Open access • Book Chapter • DOI:10.1007/978-1-4615-4215-5_33

Cryogenic Fuel Tank Draining Analysis Model — Source link

Donald Greer

Institutions: Neil A. Armstrong Flight Research Center

Published on: 01 Jan 2000 - Advances in cryogenic engineering (Springer, Boston, MA)

Topics: Fuel tank, Cryogenic fuel, Ullage, Slosh dynamics and Cryogenics

406013
9P

REPRINT/IN-05
NMF/ISS ACT
142 126

CRYOGENIC FUEL TANK DRAINING ANALYSIS MODEL

Donald Greer

Research Engineer, Fluid Dynamics
NASA Dryden Flight Research Center
Edwards, California 93523

ABSTRACT

One of the technological challenges in designing advanced hypersonic aircraft and the next generation of spacecraft is developing reusable flight-weight cryogenic fuel tanks. As an aid in the design and analysis of these cryogenic tanks, a computational fluid dynamics (CFD) model has been developed specifically for the analysis of flow in a cryogenic fuel tank. This model employs the full set of Navier-Stokes equations, except that viscous dissipation is neglected in the energy equation. An explicit finite difference technique in two-dimensional generalized coordinates, approximated to second-order accuracy in both space and time is used. The stiffness resulting from the low Mach number is resolved by using artificial compressibility. The model simulates the transient, two-dimensional draining of a fuel tank cross section. To calculate the slosh wave dynamics the interface between the ullage gas and liquid fuel is modeled as a free surface. Then, experimental data for free convection inside a horizontal cylinder are compared with model results. Finally, cryogenic tank draining calculations are performed with three different wall heat fluxes to demonstrate the effect of wall heat flux on the internal tank flow field.

NOMENCLATURE

CFD	computational fluid dynamics
He/LH2	helium and liquid hydrogen
K	kelvin
J/m^2-s	joules per meter squared per second
m	meter
m/s	meters per second
q	speed, meters per second

406013
9P

REPRINT/IN-05
NMF/ISS ACT.
142 126

CRYOGENIC FUEL TANK DRAINING ANALYSIS MODEL

Donald Greer

Research Engineer, Fluid Dynamics
NASA Dryden Flight Research Center
Edwards, California 93523

ABSTRACT

One of the technological challenges in designing advanced hypersonic aircraft and the next generation of spacecraft is developing reusable flight-weight cryogenic fuel tanks. As an aid in the design and analysis of these cryogenic tanks, a computational fluid dynamics (CFD) model has been developed specifically for the analysis of flow in a cryogenic fuel tank. This model employs the full set of Navier-Stokes equations, except that viscous dissipation is neglected in the energy equation. An explicit finite difference technique in two-dimensional generalized coordinates, approximated to second-order accuracy in both space and time is used. The stiffness resulting from the low Mach number is resolved by using artificial compressibility. The model simulates the transient, two-dimensional draining of a fuel tank cross section. To calculate the slosh wave dynamics the interface between the ullage gas and liquid fuel is modeled as a free surface. Then, experimental data for free convection inside a horizontal cylinder are compared with model results. Finally, cryogenic tank draining calculations are performed with three different wall heat fluxes to demonstrate the effect of wall heat flux on the internal tank flow field.

NOMENCLATURE

CFD	computational fluid dynamics
He/LH2	helium and liquid hydrogen
K	kelvin
J/m^2-s	joules per meter squared per second
m	meter
m/s	meters per second
q	speed, meters per second

q_i	initial speed
T	temperature, K
Tavg	average temperature, K
T_i	initial temperature, K
u, v	velocity, meters per second
ρ	density, kilograms per meter cubed
ρ_i	initial density, kilograms per meter cubed

INTRODUCTION

Developing analytical models for thermodynamic predictions of cryogenic fuel tanks has not received a large amount of attention. However, a number of people have done work in this field. Grayson and Navickas[1] performed an axisymmetric Navier-Stokes analysis of a liquid hydrogen propellant tank. Their analysis treated the cryogenic liquid-pressurization gas interface as a free boundary and therefore the ullage region of the tank was not modeled. Their results show that sloshing is an important factor in predicting tank thermal gradients. Zhou and Graebel[2] developed an axisymmetric model for a cylindrical tank draining process. They used a boundary integral method and assumed the fluid was incompressible and inviscid. Heat transfer was not considered, as they were interested in the free surface motion near the tank drain hole. Several thermodynamic models[3,4,5] have been developed for analyzing cryogenic tanks. These models are based upon quasi-steady-state solutions of the first law of thermodynamics.

The computational fluid dynamic (CFD) model presented herein simulates the two-dimensional cross section of a cryogenic tank draining process. The CFD model is developed in generalized curvilinear body-fitted coordinates to allow for a variety of cross-sectional shapes. The time-dependent Navier-Stokes equations are modeled for both the cryogenic liquid and the pressurization gas. The interface between the pressurization gas and cryogenic liquid is modeled as a free surface to enable the prediction of slosh wave dynamics. The energy transfer across the interface is calculated, but mass transfer is neglected. To reduce stiffness and decrease the computational time the method of artificial compressibility developed by Chorin[6] is used. The development of the finite difference equations utilized a volume integral procedure as discussed by Lick[7]. The mathematical details of the algorithms development are presented in the Greer reference[8]. Use of trade names or names of manufacturers in this document does not constitute an official endorsement of such products or manufacturers, either expressed or implied, by the National Aeronautics and Space Administration.

MODEL COMPARISON TO EXPERIMENTAL DATA

A series of calculations were made to model the free convection process inside a horizontal cylinder. Experimental data from Martini and Churchill[9,10] is available for comparison. Briefly, the experiment of Martini and Churchill was for a horizontal cylinder which was 1 meter long and had an inside diameter of 0.1 meter. Each hemispherical side of the cylinder was held at constant but separate temperatures using a dual-sided water bath. After steady state was reached, temperature and velocity measurements were made at the midspan of the cylinder. The fluid inside the cylinder was air.

An 8-sided polygon was used to simulate the experiment as shown in Figure 1. Calculations were performed for a Reynolds cell number of 5 (100 by 100 node grid), Courant-Friedrich-Levy number of 0.2, and Mach scaling factor of 250 (0.05 Mach number). Figures 2 and 3 present the velocity vectors and temperature contours. These figures show that the free convective flow travels in a boundary layer at the cylinder wall. The interior of the cylinder is relatively motionless. This is exactly what Martini and Churchill observed in their experiments. They concluded that the buoyancy forces were stronger than the viscous forces in the interior region. The temperature contours show that the fluid stratifies in the interior, which was experimentally observed. Figures 4 and 5 compare the velocity and temperature boundary layer profiles to the experimental data. The characteristics of the flow field between the model and the experimental data are in very good agreement.

FUEL TANK ANALYSIS

An 8-sided polygon was chosen as representative of a circular tank. (A circle can not be transformed into generalized coordinates because of the singular points at the corners.) The boundary and initial conditions are shown in Figure 6. The geometry for the 8-sided polygon is the same for all calculations. The drain calculations begin with the tank 70-percent full and the calculations are terminated when the tank is 30-percent full as was done for the rectangular tank. The initial and final grid geometry are shown in Figures 7 and 8. The fluids used are helium and liquid hydrogen. Calculations were performed for a Reynolds cell number of 8.2 (40 by 40 node grid), Courant-Friedrich-Levy number of 0.1, and Mach scaling factor of 100 (0.01 Mach number). The CPU time was 16 hours for each drain calculation performed on a Sun Ultra computer with a clock speed of 300MHz.

Figures 9 through 14 present velocity vectors of drain calculations for wall heat fluxes of 0, 1, and 2 J/m^2-s. The addition of heat flux at the wall causes a pair of convection vortices to form in the gaseous helium. As shown in the figures, the heat flux increases the helium flow velocities. The maximum flow velocities at the vertical side wall is 0.0017 m/s for q=0 J/m^2-s, 0.0035 m/s for q=1 J/m^2-s, and 0.0051 m/s for q=2 J/m^2-s. Figures 15 and 16 show temperature contours for the two cases with heat flux. The temperatures are symmetric and increase with higher heat flux. A secondary flow pattern develops in the gas region at the interface (near the center) for the q=2 J/m^2-s case (fig 14). This secondary flow pattern is a result of increased flow velocity inside the tank. It is caused by the wall heat flux and the geometry of decreasing tank width in the vicinity of the interface. Calculations with a rectangular geometry revealed no secondary flow patterns.

CONCLUDING REMARKS

A computational fluid dynamics model was developed to support the design, test, and analysis of cryogenic fuel tanks. This model uses a time-dependent finite difference technique in generalized coordinates. The finite difference algorithms were developed utilizing a volume integral method. To allow for the prediction of slosh wave dynamics, the interface between the liquid and the gas was modeled as a free surface. Artificial compressibility was used to decrease computational times. The model data compared well to experimental data for free convection inside a horizontal cylinder. A tank draining analysis was performed on an 8-sided polygon. Wall heat flux was found to be significant in the ullage gas, while it was insignificant in the liquid. This result is a function of the tank analysis parameters (drain rate, wall heat flux, etc.) and is not a general result for cryogenic fuel tanks. A secondary flow pattern was found to develop in the ullage gas for a large wall heat flux.

REFERENCES

1. Grayson, G. D. and Navickas, J., Interaction Between Fluid-Dynamic and Thermodynamic Phenomena in a Cryogenic Upper Stage, AIAA-93-2753, AIAA 28th Thermophysics Conference, 1993.
2. Zhou, Qiao-Nain and Graebel, W. P., "Axisymmetric Draining of a Cylindrical Tank With a Free Surface," *J. Fluid Mech.*, vol. 221, 1990, pp. 511–532.
3. Willen, Scott, Hanna, Gregory J., and Anderson, Kevin R., "Cryogenic Tank Analysis Program," *Proceedings of the Eighth Annual Thermal and Fluid Analysis Workshop*, 1997.
4. Van Dresar, Neil T. and Haberbusch, Mark S., "Thermodynamic Models for Bounding Pressurant Mass Requirements of Cryogenic Tanks," *Cryogenic Engineering Conference*, Albuquerque, New Mexico, 1993.
5. Danilowicz, R. L., Temperature Profiles in a Propellant Tank, computer program, LEW-11034, NASA Lewis Research Center, COSMIC, 1994.
6. Chorin, Alexandre Joel, "A Numerical Method for Solving Incompressible Viscous Flow Problems," *Journal of Computational Physics*, vol. 2, 1967, pp. 12–26.
7. Lick, Wilbert J., *Difference Equations from Differential Equations*, Springer-Verlag, New York, New York, ISBN 0-387-50739-6, 1989.
8. Greer, Donald Stephen, *Numerical Modeling of the Flow in a Cryogenic Fuel Tank*, Ph. D. Thesis, Mechanical and Environmental Engineering Department, University of California, Santa Barbara, 1998.
9. Martini, William R. and Churchill, Stuart W., "Natural Convection Inside a Horizontal Cylinder," *AIChE Journal*, vol. 6, No. 2, 1960, pp. 251–257.
10. Martini, William R., *Natural Convection Inside a Horizontal Cylinder*, Ph. D. Thesis, University of Michigan, Ann Arbor, Michigan, 1957.

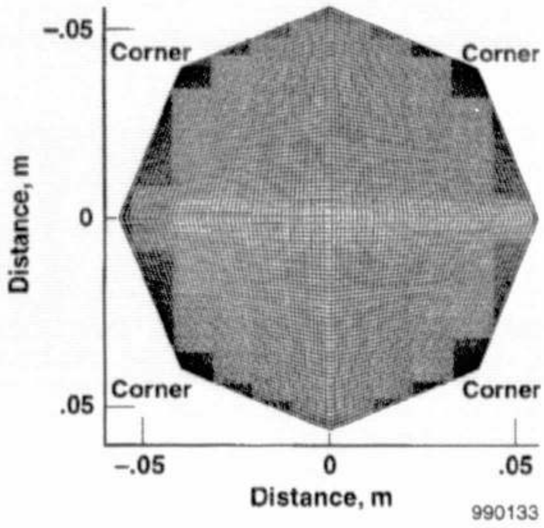

Figure 1. An 8-sided polygon for free convection inside a horizontal cylinder, 100 by 100 node grid.

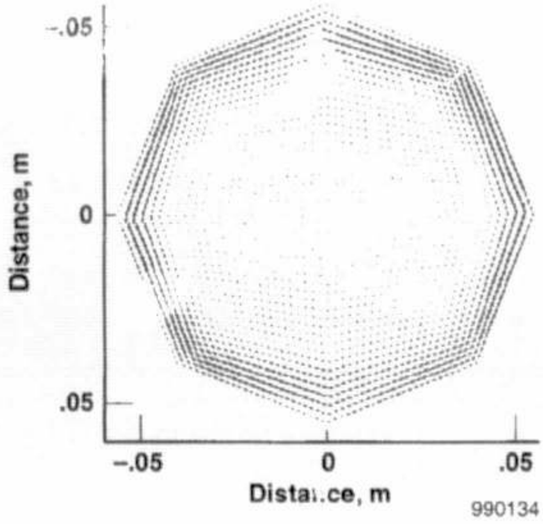

Figure 2. Velocity vectors at steady state for free convection inside a horizontal cylinder, vector skip index of 2.

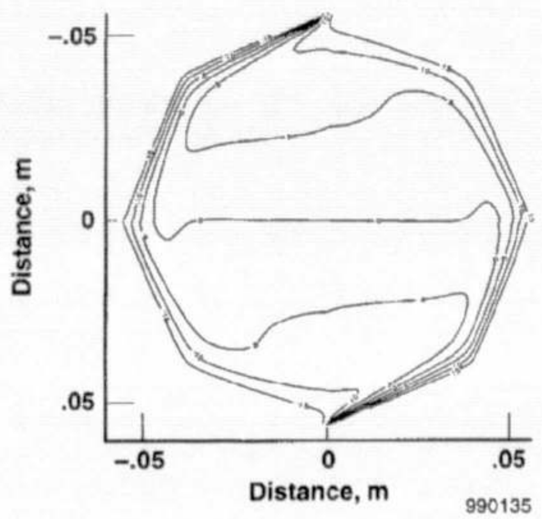

Figure 3. Temperature contours, T-Tavg, at steady state for free convection inside a horizontal cylinder.

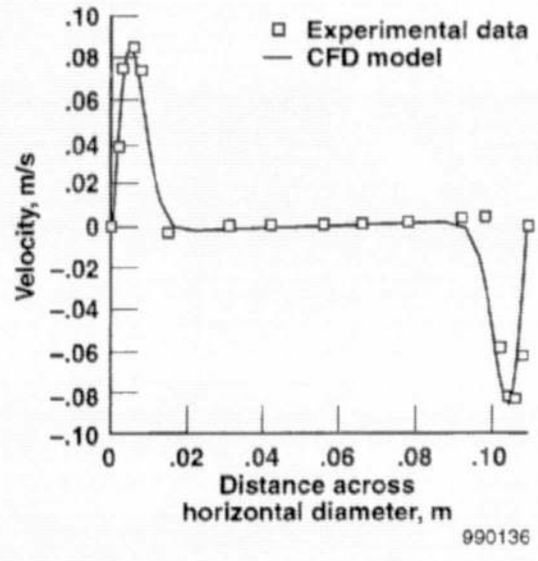

Figure 4. Comparison of velocity profiles across the horizontal diameter to experimental data of Martini and Churchill. Steady state free convection inside a horizontal cylinder.

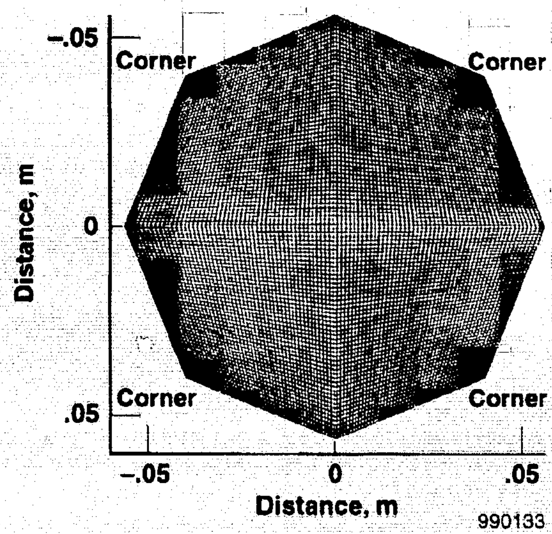

Figure 1. An 8-sided polygon for free convection inside a horizontal cylinder, 100 by 100 node grid.

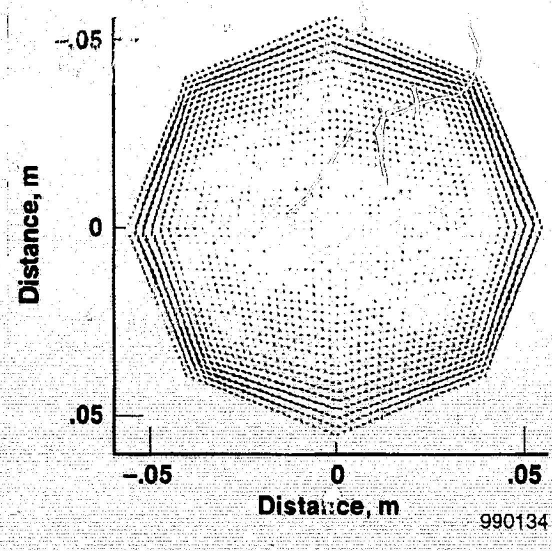

Figure 2. Velocity vectors at steady state for free convection inside a horizontal cylinder, vector skip index of 2.

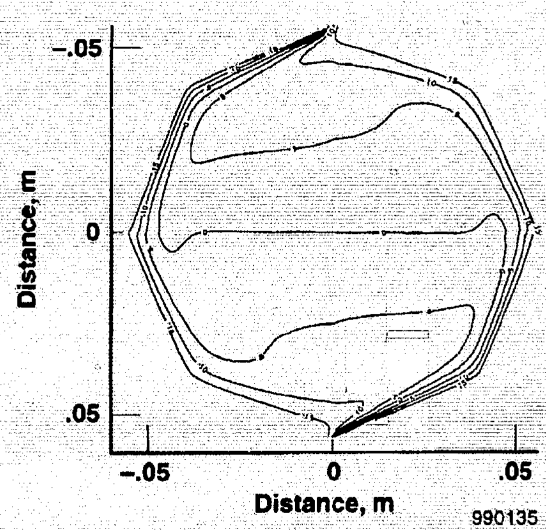

Figure 3. Temperature contours, T-Tavg, at steady state for free convection inside a horizontal cylinder.

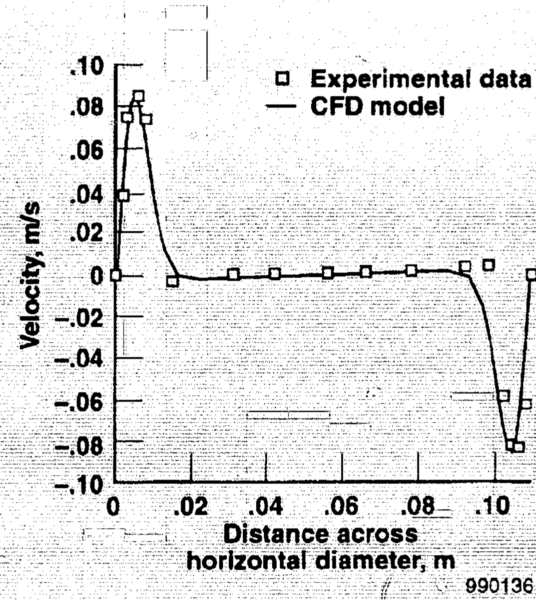

Figure 4. Comparison of velocity profiles across the horizontal diameter to experimental data of Martini and Churchill. Steady state free convection inside a horizontal cylinder.

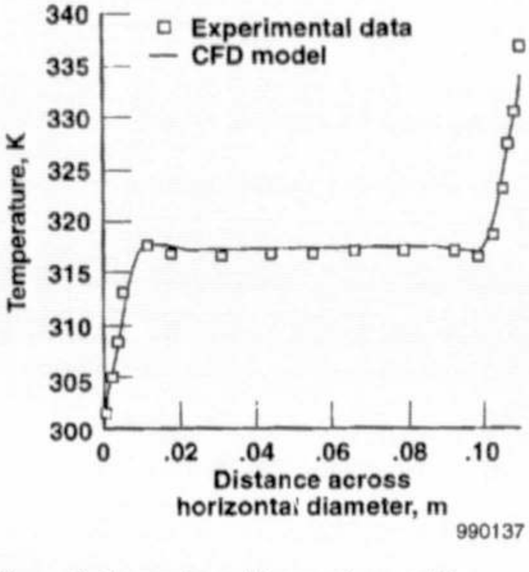

Figure 5. Comparison of temperature profiles across the horizontal diameter to experimental data of Martini and Churchill. Steady state free convection inside a horizontal cylinder.

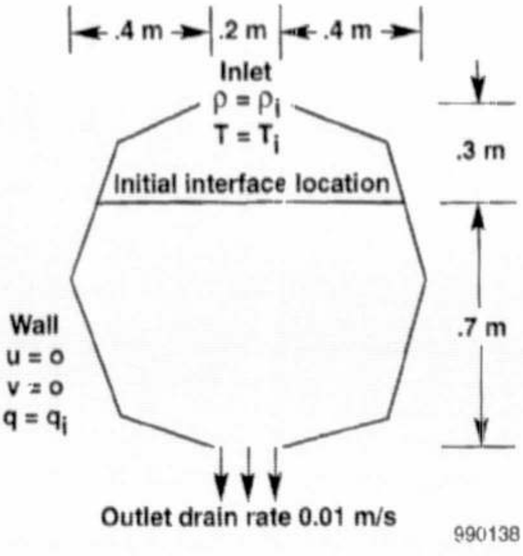

Figure 6. Geometry, boundary, and initial conditions for 8-sided polygon tank analysis. Tank is symmetric and 1 m by 1 m.

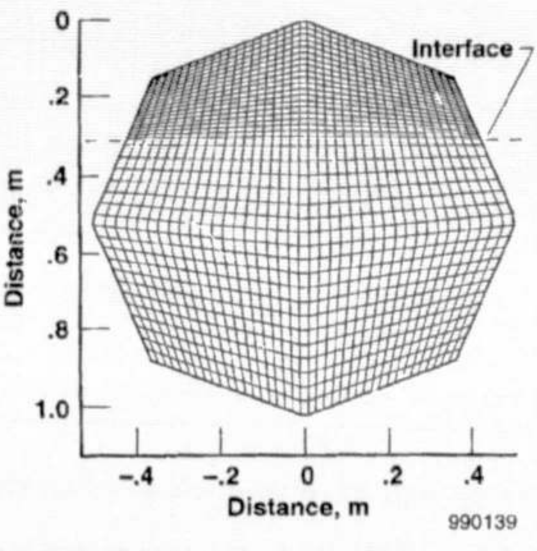

Figure 7. Initial 8-sided polygon grid. Drain time = 0 seconds.

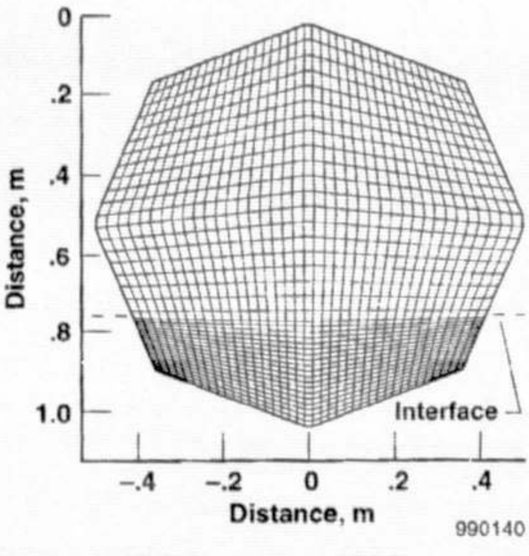

Figure 8. Final 8-sided polygon grid. Drain time = 300 seconds.

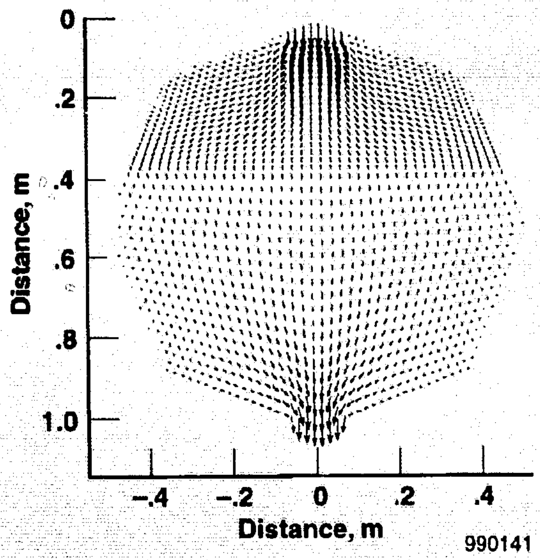

Figure 9. Velocity vectors at 50 seconds. He/LH2, q=0 J/m2-s.

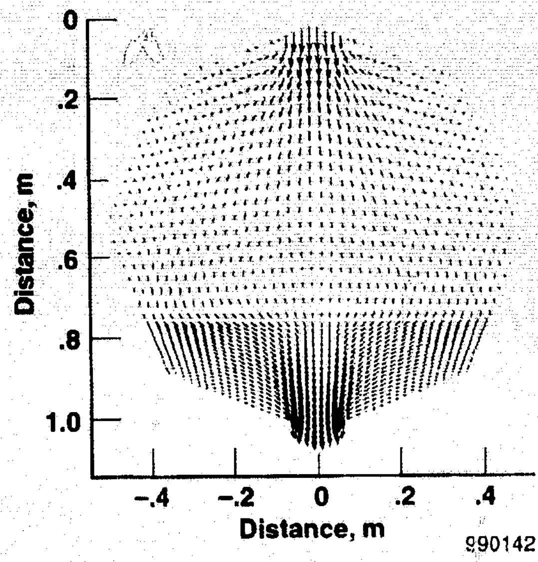

Figure 10. Velocity vectors at 300 seconds. He/LH2, q=0 J/m2-s.

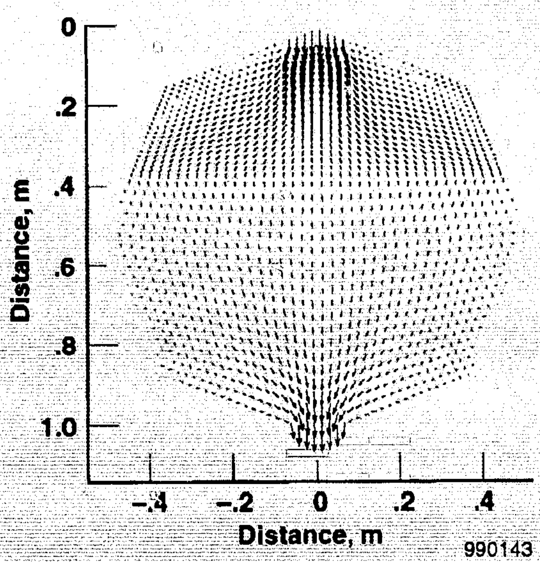

Figure 11. Velocity vectors at 50 seconds. He/LH2, q=1 J/m2-s.

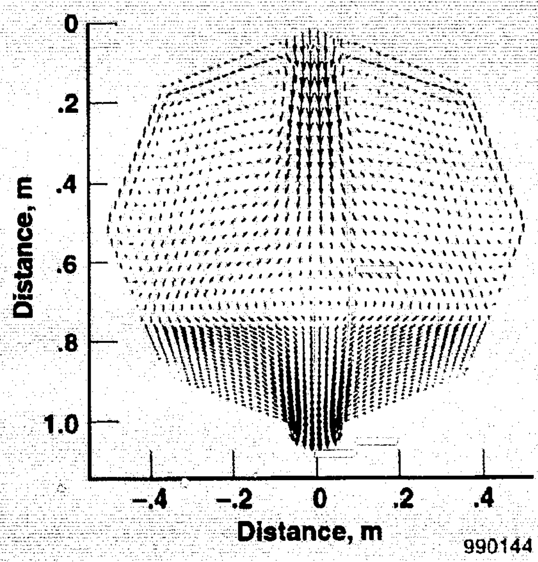

Figure 12. Velocity vectors at 300 seconds. He/LH2, q=1 J/m2-s.

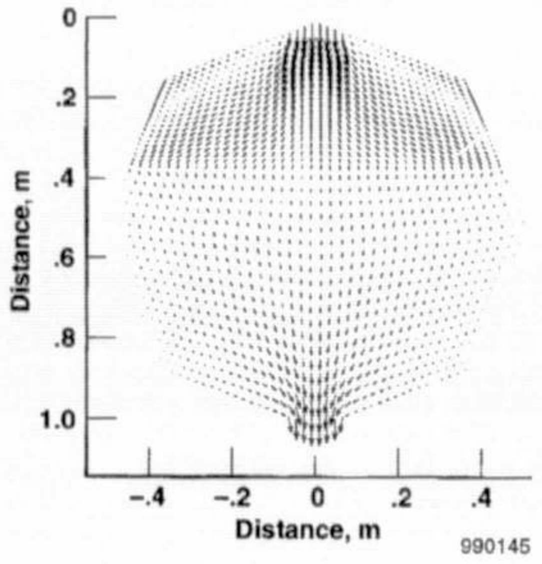

Figure 13. Velocity vectors at 50 seconds. He/LH2, q=2 J/m2-s.

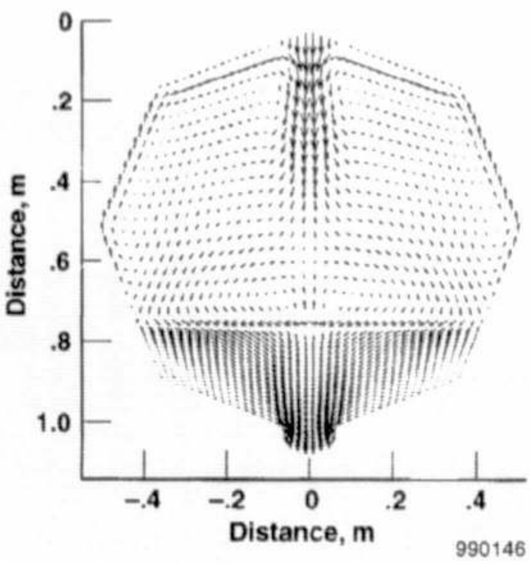

Figure 14. Velocity vectors at 300 seconds. He/LH2, q=2 J/m2-s.

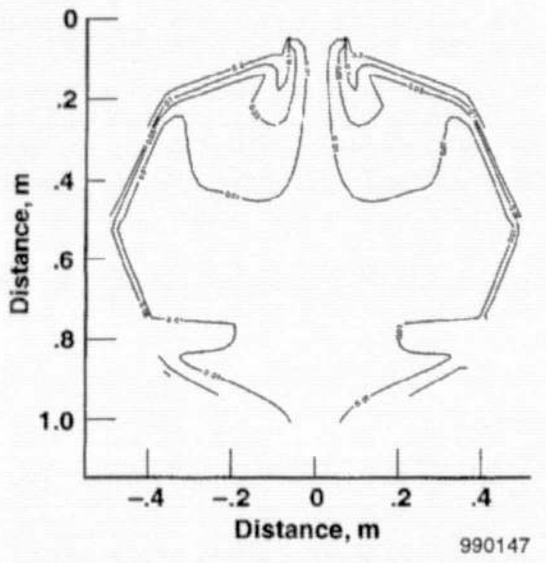

Figure 15. Temperature contours, $T - T_i$, at 300 seconds. He/LH2, q=1 J/m2-s.

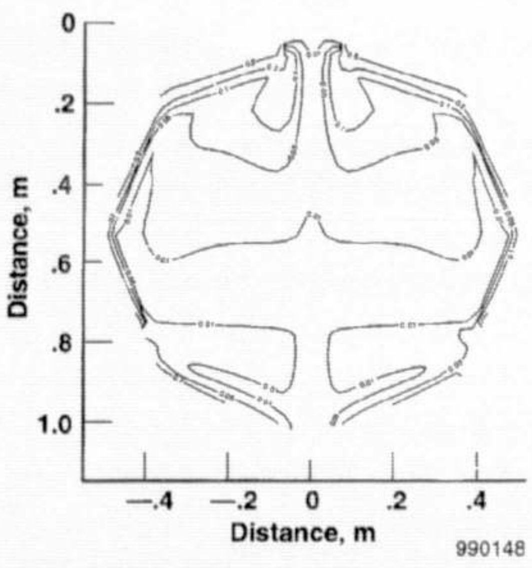

Figure 16. Temperature contours, $T - T_i$, at 300 seconds. He/LH2, q=2 J/m2-s.

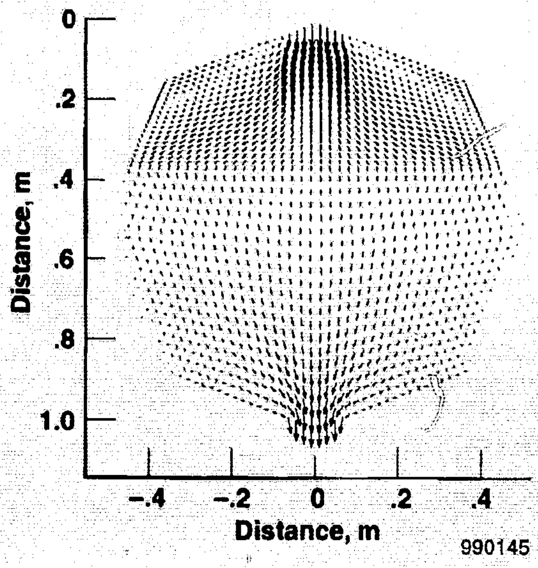

Figure 13. Velocity vectors at 50 seconds. He/LH2, q=2 J/m2-s.

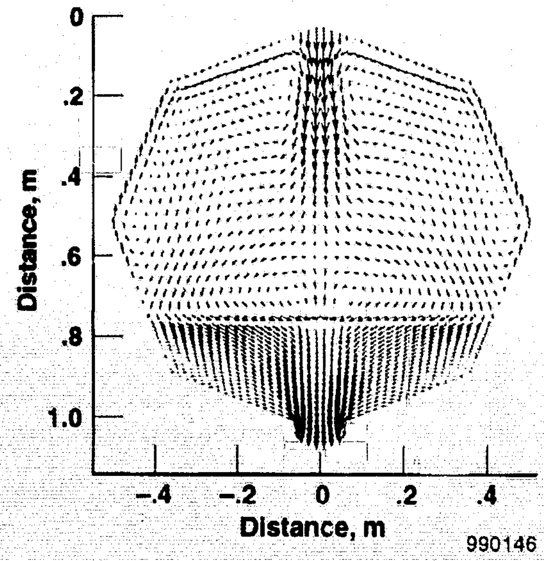

Figure 14. Velocity vectors at 300 seconds. He/LH2, q=2 J/m2-s.

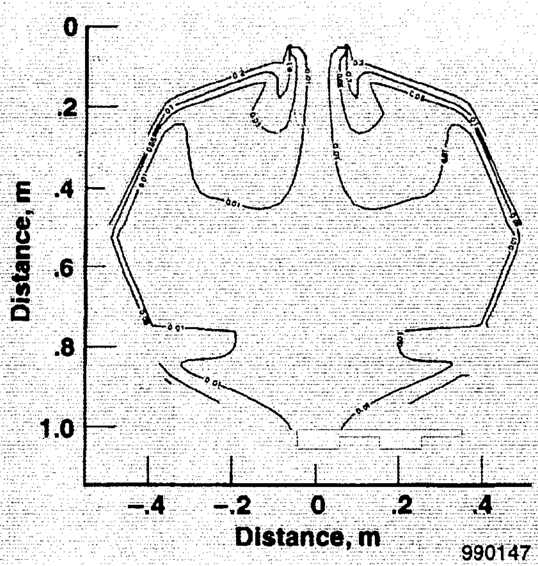

Figure 15. Temperature contours, $T - T_i$, at 300 seconds. He/LH2, q=1 J/m2-s.

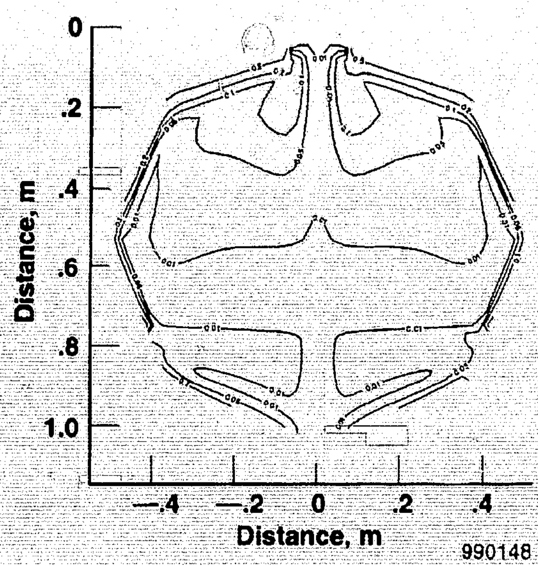

Figure 16. Temperature contours, $T - T_i$, at 300 seconds. He/LH2, q=2 J/m2-s.