REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-018

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

4. TITLE AND SUBTITLE

SOLVE FLUID FLOW PROBLEMS WITH "PHOENICS"

5. FUNDING NUMBERS

6. AUTHOR(S)

Dr. Cal Kodres and Dr. Gene Cooper

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESSE(S)

Naval Facilities Engineering Service Center
Port Hueneme, CA 93043-4328

8. PERFORMING ORGANIZATION REPORT NUMBER

TDS-2001-ENV

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESSES

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)*

Save time and money by using the computational fluid dynamics program "PHOENICS" for design, optimization, and troubleshooting of fluid flow systems.

14. SUBJECT TERMS

PHOENICS, computational fluid dynamics (CFD), high velocity, low velocity

15. NUMBER OF PAGES

6

16. PRICE CODE

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

Technical Data Sheet

Naval Facilities Engineering Service Center
Port Hueneme, California 93043-4328

November 1993 TDS-2001-ENV

SOLVE FLUID FLOW PROBLEMS WITH "PHOENICS"

Save time and money by using the computational fluid dynamics program "PHOENICS" for design, optimization, and troubleshooting of fluid flow systems.

Computational Fluid Dynamics (CFD) is a computer modeling technique for mathematical analysis of engineering and environmental processes. This technology increases the efficiency and effectiveness of conceptual design and improves the reliability of prototyping. CFD also serves as a practical alternative in areas where physical experiments are prohibitive.

CFD is applicable to nearly all fluid flow problems. Examples of just a few applications include:

- Drag and moments caused by currents past complex structures or ships at anchor.

- Viscous flow around ships or fully submerged appended bodies.

- Turbulent ship airwake prediction for helicopter approach.

- Movement of sediment in harbors.

- Computation of blast wave forces on buildings and the influence that berms, walls, and barriers have on the strength and propagation of blast waves.

- Propagation of fire through a building, or the spread of fire retarding agent.

- Circulation of air in paint hangars, or thermal stratification in hangars, or more generally, the thermodynamic field and circulation of air in any space with a ventilation system.

- Air flow past a building for applications including natural ventilation of buildings, estimation of forces on tents and soft structure buildings, and prediction of vulnerability from chemical, biological, or radiation attack.

- Flow and temperature differences between power station intake and discharge ports in a bay.

- Dispersion of hot water or foreign fluids released into a river or bay.

- Dispersion of particles released from a smoke stack.

For the past decade, the Naval Facilities Engineering Service Center (NFESC) has been using a CFD program called PHOENICS. PHOENICS was developed by Professor D. B. Spalding of the Imperial College, London, and is widely accepted as the leading, general purpose, CFD program in the world.

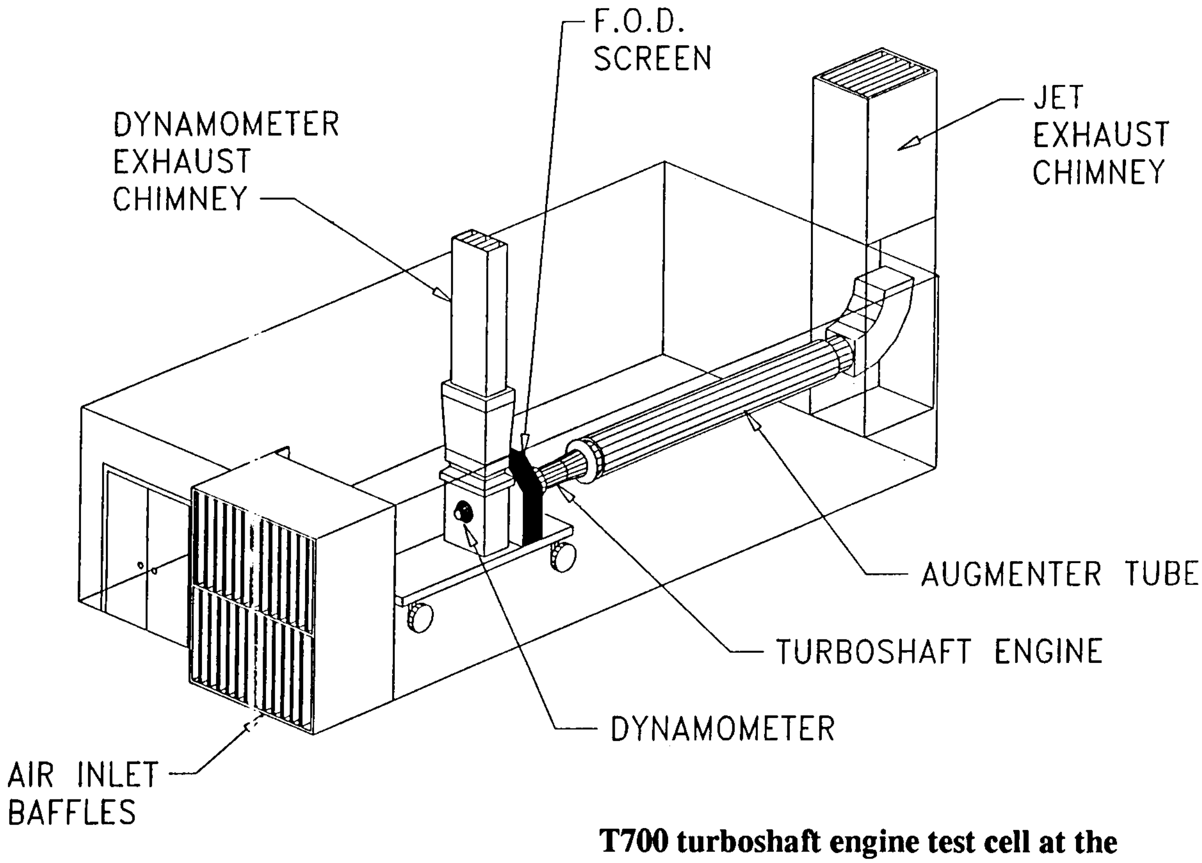
Solving fluid flow problems with PHOENICS is a three-step process. First, a grid is constructed, modeling the flow geometry. Second, boundary and initial conditions are stipulated: temperatures, pressures, and/or velocities. Finally, the PHOENICS program is run on a computer to determine all the unknown fluid properties. Once the model has been developed, different scenarios are examined by changing the geometry or changing the boundary conditions and rerunning the PHOENICS program.

PHOENICS is capable of simultaneously analyzing the following flow conditions:

- Transient or steady state flows
- 1-, 2,- or 3-dimensional flows
- Chemically reactive or inert materials; exothermic and endothermic reactions
- Complex geometries and boundary conditions
- Single-, double- or multi-phase flows
- Laminar or turbulent flows
- Incompressible or compressible flow
- Parabolic or elliptic solutions
- Heat transfer (conduction, convection, radiation)

EXAMPLE OF A HIGH VELOCITY PROBLEM

A PHOENICS CFD model was developed by NFESC for evaluating aerothermal performance and for troubleshooting a new generation of turboshaft engine test cells. This is an environment with high velocity flows and high heat transfer rates. The modeling was complicated by a nonsymmetric geometry.



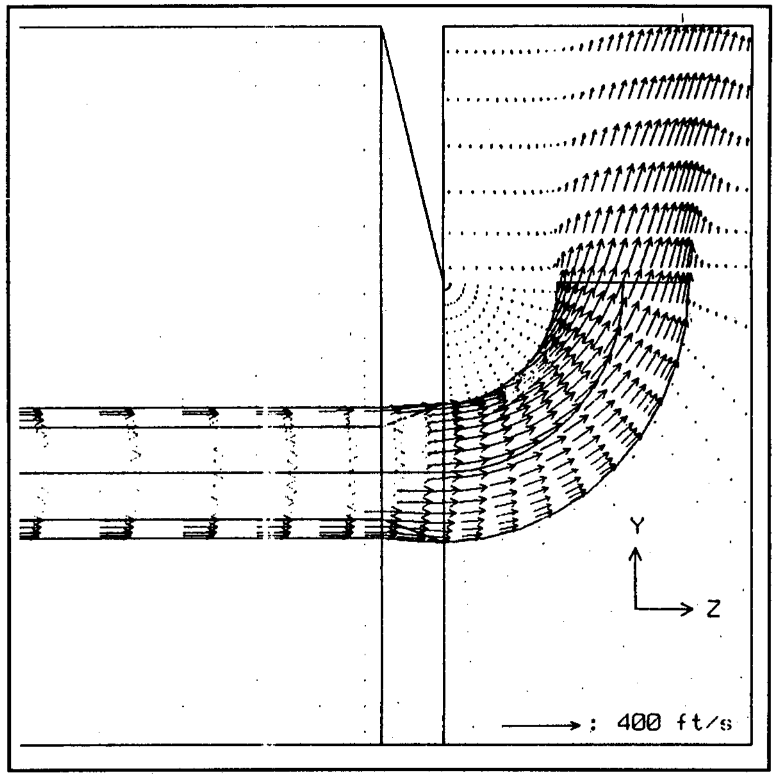
T700 turboshaft engine test cell at the Marine Corps Air Facility, Camp Pendleton, California.

There are many turboshaft engine test cell design problems that are more efficiently analyzed using CFD. For example, the designer can predetermine:

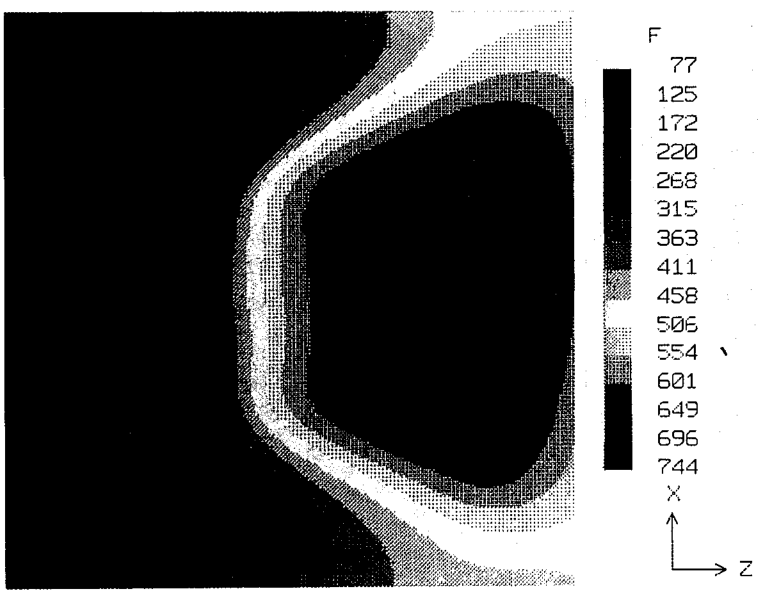
- Velocities and temperatures of engine exhaust gases that are controlled to protect augmenter tube components and to limit noise propagation.

- Velocity and direction of the air entering the engine that must remain within tolerances to achieve accurate simulation of flight performance.

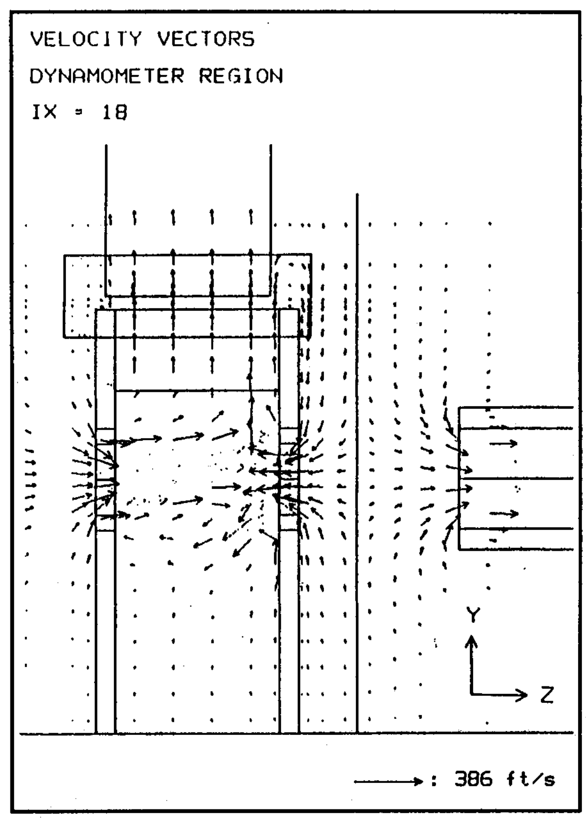
The first opportunity to use the PHOENICS model for troubleshooting occurred during the initial calibration of the test cell. High air temperatures were being measured inside the cell. Spillover of the dynamometer was suspected; some of the dynamometer exhaust was possibly entering the interior of the test cell.



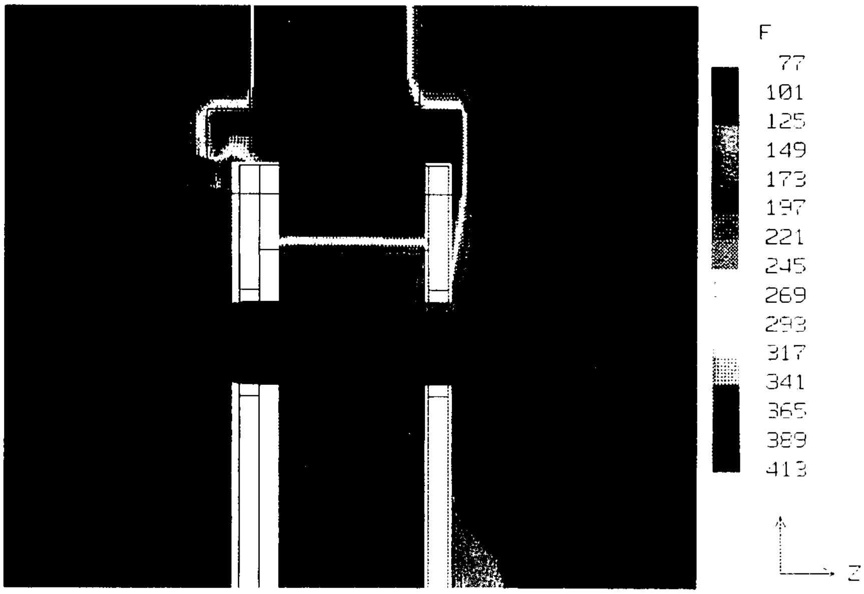
Velocities of the engine exhaust leaving the augmeter tube and passing out the chimney.



Gas temperatures in the exit plane of the chimney.



Vectors showing air velocities and directions at the inlet to the engine. This is a side view that shows both the engine inlet and the dynamometer. Note that the dynamometer and its exhaust stack are not perfectly aligned. Some of the dynamometer exhaust is escaping out of the back. A solid plate is mounted between the dynamometer and the engine to prevent dynamometer overflow from reaching the engine inlet.

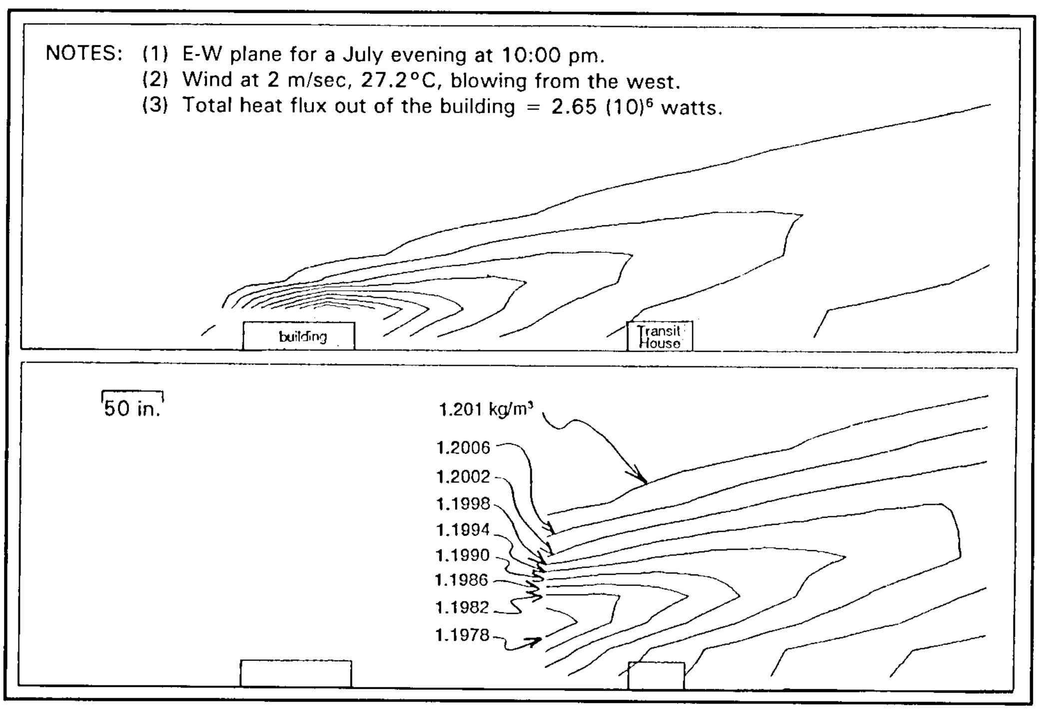


The overflow problem was examined using the PHOENICS model. For this study, alignment of the dynamometer and its exhaust chimney was the major parameter.

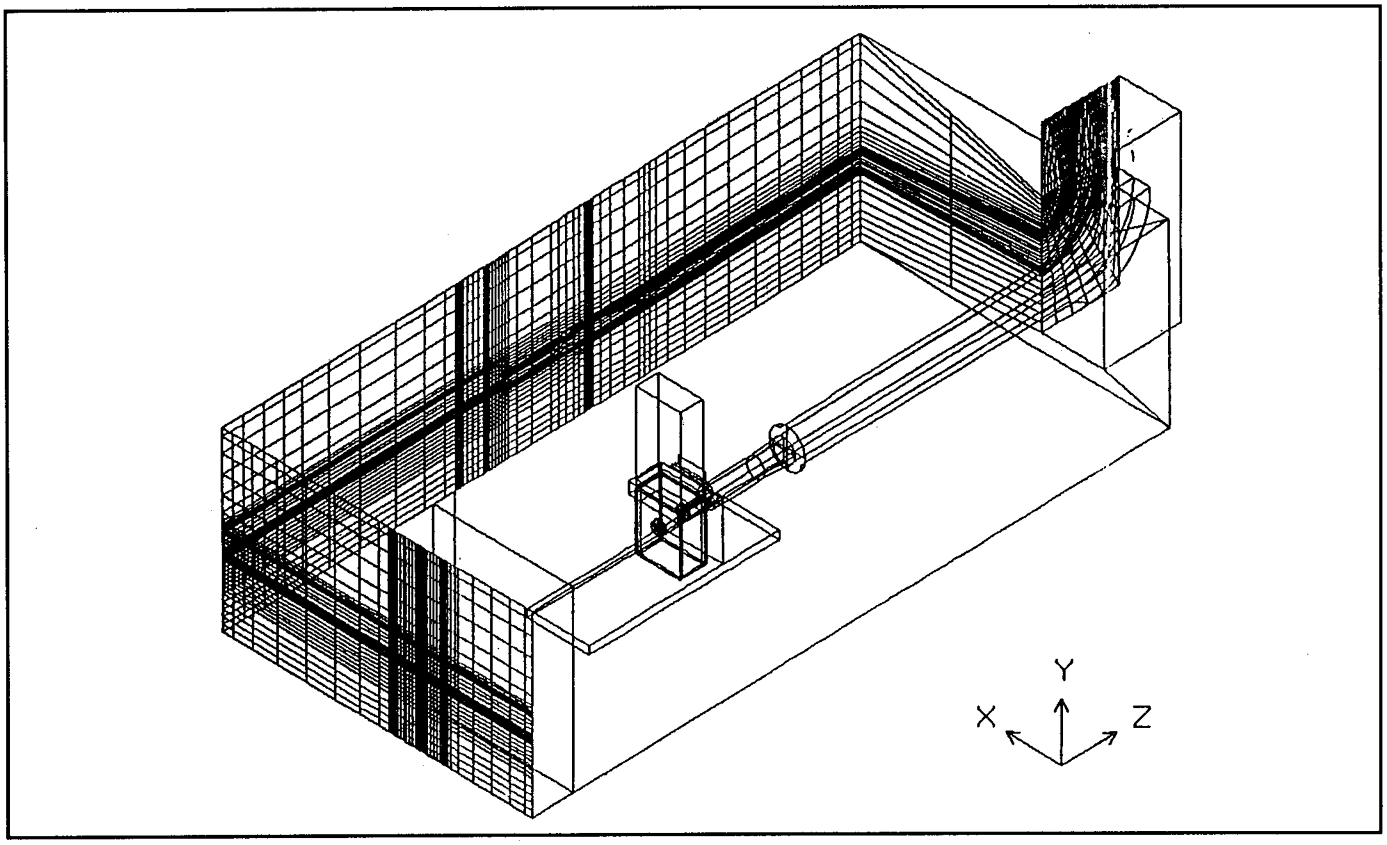
Gas temperatures in the vicinity of the dynamometer. This is the same view shown on the previous figure. The overflow is shown as a high temperature "red" region at the back of the dynamometer.

EXAMPLE OF A LOW VELOCITY PROBLEM

NFESC was asked to calculate density variations in the atmosphere above the Naval Observatory, caused when the wind is heated by the walls of nearby buildings. The Observatory was concerned that refraction of the incoming light was introducing uncorrectable errors into celestial observations. PHOENICS was again used. This problem is the other extreme: very low velocities and very low heat transfer rates. But a very high accuracy was required. The major complication was the modeling of the buoyancy of the heated air.

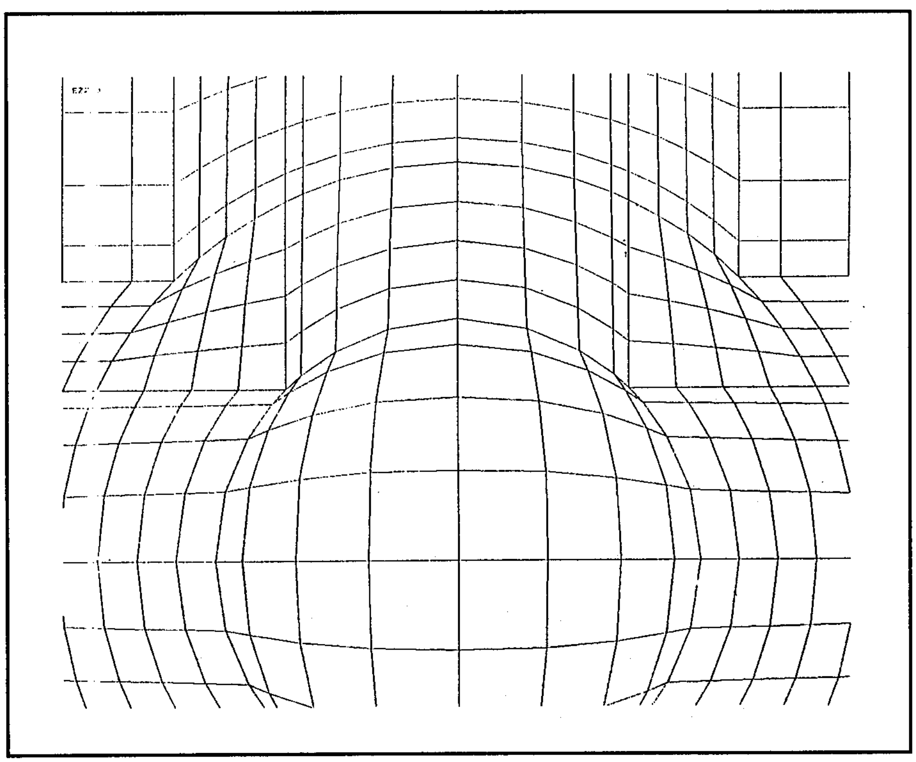


Densities of the air above the Naval Observatory. The wind is blowing past the building (where it is being heated) to the Transit House (where the telescope is located). Note that the hot air is rising as it moves towards the Observatory.

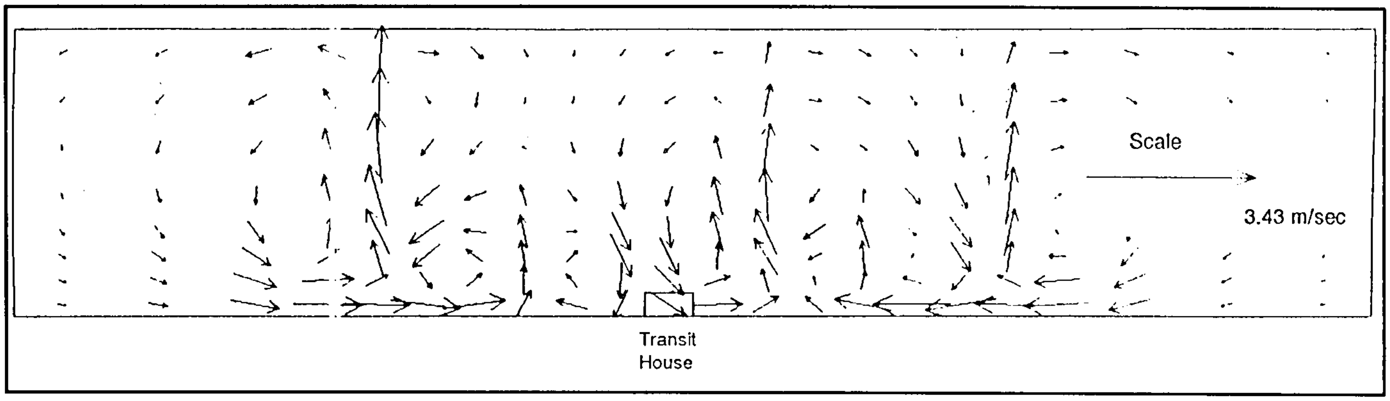


Computational grid constructed to model the Camp Pendleton turboshaft engine test cell. (To keep from cluttering this figure, the grid through the interior of the cell is now shown.)

The test cell was approximated using a 40 x 35 x 77 computational grid. The grid was clustered in areas of geometric change, areas of expected high flow gradients, and areas of heat transfer, and was fitted to circular regions such as the nozzle of the engine.



Grid modified to fit the nozzle of the engine.



Convection currents in the vicinity of the Naval Observatory for a day in July. This view shows a plane perpendicular to the direction of a 2 m/sec wind.

For further information on PHOENICS capabilities and specific applications, contact:

Dr. Cal Kodres
Code ESC421
Naval Facilities Engineering Service Center
DSN: 551-1656; Comm: (805) 982-1656

or

Dr. Gene Cooper
Code ESC64
Naval Facilities Engineering Service Center
DSN: 551-1370; Comm: (805) 982-1370

or

Jerry Dummer
Code ESC101
Naval Facilities Engineering Service Center
DSN: 551-1599; Comm: (805) 982-1599