TITLE: THE THREE-DIMENSIONAL TWO-FLUID NUMERICAL TREATMENT OF A REACTOR VESSEL IN TRAC

AUTHOR(S): D. R. Liles

SUBMITTED TO: American Nuclear Society National Topical Meeting on Computational Methods in Nuclear Engineering, Williamsburg, VA, April 23-25, 1979
A three-dimensional two-fluid finite difference model has been used in TRAC (Transient Reactor Analysis Code) to represent a pressurized water reactor vessel. Mesh cells may be blocked off completely to represent large flow obstructions such as downcomer walls. The hydrodynamic volumes and flow areas may also be reduced in order to provide a porous matrix simulation of smaller scale structure.

The finite difference equations are semi-implicit so that stability time scales are associated with material movement and not wave propagation. The block matrix structure is reduced during the implicit pass to a single element seven stripe system which is easily solved iteratively.

This procedure has successfully performed numerous simulations of both full sized reactor accidents and smaller scale experiments. It has proven to be a useful feature of the TRAC effort.
THE THREE-DIMENSIONAL TWO-FLUID
NUMERICAL TREATMENT OF A REACTOR VESSEL IN TRAC

INTRODUCTION

The TRAC (Transient Reactor Analysis Code) computer program models a Pressurized Water Reactor (PWR) vessel with a full cylindrical coordinates two-fluid finite difference representation. The purpose of going to this detail in a systems calculation is to try to better understand the three dimensional effects present during reactor accident transients. Such effects as downcomer bypass, preferential rewetting in lower power zones and sloshing oscillations in the lower plenum can be handled in a natural manner with the present representation.

A staggered semi-implicit finite difference scheme is employed with flow regime dependent constitutive equations. A porous matrix representation is used to represent the small scale internal structure while complete flow areas may be blocked out to provide a downcomer model.

The present code has successfully run quite a number of transients representing both full size PWR simulations and smaller scale experiments. Typical compute times on a CDC 7600 for the 3-D portion of the code are 2 ms per mesh-cell per time-step. Normal time steps range from of several milliseconds up to tens of milliseconds. In the present paper the TRAC subroutine names are mentioned to facilitate an inspection of the code. No formal results are provided in this paper due to the restricted length of the manuscript.

THREE-DIMENSIONAL VESSEL HYDRODYNAMICS

The field equations, finite difference scheme, solution procedure, and constitutive relations for the three-dimensional, two-fluid, hydrodynamics package are discussed in this section. Subroutines TF3D (Two-Fluid 3-Dimensional), ITR1 (an iteration subroutine), and FF3D (a subroutine that provides a final back-substitution) are the main subroutines of this package.

TF3D includes a constitutive package to provide wall and interfacial shears and the interfacial heat transfer. The wall heat transfer coefficients and temperatures are provided by HTCOR. TF3D also sets up the basic staggered finite difference scheme for the three-dimensional representation of the core, linearizes the algebraic equations, and provides a forward elimination. This information is then used in the iteration routine ITR1 and finally new pressures, void fractions, velocities (both vapor and liquid), and fluid temperatures (liquid and vapor) are computed in FF3D.
Differential Equations

The field equations\textsuperscript{1,2} describing the two-phase, two-fluid flow are given below.

\textbf{Mixture Mass Equation}

\[
\frac{\partial \rho_m}{\partial t} + \nabla \cdot [\alpha \rho_g \vec{V}_g + (1-\alpha) \rho_l \vec{V}_l] = 0 \tag{1}
\]

\textbf{Vapor Mass Equation}

\[
\frac{\partial (\alpha \rho_g)}{\partial t} + \nabla \cdot (\alpha \rho_g \vec{V}_g) = 0 \tag{2}
\]

\textbf{Vapor Equation of Motion}

\[
\frac{\partial \vec{V}_g}{\partial t} + \vec{V}_g \cdot \nabla \vec{V}_g = -\frac{c_i}{\rho_g} \frac{\nabla |\vec{V}_g|}{|\vec{V}_g|} - \frac{1}{\rho_g} \nabla p - \frac{\Gamma}{\alpha \rho_g} (\vec{V}_g - \vec{V}_{ig}) - \frac{c_{\omega g}}{\alpha \rho_g} \vec{V}_g |\vec{V}_g| + g \tag{3}
\]

\textbf{Liquid Equation of Motion}

\[
\frac{\partial \vec{V}_l}{\partial t} + \vec{V}_l \cdot \nabla \vec{V}_l = \frac{c_i}{(1-\alpha) \rho_l} \frac{\nabla |\vec{V}_l|}{|\vec{V}_l|} - \frac{1}{\rho_l} \nabla p + \frac{\Gamma}{(1-\alpha) \rho_l} (\vec{V}_l - \vec{V}_{il}) - \frac{c_{\omega l}}{(1-\alpha) \rho_l} \vec{V}_l |\vec{V}_l| + \vec{g} \tag{4}
\]

\textbf{Mixture Energy Equation}

\[
\frac{\partial [(1-\alpha) \rho_l e_l + \alpha \rho_g e_g]}{\partial t} + \nabla \cdot [(1-\alpha) \rho_l e_l \vec{V}_l + \alpha \rho_g e_g \vec{V}_g] = -p \nabla \cdot [(1-\alpha) \vec{V}_l + \alpha \vec{V}_g] + q_{\omega g} + q_{\omega l} \tag{5}
\]

\textbf{Vapor Energy Equation}

\[
\frac{\partial (\alpha \rho_g e_g)}{\partial t} + \nabla \cdot (\alpha \rho_g \vec{V}_g e_g) = -p \frac{\partial \alpha}{\partial t} - p \nabla \cdot (\alpha \vec{V}_g) + q_{\omega g} + q_{\omega l} + \eta_{sy} \tag{6}
\]
It is well known that the set of field equations described above have complex characteristics which represent a partially elliptic system. This system can be solved numerically with the present interfacial shear relations as long as the mesh size is not too small. It is recommended that nothing smaller than one centimeter (preferably larger) be used as a length scale for the mesh. This will normally be sufficient to keep the solution from developing instabilities. Mesh sizes smaller than one-centimeter may be used singly (for example, to represent radial downcomer gaps) as long as adjacent mesh cells are larger. The larger mesh cells will damp out any exponential growth in the adjoining smaller cells. Semiscale, LOFT, and full size PWRs have been modeled and run for thousands of cycles with no apparent difficulties which could be traced to unbounded growth of any of the variables.

**FINITE DIFFERENCE EQUATIONS**

The momentum equations are separated into the three coordinate components. Only the vapor equation will be discussed with the understanding that the liquid momentum equation is treated in an analogous manner. The three components of the vapor-momentum differential equation are:

**Axial (z) Component**

\[
\frac{\partial V_z}{\partial t} = - (V_x \frac{\partial V_z}{\partial x} + V_y \frac{\partial V_z}{\partial y} - g \frac{\partial V_z}{\partial z}) - \frac{1}{\rho g} \frac{\partial p}{\partial z} - \frac{c_{iz}}{\rho g} (V_{gz} - V_{i,z}) V_g - V_L \left| - \frac{r}{\rho g} (V_{gz} - V_{igz}) - \frac{c_{wz}}{\rho g} V_{gz} \right| + g
\]

**Radial (r) Component**

\[
\frac{\partial V_r}{\partial t} = - (V_x \frac{\partial V_r}{\partial x} + V_y \frac{\partial V_r}{\partial y} - \frac{\partial V_r}{\partial r}) - \frac{1}{\rho g} \frac{\partial p}{\partial r} - \frac{c_{ir}}{\rho g} (V_{gr} - V_{i,r}) V_g - V_L \left| - \frac{r}{\rho g} (V_{gr} - V_{igr}) - \frac{c_{wr}}{\rho g} V_{gr} \right| + g
\]

**Azimuthal (θ) Component**

\[
\frac{\partial V_{\theta}}{\partial t} = - (V_x \frac{\partial V_{\theta}}{\partial x} + V_y \frac{\partial V_{\theta}}{\partial y} + \frac{1}{\rho} \frac{\partial p}{\partial \theta}) - \frac{1}{\rho g} \frac{\partial p}{\partial \theta} - \frac{c_{i\theta}}{\rho g} (V_{g\theta} - V_{i\theta}) V_g - V_L \left| - \frac{r}{\rho g} (V_{g\theta} - V_{ig\theta}) - \frac{c_{w\theta}}{\rho g} V_{g\theta} \right| + g
\]

In the staggered scheme used in TRAC, the velocities are located on the mesh cell surfaces at the locations shown in Fig. 2 while the volume properties, \( p, \alpha, T, e, \rho \), etc., are located at the mesh-cell centers. The
scalar field equations are written over a given mesh cell while the momentum equations are staggered between mesh cells in the three component directions.

To write out the difference scheme for each of the momentum equations is a rather lengthy process due to the cross derivative terms. Therefore, only a typical scalar field equation will be considered.

The convective terms in the finite difference relations for the scalar field equations are written in conservation form. The finite difference form of the overall mixture mass equation is

\[ \rho_m^{n+1} = \rho_m^n + (\Delta t/v_{ol}) \left\{ FA_{z-\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right] \right\}_{z-\frac{1}{2}} 
- FA_{z+\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right]_{z+\frac{1}{2}} 
+ FA_{x+\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right]_{x-\frac{1}{2}} 
+ FA_{x-\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right]_{x+\frac{1}{2}} 
+ FA_{\theta+\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right]_{\theta-\frac{1}{2}} 
- FA_{\theta-\frac{1}{2}} \left[ \left( (1-\alpha) \rho_k^n \right) v_{k+1}^{n+1} + (\alpha \rho_g^n) v_{g}^{n+1} \right]_{\theta+\frac{1}{2}} \right\} 
\]

where \( v_{ol} \) is the hydrodynamic cell volume and \( FA \) is the flow area at the mesh cell edge. The other scalar equations are differenced similarly.

All of the field equations (1-6) have additional source terms to allow piping to be connected anywhere in the mesh. These sources in the scalar equations contain both an explicit and an implicit term. The implicit term is iterated with the rest of the new time variables in order to provide a consistent (in terms of time differencing) procedure for providing one-dimensional connections to the vessel. The source terms appearing in the mass and energy equations are given below. Subscripts \( p \) and \( v \) refer to pipe and vessel quantities, respectively.

**Overall Mass Continuity Source Term**

\[ \rho_m^n \left( FA \frac{v_{m}^{n+1}}{v_{p}} \right) \]

**Vapor Mass Continuity Source Term**

\[ \left[ (\alpha \rho_g^n) FA \frac{v_{m}^{n+1}}{v_{p}} \right] + \left[ (1-\alpha) \frac{\rho_g \rho_k}{\rho_m} FA \frac{v_{k}^{n+1}}{v_{p}} \right] \]
Overall Energy Source Term

\[
[(\rho_{m} e_{m})^{n} FA v_{m}^{n+1}]_{p} + [\alpha (1-\alpha) \frac{\rho_{m} \rho_{g}}{\rho_{m}} (e_{g} - e_{l}) FA v_{r}^{n}]_{p} \\
+ P_{v} (V_{m}^{n+1} FA)_{p} + P_{v} \frac{\rho_{m} \rho_{g}}{\rho_{m}} FA v_{r}^{n}]
\]

Vapor Energy Source Term

\[
[(\alpha c_{g} e_{g})^{n} FA v_{m}^{n+1}]_{p} + [\alpha (1-\alpha) \frac{\rho_{m} \rho_{g}}{\rho_{m}} e_{g} FA v_{r}^{n}]_{p} \\
+ P_{v} (\alpha^{n} FA v_{m}^{n+1})_{p} + P_{v} \frac{\rho_{m} \rho_{g}}{\rho_{m}} FA v_{r}^{n}
\]

The momentum source terms are complicated due to the staggered differencing and the fact that pipes may enter at an arbitrary angle. For the present version, we have assumed that the pipe enters normal to the vessel mesh cell face. The basic forms for the liquid and vapor momentum source terms are:

Liquid Momentum Source Term

\[
(V_{l}^{2} FA_{l})^{n}/FA_{V}
\]

Vapor Momentum Source Term

\[
(V_{g}^{2} FA_{g})^{n}/FA_{V}
\]

where

\[
V_{l} = V_{m} - \frac{\alpha c_{g}}{\rho_{m}} v_{r}
\]

(11)

and

\[
V_{g} = V_{m} + (1-\alpha) \frac{\alpha c_{g}}{\rho_{m}} v_{r}
\]

(12)

If structure exists in the mesh cell, the hydrodynamic flow areas (FA) and volumes (vol) are reduced from their geometric mesh cell values. Thus, FA may be less than or equal to the geometric mesh cell area and vol may be less than or equal to the geometric mesh cell volume. Flow areas may also be set identically equal to zero. If this is the case, all fluxes across that plane are suppressed along with the individual velocities of each phase. This procedure allows large obstacles such as the downcomer walls to be properly modeled. The user is allowed complete freedom to specify the flow and volume restrictions.
The finite difference equations thus formed are semi-implicit, since the pressure gradient terms in the vapor and liquid momentum equations are treated at the new time. A Courant stability criterion of the form

\[ \frac{|\mathbf{V}|}{L} = \max \left( \frac{V_{gz}}{\Delta z}, \frac{V_{g\theta}}{\Delta \theta}, \frac{V_{g\tau}}{\Delta \tau}, \frac{V_{lz}}{\Delta z}, \frac{V_{l\theta}}{\Delta \theta}, \frac{V_{l\tau}}{\Delta \tau} \right) \]

is necessary where

\[ \frac{|\mathbf{V}| \Delta t}{L} < 1 \]

In order to solve the system of finite difference equations, a linearization procedure is carried out. All of the scalar equations are reduced to a linear system in \( V_L, V_g, T_L, T_g, \alpha, \) and \( p \). This is accomplished by using the thermal equations of state:

\[ \rho_L = \rho_L(p, T_L) \]
\[ \rho_g = \rho_g(p, T_g) \]

the caloric equations of state:

\[ e_L = e_L(p, T_L) \]
\[ e_g = e_g(p, T_g) \]

and the definitions for \( \rho_m \) and \( e_m \).

A further reduction in the system is accomplished by observing that the finite difference vapor and liquid momentum equations yield equations of the form

\[ V^{n+1} = V^n + \left[ \text{conv}^n + \frac{1}{\rho_L^n} \nabla p^{n+1} + \text{FRIC} \right] \Delta t \]  

(13)

where \( \text{conv} \) designates the explicit convection terms and \( \text{FRIC} \) includes both the wall and interfacial shears. Equation (13) indicates that changes in \( V \) are linearly dependent (after an explicit pass on the explicit parts of the momentum equations) on changes in pressure. The system of variables may therefore be further reduced to \( T_L, T_g, p, \) and \( \alpha \) and solved by a Block Gauss-Seidel method. Reference 3 provides a much more detailed description of the basic Newton Block Gauss-Seidel numerical technique.

One improvement to the method proposed in Ref. 3 has been implemented to reduce the computing cost. The linear system that results from this method is a block seven-stripe matrix. In performing the Gauss-Seidel operation, if the nonlinear terms are not updated, the matrix coefficients remain constant for the time step. In this case a Gauss elimination technique can be applied once at each time step to the seven-stripe block array which allows its reduction
to a seven-stripe single-element array in which the unknowns are the vessel pressures. This results in a much faster iteration. The actual iteration is performed in subroutine ITR1. When the vessel pressures are obtained to a specified convergence criterion, a back-substitution in subroutine FF3D is performed to unfold $T_k, T_v, \alpha$ and the velocities for each phase. A call to THERMO in FF3D then updates all of the thermodynamic properties and their derivatives in preparation for the next time step.

CONSTITUTIVE EQUATIONS

The field equations (1-6) require certain auxiliary or constitutive equations to effect closure. It has already been mentioned that thermal and caloric equations of state for each phase are required. In addition, the liquid and vapor wall shear, interfacial drag, wall heat transfer, interfacial heat transfer, the net vaporization rate, and a specification for the interfacial velocities are necessary.

In the present version of the code, the vaporization thrust terms in the momentum equations are neglected. Future versions of the vessel module will specify an interfacial velocity in the three coordinate directions and these terms, which are not always second order (in the core region for example), will then be taken into account. The wall heat transfers $q_{wg}$ and $q_{wl}$ are accounted for in the standard way, i.e., $q = h_a (T - T_{wall})$. The surface areas represent an actual estimate of the total wall surface area wetted by each phase, while $h_{wg}$ and $h_{wl}$ are based on heat transfer correlations from the literature. In many two-phase flow situations the walls are totally wetted by the liquid phase, in which case wall heat transfer to the vapor is zero.

The wall shear coefficients $c_{wg}$ and $c_{wl}$ are defined as

$$c_{wg} = \alpha \frac{c_{fg}}{2D_h}$$

$$c_{wl} = (1-\alpha) \frac{c_{fl}}{2D_h}$$

where $c_{fg}$ and $c_{fl}$ are, respectively, the vapor and liquid wall friction factors and $D_h$ is the hydraulic diameter. The standard Harwell correlation is employed to provide the wall friction factors for two-phase flow. These factors go to the appropriate single-phase values for $\alpha = 0$ and $\alpha = 1$. The average cell vector velocity is used to define the mesh cell Reynolds number and the two-phase multiplier is calculated using cell-centered quantities. A total friction factor is calculated from the information above and is ascribed completely to the liquid momentum equation until a vapor fraction of 0.9 is reached. From an $\alpha$ of 0.9 to 0.9999, the shear is assigned with linear weighting to both the liquid and vapor. Beyond a vapor fraction of 0.9999, a pure vapor drag coefficient is calculated (laminar or Blasius) and assigned totally to the vapor momentum equation. If the vapor fraction is less than 0.0001, a single-phase liquid correlation (laminar or Blasius) is used.

A single friction coefficient is generated for both the outer radial and upper axial cell face from this procedure. However, the hydraulic diameter used in the radial and axial directions will, in general, vary depending on the geometry. These hydraulic diameters are calculated from...
\[ D_h = \frac{4FA_i}{P_i}, \text{ where } i = 0, z, r \]

and where the wetted perimeter (\( P_i \)) normal to direction \( i \) includes the surface area of any rods, wall heat slabs, or flow boundaries. If there is no solid material in a mesh cell, the wall shear is zero. A similar procedure is used to arrive at a wall shear in the theta direction. In this case, however, vector velocities and properties on the appropriate theta face (rather than the cell-centered averages) are used in order to achieve theta symmetry where such symmetry should exist.

The basic finite difference scheme will properly calculate classical Bourda losses at an expansion but overpredicts the losses at a contraction. Provisions have been made for the user to specify an additional constant hydraulic loss factor in any of the coordinate directions and at any mesh face to compensate for this effect.

The flashing rate 17 is determined from a simplified thermal energy jump condition. In both the vapor continuity equation and the vapor thermal energy equation, the potentials \( T_S - T_R \) and \( T_S - T_R \) are evaluated at the new time level while \( H_{gA_i} \) and \( H_{gA_i} \) are evaluated at the old time.

The interfacial heat transfers during boiling and the interfacial shear are calculated in conjunction with a simple flow regime map. This flow regime map, while originally developed for vertical pipe flow, is the simplest prescription that provides a rational means for defining the constitutive equations.

If the void fraction is less than or equal to 0.25, a bubbly flow is assumed. The interfacial surface area in this regime is calculated in conjunction with a critical bubble Weber number \( \text{Web} \). A value of \( \text{Web} = 25 \) is used in the present code version. This value was chosen on the basis of comparisons between TRAC predictions and experimental results for low subcooling (i.e., shear dominated) Creare downcomer tests. TRAC results for these tests are not very sensitive to the Weber number in the range \( 25 \leq \text{Web} \leq 100 \). The expression relating interfacial surface area with \( \text{Web} \) is:

\[ \frac{\rho_v V_r^2 D_b}{\sigma} = \text{Web} \]

or

\[ D_b = \frac{\text{Web} \sigma}{\rho_v V_r^2} \tag{16} \]

where \( D_b \) is the bubble diameter. For this diameter, and assuming a uniform bubble distribution within the mesh cell volume (vol), the number of bubbles is

\[ \text{CNB} = \frac{6 \alpha \text{vol}}{\pi D_b^3} \tag{17} \]
and the interfacial area is

\[ A_i = 6 \alpha \text{vol} \rho_k V_r^2/Re_b \sigma. \]  

The liquid side interfacial heat transfer coefficient is taken as the larger of an approximate formulation of the Plesset-Zwick bubble growth model\(^6,9\)

\[ \text{Nu} = 0.95493 (T_k - T_s) \rho_k \frac{\partial T_k}{\partial T_k} / [\rho_s (h_s g - h_k g)] \]  

and a sphere convection coefficient\(^10\)

\[ \text{Nu} = 2.0 + 0.74 \text{Re}_b^{0.5}, \]  

where

\[ \text{Re}_b = \rho_k V_r D_b / \nu_k. \]  

The interfacial shear coefficient is provided by a rather standard set of formulas for a sphere:\(^11\)

\[ C_i = \frac{C_b \alpha}{2 D_b} \]  

where

\[ C_b = \begin{cases} 240 & \text{for } \text{Re}_b < 0.1, \\ 24/\text{Re}_b & \text{for } 0.1 \leq \text{Re}_b \leq 2, \\ 9.35/\text{Re}_b^{0.68} & \text{for } \text{Re}_b > 2. \end{cases} \]

If the cell-average mass flux is less than 2 000 kg/m\(^2\)·s and the vapor fraction is between 0.25 and 0.5, the flow enters the slug regime. At the maximum \(\alpha\) of 0.5, 40% of the vapor is assumed to exist in the form of trailing bubbles with the remainder contained in the slug. These bubbles probably contribute the majority of the interfacial heat transfer and the liquid side coefficient is calculated from the relations for the entrained bubbles. If the mass flux is greater than 2 700, all of the vapor is assumed to exist in bubbly form. Linear interpolation in mass flux is used in the range 2 000 to 2 700. In the slug regime the interfacial drag is volume averaged between the slug and the trailing bubbles with a constant drag coefficient of 0.44 used for the slug.

In the vapor fraction range of 0.75 to 1.0, an annular or annular mist regime is employed. An approximation to the Wallis entrainment correlation\(^12\) is used to estimate the fraction of liquid that is in droplet form:

\[ E = 1 - \exp[-0.125(J'_g - 2.1)] \]  

where

\[ J'_g = 10^4 \alpha \frac{\nu_k \mu_k}{\sigma} \left( \frac{\rho_s}{\rho_k} \right)^{1/2}. \]
The remainder of the liquid is in a film or sheet. The interfacial shear and heat transfer are volume averaged of the film and droplet relations in the annular mist regime. The wetted surface area of the mesh cell is determined from the rod or slab heat transfer area in the cell and that portion of the geometric flow area which is blocked off. If the cell is in a region devoid of any structure, the geometric surface area is employed as a scaling factor. This is, of course, artificial but in a realistic PWR simulation very few, if any, of the mesh cells are completely free of metal structure. The total interfacial surface area is determined by the sum of the areas contained in the wetted film and the droplets. A critical Weber number equal to 5 for the drops is used with a calculation procedure that is similar to that for bubbly flow. This value of the Weber number is appropriate for accelerating drops. For those cases where sensitivity to Wed was tested, the results were not strongly influenced by Wed in the range $2 \leq \text{Wed} \leq 12$. The liquid side heat transfer coefficient is simply

$$h_{\text{liq}} = c \frac{k_L}{D_d}$$

(23)

where $c$, a constant, has been adjusted to drive the drops to equilibrium under a variety of flow conditions. In the present code, $c = 15000$ which implies a thermal boundary layer in the drops that is about a thousandth of the drop diameter. In the film a correlation

$$Nu = 0.0073 \text{Re}$$

(24)

is employed to predict $h_{\text{fi}}$. The Wallis annular flow model determines the shear for a wavy film while the same drag correlations used for a bubble are employed if droplets exist. The droplet Reynolds number is defined as

$$\text{Re}_d = \frac{\rho_g V_{rd} D_d}{\nu_g}$$

(25)

Since the actual relative velocity calculated is based on a shear that has been averaged between the film and drop correlations, a separate function is used for $V_{rd}$:

$$V_{rd} = 1.4 a [9.8 \sigma (\rho_L - \rho_g) / \rho_g^2]^{1/2}$$

(26)

In the interpolated regime defined in Fig. 1, a linear interpolation in vapor fraction is made between the conditions that would exist if the vapor fraction were at 0.75 in the annular or annular-mist topology, and the conditions that would exist if the flow were in the bubbly or bubbly-slug regime at a void fraction of 0.5. This makes the correlation for the interfacial shear, interfacial heat transfer, and surface area a continuous function of vapor fraction, relative velocity, mass flux, and the various fluid thermodynamic and transport properties.

We now discuss the vapor side heat transfer coefficient and the liquid heat transfer coefficient during condensation. The vapor heat transfer coefficient is the simple function $h_{\text{vg}} = c$, where $c = 1 \times 10^4$. This implies that the rate for boiling or condensation is determined mainly by the liquid side coefficient with a vapor coefficient designed to drive the vapor toward the saturation temperature. The formulation for the total liquid heat transfer coefficient $h_{\text{liq} \cdot \text{Ai}}$ used for boiling seems to provide too high a
coefficient during condensation. It is anticipated that a condensation rate based on a film model might be more appropriate. Therefore, for condensation the interfacial area is calculated from

$$A_i = \frac{\text{vol}}{\Delta z} c,$$

where the coefficient $c$ accounts for a rough interface and is equal to 5. The specific heat transfer coefficient is the same as the film coefficient used in the annular boiling regime. This model has performed well for the subcooled Creare downcomer tests; it is admittedly simple and may be improved in future versions of the code if additional testing indicates that this is desirable.

RESULTS AND CONCLUSIONS

A large number of test problems have been run with the code simulating both full-size reactor problems and smaller scale experiments.\textsuperscript{14,15} Although most of the comparisons between the TRAC 3-D model and the data have shown good agreement, certain model improvements are being initiated. The principal change involves adding an additional field to allow for the separate handling of droplets and liquid film. It is felt that this will provide a better representation of the extremely complicated physical phenomena which occurs during a PWR reflood. The present three-dimensional vessel module, however, is quite useful as presently constituted in handling all phases of a LOCA scenario.
STANDARD NOMENCLATURE

**Independent Variables**

- \( r \): Radial coordinate in cylindrical geometry.
- \( t \): Time.
- \( \theta \): Azimuthal coordinate in cylindrical geometry.
- \( z \): Axial coordinate in cylindrical geometry.

**Other Variables**

- \( A \): Area.
- \( c \): Shear or friction coefficient in two-fluid equations.
- \( c_p \): Specific heat at constant pressure.
- \( c_v \): Specific heat at constant volume.
- \( D \): Diameter.
- \( e \): Specific internal energy.
- \( f \): Friction factor in drift-flux equations.
- \( FA \): Flow area.
- \( g \): Acceleration due to gravity.
- \( G \): Mass flux \((\rho V)\).
- \( h_{kg} \): Specific enthalpy or heat transfer coefficient.
- \( k \): Thermal conductivity, form loss coefficient, or pipe roughness.
- \( K \): Wall shear coefficient in drift-flux equations.
- \( m \): Mass.
- \( N u \): Nusselt number.
- \( p \): Pressure.
- \( q \): Heat generation rate.
- \( q'' \): Heat flux.
- \( q''' \): Volumetric heat generation rate.
- \( R \): Radius.
- \( Re \): Reynolds number.
- \( T \): Temperature.
**Other Variables (cont.)**

**V** Velocity.

**vol** Hydrodynamic cell volume.

**We** Weber number.

**x** Quality.

**α** Vapor volume fraction or absorptivity.

**Γ** Net volumetric vapor production rate due to phase change.

**δ** Mean-fuel surface roughness.

**Δ** Increment.

**ε** Emissivity.

**μ** Viscosity.

**ρ** Microscopic density.

**σ** Surface tension.

**τ** Shear stress.

**φ^2** Two-phase friction factor multiplier.

**Subscripts**

**b** Bubble.

**c** Cladding.

**d** Droplet.

**f** Fuel or friction.

**g** Vapor (gas) field.

**h** Hydraulic.

**i** Interface (liquid-vapor) quantity or one-dimensional cell index in heat transfer equations.

**j** One-dimensional cell index in hydrodynamics equations.

**l** Liquid field.

**l & g** Liquid to vapor.

**m** Mixture quantities.

**r** Relative quantities.

**r, θ, z** Cylindrical coordinate directions.
Subscripts (cont.)

\( r \pm \frac{1}{2} \)
Mesh cell boundary indices.

\( s \pm \frac{1}{2} \)
Saturation quantities.

\( sp \)
Single-phase quantities.

\( ss \)
Steady-state quantities.

\( t \)
Transient quantities.

\( tp \)
Two-phase quantities.

\( w \)
Wall quantities.

Superscripts

\( k \)
Iteration count index.

\( n, n+1 \)
Time step boundary indices.
REFERENCES


