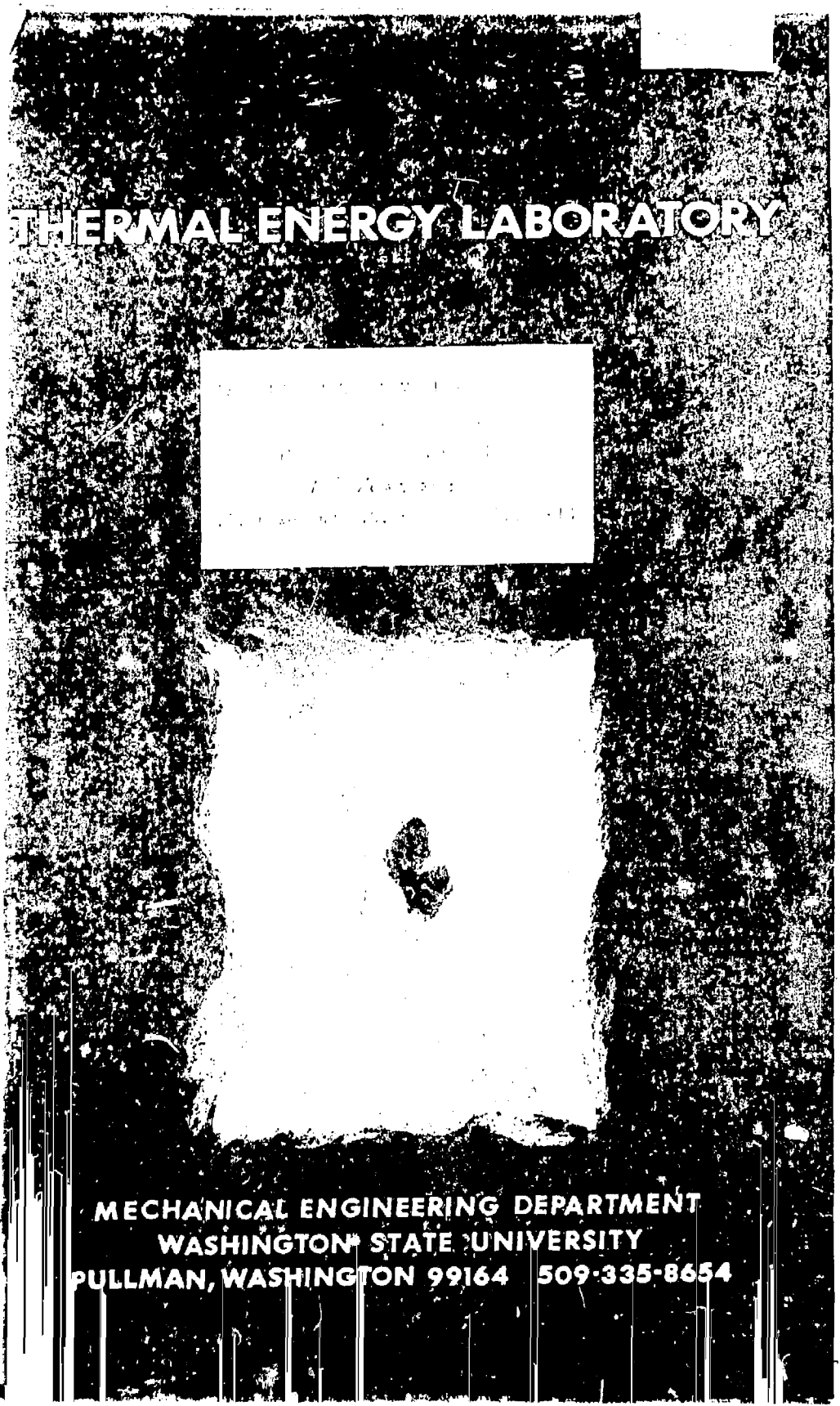
THERMAL ENERGY LABORATORY
MECHANICAL ENGINEERING DEPARTMENT
WASHINGTON STATE UNIVERSITY
PULLMAN, WASHINGTON 99164 509-335-8654

MASTER

TWO DIMENSIONAL NUMERICAL MODEL FOR STEAM-WATER FLOW IN A SUDDEN CONTRACTION

C. T. Crowe and H. N. Choi

Washington State University, Pullman, WA, USA

ABSTRACT

A computational model developed for two-dimensional dispersed two-phase flows is applied to steam-water flow in a sudden contraction. The calculational scheme utilizes the cellular approach in which each cell is regarded as a control volume and the droplets are regarded as sources of mass, momentum and energy to the conveying (steam) phase. The predictions show how droplets channel in the entry region and affect the velocity and pressure distributions along the duct.

NOMENCLATURE

A	cross-sectional area (m^2)
A_d	droplet area (m^2)
C_D	drag coefficient
c_d	specific heat of droplet (kJ/kg K)
d	droplet diameter (m)
G	mass flux (kg/s)
L	latent heat of vaporization (kJ/kg)
M^x	momentum flux in x-direction (kg m/s^2)
$\Delta \dot{m}_d$	net mass efflux rate of droplets (kg/s)
ΔM_d^x	net x-momentum efflux of droplets from cell (kg m/s^2)
m_d	mass of droplet (kg)
P	pressure (Pa)
Re	Reynolds number
S_p^x	momentum source in x-direction (kg m/s^2)
T	temperature (K)
t	time (s)
$\vec{U}$	vapor velocity (m/s)
u	x-component of vapor velocity (m/s)
$\vec{v}$	droplet velocity (m/s)
X_i	fraction of droplet mass entering at port i

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Department of Energy, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

x_i space coordinate (m)

Y_i fraction of droplet mass associated with initial diameter d_i

n_i number flow rate of droplets along trajectory i (s^{-1})

μ coefficient of viscosity (Ns/m^2)

ρ density (kg/m^3)

τ characteristic time (s)

Subscripts

d droplet

I,J index to indentify grid node

v vapor

INTRODUCTION

The flow of steam-water mixtures through ducts is of interest in power systems using water as the energy transfer medium such as fossil fuel and nuclear power plants. Also, the utilization of geothermal resources is dependent on the conveyance of steam-water mixtures through ducts.

The design of piping systems to carry steam-water mixtures is primarily empirical. The losses associated with bends, tees and other fittings are estimated using results from similar designs, or predicted based on a correlation parameter established by experimentation. The complexity of the phenomena constituting the flow of steam-water mixtures has precluded the development of analytic models to predict flow characteristics and head losses.

The purpose of this paper is to illustrate the application of the Particle-Source-In-Cell (PSI-Cell) model to analyze disperse steam-water (vapor-droplet) flow in a sudden contraction section. The generality of the model suggests its adaptability to other configurations and ultimately a better understanding of, and predictive capability for, steam-water flow in ducts.

BASIC CONCEPT

In order to apply the PSI-Cell model, it is first necessary to sub-divide the flow field into a series of cells, as shown in Fig. 1. Each cell is regarded as a control volume for the vapor phase. As droplets traverse a given cell in the flow field, they may be:

1. evaporating or condensing, resulting in a source (or sink) of vapor mass to the fluid in the cell,
2. accelerating or decelerating, resulting in a momentum augmentation or deficiency in the vapor in the cell in the direction of droplet motion.

Finite-difference equations for mass and momentum conservation are written for each cell, incorporating the contribution due to the droplets. The continuum flow field thus is analyzed utilizing the Eulerian approach, which is the most straightforward approach for analyzing continuum flows. The entire flow-field solution is obtained by solving the system of algebraic equations for each cell.

The droplet trajectories, size and temperature history are obtained by integrating the equations of motion for the droplets in the vapor flow field. Solving for the droplet velocity, size and temperature along droplet trajectories is done using the Lagrangian approach, which is the most straightforward approach

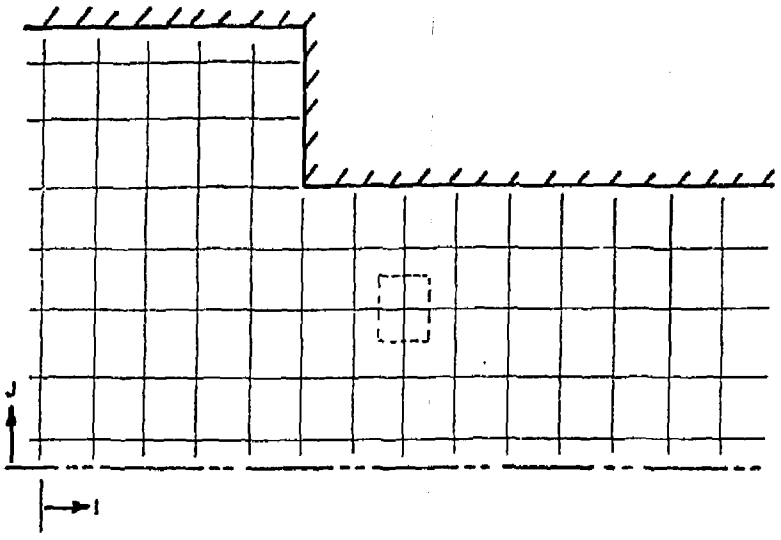

Fig. 1. Flow Field Grid System

for the droplet phase. Recording the mass and momentum of the droplets on crossing cell boundaries provides the droplet source terms for the vapor flow equations.

The flow of steam-water mixture is assumed adiabatic. Also, the velocities are sufficiently low that the kinetic energy can be neglected compared to the thermal energy. Hence, the flow is taken as isoenthalpic. Best-fit curves are used to relate the steam density and temeprature to the local pressure along the vapor saturation line.

The complete solution for a vapor-droplet flow field is executed as follows: The calculation is begun by solving the vapor flow field assuming no droplets are present. Using this flow field, droplet trajectories together with size and temperature histories along the trajectories are calculated. The mass and momentum source terms for each cell throughout the flow field then are determined. The vapor flow field is solved again, incorporating these source terms. The new vapor flow field is used to establish new droplet trajectories and temperature histories, which constitute the effect of the vapor phase on the droplets. Calculating new source terms and incorporating them into the vapor flow field equations constitutes the effect of the droplet cloud on the vapor phase, thereby completing the cycle of mutual interaction or "two-way" coupling. After several iterations, the flow field equations are satisfied to within a predetermined value and the solution which accounts for the mutual interaction of the droplets and vapor is obtained.

VAPOR FLOW EQUATIONS

The basic equations governing the droplet-vapor flow field for the computational model described here are derived using the principles of conservation of mass and momentum. The flow field of interest is subdivided into rectangular cells, and each cell is treated as a control volume for the analysis. The technique adopted here is an extension of the TEACH program [1] developed at Imperial College of Science and Technology in London.

Mass Conservation Equation

Refer to the cell shown in Fig. 2, which encloses a typical node (I,J). The four faces of this cell are identified as points on a compass, -- N, S, E, W. The continuity equation for the steady flow of a vapor-droplet mixture through this cell (control volume) is

$$G_N - G_S + G_E - G_W + \dot{\Delta m}_d = 0 \qquad (1)$$

where G_i is the vapor mass flow rate through face "i" and $\dot{\Delta m}_d$ is the net mass efflux rate of the droplets for the cell. The vapor mass flux through the west face is given by

$$G_W = u_W A_W \frac{\rho_{I,J} + \rho_{I-1,J}}{2} \qquad (2)$$

where u_W is the velocity component in the x-direction on the west face and A_W is the cross-sectional area of the cell. A linear variation in density between the (I,J) and (I-1,J) nodes is assumed. The vapor mass flow through the remaining faces is calculated in the same way.

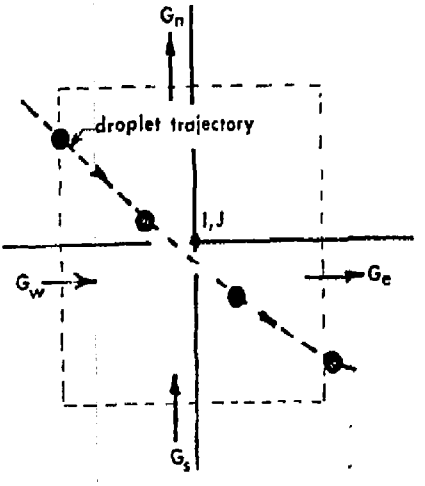

Fig. 2. Typical Computational Cell

One notes that the droplet mass term, $\dot{\Delta m}_d$, in Eqn. 1 can be regarded as a source (or sink) of mass to the gaseous phase, which is the basis of the PSI-Cell model.

Momentum Equations

Having located the cells for the continuity equation as enclosing the nodal points, the cells for the momentum equations must be located between nodes. Referring to Fig. 2, one notes that the u component velocity in the continuity equation is not the velocity at the nodal point, but the velocity halfway between nodes. Consequently, the control volume used to solve for u from the x-momentum equation is displaced from that used for the continuity equation and lies between grid lines I and I-1. The velocity, however, is identified by the node I,J.

The momentum equation in the x-direction for steady flow requires that

$$-M^x_W + M^x_E - M^x_S + M^x_N + \Delta M^x_p = (P_{I-1,J} - P_{I,J})\, A_W + S^x_p \qquad (3)$$

where M^x_i is the momentum of the vapor in the x-direction across the "i-th" face. ΔM^x_p is the net efflux of x-momentum from the cell due to the droplets , $P_{I,J}$ is the pressure at node (I,J) and S^x_p arises from the variation in effective viscosity in the flow field [2] and from body forces such as gravity.

The momentum flux is due to two mechanisms: convection and diffusion. The momentum flux in the x-direction across the "i-th" face can be written as

$$M^x_i = G_i u - \mu A_i \, (\partial u / \partial x_i) \qquad (4)$$

where A_i is the area of face "i" and x_i is the spatial coordinate in the i-direction (normal to the area A_i). This differential expression is expressed in a finite difference form depending on the magnitude of the cell Reynolds number. If the magnitude of the cell Reynolds number is two or greater, upwind differencing is used; if not, central differencing is used.

Substituting the finite difference equations for momentum flux into Eqn. (3) yields

$$u\Sigma a_k = \Sigma a_i u_i + A_W (P_{I-1,J} - P_{I,J}) - \Delta M_p^x + S_p^x \qquad (5)$$

where a_i are the resulting velocity coefficients, and the summation is performed over all faces of the cell. Note that the net efflux of particle momentum can be regarded as a body force acting on the gaseous phase.

The cell used for the y-momentum equation is located below and midway between the nodes (I,J) and (I,J-1). Equivalent finite-difference equations are derived for the y-component of velocity v as a function of the velocities v_i, the pressure gradient and the droplet momentum efflux in the y-direction. The resulting velocity component is identified by the node (I,J).

Turbulence Equations

Two more equations are used in the computer model to account for turbulence and to determine the effective viscosity. The turbulence field is described by the local intensity of turbulence and the dissipation rate. The finite-difference form of the equations is incorporated into the program to yield the kinetic energy of turbulence and the dissipation rate as related to the local mean-velocity field according to the scheme proposed by Launder and Spalding [3]. Having evaluated the turbulence intensity and dissipation rate in a cell, the effective viscosity is determined, using the Prandtl-Kolmogorov formula.

No attempt is made to include the effect of the droplets on the turbulence field, in that little is known as to the quantitative effect of particle size and concentration on the turbulence parameters.

Solution of the Vapor Flow Equations

The solution scheme is initiated by assuming a pressure and velocity field. The initial density and temperature field correspond, though the vapor saturation conditions, to the assumed pressure field. The momentum equations are then solved to yield a new velocity field. The new velocities are then used to evaluate mass fluxes and solve for a new pressure field. The density and temperature are updated to correspond to the new pressure field. The cycle is repeated by returning to the momentum equations to solve for a new velocity field. The solution is complete when the finite difference equations are satisfied to within a predetermined value. The reader is referred to the original reference [1] for more complete details on the solution scheme.

DROPLET EQUATIONS

In order to evaluate the source terms in the vapor phase flow equations due to the presence of droplets, it is necessary to establish droplet trajectories, size and temperature histories. This is accomplished by integrating the droplet equation of motion and using equations for droplet temperature and size. The velocity, pressure and temperature field of the gas is used in these calculations.

The equation of motion of a droplet is given by

$$m_d \frac{d\vec{v}}{dt} = C_D \rho_g (\vec{U} - \vec{v}) |\vec{U} - \vec{v}| \frac{A_d}{2} \quad (6)$$

where C_D is the drag coefficient and A_d is the droplet area. The other terms contributing to aerodynamic forces on the droplet -- namely, the pressure gradient, virtual mass and Basset term -- are neglected because they are of the order of the vapor/droplet material density ratio which is small. The Saffman lift and Magnus forces also are neglected because the droplets are not in a high-shear region of the gas flow.

The drag coefficient for a droplet depends primarily on the Reynolds number based on the gas-droplet relative velocity,

$$Re = \rho_g \frac{|\vec{U} - \vec{v}| d}{\mu} \quad (7)$$

where d is the droplet diameter. For a non-evaporating droplet, the drag coefficient can be represented reasonably well by [4]

$$C_D = \left(\frac{24}{Re}\right) (1 + 0.15\ Re^{0.687}) \quad (8)$$

for Reynolds numbers up to 1,000. Evaporation can reduce the drag coefficient due to mass flux from the surface, but its effect is negligible for the application considered here.

Considerable economy in computing time is realized if the droplet trajectory equation is integrated once analytically. Rewriting the above equation for droplet motion, one has

$$\frac{d\vec{v}}{dt} = \left(\frac{18\mu f}{\rho_d d^2}\right) (\vec{U} - \vec{v}) \quad (9)$$

where $f = C_D\ Re/24$. Integrating the equation, assuming the gas velocity is constant over the time of integration, yields

$$\vec{v} = \vec{U} + (\vec{U} - \vec{v}_o) \exp\left(\frac{-\Delta t}{\tau}\right) \quad (10)$$

where $\vec{v}_o$ is the initial droplet velocity, Δt is the time interval and τ is the characteristic time defined by

$$\tau = \frac{\rho_d d^2}{18 \mu f} \quad (11)$$

After determining the new droplet velocity at time Δt, the droplet position at time Δt is determined from

$$\vec{x}_d = \vec{x}_{d,o} + (\vec{v} + \vec{v}_o) \frac{\Delta t}{2} \quad (12)$$

where $x_{d,o}$ is the droplet position at the beginning of the time increment.

Because the steam-water mixture constitutes a single component, the temperature at the droplet surface must be the local vapor temperature. Hence, there is no (or negligible) heat transfer from the droplet surface to the vapor phase. Thus, the energy associated with a decrease in droplet temperature is absorbed by a change of phase at the vapor-droplet interface, therefore we have

$$\frac{dm_d}{dt} = \frac{m_d c_d}{L} \frac{dT_v}{dt} \tag{13}$$

or

$$m_d = m_{d_o} \exp\left(\frac{c_d \Delta T_v}{L}\right) \tag{14}$$

where m_{d_o} is the droplet mass at the beginning of the time step and ΔT_v is the change in vapor (and droplet) temperature over the time step.

DROPLET SOURCE TERMS

In the philosophy of the PSI-Cell approach, the droplets are regarded as sources of mass and momentum to the conveying vapor phase. The source terms are incorporated into the gas flow equations, providing the influence of the droplets on the vapor velocity and temperature fields.

As shown in the previous section, once the vapor flow field has been established, the droplet mass and momentum equations can be integrated using the Lagrangian approach to yield the droplet trajectories and the droplet diameter history along each trajectory.

The entry of the droplets is represented by a finite number of entry ports. The mass of droplet size, d_i, which enters per unit time at port j is given by

$$\dot{m}_{p_j}(d_i) = \dot{m}_p X_j Y_i \tag{15}$$

where $\dot{m}_p$ is the total droplet mass inflow rate. The number flow rate of droplets of initial diameter d_i along a given trajectory is determined by

$$\dot{n}_j(d_i) = 6\dot{m}_p X_j Y_i / m_d \tag{16}$$

If the Weber number, based on the relative velocity, exceeds six [5], breakup can occur. The droplet size is readjusted to yield a Weber number of six and the number flow rate changed accordingly.

The source term in the continuity equation, Δm_d, is the net efflux rate of droplet mass from a computational cell (control volume). The mass source due to trajectory "j" is given by

$$(\Delta \dot{m}_d)_j = \dot{n}_j (m_{out} - m_{in}) \tag{17}$$

where m is the mass of an individual droplet traveling along trajectory "j". The net source term for a cell is obtained by summing over all trajectories which traverse that cell.

The corresponding source terms for momentum are obtained in the same fashion.

APPLICATION

The numerical model is applied to a steam-water flow into a sudden contraction as shown in Fig. 1. The flow is assumed to enter the section at 20 MPa with a quality of 90%. The upstream tube diameter is 2.5 cm and the velocity is 20 m/s. The contraction ratio is .62. The initial droplet diameter is taken as 50 micrometers.

The predicted droplet trajectories are shown in Fig. 3. Sixteen starting

locations were used to represent the incoming droplet phase. One notes that the droplets tend to channel, leaving a droplet-free region adjacent to the wall in the sudden contraction section. This region is delineated by the limiting droplet trajectory. Droplets near the periphery of the duct impinge on the upstream face, after which it is assumed they no longer influence the flow field.

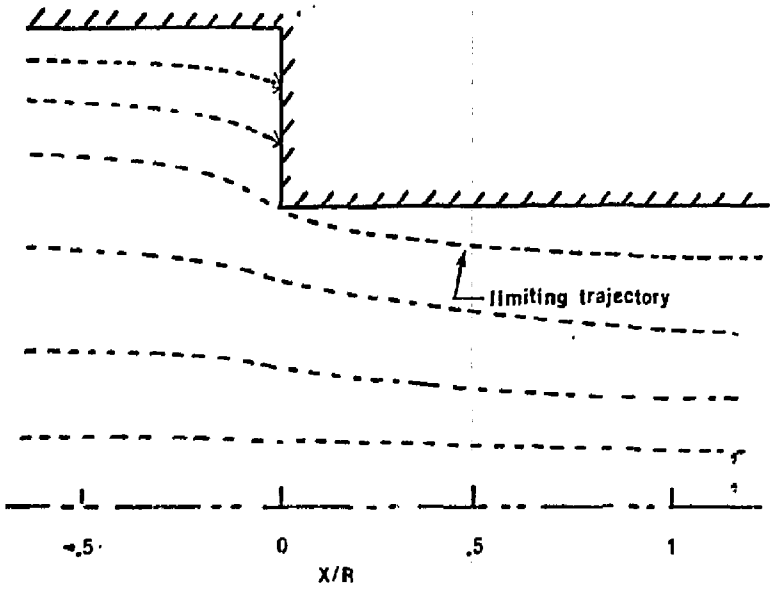

Fig. 3. Droplet Trajectories Near Transition Region

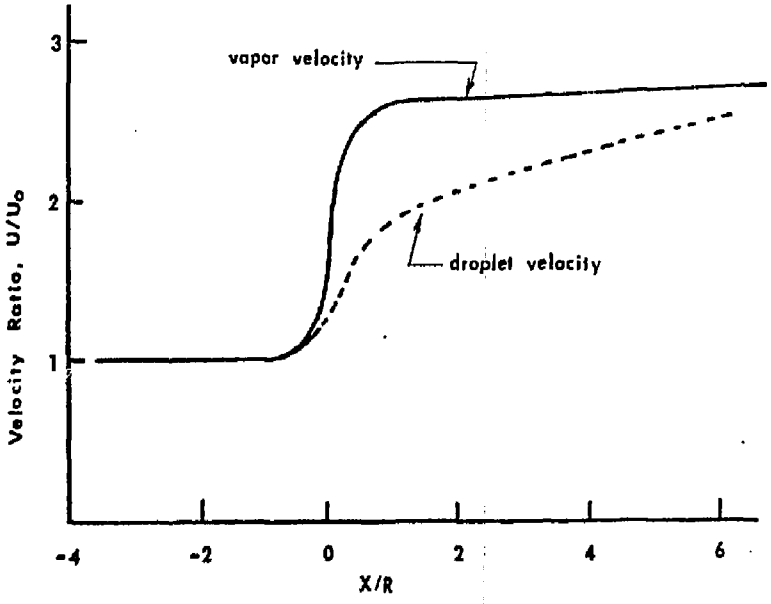

Fig. 4. Droplet-Vapor Velocity Distribution Near Centerline

The predicted droplet velocity for a droplet near the centerline is shown in Fig. 4. One notes the droplets cannot maintain kinetic equilibrium with the vapor in the transition region. The droplet velocity continues to increase as the mixture proceeds downstream.

The predicted pressure coefficient distribution across the transition is shown in Fig. 5. The pressure coefficient is referenced to the upstream pressure and based on the upstream kinetic pressure. The pressure drop is larger for the two-phase mixture, as expected, because of the augmented pressure needed to balance the droplet momentum sink. Also, the trend toward pressure recovery corresponds with preliminary data currently being analyzed by the authors.

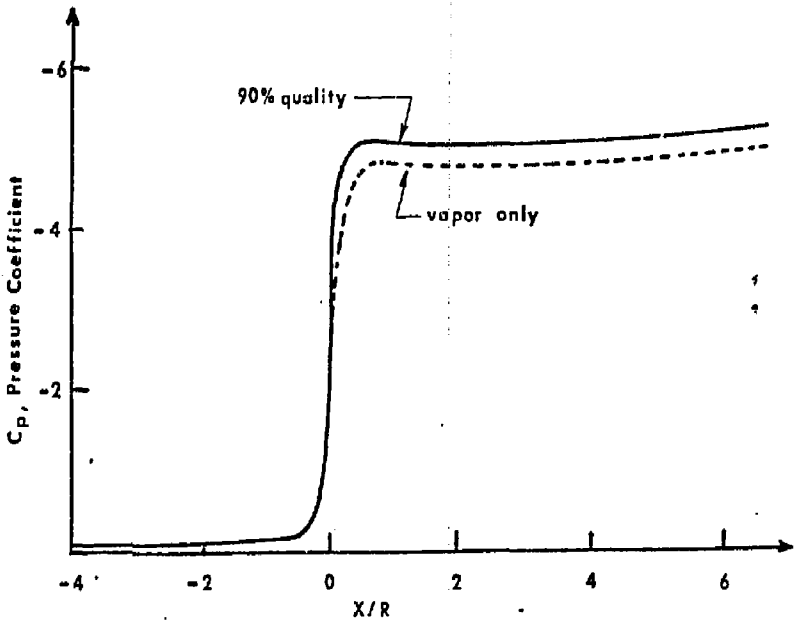

Fig. 5. Pressure Coefficient Distribution

At the present time, the solution scheme is being extended to low quality mixtures ($\sim$10%).

CONCLUSION

The PSI-Cell model represents a viable numerical model for two-dimensional single-component two-phase flows. The solution scheme provides a means whereby mass, momentum and energy coupling between phases can be adequately accounted for.

ACKNOWLEDGEMENTS

The support of the Geothermal Group of the Lawrence Livermore Laboratory and contract monitor, H. Weiss, is greatfully acknowledged.

REFERENCES

1. Gosman, A.D. and Fun, W.M., "Calculation of Recirculative Flows", Lecture Notes, Imperial College of Science and Technology, London, December 1973.

2. Wormeck, J.J., "Computer Modeling of Turbulent Combustion in a Longwell Jet-Stirred Reactor", Ph.D. Thesis, Washington State University, 1976.

3. Launder, B.E. and Spalding, D.B., *Mathematical Models of Turbulence*, Academic Press, 1972.

4. Wallis, G.B., *One-Dimensional Two-Phase Flow*, McGraw-Hill Book Co., 1969.

5. Lane, W.R., "Shatter of Drops in Streams of Air," *Ind. Engr. Chem*, Vol.43, 1951, pp. 1312-1317.

Dist

TID 2

TIC 2

NOTICE

"This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Department of Energy, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately-owned rights."