menthe
Chief - Aerodynamics

Phone: (203) 654-5306

Fax: (203) 654-2659

To: B. I. Mahaffey

Fax: 205 830 26 28

Number of pages: 1/1

Message:

Attached are approximate equations
for computation of the average flow
behind a static propeller. Make sure
the dimension (units) of the density and power
are such that the velocity has the correct
dimension (units). Hope this is of some help.

Also for ref. $C_p = P / \rho n^3 D^5$

$$C_T = T / \rho n^2 D^4$$

Bob

10/14/92

Approximate Equations for Calculation
of Average Flow behind a Static ($V=0$) Propeller.

$$a_p = \text{axial induced velocity at the propeller disk}$$

$$= \sqrt{\frac{T}{2\rho A_p}} \quad \begin{array}{l} T = \text{Thrust}, \rho = \text{density} \\ A_p = \pi D^2/4, D = \text{Prop Diameter} \end{array}$$

$$a_z/a_p = 1 + \frac{2z/D}{\sqrt{1 + (2z/D)^2}}, \quad \begin{array}{l} z = \text{distance from Prop} \\ \quad + \text{downstream} \\ \quad - \text{upstream} \end{array}$$

$$u_p = \text{circumferential induced velocity at the prop. disk}$$

$$= \frac{P}{\rho A_p a_p \pi n \bar{D}} \quad \begin{array}{l} P = \text{Power} \\ n = \text{rev/unit time} \\ \bar{D} = 0.75 \pi D \end{array}$$

$$u_z = 0 \quad -\infty < z < 0 \quad (\text{upstream})$$

$$u_z/u_p = \sqrt{a_z/a_p} \quad 0 \leq z < \infty \quad (\text{downstream})$$

DISTRIBUTION QUESTIONNAIRE

The Naval Civil Engineering Laboratory is revising its primary distribution lists.

SUBJECT CATEGORIES

1 SHORE FACILITIES

- 1A Construction methods and materials (including corrosion control, coatings)
- 1B Waterfront structures (maintenance/deterioration control)
- 1C Utilities (including power conditioning)
- 1D Explosives safety
- 1E Aviation Engineering Test Facilities
- 1F Fire prevention and control
- 1G Antenna technology
- 1H Structural analysis and design (including numerical and computer techniques)
- 1J Protective construction (including hardened shelters, shock and vibration studies)
- 1K Soil/rock mechanics
- 1L Airfields and pavements
- 1M Physical security

2 ADVANCED BASE AND AMPHIBIOUS FACILITIES

- 2A Base facilities (including shelters, power generation, water supplies)
- 2B Expedient roads/airfields/bridges
- 2C Over-the-beach operations (including breakwaters, wave forces)
- 2D POL storage, transfer, and distribution
- 2E Polar engineering

3 ENERGY/POWER GENERATION

- 3A Thermal conservation (thermal engineering of buildings, HVAC systems, energy loss measurement, power generation)
- 3B Controls and electrical conservation (electrical systems, energy monitoring and control systems)
- 3C Fuel flexibility (liquid fuels, coal utilization, energy from solid waste)

- 3D Alternate energy source (geothermal power, photovoltaic power systems, solar systems, wind systems, energy storage systems)

- 3E Site data and systems integration (energy resource data, integrating energy systems)

- 3F EMCS design

4 ENVIRONMENTAL PROTECTION

- 4A Solid waste management
- 4B Hazardous/toxic materials management
- 4C Wastewater management and sanitary engineering
- 4D Oil pollution removal and recovery
- 4E Air pollution
- 4F Noise abatement

5 OCEAN ENGINEERING

- 5A Seafloor soils and foundations
- 5B Seafloor construction systems and operations (including diver and manipulator tools)
- 5C Undersea structures and materials
- 5D Anchors and moorings
- 5E Undersea power systems, electromechanical cables, and connectors
- 5F Pressure vessel facilities
- 5G Physical environment (including site surveying)
- 5H Ocean-based concrete structures
- 5J Hyperbaric chambers
- 5K Undersea cable dynamics

ARMY FEAP

- BDG Shore Facilities
- NRG Energy
- ENV Environmental/Natural Responses
- MGT Management
- PRR Pavements/Railroads

TYPES OF DOCUMENTS

D - Techdata Sheets; R - Technical Reports and Technical Notes; G - NCEL Guides and Abstracts; I - Index to TDS; U - User Guides; ☐ None - remove my name

Old Address:

Telephone No.: _____

New Address:

Telephone No.: _____

INSTRUCTIONS

The Naval Civil Engineering Laboratory has revised its primary distribution lists. To help us verify our records and update our data base, please do the following:

- Add - circle number on list
- Remove my name from all your lists - check box on list.
- Change my address - line out incorrect line and write in correction (DO NOT REMOVE LABEL).
- Number of copies should be entered after the title of the subject categories you select.
- Are we sending you the correct type of document? If not, circle the type(s) of document(s) you want to receive listed on the back of this card.

Fold on line, staple, and drop in mail.

DEPARTMENT OF THE NAVY
Naval Civil Engineering Laboratory
560 Laboratory Drive
Port Hueneme CA 93043-4328

Official Business
Penalty for Private Use, \$300

BUSINESS REPLY CARD

FIRST CLASS PERMIT NO. 12503 WASH D.C.

POSTAGE WILL BE PAID BY ADDRESSEE

NO POSTAGE
NECESSARY
IF MAILED
IN THE
UNITED STATES

COMMANDING OFFICER
CODE L34
560 LABORATORY DRIVE
NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME CA 93043-4328

NCEL DOCUMENT EVALUATION

You are number one with us; how do we rate with you?

We at NCEL want to provide you our customer the best possible reports but we need your help. Therefore, I ask you to please take the time from your busy schedule to fill out this questionnaire. Your response will assist us in providing the best reports possible for our users. I wish to thank you in advance for your assistance. I assure you that the information you provide will help us to be more responsive to your future needs.

R. N. Storer

R. N. STORER, Ph.D., P.E.
Technical Director

DOCUMENT NO. _____ TITLE OF DOCUMENT: _____

Date: _____ Respondent Organization: _____

Name: _____ Activity Code: _____
Phone: _____ Grade/Rank: _____

Category (please check):

Sponsor _____ User _____ Proponent _____ Other (Specify) _____

Please answer on your behalf only; not on your organization's. Please check (use an X) only the block that most closely describes your attitude or feeling toward that statement:

SA Strongly Agree A Agree O Neutral D Disagree SD Strongly Disagree

	SA	A	N	D	SD		SA	A	N	D	SD
1. The technical quality of the report is comparable to most of my other sources of technical information.	()	()	()	()	()	6. The conclusions and recommendations are clear and directly supported by the contents of the report.	()	()	()	()	()
2. The report will make significant improvements in the cost and or performance of my operation.	()	()	()	()	()	7. The graphics, tables, and photographs are well done.	()	()	()	()	()
3. The report acknowledges related work accomplished by others.	()	()	()	()	()	Do you wish to continue getting NCEL reports? ☐ YES ☐ NO Please add any comments (e.g., in what ways can we improve the quality of our reports?) on the back of this form.					
4. The report is well formatted.	()	()	()	()	()						
5. The report is clearly written.	()	()	()	()	()						

Comments:

Fold on line, staple, and drop in mail.

DEPARTMENT OF THE NAVY
Naval Civil Engineering Laboratory
560 Laboratory Drive
Port Hueneme CA 93043-4328

Official Business
Penalty for Private Use, \$300

BUSINESS REPLY CARD

FIRST CLASS PERMIT NO. 12503 WASH D.C.

POSTAGE WILL BE PAID BY ADDRESSEE

NO POSTAGE
NECESSARY
IF MAILED
IN THE
UNITED STATES

COMMANDING OFFICER
CODE L03
560 LABORATORY DRIVE
NAVAL CIVIL ENGINEERING LABORATORY
PORT HUENEME CA 93043-4328

**END
FILMED**

DATE:

8-93

DTIC