Core Thermal Model: COBRA-IV Development and Applications

by

**C. W. Stewart, C. A. McMonagle,
M. J. Thurgood, T. L. George,
D. S. Trent, J. M. Cuta, G. D. Seybold**

January 1977

**Prepared for the
Nuclear Regulatory Commission**

Battelle

Pacific Northwest Laboratories

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the Energy Research and Development Administration, 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.

PACIFIC NORTHWEST LABORATORY
operated by
BATTELLE
for the
ENERGY RESEARCH AND DEVELOPMENT ADMINISTRATION
Under Contract EY-76-C-06-1830

Printed in the United States of America
Available from
National Technical Information Service
U.S. Department of Commerce
5285 Port Royal Road
Springfield, Virginia 22151
Price: Printed Copy \$____; Microfiche \$3.00

*Pages	NTIS Selling Price
001-025	\$4.50
026-050	\$5.00
051-075	\$5.50
076-100	\$6.00
101-125	\$6.50
126-150	\$7.00
151-175	\$7.75
176-200	\$8.50
201-225	\$8.75
226-250	\$9.00
251-275	\$10.00
276-300	\$10.25

CORE THERMAL MODEL:
COBRA-IV DEVELOPMENT
AND APPLICATIONS

by
C. W. Stewart
C. A. McMonagle
M. J. Thurgood
T. L. George
D. S. Trent
J. M. Cuta
G. D. Seybold

January 1977

BATTELLE
Pacific Northwest Laboratories
Richland, Washington 99352

CONTENTS

	<u>Page</u>
LIST OF FIGURES	iv
LIST OF TABLES	vi
1.0 INTRODUCTION	1
2.0 SUMMARY AND CONCLUSIONS	5
3.0 DEVELOPMENTS IN THE BASIC MATHEMATICAL MODEL	9
3.1 ACE TRANSIENT SOLUTION METHOD	10
3.2 COMPRESSIBILITY	18
3.3 TWO-PHASE MODEL IMPROVEMENTS	20
3.4 CODE MODIFICATIONS TO FACILITATE ADVANCED APPLICATIONS	25
3.4.1 Generalized Boundary Conditions	25
3.4.2 Graphical Analysis Capabilities	27
4.0 BLOWDOWN ANALYSIS	31
4.1 SEMISCALE CORE BLOWDOWN CALCULATIONS	31
4.2 PWR BLOWDOWN CALCULATIONS	38
4.3 CALCULATIONAL COMPARISON WITH EDWARDS' BLOWDOWN EXPERIMENT	45
5.0 REFLOOD ANALYSIS	51
5.1 ISOTHERMAL CALCULATIONS	52
5.2 REFLOOD CALCULATIONS WITH HEATED CORE	59
6.0 DATA COMPARISONS AND APPLICATIONS FOR THE COBRA-DF COMPUTER PROGRAM	69
6.1 ONE-DIMENSIONAL TRANSIENT DATA COMPARISONS.	69
6.2 DOWNCOMER MODELING.	73
REFERENCES	83

LIST OF FIGURES

	<u>Page</u>
3-1 Placement of Major Variables on the Computational Mesh	12
3-2 Example of Application of General Boundary Conditions	28
4-1 Semiscale Core	32
4-2 Subchannel Model of the Semiscale Core	34
4-3 Comparison of Experimental and Calculated Core Inlet Density	35
4-4 Comparison of Experimental and Calculated Inlet Mass Flow Rate	36
4-5 Calculated Exit Mass Flow Rate for Channels 1 and 5	37
4-6 Model of PWR Geometry for COBRA Calculations	39
4-7 Calculated Inlet Mass Velocities for Channels 1 and 6	40
4-8 Calculated Exit Mass Velocities for Channels 1 and 6	41
4-9 Calculated Flow Field in the PWR Core During the Blowdown Transient	42
4-10 Normalized Flow Field for PWR Core with and without Flow Blockage	44
4-11 Initial Conditions for Edwards Blowdown Transient	46
4-12 Pressure Versus Time at Gauge Station 1 During First 10 Seconds of Transient	48

LIST OF FIGURES (cont'd)

	<u>Page</u>
4-13 Pressure Versus Time at Gauge Station 1 for Full Transient	48
4-14 Pressure Versus Time at Gauge Station 4 During First 10 Seconds of Transient	49
4-15 Pressure Versus Time at Gauge Station 4 for Full Transient	49
4-16 Pressure Versus Time at Gauge Station 7 During First 10 Seconds of Transient	50
4-17 Pressure Versus Time at Gauge Station 7 for Full Transient	50
5-1 PWR Core Model (Illustrating Nonuniform Core Liquid Level)	53
5-2 Water Level Oscillations with Uniform Initial Core Level, Case 1	57
5-3 Water Level Oscillations with Nonuniform Initial Core Level, Case 2	57
5-4 Water Level Oscillations with 8-Channel Core, Uniform Start, Case 3	58
5-5 Water Level Oscillations with 8-Channel Core, Nonuniform Start, Case 4	58
5-6 Initial Core Temperature Contours	62
5-7 Liquid Level and Velocity -- Uniform Core Power	63
5-8 Liquid Level and Velocity -- Nonuniform Core Power	64
5-9 Heat Transfer -- Nonuniform Core Power	65
5-10 Axial Surface Temperature Profile -- Uniform Core Power	66

LIST OF FIGURES (cont'd)

	<u>Page</u>
5-11 Axial Surface Temperature Profile -- Nonuniform Core Power	67
6-1 Oscillatory Power Transient	72
6-2 Flow Transient	74
6-3 Power Transient	74
6-4 Creare Flat Plate Test Apparatus	80
6-5 COBRA Model of Creare Apparatus	80
6.6 Annulus Penetration for Steam-Water Countercurrent Flow.	82

LIST OF TABLES

	<u>Page</u>
5-1 Reflood Input Summary	54
5-2 Isothermal Reflood Test Cases	55
5-3 Fuel Rod Data	60

CORE THERMAL MODEL: COBRA-IV
DEVELOPMENT AND APPLICATIONS

1.0 INTRODUCTION

The objective of the Core Thermal Model Development program currently in progress at the Battelle Pacific Northwest Laboratory is to develop numerical simulation methods for evaluating thermal hydraulic performance of water-cooled nuclear reactor cores and other components during postulated accident conditions. The primary thrust of this program to date has been continued development of the COBRA computer code as a tool for multidimensional transient simulation applicable to either PWR or BWR cores.

The work reported is part of an overall Reactor Safety Code Development program sponsored by the Analysis Development Branch of Reactor Safety Research, U.S. Nuclear Regulatory Commission. The main goal of work to date has been development of the COBRA computer code as a working tool for multidimensional PWR reflood simulation. This effort has produced COBRA-IV⁽¹⁾ which, although based on homogeneous equilibrium assumptions, has the necessary numerical structure to address complex hydrodynamic phenomena. For this reason, COBRA-IV (HEM)^{*} is now considered the foundation code version that forms the base for continued code development.

COBRA-IV incorporates numerous numerical features which significantly extend computational capabilities beyond that of the widely used COBRA-III-C. One unique feature of COBRA-IV is that it maintains the subchannel connection logic used by COBRA-III-C but applies this logic in a manner similar to the finite-difference methods common in numerical hydrodynamics. Thus, by proper choice of subchannel geometry and other parameters, one can set up a

* COBRA-IV (HEM) is currently available in the Argonne Code Center.

finite-difference grid network appropriate for free field hydrodynamics applications. Yet, the subchannel logic permits modeling of the geometric complexities typical of a reactor core in the same fashion as COBRA-III-C.

The transient algorithm of COBRA-IV permits multidimensional simulation of core flow reversals, expulsions, and complete blockages. One- and three-dimensional simulations of the Semi-scale thermal-hydraulic behavior during blowdown have been conducted and compared to data. Simulations have also been performed for generic PWR core blowdown with and without localized flow blockages.

The COBRA-IV version currently available in the Argonne Code Center bases thermodynamic properties on a system reference pressure which allows for pressure-time variation of the system as a whole but does not allow for spatial pressure variation within the system to influence local properties. Recent developments allow use of local thermodynamic conditions, permitting application of COBRA-IV in simulating sonic phenomena.

Continuing development of COBRA-IV includes relative phase motion constitutive modeling, thermal nonequilibrium, and bottom/top quench fuel rod heat transfer modeling. A multidimensional drift flux model has been formulated by using available space-time averaging concepts. The model has features necessary to account for both axial and lateral relative phase velocity but requires appropriate constitutive correlations.

Currently, testing is in progress using both three- and five-equation approaches and initial results are presented here. Incorporation of relative velocity and thermal nonequilibrium capabilities presents the possibility of using COBRA-IV for PWR downcomer flow simulation. Initial attempts in modeling these phenomena and preliminary results are reported. Success of this approach will depend substantially upon defining suitable constitutive models derived from experimental programs.

This report documents the more significant achievements of the Core Thermal Model Development program to date and through various applications demonstrates the capabilities of COBRA-IV. In addition, general areas of future code development are indicated.

2.0 SUMMARY AND CONCLUSIONS

The principal objective of the PNL Core Thermal Model Development Program is the continued development of the COBRA computer code as a tool for simulating multidimensional transient reactor vessel thermal-hydraulic phenomena during postulated accident conditions. Earlier versions of COBRA were unable to address the PWR reflood phenomena because of fundamental limitations of the numerical solution algorithm. This limitation necessitated reformulating the solution approach to a pressure velocity technique incorporating a degree of implicitness which permits COBRA to address a wide range of complex phenomena, including the sharp density gradients associated with reflood. This new code version is designated COBRA-IV and possesses significant additional capabilities over COBRA-III-C, including the ability to simulate core flow reversals, expulsions, and complete blockages. COBRA-IV retains the subchannel logic used by COBRA-III-C which is convenient for core analysis, yet with a proper choice of subchannel geometry and other parameters, is equivalent to finite differencing techniques commonly used in free field hydrodynamics.

The basic version, COBRA-IV (HEM), has been documented and made available through the Argonne Code Center. This version of the code contains the necessary numerical structure to accommodate various modeling modifications needed to investigate LOCA conditions and is currently being used as the base for code improvements.

The ACE solution procedure described in Section 3 deals effectively with the reflood phenomena where a liquid-vapor interface must be maintained. The viability of the approach is demonstrated by the U-tube manometer simulations which were in excellent agreement with the exact solution. Further demonstration calculations were performed simulating two-dimensional isothermal and heated reflood oscillations. Simulations were also conducted for Semiscale blowdown test S-02-3 and results found to compare

reasonably well with test data. Similar computations were performed for a generic PWR core with and without a local flow blockage. Results from these simulations indicated that the blowdown phenomenon is substantially one-dimensional for uniform pressure boundary conditions and that overall flow behavior was affected little by the blockage.

Methods required for resolution of sonic phenomena have recently been incorporated in the COBRA-IV solution algorithm and the Edwards' blowdown test (NRC Standard Problem Number 1) results were used for computational comparisons. Although this simulation was conducted using homogeneous two-phase flow equilibrium assumptions, the code results compared quite well with experimental data.

Work has begun on incorporating relative phase motion and thermal nonequilibrium techniques in COBRA-IV. Initial test results have been obtained for both three- and five-equation models. These tests included simulation of transient two-phase flow experiments and comparisons with unwrapped downcomer data obtained by Creare. Downcomer numerical experiments to date indicate that the five-equation model approach appears to be viable but that ultimate success will depend substantially on obtaining appropriate constitutive correlations.

Numerical testing has illustrated that techniques currently available in COBRA-IV or under development have a good potential for realistically simulating many complex two-phase flow phenomena. The future direction of the Core Thermal Model Development Program lies in constructing a reactor vessel numerical simulation tool which will permit analysis of various LOCA phenomena from the initiation of blowdown through reflood. In view of the complex phenomena postulated, this appears to be quite an ambitious task. And indeed, resolution of quantities during the transient will obviously be compromised by the necessity of reasonable approximations and computational limitations. However, based on progress

to date, it appears that extending COBRA-IV to such applications is entirely feasible and that it will be possible to perform computations that will reasonably simulate reactor vessel thermal-hydraulic conditions during postulated accident situations.

3.0 DEVELOPMENTS IN THE BASIC MATHEMATICAL MODEL

The primary objective of the PNL Core Thermal Model Development of multidimensional transient thermal-hydraulic analysis capability is for simulating loss of coolant accident phenomena in water-cooled nuclear reactor vessels. The complex thermal-hydraulic behavior of LOCA blowdown and reflood transients is requiring significant code development to adequately model the phenomena involved.

Low pressure reflood modeling required reformulation of the transient numerical solution method. The major feature of this reformulation was the fully implicit coupling of the energy equation to obtain computational stability at an oscillating steam-water interface that occurs during reflood. This new solution method, called ACE (for Advanced Continuous-fluid, Eulerian), is discussed in Section 3.1.

In addition to reformulating the numerical transient solution method, two basic simplifying assumptions in the code are not fully justifiable for reflood and certain blowdown calculations. These assumptions are (1) a uniform reference pressure in the equation of state for fluid property computation, and (2) homogeneous two-phase flow with thermal equilibrium between the phases.

Assuming a uniform reference pressure precludes analyzing the effects of local pressure variations on fluid properties or vapor compressibility. Rapid depressurization may cause expansion waves to propagate back through the vessel, significantly affecting the flow behavior during blowdown. This phenomenon can be modeled using a local pressure, enthalpy, and density equation of state in the ACE solution method and is discussed in Section 3.2.

Homogeneous two-phase flow assumptions are obviously not applicable where relative motion between the liquid and vapor phases must be considered, such as in flooding and the interaction

between superheated steam in the core and subcooled water injected by ECC systems. Section 3.3 summarizes mathematical model developments for use in COBRA-IV that will permit two-phase flow analysis with unequal phase velocities and thermal nonequilibrium.

Along with the more fundamental alterations required for LOCA analyses, several improvements in the code mechanics have been completed. Modeling the complex geometries of reactor vessel internals, which include the lower plenum, downcomer, upper plenum, upper head, cold and hot leg outlet piping, etc., requires a flexible means of specifying system geometry and boundary conditions. A generalized boundary condition specification capability has been developed in the code to facilitate the modeling of such systems. Similarly, the nature of the calculational results of blowdown and reflood analyses requires more versatile methods of presenting and summarizing computer results. Auxiliary programs have been developed to present results in various graphical modes. These programs, and the generalized boundary conditions capability, are discussed in Section 3.4.

3.1 ACE TRANSIENT SOLUTION METHOD

The ACE method is an extension of the basic ICE technique.⁽¹⁾ Conventional applications of ICE and its successors employ an explicit energy equation where flows and energies from the previous time step are used to form the convective terms. In many low speed two-phase flow applications, explicit treatment of the convective terms can lead to severe computational difficulties if steep density gradients exist, such as the oscillating steam-water interface typical of reflood. The ACE method eliminates this problem by combining the energy equation implicitly with the continuity equation through the equation of state. The resulting expression for flow divergence is solved simultaneously with the explicit momentum equations.

The full set of subchannel conservation equations is described in Reference 2. To illustrate the method, the following abbreviated forms of the equations will be used for clarity:

Mass

$$\bar{A}_j \frac{\Delta x}{\Delta t} (\rho_j - \rho_j^n) + m_{j+1/2} - m_{j-1/2} + \Delta x [D_c]^T w_j = 0 \quad (3.1)$$

Energy

$$\bar{A}_j \frac{\Delta x}{\Delta t} (\rho h_j - \rho h_j^n) + m h_{j+1/2}^* - m h_{j-1/2}^* + \Delta x [D_c]^T w h_j^* = Q_j^n \quad (3.2)$$

Axial Momentum

$$m_{j+1/2} = m_{j+1/2}^n - \Delta t \frac{\bar{A}_{j+1/2}}{\Delta x} (P_{j+1} - P_j) - \Delta t F_{j+1/2}^n \quad (3.3)$$

Lateral Momentum

$$w_j = w_j^n + \Delta t \frac{S}{\ell} [D_c] P_j - \Delta t G_j^n \quad (3.4)$$

All explicit terms have been lumped into the variables Q^n , F^n , and G^n , respectively. All other variables not specifically superscripted by the symbol n are assumed to be new time, implicit, values. The placement of variables on a computational cell is shown in Figure 3-1.

In addition, an equation of state is used which assumes a uniform reference pressure, P^* , over the entire computational mesh for fluid property computation. The reference pressure is specified by code input and may be made a function of time. The use of the reference pressure concept is justified if the spatial pressure changes are small compared to the reference pressure or if fluid properties are assumed independent of pressure.

FIGURE 3-1. Placement of Major Variables on the Computation Mesh

$$\rho = \rho(h, P^*) \quad (3.5)$$

By inverting the state equation, all the enthalpies in equation (3.2) can be expressed in terms of the specific volume, v . For example,

$$h_{j+1/2}^* = h_o + \left. \frac{\partial h}{\partial v} \right|_P (v_{j+1/2}^* - v_o) \quad (3.6)$$

Where h_o and v_o define some arbitrary reference state sufficiently close to h^* and v^* so that $\frac{\partial h}{\partial v}$ can be assumed constant between the two states.

Applying equation (3.6) to the energy equation (3.2) yields the following result:

$$\begin{aligned} & m v_{j+1/2}^* - m v_{j-1/2}^* + \Delta x [D_c]^T w v_j^* - \left. \frac{\partial v}{\partial h} \right|_P Q_j^n \\ & = \left(v_o - \left. \frac{\partial v}{\partial h} \right|_P h_o \right) \left[\bar{A}_j \frac{\Delta x}{\Delta t} (\rho_j - \rho_j^n) + m_{j+1/2} - m_{j-1/2} \right. \\ & \quad \left. + \Delta x [D_c]^T w_j \right]. \end{aligned} \quad (3.7)$$

From continuity, the right-hand side of equation (3.7) reduces to zero. The left side relates the flow field divergence to the volumetric expansion due to heat addition. This "divergence equation" is the basis of the ACE solution method.

The solution scheme begins a time step by evaluating the explicit terms Q , F , and G and obtaining initial estimates for the flows, $\hat{m}$ and $\hat{w}$, based on pressures and flows from the previous time step. These tentative flows will not in general satisfy equation (3.7) but will yield a residual error, E , in the divergence equation. That is,

$$\tilde{m}v_{j+1/2}^* - \tilde{m}v_{j-1/2}^* + \Delta x [D_c]^T wv_j^* - \left. \frac{\partial v}{\partial h} \right|_p Q_j^n = E_j. \quad (3.8)$$

The objective of the solution at this point is to reduce this residual error to zero in all computational cells. This is done by adjusting the pressures and flows in each cell sequentially in an iterative loop using a technique similar to that employed by Hirt and Cook.⁽³⁾

The pressure change, ΔP_j , necessary to reduce E_j to zero in any computational cell is determined by using the total derivative of E with respect to pressure. That is,

$$\Delta P_j = \frac{-E_j}{dE/dP}, \quad (3.9)$$

The total derivative is formed as follows, holding the specific volume constant.

$$\frac{dE}{dP_j} = \frac{\partial E}{\partial \tilde{m}_{j+1/2}} \frac{d\tilde{m}_{j+1/2}}{dP_j} + \frac{\partial E}{\partial \tilde{m}_{j-1/2}} \frac{d\tilde{m}_{j-1/2}}{dP_j} + [D_c]^T \frac{\partial E}{\partial w_j} \frac{dw_j}{dP_j} \quad (3.10)$$

The momentum equations (3.3) and (3.4) provide linear relations for the flow differentials:

$$\frac{d\tilde{m}_{j+1/2}}{dP_j} = \frac{\Delta t}{\Delta x} \bar{A}_{j+1/2} \quad (3.11)$$

$$\frac{d\tilde{m}_{j-1/2}}{dP_j} = \frac{\Delta t}{\Delta x} \bar{A}_{j-1/2} \quad (3.12)$$

$$\frac{dw_j}{dP_j} = -\Delta t \frac{S}{\ell} [D_c] \quad (3.13)$$

The partial derivatives of E with respect to the flows are found from equation (3.8). The final result is

$$\frac{dE}{dP} = \frac{\Delta t}{\Delta x} \left[v^* \bar{A}_{j+1/2} + v^* \bar{A}_{j-1/2} + \Delta x \frac{S}{\ell} [D_c]^T [D_c] v_j^* \right]. \quad (3.14)$$

After the necessary pressure change is computed by equation (3.9), the remaining steps serve to update the flows, density, and enthalpy in the cell. The new flows are found via the momentum derivatives, (3.11) through (3.13). These flows are then used in the continuity equation (3.1) to find the new density (and specific volume) and, finally, a new enthalpy is found from the equation of state using the new density and the reference pressure. This procedure is repeated over all computational cells until the maximum error, E, in the mesh is less than a specified error criterion. The solution is then considered to be converged and the calculations for the time step are complete.

To maintain computational stability it is important that the density is continually updated inside the iteration and that no acceleration is used to attempt to speed the convergence. The pressure and flow changes calculated by equation (3.9) for cell j guarantee satisfaction of continuity in that particular cell but will disturb the mass balance in neighboring cells. Although this disturbance becomes smaller as the iteration proceeds, a finite error remains at convergence. Therefore, a new density calculated after iteration terminates will contain this residual error. This residual error is unacceptable in some cases, such as when the cell density is very near saturation. The error in the density is very near saturation. The error in the density can be eliminated by computing the new density immediately after the pressure and flow adjustment specified by equation (3.9). This does not imply that the small residual error disappears; rather it shows up as a small deviation in velocity which is normally considerably less critical than density deviations. For the same reason, no acceleration

coefficient is used since it would modify the pressure change and introduce a continuity error. In practice, the use of acceleration has provided only a 20 percent savings in computation time at best. This is considered a small price to pay for the absence of extraneous density deviations.

Large hydraulic resistances are encountered in many applications. A loss coefficient on the order of 10 is required to represent PWR inlet nozzles and associated hardware. A value of 700 has been proposed to model the resistance of the primary system from the upper plenum to the cold legs through the steam generator and pumps.

The pressure drop, ΔP_f , across a computational cell due to a loss coefficient of this type is normally calculated explicitly. For example,

$$\Delta P_f = K |u_j^n| m_j^n. \quad (3.15)$$

It can be shown that a numerical stability criterion for an explicit momentum equation with explicit hydraulic resistance is

$$\Delta t < \frac{\Delta x}{(1+K) |u|} \quad (3.16)$$

where Δt is the maximum time step and K is the maximum loss coefficient considered. In combination with high velocities, this limitation can clearly be prohibitive.

This severe time step restriction can be circumvented by treating the flow resistance semi-implicitly using new-time flows to compute the pressure drop. Thus, equation (3.15) is rewritten as

$$\Delta P_f = K |U_j^n| m_j^{n+1} \quad (3.17)$$

The linearized definition for the pressure losses applied to the momentum equations (3.3) and (3.4) results in the following expressions:

$$m_{j+1/2} = \left[m_{j+1/2}^n - \Delta t F_{j+1/2} - \bar{A}_{j+1/2} \frac{\Delta t}{\Delta x} (P_{j+1} - P_j) \right] \frac{1}{(1 + \Delta t K |u|_{j+1/2}^n)} \quad (3.18)$$

$$w_j = \left\{ w_j^n - \Delta t G_j + \frac{S}{\ell} \Delta t [D_c] P_j \right\} \frac{1}{(1 + \Delta t C |v|_j^n)} \quad (3.19)$$

When these new momentum pressure relations are used in place of (3.3) and (3.4) in calculating the pressure cell pressure correction, the total derivative of E with respect to P takes the form

$$\begin{aligned} \frac{dE}{dP} = \frac{\Delta t}{\Delta x} & \left\{ \left(\frac{v^* \bar{A}}{1 + \Delta t K |u^n|} \right)_{j+1/2} + \left(\frac{v^* \bar{A}}{1 + \Delta t K |u^n|} \right)_{j-1/2} \right. \\ & \left. + \Delta x \frac{S}{\ell} [D_c]^T \left(\frac{v^*}{1 + \Delta t C |v^n|} \right)_j [D_c] \right\} \quad (3.20) \end{aligned}$$

Other than substituting (3.19) above for equation (3.14) and modifying the flow pressure derivatives (3.11) through (3.13), the solution procedure is not altered. This approach removes any stability limitations arising from the use of large hydraulic resistances.

Since the basic numerical scheme is explicit, the maximum time step is limited by a Courant-like limitation

$$\Delta t \left(\frac{|u|}{\Delta x} + \frac{|v|}{\ell} \right) = C \leq 1 \quad (3.21)$$

where C is the Courant number definition. That is, no particle of fluid may traverse farther than the equivalent of the computational cell dimensions in any one time step. Because fluid velocities may vary widely during a calculation, an automatic time step adjuster is included in COBRA-IV. The user inputs the desired Courant number to be maintained during the transient and the time step is recalculated with equation (3.21) each step based on the maximum axial and lateral velocities. The newly computed time step is averaged with the previous one weighted heavily toward the smaller as

$$\Delta t = \frac{2\Delta t_{\text{OLD}}\Delta t_{\text{NEW}}}{\Delta t_{\text{OLD}} + \Delta t_{\text{NEW}}} \quad (3.22)$$

so that the step size may never increase by more than a factor of two in any one computation cycle.

3.2 COMPRESSIBILITY

This section describes an extension of ACE for calculating flows with steep density gradients at all speeds. While the ACE method is able to compute the motion of density gradients approaching liquid-vapor interface conditions, it uses a uniform reference pressure equation of state which does not consider the effects of local pressure variations on fluid properties or vapor compressibility. This condition is eliminated by employing a local pressure, enthalpy, density equation of state while retaining the implicit energy equation.

The implicit part of the algorithm iterates on the residual error in the energy equation, rather than the original divergence equation (3.8) in the ACE method. That is, the residual error, E , is expressed as

$$E = \bar{A}_j \frac{\Delta x}{\Delta t} \left(\tilde{\rho} h_j - \rho h_j^n \right) + \tilde{m} h_{j+1/2}^* - \tilde{m} h_{j-1/2}^* + \Delta x [D_c]^T w_h^* - Q_j^n - \bar{A}_j \frac{\Delta x}{\Delta t} \left(P_j - P_j^n \right) \quad (3.23)$$

where all implicit terms are the current iterate values as indicated by the tilde superscript. Note the inclusion of the pressure derivative term.

The objective of the iteration is to reduce this residual error to zero in all cells by simultaneously adjusting the pressures and flows in each computational cell.

To find the pressure change necessary to reduce E for the cell, the total derivative of E with respect to pressure is calculated by

$$\begin{aligned} \frac{dE}{dP} = & \frac{\partial E}{\partial \tilde{h}} \frac{d\tilde{h}}{dP} + \frac{\partial E}{\partial P} + \frac{\partial E}{\partial \tilde{m}_{j+1/2}} \frac{d\tilde{m}_{j+1/2}}{dP} + \frac{\partial E}{\partial \tilde{m}_{j-1/2}} \frac{d\tilde{m}_{j-1/2}}{dP} \\ & + \frac{\partial E}{\partial [P] T_w^*} \frac{d[D] T_w^*}{dP} + [D]^T \frac{\partial E}{\partial \tilde{w}_j} \frac{d\tilde{w}_j}{dP} . \end{aligned} \quad (3.24)$$

The enthalpy derivative is calculated from the equation of state

$$\frac{d\tilde{h}}{dP} = \left. \frac{\partial h}{\partial P} \right|_{\rho} + \left. \frac{\partial h}{\partial \rho} \right|_P \frac{d\rho}{dP} \quad (3.25)$$

where the density derivative is found by combining the continuity and momentum equations. The flow derivatives are easily found from the momentum equations. Thus, the residual energy error is a function of cell pressure only as in the original ACE procedure. The final form of dE/dP is

$$\begin{aligned} \frac{dE}{dP} = \frac{\Delta t}{\Delta x} & \left[\left(\bar{A}_{j+1/2} - \rho_j \left. \frac{\partial h}{\partial \rho} \right|_P \right) + \bar{A}_{j-1/2} \left(h_{j-1/2}^* - \rho_j \left. \frac{\partial h}{\partial \rho} \right|_P \right) \right. \\ & \left. + \Delta x^2 \frac{s}{\ell} [D_c]^T \left(h_j^* - \rho_j \left. \frac{\partial h}{\partial \rho} \right|_P \right) [D_c] \right] + \bar{A}_j \frac{\Delta x}{\Delta t} \left(\left. \frac{\partial h}{\partial P} \right|_\rho - 1 \right). \quad (3.26) \end{aligned}$$

As in ACE, the cell pressure change, ΔP , needed to reduce the energy error is calculated according to $\Delta P = -\frac{E}{\frac{dE}{dP}}$. The pressures are then updated and the flows $\tilde{m}$ and $\tilde{w}$ are adjusted via the momentum equations. The new density, ρ , is calculated from continuity using the new flows. The new enthalpy, $\tilde{h}$, is found through the equation of state using the updated $\tilde{P}$ and $\tilde{\rho}$. This procedure is repeated over all cells until the energy error, E , in all cells is simultaneously less than some specified value.

The increase in computation time by including local property effects has been minimized by using a full property table for the equation of state. Enthalpies are found by a direct entry routine that is quite accurate and considerably faster than evaluating approximating functions for the equation of state.

3.3 TWO-PHASE MODEL IMPROVEMENTS

The physical models described in the preceeding sections have addressed homogeneous two-phase flows in thermodynamic equilibrium. This formulation has been demonstrated to be a reasonable approximation for many important problems; however,

there are also many situations for which the homogeneous-equilibrium approximation is not sufficient to describe physical processes. Therefore, development of an extension of COBRA, called COBRA-IV (DF), is in progress to increase the code's capability to consider both relative phase motion and thermal nonequilibrium.

Different versions of COBRA-IV (DF) contain the "three" and "five" conservation equation models. In the three-equation formulation, relative motion between the liquid and vapor phases is described by the specification of empirical correlations for vapor drift. However, this model requires the two-phase mixtures to be in equilibrium. The five-equation model is more general because it allows liquid and vapor to be in nonequilibrium in addition to considering unequal phase velocities.

The discussion in this section will be limited to the three- and five-equation models and their respective constitutive equation requirements. The numerical solution algorithm is essentially the same as that presented in Section 3.1.

The five-equation formulation employs an equation of state, mass and energy conservation equations for each phase, and a momentum conservation equation for a two-phase mixture. It allows a multidimensional treatment, unequal phase velocities, and thermal nonequilibrium. The equations are:

Continuity

vapor

$$\begin{aligned} \frac{\Delta x}{\Delta t} \bar{A}_j \left(\alpha \rho_{v_j}^{n+1} - \alpha \rho_{v_j}^n \right) + \left\{ \left(\frac{\alpha \rho_v}{\rho_m} \right)^* m_{j+1/2} - \left(\frac{\alpha \rho_v}{\rho_m} \right)^* m_{j-1/2} \right. \\ \left. + \left(\frac{\alpha \rho_v \rho_l}{\rho_m} \right)^* (U_{gj} A)_{j+1/2} - \left(\frac{\alpha \rho_v \rho_l}{\rho_m} \right)^* (U_{gj} A)_{j-1/2} \right\} \\ + \Delta x [D_c]^T \left\{ \left(\frac{\alpha \rho_v}{\rho_m} \right)^* w_j + \left(\frac{\alpha \rho_v \rho_l}{\rho_m} \right)^* (v_{gj} S)_j \right\} = \Gamma_j \end{aligned} \quad (3.27)$$

liquid

$$\begin{aligned}
& \frac{\Delta x}{\Delta t} \bar{A}_j \left\{ (1-\alpha) \rho_{\ell j}^{n+1} - (1-\alpha) \rho_{\ell j}^n \right\} + \left\{ \left(\frac{(1-\alpha) \rho_{\ell}}{\rho_m} \right)^* m_{j+1/2} \right. \\
& - \left(\frac{(1-\alpha) \rho_{\ell}}{\rho_m} \right)^* m_{j-1/2} - \left(\frac{\alpha \rho_v \rho_{\ell}}{\rho_m} \right)^* (U_{gj}^A)_{j+1/2} \\
& + \left. \left(\frac{\alpha \rho_v \rho_{\ell}}{\rho_m} \right)^* (U_{gj}^A)_{j-1/2} \right\} + \Delta x [D_c]^T \left\{ \left(\frac{(1-\alpha) \rho_{\ell}}{\rho_m} \right)^* w_j \right. \\
& - \left. \left(\frac{\alpha \rho_v \rho_{\ell}}{\rho_m} \right) (V_{gj}^S)_j \right\} = -\Gamma_j
\end{aligned} \tag{3.28}$$

Energy

vapor

$$\begin{aligned}
& \frac{\Delta x}{\Delta t} \bar{A}_j \left[(\alpha \rho_v h_v)_j^{n+1} - (\alpha \rho_v h_v)_j^n \right] + \left\{ \left(\frac{\alpha \rho_v h_v}{\rho_m} \right)^* m_{j+1/2} \right. \\
& - \left(\frac{\alpha \rho_v h_v}{\rho_m} \right)^* m_{j-1/2} + \left(\frac{\alpha \rho_v \rho_{\ell} h_v}{\rho_m} \right)^* (U_{gj}^A)_{j+1/2} \\
& - \left. \left(\frac{\alpha \rho_v \rho_{\ell} h_v}{\rho_m} \right)^* (U_{gj}^A)_{j-1/2} \right\} + \Delta x [D_c]^T \left\{ \left(\frac{\alpha \rho_v h_v}{\rho_m} \right)^* w_j \right. \\
& + \left. \left(\frac{\alpha \rho_v h_v \rho_{\ell}}{\rho_m} \right)^* (V_{gj}^A)_j \right\} = Q_{v_j} + \Gamma_j h_{g_j}
\end{aligned} \tag{3.29}$$

liquid

$$\begin{aligned}
& \frac{\Delta x}{\Delta t} \bar{A}_j \left\{ [(1-\alpha) \rho_{\ell} h_{\ell}]_j^{n+1} - [(1-\alpha) \rho_{\ell} h_{\ell}]_j^n \right\} + \left\{ \left(\frac{(1-\alpha) \rho_{\ell} h_{\ell}}{\rho_m} \right)^* m_{j+1/2} \right. \\
& - \left(\frac{(1-\alpha) \rho_{\ell} h_{\ell}}{\rho_m} \right)^* m_{j-1/2} - \left(\frac{\alpha \rho_v \rho_{\ell} h_{\ell}}{\rho_m} \right)^* (U_{gj}^A)_{j+1/2} \\
& + \left. \left(\frac{\alpha \rho_v \rho_{\ell} h_{\ell}}{\rho_m} \right)^* (U_{gj}^A)_{j-1/2} \right\} + \Delta x [D_c]^T \left\{ \left(\frac{(1-\alpha) \rho_{\ell} h_{\ell}}{\rho_m} \right)^* w_j \right. \\
& - \left. \left(\frac{\alpha \rho_v \rho_{\ell} h_{\ell}}{\rho_m} \right) (V_{gj}^S)_j \right\} = -\Gamma_j
\end{aligned}$$

$$\begin{aligned}
& + \left(\frac{\alpha \rho_v \rho_\ell h_\ell}{\rho_m} \right)^* (U_{gj} A)_{j-1/2} \Bigg\} + \Delta x [D_c]^T \left\{ \left(\frac{(1-\alpha) \rho_\ell h_\ell}{\rho_m} \right)^* w_j \right. \\
& \left. - \left(\frac{\alpha \rho_v \rho_\ell h_\ell}{\rho_m} \right)^* (V_{gj} S)_j \right\} = Q_\ell - \Gamma_j h_f
\end{aligned} \tag{3.30}$$

Mixture Momentum

axial

$$\frac{\Delta x}{\Delta t} \left(m_{j+1/2}^{n+1} - m_{j+1/2}^n \right) + \bar{A}_{j+1/2} \left(P_{j+1} - P_j \right) = \Delta x F_{j+1/2}^n \tag{3.31}$$

lateral

$$\frac{\ell \Delta x}{\Delta t} \left(w_j^{n+1} - w_j^n \right) - s \Delta x [D_c] P_j = G_j^n \Delta x \tag{3.32}$$

where the terms U_{gj} and V_{gj} are the axial and lateral vapor drift velocities and Γ is the vapor generation rate. As before, explicit terms in the momentum equations have been grouped into the variables F^n and G^n . However, it should be noted that momentum fluxes due to vapor drift are included in these expressions.

Several approximations have been made in the development of these equations. The phase densities are assumed to be a function of enthalpy and a reference pressure, $P^*(t)$, rather than the local pressure, $P(x, t)$. In the energy equation, potential and kinetic energy changes are neglected and $\frac{\partial P}{\partial t}$ terms are assumed negligible with respect to heat addition from surface heat flux.

The primary dependent variables in this equation set were chosen to be:

P - pressure
 m_m - axial flow rate
 w_m - lateral flow rate
 ρ_m - mixture density
 α - void fraction
 h - dispersed phase enthalpy

with an equation of state to relate ρ , P^* , and h .

Among the most important constitutive equation requirements are the specification of U_{gj} , V_{gj} , and the vapor generation rate, $\dot{\Gamma}$. Other correlation requirements, such as friction factors, etc., are the same as those reported for COBRA-IV.

The three-equation formulation is a simplification of the five-equation model. It retains the capability to simulate two-phase motion, but the liquid and vapor phases are assumed to be in equilibrium, thereby reducing the required number of equations. Although this model is not as general as the preceding case, it is valuable because thermal equilibrium is a good approximation for many important two-phase problems. Further, it is computationally less timeconsuming than the five-equation model.

Mixture conservation equations for mass, energy, and axial and lateral momentum are used to describe thermal-hydraulic behavior. The axial and lateral mixture momentum equations are identical to equations 3.31 and 3.32 of the five-equation set. The mixture continuity is given by equation 3.1 and the mixture energy equation, obtained by adding the respective phase conservation equations, is shown below.

Mixture energy

$$\begin{aligned} & \frac{\Delta x}{\Delta t} \bar{A}_j \left(\rho_m h^{n+1} - \rho_m h^n \right) + m h_{j+1/2}^* - m h_{j-1/2} \\ & + \left(U_{gj} A \frac{\alpha \rho_g \rho_\ell^*}{\rho_m} h_{fg} \right)_{j+1/2} - \left(U_{gj} A \frac{\alpha \rho_g \rho_\ell^*}{\rho_m} h_{fg} \right)_{j-1/2} \\ & + \Delta x [D_c]^T \left\{ w h_j^* + \frac{\alpha \rho_g \rho_\ell}{\rho_m} v_{gj} s h_{fg} \right\} = Q^n \end{aligned} \quad (3.33)$$

The primary dependent variables for this model are P , m_m , w_m , ρ_m .

Approximations to the conservation equations and constitutive equation requirements are similar to those previously described; however, it is not necessary to specify the vapor generation rate with this formulation.

3.4 CODE MODIFICATIONS TO FACILITATE ADVANCED APPLICATIONS

3.4.1 Generalized Boundary Conditions

The reactor core vessel geometries encountered in blowdown and reflood transient calculations are more complex than the basic rod bundle/core geometry for which the subchannel analysis technique used in COBRA was devised. These complex geometries include the lower plenum and downcomer in blowdown and reflood analysis and the upper plenum, upper head, and associated tubes and pipes in analysis of UHI systems. To facilitate modeling systems, a generalized boundary conditions specification capability has been developed for the code.

This capability permits the user to specify, through input parameters, local boundary conditions in any cell of the computational mesh. The specifiable boundary conditions are

- flow rate at a cell boundary (including zero flow)
- enthalpy and pressure
- injected flow rate and enthalpy
- pressure and enthalpy sink or source.

Thus, the user can model specific flow paths by blocking flow at appropriate levels in a channel and/or between channels, and can specify mass injected from a cold leg or lost from a hot leg by specifying a flow and enthalpy or source pressure and enthalpy in the appropriate cell. The boundary conditions can be different in each channel, and each boundary condition may be time-dependent. An initial liquid level can also be specified for each channel with the liquid either above or below a cell boundary.

The capabilities of the new boundary specification are demonstrated by the sketch shown in Figure 3-2. The model consists of three parallel channels. Using only the normal input parameters in COBRA, it would have been possible to specify only the inlet conditions of M_1 , h_1 in channel 1 and zero inlet flow for channels 2 and 3. Through the new input parameters it is possible to specify not only the inlet conditions, but also the flow path around the two U shaped bends, the injection of M_2 , h_2 at the second axial node of channel 3, the flow entering from the pressure source at the top of channel 3, and the flow exit through the pressure source in the next-to-last axial node of channel 1.

The generalized boundary conditions have enabled the user to model more complex geometries from input. The capability has been used in COBRA to model several problems and, although the procedure

is still being tested, the mechanics appear to be functioning properly. Development is continuing, and it is expected that revisions will be made as the need arises.

3.4.2 Graphical Analysis Capabilities

One difficulty encountered in analyzing COBRA calculations for transient phenomena such as PWR blowdown and reflood is presenting the results in an understandable and useable manner. To deal with this problem, two independent plot graphics packages have been developed and used to analyze the results of PWR blowdown and reflood transient calculations.

The first package, REHASH, was designed to accept information generated by the host code, COBRA, as input for plotting. The package was written to accommodate two modes of plotting, CALCOMP and line printer plots. REHASH has the capability to plot the following quantities as a function of time:

1. Axial flow rate (lb/sec)
2. Density (lb/ft³)
3. Axial velocity (ft/sec)
4. Axial mass flux (million lb/hr-ft²)
5. Pressure (psi)
6. Rod surface temperature (F)
7. Heat flux (million Btu/hr-ft²)
8. Surface coefficient (Btu/hr-ft²)
9. Crossflow (lb/sec)

FIGURE 3-2. Example of Application of General Boundary Conditions

10. Lateral velocity (ft/sec)
11. Lateral mass flux (million lb/hr-ft²)
12. Void fraction
13. Equilibrium quality
14. Liquid level (inches)

The second package, EYES, is still in development, although a number of options have been exercised. The package is written for four modes of plotting: CALCOMP, 16 mm movies, 35 mm slides, and microfiche. Sixteen and thirty-five millimeter modes include color options. The capabilities within EYES include PWR reflood movie production that has been successfully demonstrated.

The package has been developed for three-dimensional conical projection graphics, stereo projection graphics, and for sophisticated, four-sided cell, contour plotting, all of which will be valuable in interpreting the results of various PWR transient calculations.

4.0 BLOWDOWN ANALYSIS

PWR core blowdown studies have been performed with COBRA for Semiscale core blowdown test S-02-3 and for a representative PWR core with and without a localized flow blockage. These calculations were performed using the ACE solution method, with the assumptions of uniform reference pressure for the equation of state and homogeneous two-phase flow in thermal equilibrium. These numerical simulations are discussed in Sections 4.1 and 4.2. The results demonstrate that COBRA can reasonably describe the general flow behavior within the semiscale core and provide insight into the thermal-hydraulic behavior during blowdown within a PWR core.

Calculations made using COBRA were compared with the results of the Edwards' blowdown test. The ACE solution method was employed with the local pressure, density, enthalpy equation of state to calculate the effects on the thermal-hydraulic solution of the sonic wave propagation through the fluid. The results are discussed in Section 4.3.

4.1 SEMISCALE CORE BLOWDOWN CALCULATIONS

The Semiscale facility is being used to experimentally investigate the system response to PWR accident conditions. The facility contains a core, pressure vessel, pump, steam generator, and associated piping. The computations performed with the COBRA code modeled only the Semiscale core. The simulation considered the first 10 seconds of blowdown for test S-02-3. Required boundary conditions and values for COBRA input parameters were supplied, whenever possible, from experimental results.⁽⁴⁾

A one-eighth section-of-symmetry from the 40-rod core was modeled for the calculations by six interconnected flow channels and 39 axial nodes. Figure 4-1 shows the axial representation

FIGURE 4-1. Semiscale Core

used for the core. The model, 190 inches overall, considers the core from the lower plenum to the hot legs and contains a 65-inch electrical heated length with unheated portions above and below. Figure 4-2 displays a detailed description of the subchannel model used in the computation. It consists of six rods and six interconnected flow channels to provide a multi-dimensional geometric representation of a symmetrical section of the Semiscale core.

Computational boundary conditions for core pressure drop, system pressure and the enthalpy of fluid entering the core were determined directly from experimental measurements. Additionally, the rod heat flux was supplied from the results of an inversion calculation which utilized experimentally measured rod temperatures.⁽⁵⁾ Three results from this computation will be discussed:

- data comparisons for inlet density
- data comparison for inlet flow rate
- computed results for exit flow rate.

Figure 4-3 presents a comparison of the COBRA prediction and experimentally determined core inlet density and Figure 4-4 compares the computed and measured core inlet flow rate. In general, both show reasonable agreement. Unfortunately, uncertainties in some important boundary conditions make a detailed evaluation of the computed results impossible.

Flow measurements were not made at the core exit and it is, therefore, impossible to compare with experimental values at this location. However, it is important to understand the flow behavior at this location so the computed exit mass flux for subchannels 1 and 5 are presented in Figure 4-5. Although some variations are evident, the calculated flow behavior is, as expected, primarily one-dimensional.

FIGURE 4-2. Subchannel Model of the Semiscale Core

FIGURE 4-3. Comparison of Experimental and
Calculated Core Inlet Density

FIGURE 4-4. Comparison of Experimental and Calculated Inlet Mass Flow Rate

FIGURE 4-5. Calculated Exit Mass Flow Rate for Channels 1 and 5

4.2 PWR BLOWDOWN CALCULATIONS

Numerical simulations of coolant behavior in a representative PWR core during the blowdown portion of a LOCA were undertaken to investigate the significance of multidimensional flow redistributions during blowdown. The effects of core radial power distribution on the flow redistributions were examined. The effect of a regional flow blockage was investigated in a second calculation.

The COBRA model of the PWR core consisted of a one-eighth sector of the core with six flow channels and 14 axial nodes. Figure 4-6 illustrates a lateral cross section of the PWR sector and the geometric representation used in COBRA for these calculations. Boundary conditions required for the computation were obtained from the results of a RELAP-4 calculation. These one-dimensional boundary conditions were similar to those used in the Semiscale computations, except that the COBRA fuel rod conduction model was used here instead of specifying the surface heat flux as was done in the previous case.

Results for the blowdown simulation without flow blockage are shown in Figures 4-7 through 4-9. Figures 4-7 and 4-8 are the computed inlet and exit mass flux for flow channels 1 and 6. Figure 4-9 shows a flow field vector display of normalized mass flux during the initial core flow reversal. The blowdown transient is initiated from steady-state normal operating conditions, and all flows are positive at time $t = 0$. By 0.126 seconds, a flow stagnation plane is beginning to form and an important flow redistribution, caused by nonhomogeneous voiding in the hottest regions of the core, forces fluid toward the cooler outer periphery. By approximately 0.25 seconds, the system pressure has dropped sufficiently so that void production is more homogeneous throughout the core and the flow pattern tends toward one-dimensional behavior. Also, a plane of stagnation is evident near the midline of the core, where water flashing to steam causes coolant to be forced out both

FIGURE 4-6. Model of PWR Geometry
for COBRA Calculations

FIGURE 4-7. Calculated Inlet Mass Velocities
for Channels 1 and 6

FIGURE 4-8. Calculated Exit Mass Velocities
for Channels 1 and 6

FIGURE 4-9. Calculated Flow Field in the PWR Core During the Blowdown Transient

ends of the core. Results at 0.5 seconds are similar to those at 0.25 seconds but with lower flow rates. Finally, at 0.66 seconds all flows are negative and, again, very nearly one-dimensional.

Although the contributions of flow redistribution may be important during initial depressurization, these calculations indicate that PWR blowdown hydrodynamics are primarily one-dimensional, controlled by core wide phenomena, particularly voiding, with relatively small flow redistributions caused by the radial power profile. It is important to note, however, that these calculations were made with the one-dimensional boundary condition of uniform radial pressures at the core inlet and exit, and therefore flow redistributions caused by pressure variations at the boundaries will not be run in these calculations. These results indicate only that the radial power distribution by itself is probably not responsible for major flow redistributions.

The second PWR blowdown simulation incorporates all conditions present in the first, with the addition of a flow blockage. The blockage used in this computation does not necessarily represent the type or magnitude of blockage expected during a LOCA; it was arbitrarily selected for this simulation solely to investigate the qualitative character of possible multidimensional flow redistribution. This blockage was introduced only in Channel 1 and it is present for the entire computation, i.e., it is not introduced as a function of time. The flow area remains constant at the nominal value from the core inlet to an axial position of six feet. Between six and seven feet the area of channel 1 is reduced by 40 percent, and between seven and eight feet the flow area expands back to the nominal value. The fuel rod conduction model was not modified to include effects of rod swelling, and the only impact of the blockage on heat transfer characteristics would be through changes in thermal-hydraulic parameters contained in heat transfer correlations.

Figure 4-10 presents a comparison of a normalized velocity flow field at 0.0 seconds for the blocked and unblocked cases, respectively. (Note: The normalization factor is different for

FIGURE 4-10. Normalized Velocity Flow Field for PWR Core
With and Without An Arbitrary Flow Blockage

the two plots and meaningful comparisons of vector length can be made only within each figure. The relative angle of the vectors from the vertical can be compared meaningfully between the two figures, however.)

It is evident that the blockage disturbs the nominal flow field. Fluid is forced out of channel 1 in the vicinity of the blockage, with subsequent flow recovery downstream. Flow redistributions in the vicinity of the blockage during blowdown are qualitatively similar to those shown for steady state. Although a blockage of this magnitude causes local flow redistributions, the overall character of blowdown in this simulation remains controlled by void production. Thus, the blockage does not change the general hydraulic characteristics calculated for the PWR blowdown simulation without the blockage.

4.3 CALCULATIONAL COMPARISON WITH EDWARDS' BLOWDOWN EXPERIMENT

The Edwards' experiment consists of the breaking of a rupture disk at the end of a pipe containing compressed water at 1000 psi and 515°K. When the rupture disk breaks, a decompression wave propagates through the water, dropping the pressure to the saturation level. The water then flashes and the two-phase fluid and superheated steam flows out the rupture until the pressure drops to ambient conditions.

The calculations were performed in COBRA using an improved ACE solution method with the equation of state dependent upon local conditions. The test section was modeled with two channels, one representing the pipe containing pressurized water and the other representing ambient conditions external to the pipe. A diagram of the model is shown in Figure 4-11. The generalized boundary condition capability in COBRA was used to specify zero flow across the right hand boundary and zero crossflow between the two channels except in the last six axial nodes. This arrangement was used to account for area expansion effects at the exit. Dimensions and experimental conditions used in the

FIGURE 4-11. Initial Conditions for Edward's Blowdown Transient

model were as described by Edwards and O'Brien.⁽⁶⁾ As suggested by Hirt and Romero⁽⁷⁾, the initial temperature was chosen as 502° K rather than the 515° K as reported by Edwards and O'Brien, to account for the difference in the local and bulk saturation temperatures during the decompression.

The results are presented in Figures 4-12 through 4-17. Figures 4-12, 4-14, and 4-16 give the short term time-pressure history at three points (gauge stations, see Figure 4-11) along the pipe. Figures 4-13, 4-15, and 4-17 give the corresponding long term histories. As the decompression wave passes each of the gauge stations the local pressure drops to the saturation pressure, is maintained there until all of the fluid has vaporized, and then the final blowdown phase occurs when the superheated steam flows out of the pipe.

Since this example was intended primarily to demonstrate computational capability, no attempt was made to improve the corresponding experimental and COBRA results by adjusting some of the more uncertain parameters. For instance, Edwards reports reduction of the exit area by approximately 13 percent due to unclean breakage of the rupture disk. This area reduction was not modeled in COBRA. Alternate friction factors could also affect the results as shown by Hirt and Romero. In general, however, the results of this calculational comparison indicate that COBRA is capable of calculating local pressure variation effects on thermal-hydraulic behavior during a transient.

FIGURE 4-12. Pressure Versus Time at Gauge Station 1
During First 10 Milliseconds (msec) of Transient

FIGURE 4-13. Pressure Versus Time at Gauge Station 1
for Full Transient

FIGURE 4-14. Pressure Versus Time at Gauge Station 4
During First 10 Milliseconds (msec) of Transient

FIGURE 4-15. Pressure Versus Time at Gauge Station 4
for Full Transient

FIGURE 4-16. Pressure Versus Time at Gauge Station 7
During First 10 Milliseconds (msec) of Transient

FIGURE 4-17. Pressure Versus Time at Gauge Station 7
for Full Transient

5.0 REFLOOD ANALYSIS

Several nominally one-dimensional experiments have attempted to model the interaction of the downcomer liquid head, vapor generation in the core and the system hydraulic resistance during the reflood stage of a postulated loss of coolant accident (LOCA) in a PWR. However, since an actual reactor core is not hydraulically, thermally, or geometrically one-dimensional, such experiments may not be typical of the entire core. The PNL COBRA-IV computer code has been applied to multidimensional analysis of a PWR core during reflood to provide insight into important reflood phenomena.

A series of COBRA calculations has been performed in two two-dimensional PWR configurations. The first compared the isothermal oscillations of an unheated core-downcomer system with a uniform initial core liquid level to the response of the same system with a radially nonuniform initial level. In the second series, the oscillatory behavior of a two-dimensional heated PWR core was investigated with both a uniform and cosine radial power distribution.

The isothermal PWR core simulation revealed that the primary oscillations of the total mass in the system were essentially the same for both uniform and nonuniform initial liquid level. There was no significant difference in observed irregular core liquid level oscillations between the two radial core power distributions in the heated series.

Although the basic solution algorithm performed quite satisfactorily throughout the simulations, these preliminary results revealed several weaknesses in the constitutive relations and associated computation logic. Improved heat transfer modeling

and the capability to consider compressibility and liquid entrainment are required before any meaningful data comparison is possible. These and other model improvements are underway and it is intended that the reflood simulation be rerun when these capabilities become available.

5.1 ISOTHERMAL CALCULATIONS

This demonstration problem was set up to investigate multi-dimensional effects on isothermal PWR core oscillations by comparing the response of a system with a uniform initial liquid level in the core to that of a system with a nonuniform initial liquid level.

The geometry for this problem was constructed to represent a full-size PWR core, including downcomer and plenums. The actual modeling setup for this problem is shown in Figure 5-1. No attempt is made to model the plenum hardware in detail; they are included only for illustration and to provide a flow path from the downcomer to the core. The downcomer is modeled by one channel that connects to the outer core channel only between the lowest two cells. The annular arrangement of the core was chosen to provide the core channels with equal area and heated perimeter. The interior core and grid spacers with the core loss coefficients are intended to model the various inlet and exit hardware. The core exit loss coefficient of 700 is assumed to represent the resistance of the primary coolant system from the core exit through a steam generator, pump, and associated piping out to a postulated double-ended cold leg break. Other input data for the problem are shown in Table 5-1.

Besides the basic comparison of the effects of the initial core liquid level, several additional runs were made to investigate the effect of a more detailed computational mesh and to check the importance of turbulent mixing. The entire series is summarized in Table 5-2.

RADIAL NODING

AXIAL NODING

FIGURE 5-1. PWR Core Model (Illustrating Nonuniform Core Liquid Level)

TABLE 5-1. Reflood Input Summary

a) Channel Data

<u>Input Quantity</u>	<u>Channel Number</u> (2, 3, 4, 5)		
	<u>Downcomer</u>	<u>Plenums</u>	<u>Core</u>
Area (ft ²)	45.2	24.02	13.0
Wetted Perimeter (ft)	76.9	10.0	1214
Heater Perimeter (ft)	0	0	1205
Hydraulic Diameter (ft)	2.35	9.61	0.079
Number of Rods	0	--	11000

b) Connection Data (gap)

<u>Input Quantity</u>	(1-2)	(2-3)	(3-4)	(4-5)
	<u>Downcomer</u>		<u>Core</u>	
Loss coefficient	12.0	7.58	9.4	13.6
Nominal gap (ft)*	34.75	7.51	6.14	4.34
Corrected gap (ft)	34.75	2.72	2.00	1.18
s/l Parameter	12.0	9.0	6.0	3.0

c) Lumped Hydraulic Resistance

<u>Relative Location (X/L)</u>	<u>Channel Loss Coefficient</u>	
	(1)	(2, 3, 4, 5)
0.125	0.5	10.0
0.25, 0.375, 0.5, 0.625, 0.75	0.0	1.0
0.875	0.0	2.5
0.999	0.0	700.0

d) Miscellaneous

friction factor	$f = 0.316 \text{ Re}^{-.25}$ (turbulent) $f = 64 \text{ Re}^{-1}$ (laminar)
turbulent mixing	$\beta = 0.02$
number axial nodes	16
node length	1.0 ft
Courant number	0.2 - 0.3
system pressure	50 psia

*Nominal gap increases by a factor of 4 in plenum regions.

TABLE 5-2. Isothermal Reflood Test Cases

<u>Case No.</u>	<u>Type of Problem</u>	<u>Figure No.</u>	<u>Comment</u>
1	4-core channel, uniform initial level (72 inches)	5-2	Damped Oscillation, ~ 3 sec. period.
2	4-core channel, non-uniform initial level (86, 84, 60, 48 inches)	5-3	Same overall oscillation as Case 1. Higher frequency core channel oscillations.
3	8-core channel, uniform initial level (72 inches)	5-4	Same result as Case 1.
4	8-core channel, non-uniform initial level (corresponds to Case 2)	5-5	Same result as Case 2.
5	Mixing test (Case 2 run without turbulent mixing, $\beta = 0$)		Same result as Case 2. Mixing effects negligible.

It is conceivable that, with a differential initial liquid level, each core channel will tend to oscillate with a different natural frequency. Interference between the core channels could then cause the overall oscillation to attenuate much sooner than if all core channels were oscillating together. A comparison of these two cases in Figures 5-2 and 5-3, displaying the transient liquid level for the downcomer and core channels, does not support this hypothesis. Note that the downcomer level oscillations of the two cases are nearly identical even though individual core channels oscillate quite differently. The secondary oscillations within the core caused by the initial nonuniform core mass distribution are strongly damped within a few cycles but the primary motion of the total mass of the coupled core and downcomer is unaffected by the motion in the core. It is apparent that, for the isothermal case, the overall motion of the system is determined by the amount of mass in the system, not by its distribution within the core.

Cases 1 and 2 were rerun with twice as many radial divisions in the core to determine if sufficient detail was provided by the original computational mesh. The results are shown in Figures 4-5 and 5-5. The liquid level oscillations are nearly identical with the original runs (Figures 5-2 and 5-3) so the four-channel mesh is probably sufficient for these calculations.

Although turbulent momentum mixing ($\beta = 0.02$) was included in all the above calculations, it can be shown that its effect is negligible compared to frictional drag for large channels which integrate the effects of several thousand subchannels. To confirm this assertion, Case 2 was rerun once again without turbulent mixing ($\beta = 0$). The results were virtually identical to those with $\beta = 0.02$, which indicates that turbulent mixing effects are negligible in this case.

This series of multidimensional calculations provides an initial insight into the isothermal oscillatory behavior of a PWR. The geometry and physics considered in the calculations are not of sufficient detail to provide the final answer to any reflood

FIGURE 5-2. Water Level Oscillations With Nonuniform Initial Core Level, Case 2

FIGURE 5-3. Water Level Oscillations With Uniform Initial Core Level, Case 1

FIGURE 5-4. Water Level Oscillations With 8-Channel Core, Uniform Start, Case 3

FIGURE 5-5. Water Level Oscillations with 8-Channel Core, Nonuniform Start, Case 4

problem. Although basic trends revealed by these results are quite strong, the vapor generated by cooling the hot fuel surfaces also has a powerful influence on the core hydrodynamics. The investigation of these thermal effects is discussed in the next section.

5.2 REFLOOD CALCULATIONS WITH HEATED CORE

In the isothermal case it was postulated that, if each core channel had a different initial liquid level, it would tend to oscillate with a different natural frequency and possibly damp out the overall oscillation of the coupled core-downcomer system. Because the vapor generation rate also affects the oscillation frequency, the same argument can be made for a nonuniform power situation: that a nonuniform radial power profile may damp the reflood oscillations. To test this hypothesis, the isothermal calculations were extended to include a fully-coupled fuel model with a realistic surface temperature and decay power level. This model was applied to evaluate the effects of a cosine-shaped radial core power distribution on reflood oscillations. Subsequent comparisons were made with a radially uniform power case to provide further insight into full reactor core dynamics during reflood. The data for the fuel rod model are shown in Table 5-3. All rod material properties, including the gap conductance, are assumed constant. The surface heat transfer coefficient is calculated by a modified version of the RELAP heat transfer package. No attempt is made to model the propagation of a quench front or liquid entrainment. Although the RELAP package is not strictly applicable to the reflooding process, it was used because it was immediately available in COBRA and provides a reasonable variation in surface heat transfer as the fuel surface cools.

At the start of the transient, a steady-state fuel rod temperature distribution is calculated under decay power conditions with rod boundary conditions adjusted to give an average surface cladding

TABLE 5-3. Fuel Rod Data

a) Thermal Properties

	<u>Fuel</u>	<u>Clad</u>
Material	UO ₂	Stainless Steel
Density (lb/ft ³)	788	488
Heat Capacity (Btu/lb-ft°F)	0.085	0.067
Thermal Conductivity (Btu/hr-ft-°F)	2.24	11.56

b) Dimensions (Inches)

Heated Length	144
Fuel Pellet OD	0.3699
Clad OD	0.422
Clad Thickness	0.0243
Number of Pins per Channel	11,000 (44,000 total)

c) Heat Transfer

Gap Conductance (Btu/hr-ft ² -°F)	2000
Surface Conductance	Modified RELAP Heat Transfer Package
Initial Surface Temperature	Radial Power Distribution
	<u>Cosine</u> <u>Uniform</u>
Average	959 1004
Maximum	1938 1527
Minimum	494 592

d) Axial Power Profile

<u>Position from Bottom of Heated Length (ft)</u>	<u>Power Factor (Local/Average)</u>
0-1	0.456
1-2	0.629
2-3	0.876
3-4	1.140
4-5	1.380
5-6	1.520
6-12	Symmetric with 0-6

e) Radial Power Profile (Nonuniform Case)

<u>Channel Number</u>	<u>Power Factor (Local/Average)</u>
2	0.685
3	0.885
4	1.100
5 (Center)	1.330

temperature of about 1100°F. The local clad temperature varies significantly because of the assumed cosine axial and radial power profiles. A map of the initial clad surface temperature is shown in Figure 5-6 for both uniform and cosine radial power profiles.

The initially uniform liquid level fills the lower plenum to the bottom of the core. After allowing a short time for the core steam temperatures to stabilize, subcooled water begins to enter the lower plenum, reaching an equivalent core flooding rate of about 4.5 inches per second at 0.7 seconds into the simulated transient. In the absence of a flooding correlation or drift model, this lower plenum injection is the most meaningful method of introducing cooling water to the core for these calculations.

The overall core dynamic behavior and clad temperature response are presented in Figures 5-7 through 5-12 for a 25-second transient. Figure 5-7 shows the transient liquid level in downcomer and core and the core axial velocity profile for the uniform power case. Figure 5-8 illustrates the level and velocity for the cosine radial power profile. The temperature response is described in Figures 5-9 through 5-12. Figure 5-9 shows the clad temperature, surface heat transfer coefficient, and density for the first foot of core in each of the four flow channels. Figures 5-10 and 5-11 plot the axial surface temperature profiles for the inner and outer rods in the first four feet of core. The darkened areas on the small diagrams in each figure indicate the region of the core from which the data were computed.

The general system behavior for this time period appears to consist of two fairly distinct stages. The first is characterized by a violent, chaotic oscillation and a net decrease in the amount of liquid in the lower plenum. After a particularly violent expulsion at about 8.5 seconds, which overflows the downcomer and nearly empties the lower plenum, the second stage of somewhat better-behaved oscillations begins. The beginning of the second

FIGURE 5-6. Initial Core Temperature Contours

FIGURE 5-7. Liquid Level and Velocity -- Uniform Core Power

FIGURE 5-8. Liquid Level and Velocity -- Nonuniform Core Power

FIGURE 5-9. Heat Transfer -- Nonuniform Core Power

FIGURE 5-10. Axial Surface Temperature Profile -- Uniform Core Power

FIGURE 5-11. Axial Surface Temperature Profile -- Nonuniform Core Power

stage corresponds roughly with the time when most of the fuel rods in the first one foot of core have cooled nearly to saturation temperature. During this later period, liquid refills the lower plenum and penetrates the first few inches of core. The equivalent core flooding rate for both radial power profiles is about 1.6 inches per second based on core flow area and saturated liquid density.

Although the basic solution algorithm performed quite satisfactorily throughout these simulations, these results revealed several deficiencies in the constitutive relations and associated computation logic. The use of twelve-inch axial computational cells for the fuel model in conjunction with blowdown-type heat transfer logic caused a "stepwise" quenching rather than a steadily progressing quench front. This nonphysical process was so violent that the true response of the flow field was probably obscured. Also, the "reference pressure" equation of state in COBRA prevents local pressure variations from influencing other properties, preventing consideration of vapor compression, an important aspect in steam binding. Finally, the homogeneous equilibrium assumption for two-phase flow cannot consider relative phase motion. With this model, it is not possible to consider the downcomer flooding phenomenon, liquid entrainment at the quench front or the nonequilibrium condensation in the downcomer from ECC injection.

6.0 DATA COMPARISONS AND APPLICATIONS FOR THE COBRA-DF COMPUTER PROGRAM

The COBRA-DF computer program is an extended version of the COBRA-IV code⁽⁸⁾ which can perform transient calculations with relative motion between the liquid and vapor phase and also treat fluids in thermal nonequilibrium. A summary of the conservation equations used in COBRA-DF were presented in Section 3.3. In Section 6.1 the results of computations with the COBRA-DF code are compared with transient one-dimensional experiments performed by the General Electric Corporation. The COBRA-DF code is still under development and the main purpose of the data comparisons is to show how the current drift flux models are performing.

The drift flux model will be a vital analytical tool in simulating the hydraulic behavior within a PWR reactor core vessel under postulated accident conditions. Section 6.2 documents current status of simulation of the hydraulic behavior of a PWR downcomer for conditions similar to those expected during emergency core cooling injection. The thermal-hydraulic phenomena observed in small-scale experimental programs and the method currently used to characterize the vapor drift velocity within the downcomer are described. Results from COBRA-DF calculations are compared with data obtained in the Creare flat plate downcomer experiments.

6.1 ONE-DIMENSIONAL TRANSIENT DATA COMPARISONS

In this section, results of computations with the COBRA-DF computer program are compared with transient one-dimensional experiments performed by the General Electric Corporation. These experiments were conducted with Freon as the test fluid at system

pressures designed to match the ratio of the liquid and vapor densities of Freon to that of water at 1000 psia.

Three data comparisons are presented:

- a sinusoidally varying power input at constant inlet flow rate
- an exponential power decay at constant inlet flow
- a flow decay at constant power.

All computations were executed using the churn-turbulent⁽⁹⁾ vapor drift velocity and a value of the distribution parameter, C_o , between 1.10 and 1.15 adjusted to give a best fit to the experimental data.

The apparatus used in the oscillatory power input experiment⁽¹⁰⁾ included a vertical stainless steel tube with Freon-22 as the test fluid. The tube had a 0.40-inch inside diameter, 0.020-inch wall thickness, and 60.6-inch heated length with six void measurement stations along its length. The tube was directly heated by a power supply capable of furnishing constant or oscillatory heat input.

Run number 92-III-28 was used for data comparison. Operating conditions for this test were:

system pressure	= 158.3 psia
inlet subcooling	= 1.1° F
mass flux	= 337,900 lb/hr ft ²
average heat flux	= $P_{av} = 4,820$ Btu/hr ft ²
power supplied to test section	= $P_{av} (1 + 0.183 \sin \omega t)^2$
frequency of power oscillation	= 0.5 cps ($\omega = \pi$)

Heat capacity of the test section caused a phase shift of approximately 0.3 seconds and attenuated the power input to the fluid by about 60 percent.

As shown in Figure 6-1, the data and values predicted by COBRA using the 3-equation drift flux model are in good agreement. The average value of the results obtained from COBRA is very close to the GE data, although the computed results are somewhat flatter than the data. The phase angle of the computed and experimental results are also in good agreement.

The apparatus used for both the inlet flow and power decay tests⁽¹¹⁾ included a vertical five-foot directly heated stainless steel tube with a 0.55-inch inside diameter. A void meter was located 5.5 inches from the end of the heated length and the apparatus was set up so that either the power input to the test section or the inlet flow could be varied with time.

Both the inlet flow and power decay experiments began at steady-state conditions. The transients were initiated by rapidly reducing either flow or power. For the flow decay test, the inlet flow was reduced approximately 50 percent, and the power transient investigated a reduction to zero power. The primary experimentally determined variable was the average void fraction 5.5 inches above the heated region of the test section. Operating conditions for the two runs were as follows:

	<u>Flow Decay</u>	<u>Power Decay</u>
System pressure	123 psia	123 psia
Inlet subcooling	7.9 Btu/lb	9.08 Btu/lb
Initial mass flux	880,500 lb/hr-ft ²	251,700 lb/hr-ft ²
Final mass flux	439,700 lb/hr-ft ²	251,700 lb/hr-ft ²
Initial heat flux	18,488 Btu/hr-ft ²	15,360 Btu/hr-ft ²
Final heat flux	18,488 Btu/hr-ft ²	0 Btu/hr-ft ²

FIGURE 6-1. Oscillatory Power Transient

The above experiments were simulated with COBRA using one channel and 13, 26, or 52 axial nodes. The 3-equation drift flux model was used for the flow transient calculation, and the 5-equation model was employed in the power decay transient calculation.

As shown in Figure 6-2, the data and COBRA predictions agree favorably for the flow transient. The void fraction is overestimated in the regions of small void fraction and underestimated in the region of larger void fractions. One possible explanation for this is that C_o was assumed constant when it is probably a function of flow regime. Although the void fraction is not exactly predicted, the shapes of the curves are very similar. In addition, the method of characteristics analysis used by Lahey, et al.,⁽¹¹⁾ to predict the void fraction also deviated from the data in a manner similar to the COBRA predictions.

Figure 6-3 presents a comparison between COBRA predictions and the experimental data for the power decay test. The steady-state void fraction is predicted well. During the transient, agreement is good to approximately 0.75 seconds, then the computation and experimental data deviate. The predicted void decreases in a parabolic manner while the data exhibits a continuity wave phenomenon, remaining at approximately the initial value for 2.75 seconds, then reducing very rapidly to zero at 3.2 seconds.

As in the previous data comparisons, V_{gj} and C_o were also assumed independent of flow regime for this analysis. A more general specification of these parameters might enhance the predictive capability of the COBRA code for this test.

6.2 DOWNCOMER MODELING

This section presents the current status of progress in simulating the hydraulic behavior within a PWR downcomer for conditions similar to those expected during Emergency Core

FIGURE 6-2. Flow Transient

FIGURE 6-3. Power Transient

cooling injection. Included are: (1) a description of the thermal-hydraulic phenomena observed in small-scale experimental programs, (2) the method currently used to describe the vapor drift velocity within the downcomer, and (3) results of comparisons with data obtained in the Creare flat plate tests.

Emergency core cooling water is injected into the downcomer of a PWR via direct annulus injection or through cold leg injection during the latter portion of blowdown following a loss-of-coolant accident. Upon entering the downcomer, the coolant water interacts with reverse core steam flowing up the downcomer from the lower plenum. Small-scale experiments ^(12,13,14) have shown that, at sufficiently high steam flow rates, liquid may be carried out the broken leg.

As the steam flow rate is decreased, some of the injected water will penetrate to the lower plenum. Further reduction in the steam flow rate results in complete delivery of the ECC water to the lower plenum. The time at which this occurs has been termed the end of bypass. The end of bypass is of significant importance in calculating the amount of time required to reflood the core and thus has a direct effect upon the peak cladding temperature obtained in the fuel assembly. Because of the significance of this phenomenon in determining the overall effectiveness of the ECC system to limit the peak cladding temperature to acceptable levels, it is desirable to simulate it as accurately as possible.

A great deal of work has been done to define the dominant factors affecting the penetration of ECC cooling water to the lower plenum. It is not the purpose to describe the phenomena in detail here; however, the salient features will be provided and the interested reader is referred to reports ^(12,13,14,15) describing the phenomena in greater detail.

The following main conclusions have been reached from the small-scale experiments:

1. The amount of water reaching the lower plenum is a direct function of the reverse core steam flow with more water being delivered as the steam flow is decreased.
2. The ECC injection flow rate affects the amount of lower plenum delivery, with higher injection flows delivering more water at a given steam flow rate.
3. The subcooling of the injected water decreases the amount of steam available to bypass the water by condensing some of the reverse core steam flow. Increased subcooling of the ECC water results in increased penetration to the lower plenum for a given steam flow.
4. The stored energy in the vessel wall and core barrel results in the evaporation of some of the ECC water. This evaporation adds to the effect of the reverse core steam flow in bypassing the water. Therefore, at high wall superheats, less water is delivered to the lower plenum for a given reverse core steam flow than when no wall superheat is considered. As the stored energy is removed from the wall, the penetration of liquid to the lower plenum increases. Hot walls have the net effect of delaying the time at which end of bypass occurs.

The major effort thus far in modeling the downcomer with COBRA has been to predict the effect of the first two factors mentioned above, i.e., the flooding phenomenon. Once this problem has been solved, the effects of condensation and the hot wall may be taken into account using appropriate heat transfer correlations and condensation rates. This will require the use of thermodynamic nonequilibrium.

The flow behavior observed to occur in the downcomer is similar to the phenomenon known as flooding. It has been found that penetration data from small-scale experiments can be correlated using a Wallis type flooding correlation of the form^(13,16)

$$J_g^{*1/2} + M J_f^{*1/2} = C \quad (6.1)$$

where

$$J_X^* = \frac{\rho_X^{1/2} J}{[gw(\rho_f - \rho_g)]^{1/2}} \quad (6.2)$$

J^* - dimensionless groups which relate momentum fluxes to buoyancy forces

X - subscript, either f or g

- density

J - average volumetric flux density

w - annulus width

f - subscript for liquid phase

g - subscript for vapor phase

g - gravitational constant

M - slope of flooding curve

c - constant, characteristic of the steam flow at which some water begins to penetrate to the lower plenum.

Because the amount of water reaching the lower plenum increases with increasing injection rates, this effect may be accounted for by changing the value of M for various flow rates. This has been done⁽¹³⁾ and the following functional relationship has been found to correlate the data

$$M = \exp[-5.6 J_{f,in}^{*0.6}] \quad (6.3)$$

where $J_{f,in}^*$ is the dimensionless volumetric flux of the injected ECC water. The value of the constant C was found to have a value of 0.32 for best correlation of the data. To use this correlation in performing COBRA calculations, it is necessary to either (1) write the flooding correlation in terms of a drift velocity or (2) use the correlation to limit the downward liquid velocity as computed by one of the generally accepted drift velocity relationships.

To obtain a preliminary estimation of COBRA's ability to compute the downcomer phenomena, the first approach was selected. A more general specification for the drift velocity is being incorporated into COBRA where the second method, i.e., limiting the liquid penetration with the flooding correlation, will be used.

The flooding correlation can be written in terms of a drift velocity using an approach suggested by Westinghouse.⁽¹⁷⁾ The drift velocity may be defined in terms of the component fluxes as follows:

$$J_g = \frac{\alpha V_{gj}}{(1-\alpha)} + \frac{\alpha}{1-\alpha} J_f \quad (6.4)$$

where

V_{gj} - drift velocity
 α - void fraction.

The flooding correlation may also be written in terms of the component fluxes

$$J_g^{1/2} + m \left(\frac{\rho_f}{\rho_g} \right)^{1/4} J_f^{1/2} = \frac{C [g w (\rho_f - \rho_g)]^{1/4}}{\rho_g^{1/4}} = C' \quad (6.5)$$

If it is assumed that equation 6.4 is tangent to the curve defined by equation 6.5⁽¹⁶⁾, then there is a sufficient number of equations to solve for the three unknowns, J_g , J_f , and V_{gj} . Solving for V_{gj} , the following expression for the flooding drift velocity, V_{gj_f} , is obtained:

$$V_{gj_f} = \frac{(1-\alpha) (C')^2}{\alpha + m^2 (\rho_f / \rho_g)^{1/2} (1-\alpha)} \quad (6.6)$$

In deriving this expression, it was assumed that (1) there is uniformity of flow and void across the flow area, (2) the drift flux lines are tangent to the flooding curve, and (3) the flooding correlation correctly models hydraulic phenomena in the downcomer. The use of this correlation beyond its data base, however, is tentative at best.

This drift correlation was used in a COBRA simulation of the Creare flat plate downcomer. A schematic of the Creare facility is shown in Figure 6-4 and the corresponding COBRA representation is described in Figure 6-5. Channels 2 through 5 in Figure 6-5 represent the downcomer. Channel 2 is left open at the top to simulate the broken cold leg and a zero flow boundary condition is used at the top of channels 3 through 5 to simulate the top of the downcomer. Channel 1 provides a reverse core steam supply

PHYSICAL MODEL

FIGURE 6-4. Creare Flat Plate Test Apparatus

FIGURE 6-5. COBRA Model of Creare Apparatus

and channel 6 allows for water injection at the top of the downcomer. Fluid flow between channels 5 and 6 is allowed only through the opening shown.

Results of computations obtained with this model are shown in Figure 6-6 where the dimensionless water flow is plotted versus dimensionless counter current steam flow. Open symbols represent experimental data obtained at Creare and solid symbols display results obtained with COBRA-DF. Dashed lines have been added for clarity.

The COBRA calculation follows the experimental trends quite well. The flooding correlation as it is applied in COBRA, predicts the data upon which the correlation is based. In addition to predicting the liquid penetration, the pressure drop out to the break is similar to that observed in experiments: high pressure drops are associated with complete bypass and low pressure drops are associated with complete penetration. It may be concluded that COBRA-DF can compute hydraulic behavior under downcomer conditions and, given the correct drift velocity, good agreement with experimental data can be obtained.

The present drift velocity correlation is strictly limited to conditions when flooding is occurring and its application to each computational cell, as was done here, is questionable. A more realistic drift correlation is being incorporated into COBRA where the flooding correlation is used only as a limiting condition on the vapor or liquid velocity during counter-current flow situations.

FIGURE 6-6. Annulus Penetration for Steam-Water Countercurrent Flow

REFERENCES

1. F. H. Harlow and A. A. Amsden, "A Numerical Fluid Dynamics Calculation Method for All Flow Speeds." Journal of Computational Physics. 8:197-213, 1971.
2. C. W. Stewart and D. S. Rowe, "Advanced, Continuous Fluid Eulerian Computation Scheme for Flows with Large Density Gradients." ANS Transactions. 24:178, 1976.
3. C. W. Hirt and J. L. Cook, "Calculating Three-Dimensional Flows Around Structure and Over Rough Terrain." Journal of Computational Physics. 10:324-340, 1972.
4. H. S. Crapo, M. F. Jensen and K. E. Sackett, Experiment Data Report for Semiscale Mod-1 Test S-D2-3 (Blowdown Heat Transfer Test). ANCR-1233, Idaho National Engineering Laboratory, Idaho Falls, ID, September 1975.
5. D. M. Snyder, Aerojet Nuclear Company, Idaho Falls, ID, private communication, October 1975.
6. A. R. Edwards and T. P. O'Brien, "Studies of Phenomena Connected with the Depressurization of Water Reactors." Journal of the British Nuclear Society. 9(1-4): January 1970.
7. C. W. Hirt and N. C. Romero, Application of a Drift-Flux Model to Flashing in Straight Pipes. LA-6005-MS, Los Alamos Scientific Laboratory, Los Alamos, NM, July 1975.
8. C. L. Wheeler et al., COBRA-IV-I: An Interim Version of COBRA for Thermal-Hydraulic Analysis of Rod Bundle Nuclear Fuel Elements and Cores. BNWL-1962, Battelle, Pacific Northwest Laboratories, Richland, WA 99352, March 1976.
9. M. Ishii, Thermo-Fluid Dynamic Theory of Two-Phase Flow. Eyrolles, 1975.
10. G. Bijwaard, F. W. Staub and N. Zuber, A Program of Two Phase Flow Investigation. GEAP-4844, General Electric Corporation, 1965.
11. R. T. Lakey, B. S. Shiralkar, J. M. Gonzalez and L. E. Schnebly, The Analysis of Transient Heat Flux. GEAP-13249, General Electric Corporation, 1972.

12. C. J. Crowley and J. A. Block, ECC Delivery Study-Experimental Results and Discussion. Creare Technical Note TN-217, Creare, Hanover, NH, October 1975.
13. J. A. Block, P. H. Rothe, M. W. Fanning, C. J. Crowley and G. B. Wallis, Analysis of ECC Delivery. Creare Technical Note TN-231, Creare, Hanover, NH, April 1976.
14. L. J. Flanigan, R. A. Cudnik and R. S. Denning, Topical Report on Experimental Studies of ECC Delivery in 1/15-scale Transparent Vessel Model. BMI-1941, Battelle Columbus Laboratories, Columbus, OH, November 1975.
15. R. A. Cudnik, L. J. Flanigan, J. A. Carr and R. S. Denning, Reactor Safety Program Applications and Coordination, Task I- Steam-Water Mixing Program and System Hydrodynamics. BMI-NUREG-1949, Battelle Columbus Laboratories, Columbus, OH, May 1976.
16. G. B. Wallis, One-Dimensional Two-Phase Flow. McGraw-Hill Publishing Company, New York, NY, pp. 315-374.
17. R. D. Kelley et al., Application of Drift Flux to Loss of Coolant Analysis. Paper No. 74-WA/HT-40, American Society of Mechanical Engineers, New York, NY, 1974.

DISTRIBUTION

No. of
Copies

No. of
Copies

OFFSITE

	A.A. Churm ERDA Chicago Patent Group Chicago Operations Office 9800 South Cass Avenue Argonne, IL 60439	E. Ziurys Division of Magnetic Fusion Energy Energy Research and Development Administration Washington, DC 20545
3	M. Jinks, Chief Central Mail and Files Nuclear Regulatory Commission Washington, DC 20555	R.J. Neuhold Division of Reactor Development and Demonstration Energy Research and Development Administration Washington, DC 20545
311	<u>ERDA Technical Information Center for NRC-4</u>	M. Legan Argonne Code Center Argonne, IL 60439
5	Chief, Systems Engineering Branch Division of Reactor Safety Research Nuclear Regulatory Commission Washington, DC 20555	R. Singer Argonne National Laboratory 9700 South Cass Avenue Argonne, IL 60439
	J. Baublitz Division of Magnetic Fusion Energy Energy Research and Development Administration Washington, DC 20545	G. Wikhammer Advanced Engineering Branch Atomic Energy of Canada, Ltd. Chalk River, Ontario CANADA KOJIJO
	H. Cullingford Division of Magnetic Fusion Energy Energy Research and Development Administration Washington, DC 20545	B. Hao Babcock and Wilcox Company Lynchburg, VA 24505
	B. Twining Division of Magnetic Fusion Energy Energy Research and Development Administration Washington, DC 20545	J.F. Kapinos Combustion Engineering, Inc. Windsor, CT 06095
		R. Benedetti EG and G, Incorporated P.O. Box 1625 Idaho Falls, ID 83401

No. of
Copies

C.M. Rice
EG and G, Incorporated
P.O. Box 1625
Idaho Falls, ID 83401

L. Agee
Electric Power Research
Institute
3412 Hillview Avenue
P.O. Box 10412
Palo Alto, CA 94304

R.T. Fernandez
Electric Power Research
Institute
3412 Hillview Avenue
P.O. Box 10412
Palo Alto, CA 94304

K. Nilsson
Electric Power Research
Institute
3412 Hillview Avenue
P.O. Box 10412
Palo Alto, CA 94304

B.R. Sehgal
Electric Power Research
Institute
3412 Hillview Avenue
P.O. Box 10412
Palo Alto, CA 94304

C. Sullivan
Electric Power Research
Institute
3412 Hillview Avenue
P.O. Box 10412
Palo Alto, CA 94304

F. Bennet
General Atomic Company
P.O. Box 608
San Diego, CA 92112

No. of
Copies

A.E. Bergles
Iowa State University
Ames, IA 50010

J.H. Scott
Los Alamos Scientific
Laboratory
Los Alamos, NM 87544

N.E. Todreas
Massachusetts Institute of
Technology
Cambridge, MA 02139

L. Wolf
Massachusetts Institute of
Technology
Cambridge, MA 02139

D.M. Chapin
MPR Associates
Washington, DC 20036

H. Estrada
MPR Associates
Washington, DC 20036

D. Nguyen
Sandia Laboratories
Albuquerque, NM 87115

Manager
Blowdown Heat Transfer
Program
Union Carbide Corporation
Oak Ridge National
Laboratory
P.O. Box X
Oak Ridge, TN 37830

Manager
Heat Transfer-Fluid Dynamics
Department
Union Carbide Corporation
Oak Ridge National
Laboratory
P.O. Box X
Oak Ridge, TN 37830

No. of
Copies

No. of
Copies

P. Jalouk
Union Carbide Corporation
Oak Ridge National
Laboratory
P.O. Box X
Oak Ridge, TN 37830

A.A. Bishop
University of Pittsburgh
Pittsburgh, PA 15213

C.L. Caso
Westinghouse Electric
Corporation
Pittsburgh, PA 15236

R.A. Markley
Westinghouse Electric
Corporation
Pittsburgh, PA 15236

E. Novendstern
Westinghouse Electric
Corporation
Pittsburgh, PA 15236

D. Spencer
Westinghouse Electric
Corporation
Pittsburgh, PA 15236

ONSITE

ERDA Richland Operations
Office Energy Programs
Division

H.E. Ransom

EXXON Nuclear Company

D.S. Rowe

3 Hanford Engineering
Development Laboratory

M.W. Cappiello
S.A. Wood
D.M. Shearer

United Nuclear Industries

G. Smith

98 Battelle-Northwest

D.T. Aase
J.M. Bates
S.H. Bush
N.E. Carter
R.J. Cena
J.M. Creer
J.M. Cuta
D.E. Fitzsimmons
T.L. George
C.R. Hann
A.J. Haverfield
B.M. Johnson
D.K. Kreid
W.W. Laity
R.A. McCann
M.A. McKinnon
C.A. McMonagle
D.E. Oleson
L.T. Pedersen
M.S. Quigley
C.W. Stewart (60)
A.M. Sutey
M.J. Thurgood
D.S. Trent (5)
C.L. Wheeler (5)
R.D. Widrig
F.R. Zaloudek
Technical Information (3)
Technical Publications