4.0 REACTOR COOLANT SYSTEM THERMAL-HYDRAULIC MODEL

This section contains a description of the Reactor Coolant System (RCS) thermal-hydraulic models used for the primary system loops, the pressurizer, and the reactor vessel. The name of the subroutine and the name of the procedure where the model is implemented are indicated in parentheses at the beginning of each subsection.

4.1 System Representation

The RCS is typically represented by the node-flowpath network shown in Figures 4.1 and 4.2 (corresponding to a typical C-E plant and a 3-loop W-plant, respectively). The vessel is represented by four or more control volumes (nodes) and corresponding flowpaths which represent the following regions:

- Inner vessel core - upper plenum region.
- Vessel upper head region.
- Inner vessel reactor control cluster (control element assembly guide tubes) region.
- Vessel annulus - lower plenum. This region may optionally be represented by one control volume, or by up to twenty control volumes, the latter number depending on the plant geometry design, as indicated in Section 7.2.1.

The pressurizer and surge line are represented by a node and a flowpath, respectively. Each loop is represented separately by means of the multiple node-flowpath representation shown in the above figures. Each hot leg, steam generator, suction leg, reactor coolant pump, and cold leg is modeled separately. Each steam generator's primary side is represented by either two or four nodes, as discussed in Section 5.3.

The nodes enclose control volumes representing the fluid mass and energy. Flowpaths connecting the nodes represent the fluid momentum and have no volume. The

flowpaths are used to represent the fluid flow between nodes. The flowpaths are of two types: momentum paths (the mass flowrate is calculated by solving the complete momentum equation), and non-momentum paths (the mass flow is an interface with other systems, or is calculated using choked flow or orifice type equations). All pipe connections to interfacing systems, i.e., charging, letdown, safety injection, residual heat removal, drains, valve connections, etc., are modeled separately.

All frictional losses are assumed to occur in the flowpaths, with the exception of the

[

]

[

]

4.2 The Conservation Equations

The RCS thermal-hydraulic model is a non-homogeneous, non-equilibrium model formulated with five one-dimensional conservation equations of two-phase flow. The conservation equations are the following:

1. Conservation of liquid mass

$$\frac{dM_e}{dt} = \Sigma\,(1 - X_l)\,W_i + W_{cond} \qquad (4.2.1)$$

2. Conservation of mixture mass

$$\frac{dM}{dt} = \Sigma\,W_i \qquad (4.2.2)$$

3. Conservation of mixture energy

$$\frac{dE}{dt} = \Sigma W_i h_i + Q \qquad (4.2.3)$$

4. Conservation of steam total enthalpy

$$\frac{dH_s}{dt} = \Sigma X_i W_i h_{stm,i} + Q_{stm} - W_{cond} h_{stm} + W_{cond,vap} (h_{stm} - h_g) + W_{cond,surf} (h_{stm} - h_f) + W_{cond,wall} (h_{stm} - h_f) \qquad (4.2.4)$$

The summations in the above four equations are over all momentum and non-momentum paths connected to each given node.

5. Conservation of mixture momentum

$$\frac{1}{144g}\left(\frac{L}{A}\right)\frac{dW}{dt} = (P_u - P_d) - K_f \frac{\phi W|W|}{288 \rho_e g A^2} - K_g \frac{W|W|}{288 \rho g A^2} + \Delta P_{elev} + \Delta P_{pump} \qquad (4.2.5)$$

The nomenclature is as follows:

A	= flowpath area
E	= total internal energy
g	= gravity
h_g	= saturated steam enthalpy
h_f	= saturated liquid enthalpy
h_i	= path mixture enthalpy

h_{stm} = steam enthalpy

$h_{stm,i}$ = path steam enthalpy

H_s = total steam enthalpy

K_f = Reynolds number dependent friction k-factor for turbulent flow[4 1]

K_g = geometric k-factor

(L/A) = path average length over area

M = mixture mass

M_e = liquid mass

P_u = upstream nodal pressure

P_d = downstream nodal pressure

Q = total heat rate

Q_{stm} = heat rate to steam region

t = time

W_i = path flowrate

W_{cond} = total condensation rate (with negative component for boiling)

$W_{cond,vap}$ = vaporization rate (negative)

$W_{cond,wall}$ = condensation rate on wall

$W_{cond,surf}$ = condensation rate on liquid surface

X_i = flowpath quality

ϕ = two-phase frictional multiplier (Thom & Martinelli-Nelson[4 2], [4 3])

ρ = flowpath density

ρ_e = flowpath liquid density

ΔP_{elev} = elevation pressure drop

ΔP_{pump} = pump delta pressure

The mixture momentum equation is used for all momentum paths in the node-flowpath system representation, including the vessel internal paths, loops paths, and surge line.

The total heat rate Q includes:

i. Wall heat (all nodes)
ii. Pump heat (nodes downstream of pump paths)
iii. Core heat transfer to coolant (core node)
iv. Steam generator heat transfer (steam generator nodes)
v. Pressurizer heaters heat transfer to coolant (pressurizer node)
vi. Control Element Assembly (CEA) node to core node heat transfer (CEA node and core node).
vii. Upper plenum (core node) to upper head node heat transfer.

The above equations are obtained by means of standard integration procedures of the multidimensional conservation of mass, energy and momentum equations.[44],[45], and [46]

In addition, for convenience during the integration procedure, an additional conservation equation (Conservation of steam mass M_s)

$$dM_s/dt = \Sigma X_l W_l - W_{cond}$$

is integrated. The code will use the pairs M_e, M or M, M_s, depending on the non-equilibrium conditions, in the calculation of the nodal conditions (pressure and liquid and steam enthalpy).

The above thermal-hydraulic model represents single-phase liquid and steam (node solid, node empty), or two-phase conditions in every component of the system (vessel, loops, and pressurizer). The non-equilibrium formulation represents the coexistence of liquid and steam at different temperatures in every part of the system. Single-phase and two-phase flow conditions are simulated in every path of the system for all flow conditions (forced flow, asymmetric flow, density difference natural circulation,

interruption of natural circulation, etc). Other features of the RCS thermal-hydraulic model, i.e., phase separation, conservation equations for solutes and non-condensibles, are described in other sections.

Although the above formulation assumes that only two phases can coexist within a node at any given time, this assumption is used only for the calculation of[

]

4.3 Integration of the Conservation Equations

4.3.1 The Integration Method

The integration of the conservation equations is an implicit one step method. The implicit one step method is defined by backwards differentiation of the linearized conservation equations. That is, let the conservation equations be written in vector form as

$$\frac{dY}{dt} = f(Y)$$

Linearization of this equation gives

$$\frac{dY}{dt} = f(Yo) + f'(Yo)\,(Y\text{-}Yo)$$

where f '(Yo) is the Jacobian of the function f(Y) evaluated at Yo. Backwards discretization yields

$$\frac{\Delta Y}{\Delta t} = f(Yo) + f'(Yo)\ \Delta Y$$

where Δt denotes the base time step for linearization. This expression can be written in matrix form as

$$(I - \Delta t\, f'(Yo))\ \Delta Y = \Delta t\, f(Yo). \qquad (4.3.1)$$

4.3.2 The Linearized Conservation Equations

Implementation of the integration procedure Equation (4.3.1) requires definition of the linearized discretized conservation equations. They are the following (using the base time step for linearization):

[

]

The nomenclature for the discretized equations is the same described in Section 4.2, with the following additions:

Δ : Increment of the corresponding variable; e.g., ΔW for flow increment, etc.

$\frac{\partial}{\partial}$: partial derivative of the corresponding variables. E.g., $\frac{\partial P}{\partial M_e}$ is the partial derivative of pressure with respect to the liquid mass.

The system of Equations (4.3.1) is solved using the procedure described in Reference 4.7. That is, The above linear system of equations is first reduced in size by substitution of the changes in mass and energy (Equations (4.3.2)-(4.3.5)) in the discretized momentum equations, (4.3.6). This yields a linear system of equations for the changes in mass flow, whose coefficient matrix is a block type matrix like the one described in Reference 4.7. This system of equations is solved using the block inversion technique described in Reference 4.7. The solution is completed by solving for the mass and energy changes from the corresponding equations, (4.3.2)-(4.3.5).

4.4 Determination of Node State Variables

Integration of the conservation equations yields the following nodal variables:

1. Mixture mass (M) and mixture energy (U) for equilibrium states. (NON_EQ_STATE=0)

2. Mixture mass (M), liquid mass (M_e), and mixture energy (U), for the subcooled water-saturated steam non-equilibrium condition. (NON_EQ_STATE=1)

3. Mixture mass (M), steam mass (M_{stm}), and mixture energy (U), for the saturated water-superheated steam non-equilibrium condition. (NON_EQ_STATE=2)

4. Mixture mass (M), liquid mass (M_e), mixture energy (U), and total steam enthalpy (H_{stm}), for the subcooled water-superheated steam non-equilibrium state. (NON_EQ_STATE=3)

The above state variables are calculated for every node in the system. The code then determines the node thermal-hydraulic state (pressure, enthalpy, equilibrium, non-equilibrium) from the node state variables and equations of state (i.e., single-phase liquid, single-phase steam, two-phase equilibrium, two-phase non-equilibrium).

In order to determine the thermal state of a node, the code predicts, prior to integration and using the present state of the system and external fluid/heat sources or sinks, the anticipated thermal equilibrium/non-equilibrium state. If the node is single-phase, the predicted thermal state is equilibrium. If the node is two-phase, the predicted thermal state is determined by the user control flag NE_CANDIDATE(node) (non-equilibrium state to be used). After integration, the code determines the thermal state of the node using the predicted non-equilibrium state. If a consistent thermal state cannot be found with the predicted non-equilibrium state, the node switches to the non-

equilibrium/equilibrium state that yields a consistent state. The following possible thermal-hydraulic states are considered:

4.4.1 *Calculation of the Pressure and Mixture Enthalpy for Thermal Equilibrium Conditions (Single-phase liquid, single-phase steam, and saturated equilibrium two-phase).*

Given the mixture mass (M), mixture energy (E) and total volume (Vol), the node pressure (P) and mixture enthalpy (h) satisfy the following system of equations:

$$h = \frac{E}{M} + k\ Vol\ P/M \tag{4.4.1}$$

$$Vol/M = v(P,h) \tag{4.4.2}$$

where k = 144/777.98 (change of units constant) and v(P,h) is the specific volume equation of state. Solution of this system of equations using a Newton iterative solution yields P and h.

That is, substitution of Equation (4.4.1) in (4.4.2) yields

$$f(P) = 0$$

where

$$f(P) = \frac{Vol}{M} - v\left(P, \frac{E}{M} + k\ Vol\ \frac{P}{M}\right).$$

This equation is then solved using the standard Newton iterative procedure in one variable,

$$P_{i+1} - P_i = - f(P_i)/f'(P_i).$$

4.4.2 *Calculation of the Pressure and Liquid Enthalpy, for Subcooled Water-Saturated Steam Non-Equilibrium Conditions*

Given the mixture mass (M), liquid mass (M_e), mixture energy (E), and total volume (Vol), the pressure (P) and liquid enthalpy (h_e) satisfy the following system of equations:

$$M_e h_e + (M - M_e) h_g(P) = E + k \, Vol \, P \tag{4.4.3}$$

$$Vol = M_e v(P,h_e) + (M - M_e) v_g(P) \tag{4.4.4}$$

where

h_g = saturated steam enthalpy

v_g = saturated steam specific volume

$v(P,h_e)$ = specific volume equation of state.

Solution of this system of equations using a Newton iterative procedure yields P and h_e. That is, the system of Equations (4.4.3) and (4.4.4) is rewritten in the form,

$$f_1 (P,h_e) = 0$$

$$f_2 (P,h_e) = 0.$$

This nonlinear system of equations is solved using the Newton iterative procedure

$$A \begin{vmatrix} \Delta P \\ \Delta h_e \end{vmatrix} = - \begin{vmatrix} f_1 (P,h_e) \\ f_2 (P,h_e) \end{vmatrix}$$

where A is the 2 x 2 Jacobian matrix of the functions f_1, f_2. (This is the standard Newton iterative method in several variables).

4.4.3 *Calculation of the Pressure and Steam Enthalpy for Saturated Water-Superheated Steam Non-Equilibrium Conditions*

Given the mixture mass (M), steam mass (M_{stm}), mixture energy (E), and node volume (Vol), the node pressure (P) and steam enthalpy (h_{stm}) satisfy the following system of equations:

$$(M - M_{stm})\, h_f(P) + M_{stm}\, h_{stm} = E + k\, Vol\, P \qquad (4.4.5)$$

$$Vol = (M - M_{stm})\, v_f(P) + M_{stm}\, v(P,h_{stm}) \qquad (4.4.6)$$

where

$h_f(P)$ = saturated liquid enthalpy

$v_f(P)$ = saturated liquid specific volume

$v(P,h_{stm})$ = specific volume equation of state.

Solution of this system of equations using a Newton iterative procedure yields P and h_{stm}.

That is, the system of Equations (4.4.5) and (4.4.6) is rewritten in the form,

$$f_1(P,h_{stm}) = 0$$

$$f_2(P,h_{stm}) = 0.$$

This nonlinear system of equations is solved using the Newton iterative procedure

$$A \begin{vmatrix} \Delta P \\ \Delta h_{stm} \end{vmatrix} = - \begin{vmatrix} f_1(P,h_{stm}) \\ f_2(P,h_{stm}) \end{vmatrix}$$

where A is the 2 x 2 Jacobian matrix of the functions f_1, f_2.

4.4.4 *Calculation of the Pressure, Liquid Enthalpy and Steam Enthalpy for Subcooled Water-Superheated Steam Non-Equilibrium Conditions*

Given the liquid mass (M_e), total mass (M), mixture energy (E), total steam enthalpy (H_{stm}), and nodal volume (Vol), the pressure (P), liquid enthalpy (h_e), and the steam enthalpy (h_{stm}) satisfy the following system of equations:

$$M_e h_e + (M - M_e) h_{stm} = E + k \, Vol \, P \tag{4.4.7}$$

$$Vol = M_e \, v(P,h_e) + (M - M_e) \, v(P,h_{stm}) \tag{4.4.8}$$

$$h_{stm} = H_{stm}/(M-M_e) \tag{4.4.9}$$

where v(P,h) is the specific volume equation of state.

Solution of this system of equations using a Newton iterative procedure yields P, h_e, and h_{stm}.

That is, Equation (4.4.9) yields h_{stm}. Then Equations (4.4.7) and (4.4.8) can be rewritten in the form

$$f_1 (P,h_e) = 0$$

$$f_2 (P,h_e) = 0.$$

This nonlinear system of equations is solved using the Newton iterative procedure.

$$A \begin{vmatrix} \Delta P \\ \Delta h_e \end{vmatrix} = - \begin{vmatrix} f_1 (P,h_e) \\ f_2 (P,h_e) \end{vmatrix}$$

where A is the 2 x 2 Jacobian matrix of the functions f_1, f_2,

4.5 Calculation of the Derivatives for Integration of the Conservation Equations

Integration of the conservation equations requires values for $\partial P/\partial M$, $\partial P/\partial M_e$, $\partial P/\partial M_{stm}$, $\partial P/\partial E$, and $\partial P/\partial H_{stm}$, where

P = pressure

M = mixture mass

M_e = liquid mass

M_{stm} = steam mass

E = mixture energy

H_{stm} = steam total enthalpy.

(See the linearized equations in Section 4.3.)

The derivatives are calculated as in the following sections.

4.5.1 Equilibrium State

Equations (4.4.1) and (4.4.2) can be written in the form

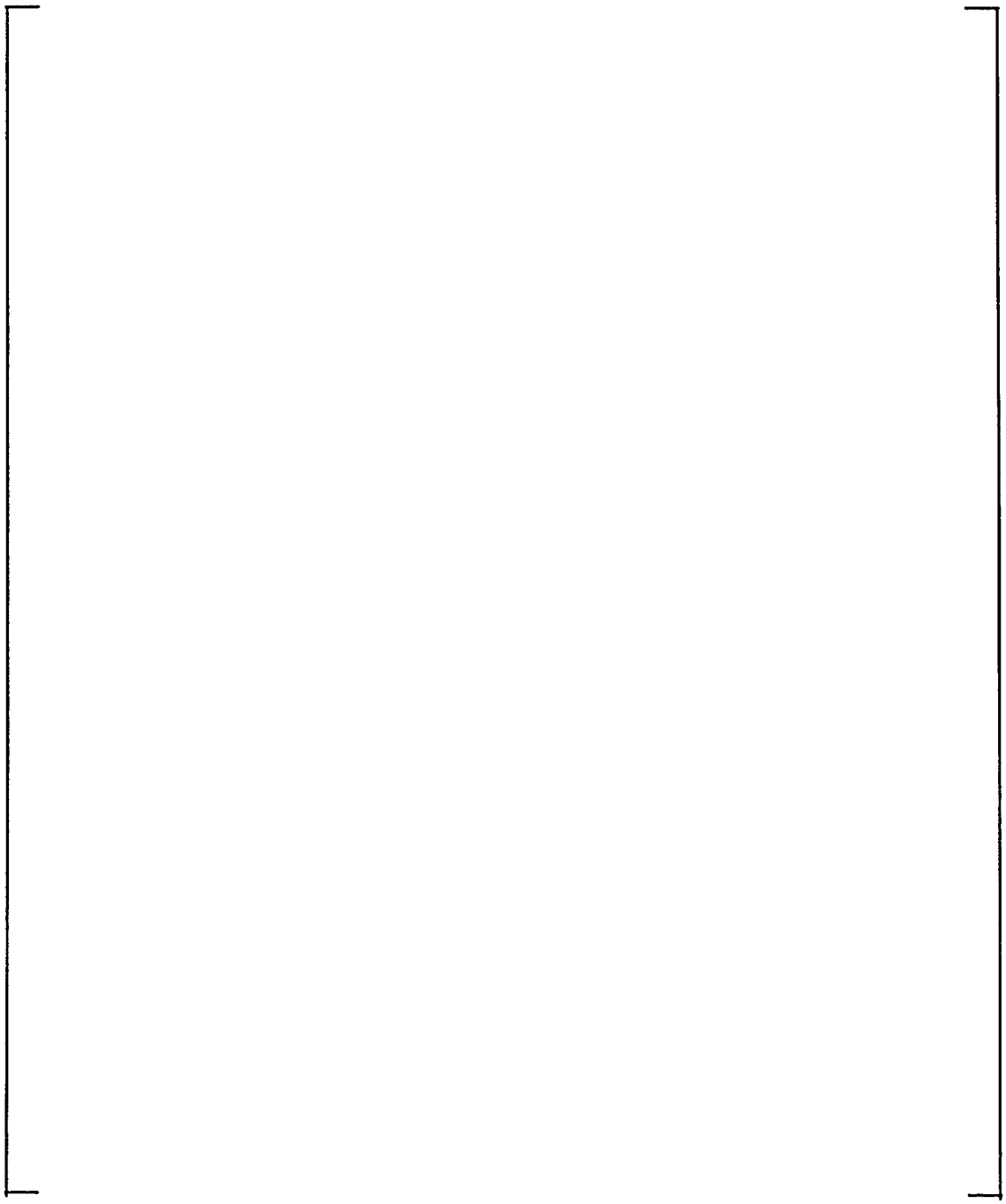
$$\text{Vol/M} - \nu\left(\text{P}, \frac{E}{M} + \text{k Vol P/M}\right) = 0$$

Assuming

[ ]

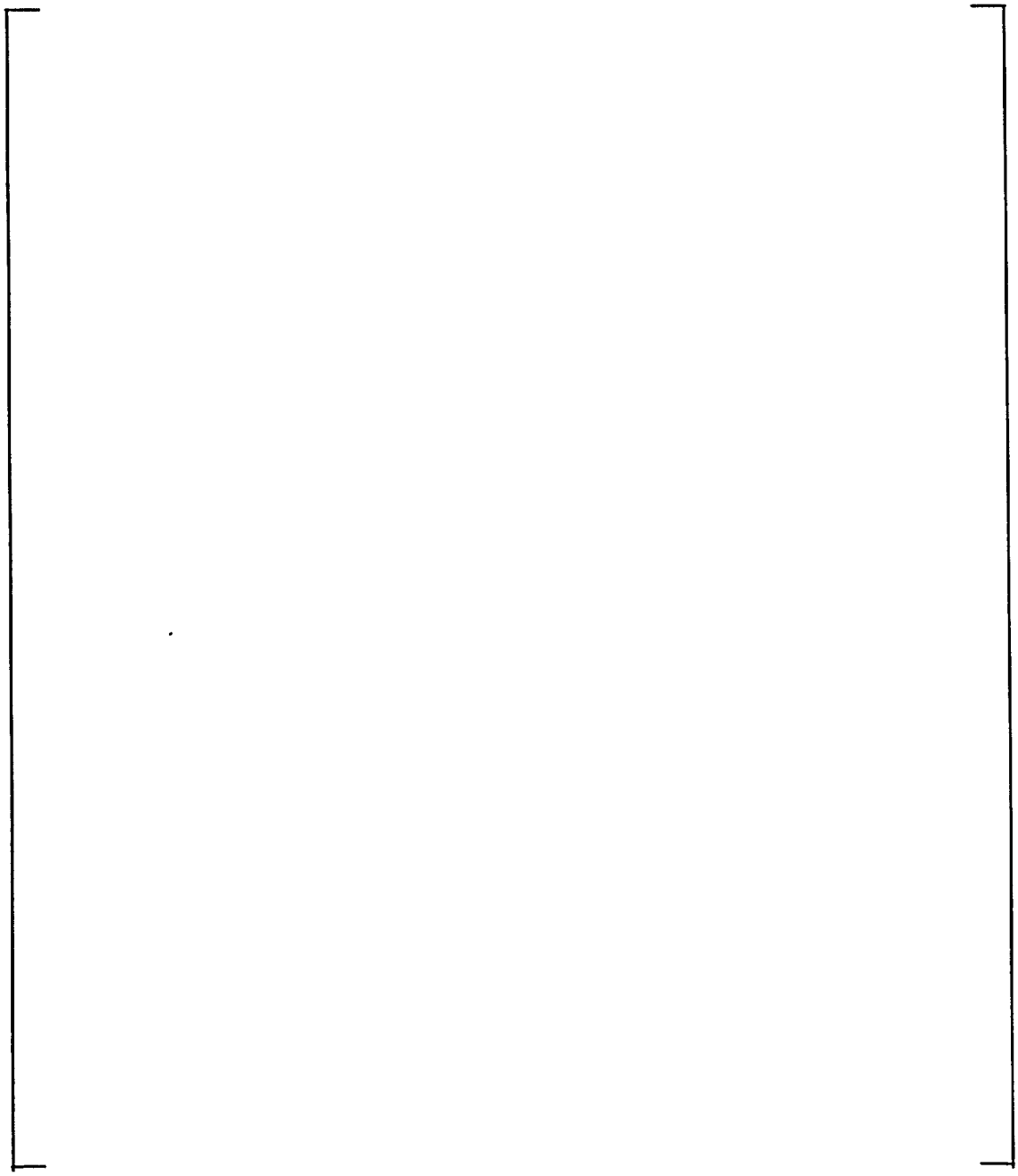
4.5.2 *Subcooled Water - Saturated Steam Non-equilibrium State*

Substitution of h_e from Equation (4.4.3) into Equation (4.4.4), and assuming



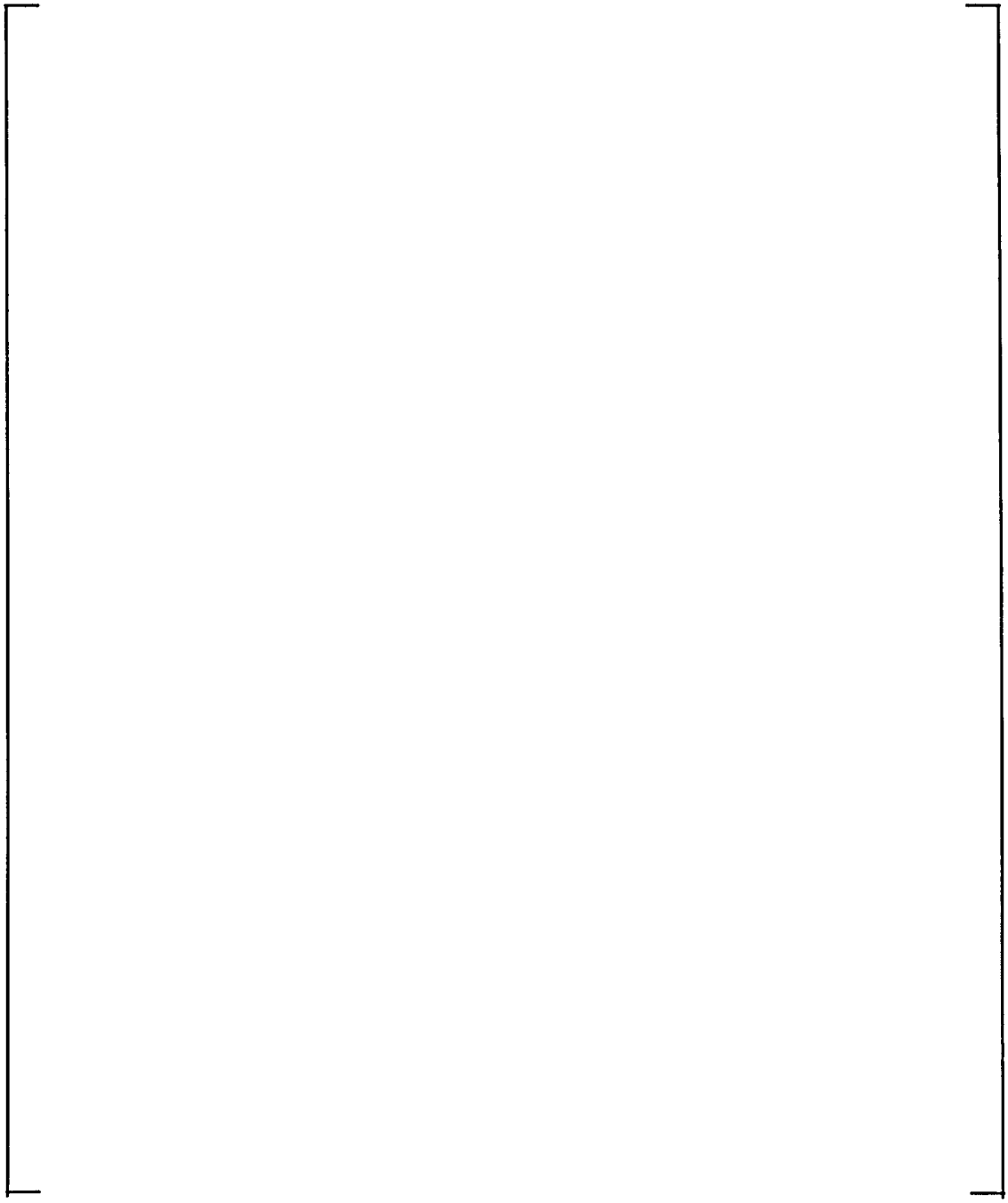
4.5.3 *Saturated Water - Superheated Steam Non-equilibrium State*

Substitution of h_{stm} from Equation (4.4.5) into Equation (4.4.6), and assuming



4.5.4 *Subcooled Water-Superheated Steam Non-equilibrium State*

Substitution of h_e and h_{stm} from Equations (4.4.7) and (4.4.9) into Equation (4.4.8), and assuming that



4.6 Steam Condensation

The steam condensation, W_{cond}, between the liquid and steam regions in a node is calculated as follows:

$$W_{cond} = W_{cond,surf} + W_{cond,inj} + W_{cond,bub} + W_{cond,vap} + W_{cond,wall} \quad (4.6.1)$$

where

$W_{cond,surf}$ = Condensation on the liquid surface of a stratified node,
$W_{cond,inj}$ = condensation on the injected subcooled water falling through steam,
$W_{cond,bub}$ = condensation of bubbles injected or produced in the liquid,
$W_{cond,vap}$ = vaporization of liquid due to heat sources and flashing,
$W_{cond,wall}$ = steam condensation on wall.

4.6.1 Condensation on Injected Liquid: Pressurizer Spray

Condensation on the pressurizer spray is of the form

$$W_{cond,inj} = \eta\, W_{spray}\, (h_f - h_{spray})/h_{fg}$$

where

h_f = liquid enthalpy
h_{fg} = evaporation heat
η = efficiency (function of pressurizer pressure and level)
W_{spray} = spray flow
h_{spray} = spray enthalpy.

The spray efficiency (η) is the product of the following terms:

[ ]

4.6.2 *Condensation on Injected Liquid: Other Than Pressurizer Spray*

For liquid entering a node from any internal or external flowpath other than the pressurizer spray, condensation on the injected fluid is of the form

$$W_{cond,\,inj} = h\,A\,(T_{stm,\,node} - T_{liq,\,path}) / h_{fg}$$

where

h = interphase heat transfer coefficient

A = interfacial heat transfer area

$T_{stm, node}$ = temperature of steam in the node

$T_{liq, path}$ = temperature of liquid delivered by the flowpath

h_{fg} = latent heat of vaporization.

In the presence of non-condensible gases, the node steam temperature in the above formulation is evaluated at the steam's partial pressure.

The heat transfer coefficient is calculated from the following correlation for[

]

where

k = liquid conductivity

r = droplet radius.

The interfacial heat transfer area for the injected fluid is calculated assuming droplet flow with spherical droplets with radius r, equal to[.

]

where

Vol = volume of the injected column of water

W = mass flowrate

t_{fall} = time for the drops to fall

α = void fraction

ρ = fluid density.

4.6.3 Condensation of Bubbles

Bubbles of steam may be dispersed within a node's liquid phase. The bubbles appear due to flashing, boiling or convection into the node's liquid phase via connecting flowpaths. Also, when the node is in a homogeneous state (Section 4.8.6), the entire steam inventory of the node is represented as bubbles. When the liquid is subcooled, the bubbles condense as follows:

$$W_{cond,\,bub} = h\,A\,(T_{sat} - T_{liq}) / h_{fg}.$$

The heat transfer coefficient is calculated from the following correlation for[

]

where

V_D = bubble relative velocity = $(g\,r_b)^{1/2}$

k_e = liquid conductivity

ρ_e = liquid density

C_{pe} = liquid specific heat

r_b = bubble radius = $2\,(\sigma / g\,\rho_e)^{1/2}$ (Ref. 4.16)

σ = surface tension = $g_c * 8.333 \times 10^{-6}\,(700 - T_{sat})$ lbm/sec^2 (Ref. 4.16)

g = gravitational constant = 32.17 ft/sec^2.

g_c = conversion factor = 32.17 lbm ft / lbf sec^2

The interfacial transfer area is the surface area for the mass of bubbles (M_B) dispersed in the liquid, assuming spherical bubbles with radius r_b .

$$A = 3 M_B / (\rho_{stm} r_b).$$

4.6.4 Condensation on Liquid Surface

The condensation on the liquid surface is of the form

$$W_{cond,surf} = h A (T_{sat} - T_{liq, surf}) / h_{fg} .$$

The terms of the above formulation are:

- h = Heat transfer coefficient, which is one of two constants for each node:
  - XFER_SURF_PON(node) when the pumps are running,
  - XFER_SURF_POFF(node), when the pumps are off.
- A = Surface area at the liquid-steam interface, determined as one of:
  - Circle chord area times pipe length, for the horizontal cylindrical-pipe nodes (N_GEOM(node) = 1) when the pumps are off:

$$A = \frac{8\,Vol}{\pi\,D}\sqrt{\frac{z}{D}\left(1-\frac{z}{D}\right)}$$

  - The node area, NODE_AREA(node), otherwise.
- T_{sat} = Saturation temperature of steam. In the presence of non-condensible gases, it is evaluated at the steam's partial pressure.
- $T_{liq, surf}$ = Liquid temperature at the surface.

4.6.5 *Condensation on Walls*

Condensation on walls is calculated for the pressurizer only. It is equal to

$$W_{cond,\,wall} = h\,A \cdot \Psi(P) \cdot (T_{sat} - T_{wall,\,stm})$$

[ ]

Condensation occurs if $T_{wall,\,stm} < T_{sat}$. Otherwise the condensation on the walls is zero.

4.6.6 *Vaporization*

The process of vaporization change-of-phase includes the effects of flashing due to depressurization and boiling due to heat sources. That is,

$$W_{cond,vap} = -(W_{flash} + W_{boil})$$

The rates of flashing and boiling are calculated from the expressions:

$$W_{flash} = -\{M_f \cdot dh_f / dt\} / h_{fg} > 0$$

$$W_{boil} = Q_e / h_{fg} > 0$$

where

M_f = saturated liquid mass

Q_e = rate of heat addition to the liquid region from walls, heaters, fuel rods, pumps, etc.

The following special cases are considered:

a. Pressurizer. [

]

b. Inner Vessel. [

]

c. All Other Nodes. [

]

d. Equilibrium Condensation Rate. [

]

4.6.7 *Condensation in the Presence of Non-Condensible Gases*

The presence of non-condensible gases in the gas/steam space affects condensation rates by

(a) decreasing the partial pressure of the steam and its saturation temperature, and

(b) reducing the condensation efficiency.

The condensation model accounts for the temperature effect by using the steam temperature or steam saturation temperature at the partial pressure, where necessary, as discussed above for the $W_{cond,inj}$, $W_{cond,surf}$ and $W_{cond,wall}$ terms of Equation (4.6.1).

The model accounts for the efficiency degradation due to non-condensibles by employing the following condensation efficiency term,

[ ]

4.7 Critical Flow Calculation

Critical (choked) flow for two-phase and subcooled liquid conditions is calculated using one of two correlations, selected by the user via the flag RCS_CRIT_MODEL. They are the homogeneous equilibrium model (HEM) correlation[4 9] and the Henry-Fauske (H-F) correlation[4 10]. Choked flow of saturated or superheated steam is determined from the Murdock and Bauman function[4 11]. When the throat pressure calculated from the correlation is less than the downstream pressure, a standard orifice equation is used.

The HEM and H-F tables are given in Appendix B. The Murdock and Bauman function is

$$W = (42.77\ A)\ (P/v)^{1/2}$$

where

W = steam flowrate (lbm/sec)
A = area (ft^2)
P = upstream pressure (psia)
v = steam specific volume (ft^3 /lbm)

The model employs the critical flow calculation in the following two ways;

- For leak paths and relief/vent valves, the flowrate is determined as above if the flow is choked, using the leak/valve area.
- Choked conditions are optionally (option flag RCS_CRIT_FLOW_CHECK) checked for all momentum governed flowpaths. The critical flow value is used if the momentum solution's mass flowrate exceeds the choked flow value.

4.8 Mixture Level Model

The height of a node's liquid inventory or two-phase mixture directly affects the flow properties (esp. flow quality) in the flowpaths that exit from the node, thereby affecting the conditions in neighboring nodes. The RCS model determines the mixture height dynamically, based on the node's geometry, mass inventories of the phases, the distribution and disengagement of bubbles in the liquid phase, and the possible presence and extent of a subcooled region. Subsections 4.8.1–4.8.5 discuss these parameters, followed by a discussion of the mixture height calculation in Subsection 4.8.6.

4.8.1 Node Geometry

For the calculation of the mixture level, each node is represented with one of the following three models:

- Lumped cylindrical node (vertical)
- Lumped cylindrical node (horizontal)
- Sectionalized node,
- Lumped segmented node.

The lumped cylindrical node is a horizontal or vertical right cylinder representation, which assumes a uniform node area and a uniform distribution of the bubbles in the liquid phase (Figure 4.3). It is used fo[

]

The sectionalized node representation models the node using piecewise uniform cylindrical sections. It represents the axial variation of temperature and void fraction

(bubble distribution) within the node (Figure 4.4). It is used for[

]

The lumped segmented node representation assumes a piecewise uniform cylindrical cross-sectional area and a uniform distribution of bubbles in the liquid phase (Figure 4.5). It is used for[ ]

4.8.2 Steam Dispersed in the Liquid Phase (Bubble Mass)

Integration of the conservation equations (Section 4.3) yields the mass and energy of liquid and steam for each node. The mixture level model calculates for each node the mass of dispersed bubbles and the level of the two-phase mixture. The mass of bubbles is calculated by discretizing the following equation for the conservation of steam dispersed in a liquid phase [4 14]:

[ ]

The discretized equation for Equation (4.8.1) is

[

]

Equation (4.8.2) is applied for all node conditions. For solid conditions (single phase liquid), the bubble mass (M_B) and steam mass (M_{st}) are both equal to zero.

4.8.3 Distribution and Release of Bubbles in Lumped Nodes

The bubble release rate W_{rele} and the steam bubble distribution are calculated for each node using either the model for the lumped nodes or the model for the sectionalized core node.

The lumped node model is used for both the lumped cylindrical nodes and lumped segmented nodes, that is, for all primary nodes except the core node. The model assumes a uniform distribution of bubbles in the liquid phase (Figures 4.3 and 4.5), and calculates the bubble release rate from the following equation [4 14]:

$$W_{rele} = V_D \rho_g A \alpha / (1-\alpha) \qquad (4.8.3)$$

where

V_D = drift velocity, given by a correlation of a fit of experimental data[4 14A]
= [ ]

P = pressure

ρ_g = saturated steam density

A = area

α = mixture void fraction.

Equation (4.8.3) was obtained directly from a drift flux formulation of liquid and steam fluxes, by setting the liquid flux at the surface is zero. That is,

$$V_D = j_g (1-\alpha) / \alpha - j_f / (1 - \alpha)$$

where

$$j_f = W_f / (\rho_f A)$$
$$j_g = W_g / (\rho_g A).$$

Setting[ ]Equation (4.8.3) follows. With the calculated W_{rele}, the nodal bubble mass is updated by Equation (4.8.2).

If the annulus is represented by a single node, then the bubble release rate for the annulus node is calculated from the equation

[

]

4.8.4 *Distribution and Release of Bubbles in Sectionalized Nodes*

The sectionalized node model is used for[

]

A quasi-steady calculation of the average void fraction and bubble release rate for each subsection is calculated from a drift flux model formulation.

The liquid flow and steam flow at the top of each section are calculated as follows

$$W_{g,i} = W_{g,i-1} + Q_i$$

$$W_{f,i} = W_{f,i-1} - Q_i$$

where

$W_{g,i}$ = steam flow at top of section i
$W_{f,i}$ = liquid flow at top of section i
Q_i = total steam production rate for section i.

The liquid and steam mass flow at the bottom of section 1 are obtained from the conditions of the path entering the[ ]

The void fraction[ ]solving the drift flux equation.

[ ] (4.8.4)

where

ρ_g = saturated steam density

ρ_f = saturated liquid density

[ ]

P = pressure, psia.

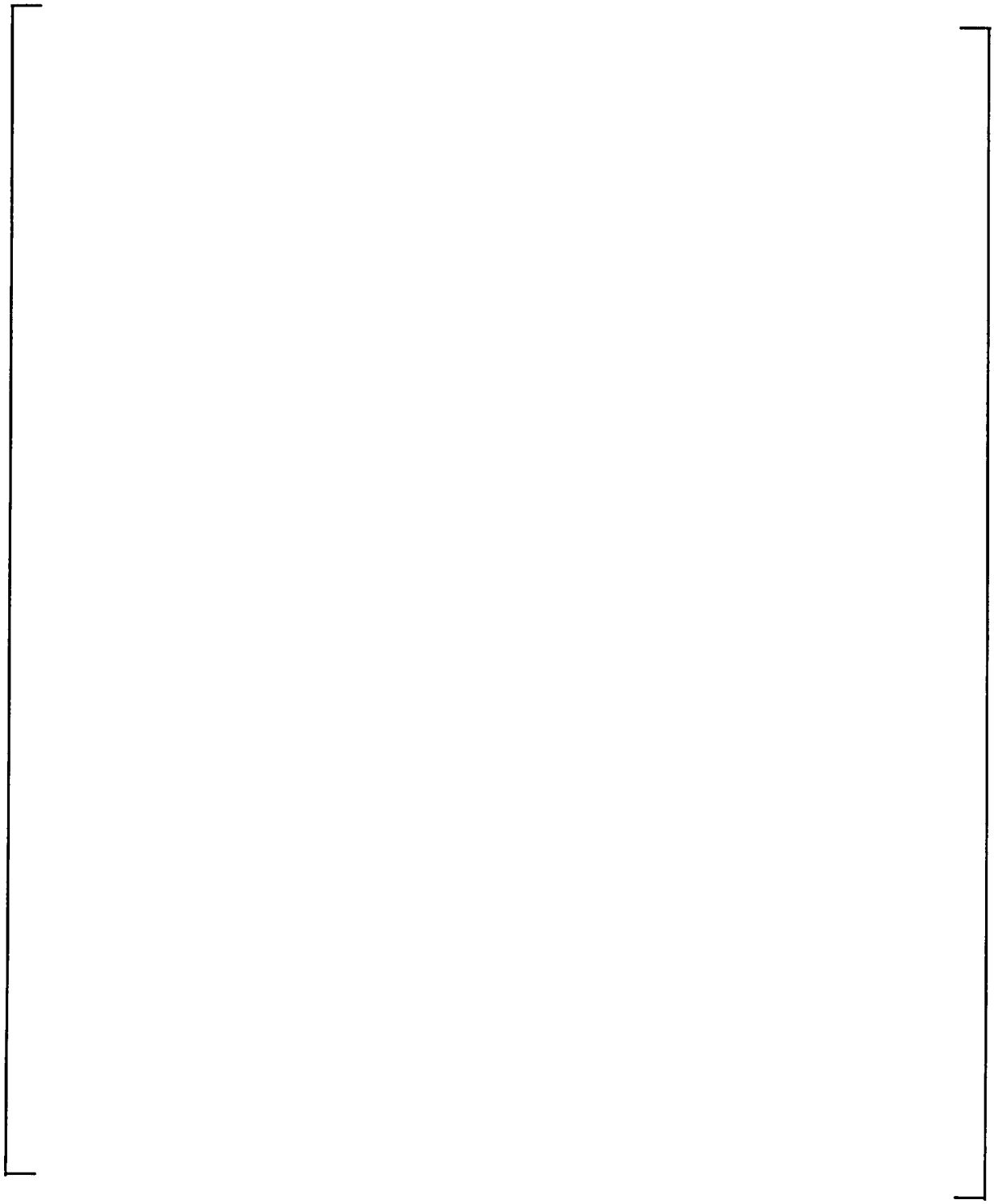
The bubble mass for [ ] is calculated from the equation

[ ]

where

[ ]

The above summation over the two-phase region is done starting at the saturation line, which marks the top of the subcooled layer (Section 4.8.5), and ending at the top of the two-phase mixture.



$$\frac{dW_{rele}}{dM_B} = \frac{1}{2} U_0 \; A_{top} \; \rho_g \cdot \frac{\alpha_{top,ss}}{M_{B,ss}} \cdot \frac{2 - \alpha_{top}}{(1 - \alpha_{top})^{3/2}}$$

4.8.5 *Height of Subcooled Layer in the Core*

The top of the subcooled liquid layer is the saturation line, the vertical location at which liquid in the core node reaches the saturated liquid state. The coolant is subcooled below the saturation line, and is saturated two-phase above it. The height of the saturation line is calculated by explicit integration of the differential equation,

[ ]

4.8.6 *Calculation of the Mixture Level*

The mixture height in each node depends on the current heterogeneous / homogeneous status of the node.

- In a node that is *heterogeneous*, the liquid and steam/gas phases are separated, except for the dispersed bubbles which disengage at a calculated rate (Sections 4.8.2, 4.8.3 and 4.8.4). In order to determine the mixture height, the volume of the two-phase mixture is obtained by summing the liquid phase volume ($M_e v_e$) and the bubble phase volume ($M_B v_g$). The mixture height is then obtained directly by geometric analysis for variable-area sectionalized nodes (core node), lumped vertically segmented nodes (pressurizer and annulus/lower plenum nodes) and lumped cylindrical nodes.

- In a node that is *homogeneous*, the liquid, steam and gas phases are dispersed as a uniform mixture throughout the extent of the node. Hence, the mixture height equals the geometric height of the node.

CENTS dynamically selects one of these two conditions in each node at each calculational time step, based on the following logic.

[ ]

[ ]

4.9 Flowpath Properties

4.9.1 Thermodynamic Path Properties

Calculation of the flowpath parameters includes calculation of:

- path quality (x)
- path mixture enthalpy (h)
- path liquid enthalpy (h_e)
- path steam enthalpy (h_s)
- path mixture specific volume (v)
- path temperature (T).

To support the calculation of these flowpath properties, the model treats each flowpath's junction(s) to its connecting node(s) as having cross-sections that are of one of four types:

[ ]

All non-momentum flowpaths are assumed by CENTS to have a [ ] connection. For momentum flowpaths, the connection types are pre-determined through input, P_GEOM(path). [

]

The path properties are calculated based on the local fluid conditions in the upstream node, at the location of the connection. These upstream conditions may be (a) single-phase, (b) two-phase mixture that completely covers the path connection or (c) liquid or two-phase mixture that covers the path only partially. (The two-phase mixture consists of the liquid and dispersed bubbles, not the free steam.) For these three scenarios, the path properties are calculated as follows:

Upstream Fluid is Single-Phase. [

]

Path Is Fully Covered by a Two-Phase Mixture (Figure 4.6). [

]

[

] The remaining properties follow directly:

$$h = (1-x)\, h_e + x h_s$$
$$v = (1-x)\, v_e + x v_s.$$

Path Is Partially Covered by Liquid or a Two-Phase Mixture (Figure 4.7). The path quality is calculated by means of an empirical slip (Von Glahn) correlation [4 12] that accounts for slip between the phases. [

] The remaining path properties (mixture enthalpy and mixture specific volume) are calculated as described above.

[

]

In order to prevent large changes in path properties which can produce instabilities, [ ]phase are implemented.

4.9.2 Free-Steam Fraction at Steam Inlet/Outlet

This model calculates the amount of steam that enters or leaves a node's free-steam space through the connected flowpaths. This information is needed in Equation (4.8.1) to calculate the bubble mass in each node.

4.10 Wall Heat

The RCS wall heat model represents the walls of the system using a lumped node representation for the walls in each thermal-hydraulic node. The wall temperature is calculated by solving an integrated heat conduction equation for each wall. The walls are dynamically coupled to the RCS and containment conditions by means of heat transfer coefficients on the RCS and containment sides, respectively. The heat capacity of the walls is included in the formulation. This model is used to simulate the dynamic response of the RCS walls (wall temperature and heat transfer to the RCS coolant and to the containment) during all plant conditions, including heatups and cooldowns.

4.10.1 Lumped Node Wall Heat

A lumped wall with heat capacity MC_P is assigned to each node, [ ]. The average wall temperature is T_{wall}. The wall is assumed to extend from the bottom to the top of each node. Level dependent wall heat transfer to the separated liquid and steam phases is calculated as follows:

$$Q_{wall,liq} = - h_{liq}A_{liq} (T_{wall} - T_{liq}) - h_{bot}A_{bot} (T_{wall} - T_{liq})$$

$$Q_{wall,stm} = - h_{stm}A_{stm} (T_{wall} - T_{stm})$$

where

$Q_{wall,liq}$ = wall heat transfer rate from liquid phase,
$Q_{wall,stm}$ = wall heat transfer rate from steam phase,
h_{bot} = heat transfer coefficient for bottom-liquid interface,
h_{liq} = heat transfer coefficient for wall-liquid interface,
h_{stm} = heat transfer coefficient for wall-steam interface,

A_{liq} = wall-liquid contact area,

A_{stm} = wall-steam contact area,

A_{bot} = node cross-sectional area (bottom),

T_{liq} = liquid temperature,

T_{stm} = steam temperature.

The wall to containment heat transfer is calculated as follows:

$$Q_{wall,cont} = h_{cont} A_{cont} (T_{wall} - T_{cont})$$

where

$Q_{wall,cont}$ = wall heat transfer rate to the containment,

h_{cont} = heat transfer coefficient for wall-containment surface

A_{cont} = wall-containment contact area,

T_{cont} = containment temperature.

The wall temperature is integrated by explicit integration of the conservation equation

$$MCp \frac{dT_{wall}}{dt} = Q_{wall,liq} + Q_{wall,stm} - Q_{wall,cont.} \quad (4.10.1)$$

The user input heat transfer coefficients are:

- $h_{liq}A$ for heat transfer to the liquid phase, N_HEAT_XFER_LIQ(node)
- $h_{stm}A$ for heat transfer to the steam phase, N_HEAT_XFER_STM(node)
- $h_{cont}A$ for heat transfer to the containment, N_HEAT_XFER_CONT(node)
- h_{bot} for heat transfer to liquid at the node bottom, N_HEAT_XFER_BOT(node).

The area A above is the total wall surface area. The input values for the overall heat transfer coefficients are calculated by the user by the expression

$$h = \frac{1}{\frac{1}{h_{surf}} + \frac{1}{k/(\Delta r/2)}}$$

where

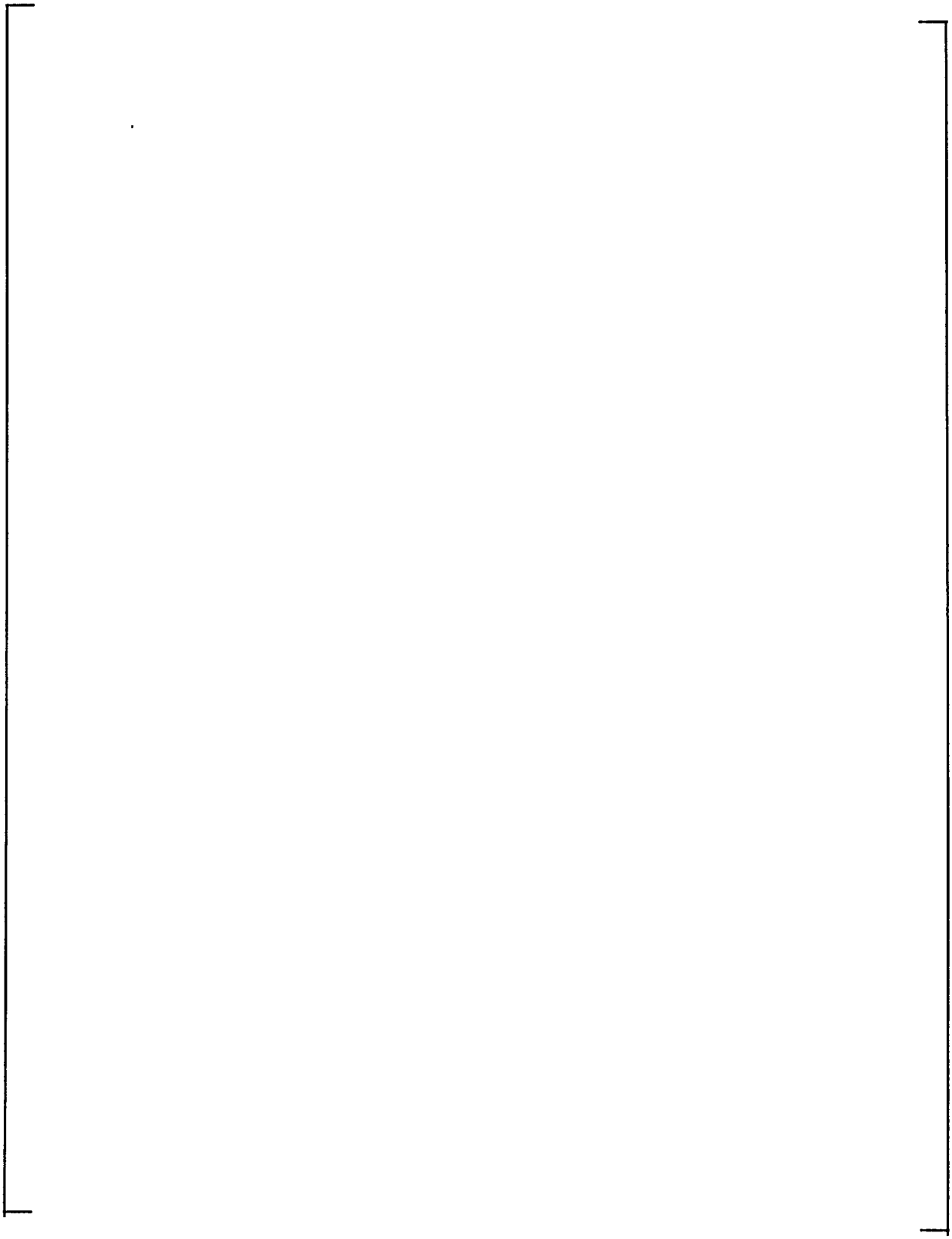
h_{surf} = surface heat transfer coefficient

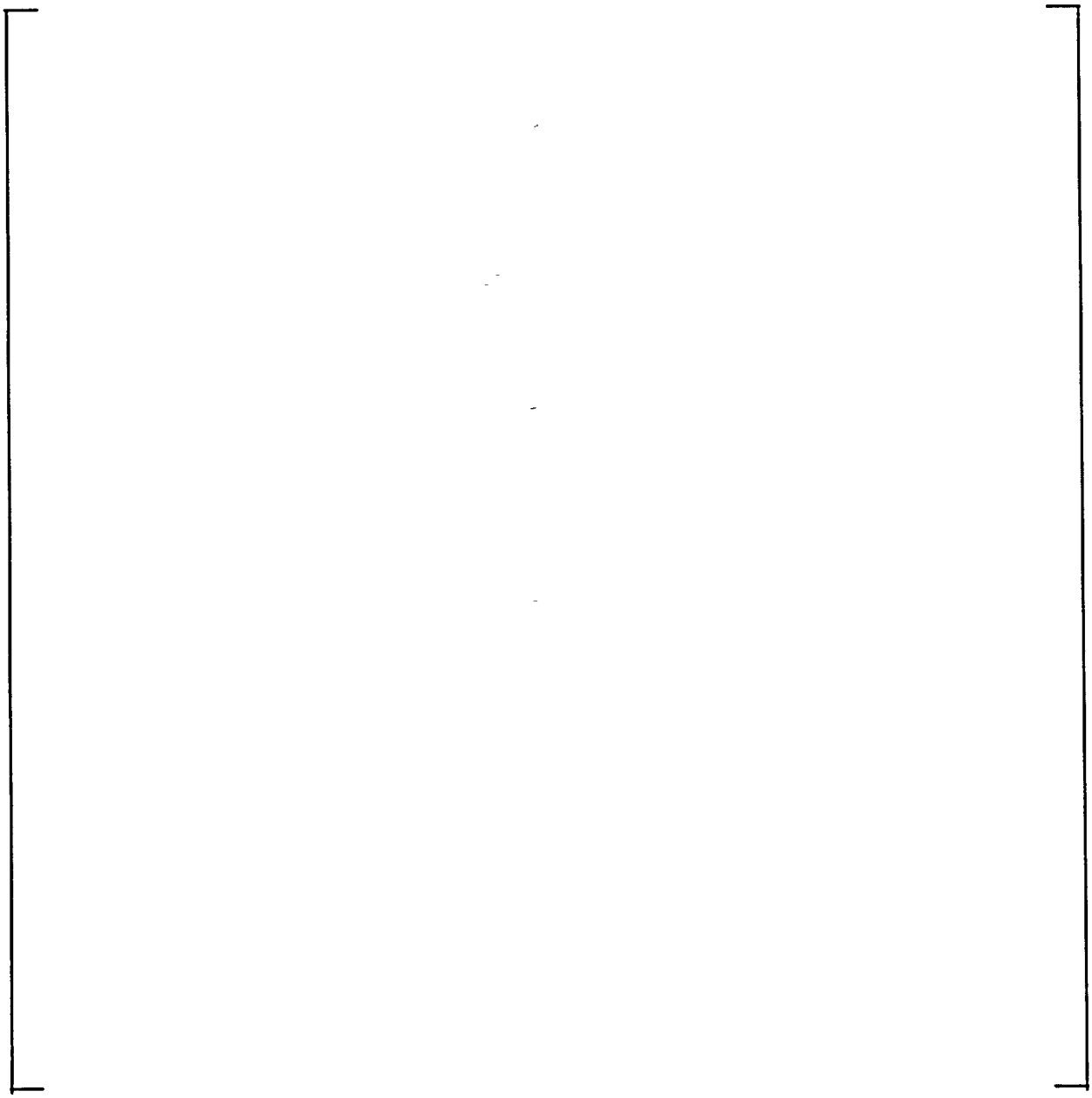
k = wall conductivity

Δr = wall thickness.

[

]





4.11 Reactor Coolant Pumps

This section includes a description of models for the reactor coolant pump (RCP) thermal-hydraulic calculations and the RCP shaft speed calculations.

4.11.1 RCP Thermal-Hydraulic Calculations

The reactor coolant pump hydraulic calculations include calculations for the hydraulic torque, pump head, and pump heat. The RCP hydraulic torque and pump head are calculated from user supplied two-phase pump homologous curves. (The default curves are from the Semiscale 1-1/2 loop system[4 13]).

RCP Hydraulic Torque The hydraulic torque is calculated from the following expression:

$$T_{hyd} = \begin{cases} \left(\frac{\beta}{\alpha^2}\right) \alpha^2 T_r \left(\rho/\rho_r\right) & \text{if } \nu/\alpha \le 1 \\ \left(\frac{\beta}{\nu^2}\right) \nu^2 T_r \left(\rho/\rho_r\right) & \text{if } \nu/\alpha \ge 1 \end{cases}$$

where

T_{hyd} = hydraulic torque
T_r = rated hydraulic torque
α = ω / ω_r
ω = Pump speed
ω_r = Rated pump speed
$\left(\frac{\beta}{\alpha^2}\right)$ = torque ratio (BA_ tables in database)
$\left(\frac{\beta}{\nu^2}\right)$ = torque ratio (BV_ tables in database)
υ = Q / Q_r
Q = volumetric mass flowrate

Q_r = rated volumetric mass flowrate

ρ = density

ρ_r = rated density

$$\beta = \frac{T_{hyd} / \rho}{T_r / \rho_r} .$$

Pump Head The pump head is calculated using difference homologous curves from the following expressions:

$$H = \begin{cases} \alpha^2 H_r \left\{ \left(\frac{h}{\alpha^2}\right) - m_h \left(\frac{h}{\alpha^2}\right)_{TP} \right\} & \text{if } | \nu/\alpha | \le 1.0 \\ \nu^2 H_r \left\{ \left(\frac{h}{\nu^2}\right) - m_h \left(\frac{h}{\nu^2}\right)_{TP} \right\} & \text{if } | \nu/\alpha | > 1.0 \end{cases}$$

where

ν and α are as defined above

H = pump head

H_r = rated pump head

m_h = degradation multiplier

h = H/H_r

$\left(\frac{h}{\alpha^2}\right)$, $\left(\frac{h}{\nu^2}\right)$ = pump head ratios (HA_ and HV_ Tables in data base)

$\left(\frac{h}{\alpha^2}\right)_{TP}$, $\left(\frac{h}{\nu^2}\right)_{TP}$ = two-phase pump head ratios, (HA_ANC and HV_ANC Tables in data base).

Pump Heat The pump heat is calculated from the equation

$$Q_{pump} = k \cdot \omega \cdot (T_{elec} - T_{fric}) / 777.98$$

where

Q_{pump} = pump heat rate, Btu/sec

ω = angular speed of pump, rad/sec

T_{elec} = electric torque, ft-lbf (Section 4.11.2)

T_{fric} = friction and windage torque, ft-lbf (Section 4.11.2)

k = fractional efficiency term (RCP_HEAT_MULT).

4.11.2 RCP Shaft Speed Calculations

The pump speed is calculated by solving the conservation of angular momentum equation

$$I \frac{d\omega}{dt} = (T_{elec} - T_{fric} - T_{hyd})g \qquad (4.11.1)$$

where

I = pump inertia

g = gravity constant.

T_{elec} = electric torque = $T_{elec,rate} * \left(\frac{Vol}{Vol_{rate}}\right)$ K(slip) (4.11.2)

T_{fric} = friction and windage torque = $k_{fw}\ \omega\ |\omega|$

k_{fw} = friction and windage torque constant (WIND_TORQ_MULT)

T_{hyd} = hydraulic torque (Section 4.11.1)

$T_{elec,rate}$ = rated electric torque

Vol = voltage

Vol_{rate} = rated voltage

K(slip) = electric torque multiplier (function of slip:
Table in database: TORQ_TAB vs. SLIP_TAB)

slip = $1 - \omega / \omega_{synch}$ (4.11.3)

ω_{synch} = 4 π freq / N (synchronous speed)

freq = frequency

N = number of poles.

For low pump speeds, the friction and windage torque is set equal to a constant larger frictional torque, to simulate engaging of the pumps anti-reverse device.

[

]

The above pump speed model provides a dynamic representation of the pump speed for all conditions during normal and abnormal operation, including startups, coastdowns, forward and reverse pump flow, RCP malfunctions (like locked rotor and sheared shaft), etc.

The pump speed equation is integrated by linearization and backwards differentiation of the pump speed equation.

That is, Equation (4.11.1) is written in the form

$$I \frac{\Delta\omega}{\Delta t} = (T_{elec} + \frac{dT_{elect}}{d\omega} \Delta\omega - T_{fric} - T_{hyd}) g.$$

Similarly for Equations (4.11.4) and (4.11.5).

The derivative $\frac{dT_{elect}}{d\omega}$ is obtained from Equations (4.11.2) and (4.11.3), and the electric torque multiplier (function of slip table) described above to obtain

$$\frac{dT_{elect}}{d\omega} = T_{elec,\,rate} * \left(\frac{Vol}{Vol_{rate}}\right) * \frac{dK(slip)}{dslip} * \frac{-1}{\omega_{synch}}.$$

The nomenclature for this equation is as described at the beginning of this section. The derivative dK(slip)/dslip is obtained from the K(slip) table.

4.12 Non-Condensibles Transport Model

4.12.1 Model Overview and Assumptions

CENTS tracks non-condensible gases as they enter and exit each node via the momentum flowpaths that connect the node to its neighbors, and via the external non-momentum flowpaths that connect the node to the interfacing systems (e.g., nitrogen supply system, safety injection system, vents to containment and to the quench tank).

In addition, an exchange of gases takes place at the liquid/gas interface as the gases dissolve or separate. Non-condensible hydrogen may also be formed in the core due to the zirconium-water reaction.

The model tracks the non-condensible gases in each node by solving the conservation of mass equation separately for each species of gas. The following are highlights of the model and its assumptions.

- Tracking of non-condensible gases is performed separately for each node and for each momentum and non-momentum flowpath of the system.
- Transport modeling is done individually for each species of gas.

  Different gases may be modeled simultaneously. The gases may be of any species, identified in the database by their molecular weights. [ ]
- The gases are modeled both in solution within the liquid phase, and as separated gases in the node's free steam/gas space.
- Separated gases are assumed to be homogeneously mixed with each other and with any steam, in the node's free steam/gas space.
- Node-to-node gas flow is determined on the basis [ ]
- Gas that enters the RCS from interfacing auxiliary systems is assumed to be at the containment temperature.

  Several mechanisms exist for separated gas to enter or leave a node. [ ]

[

]

4.12.2 Gas Transport

This and the following subsections describe the transport model for separated non-condensible gases. The exchange mechanism between separated and dissolved gas is also described here. [

]

The mass of each gas species in each node obeys the conservation equation

$$\frac{dM_{G,k}}{dt} = \sum_{\substack{\text{sinks \&} \\ \text{sources}}} W_{G,k} \qquad (4.12.1)$$

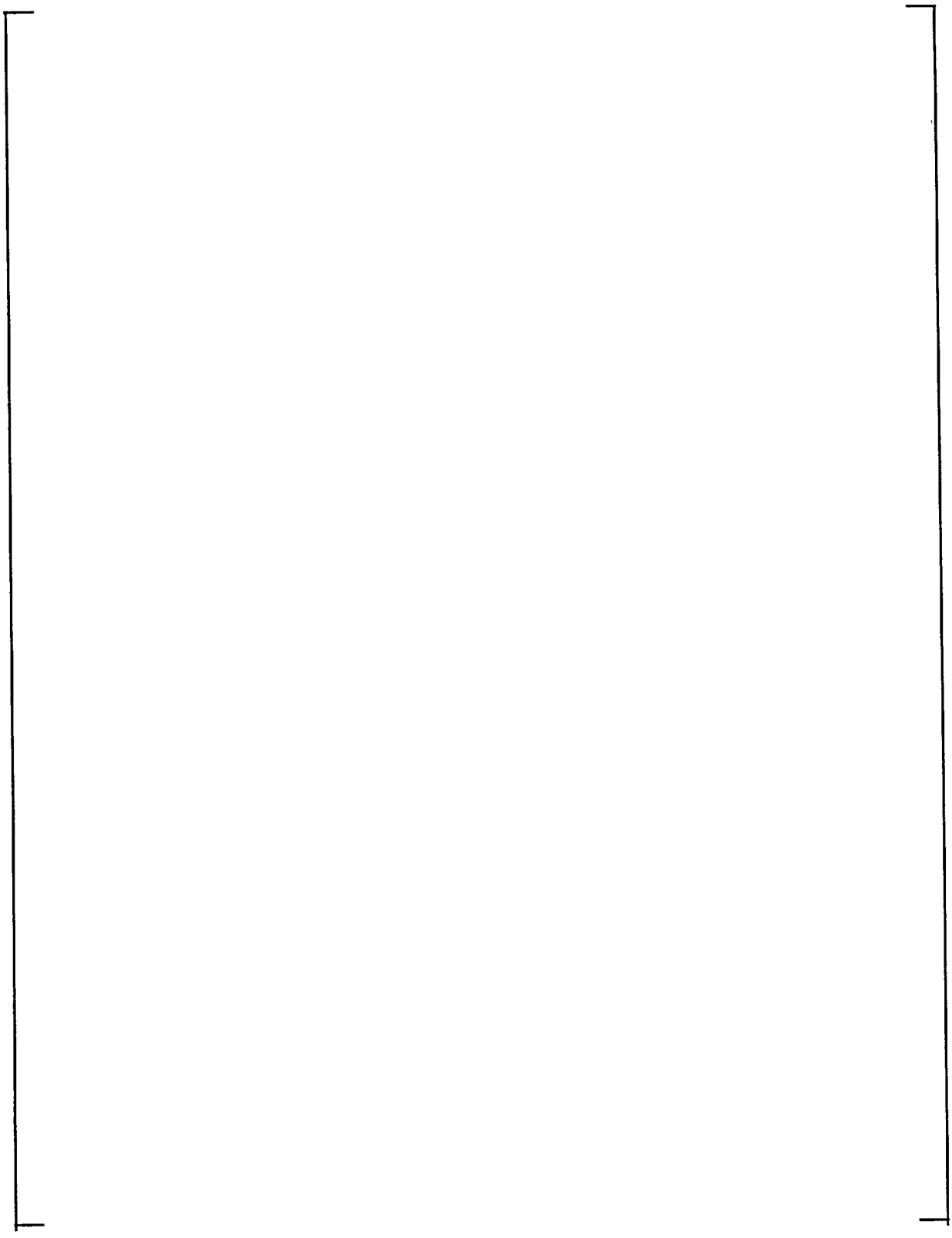
where,

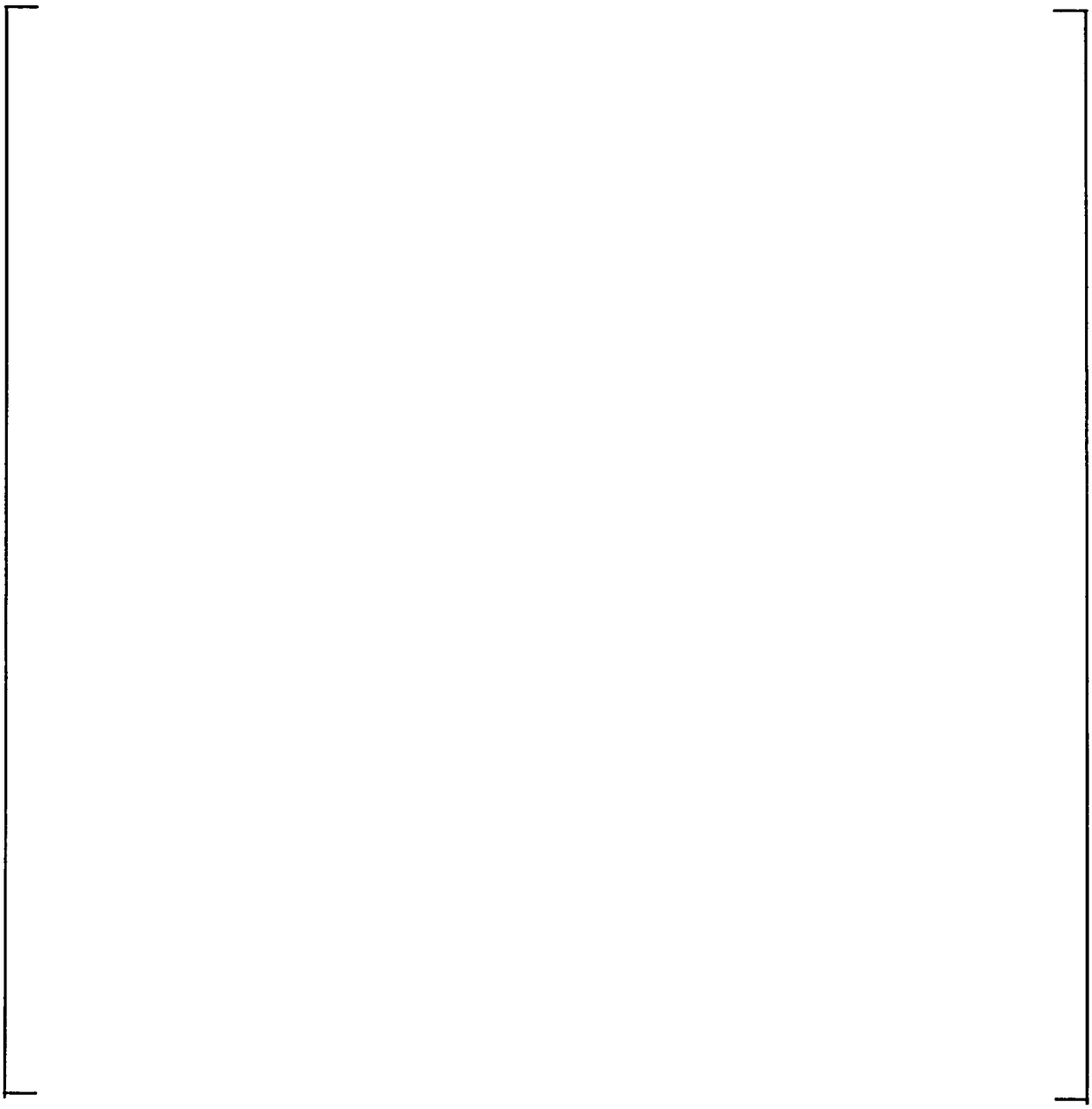
$M_{G,k}$ = Mass of gas species k (MASS_NONC_SEP).

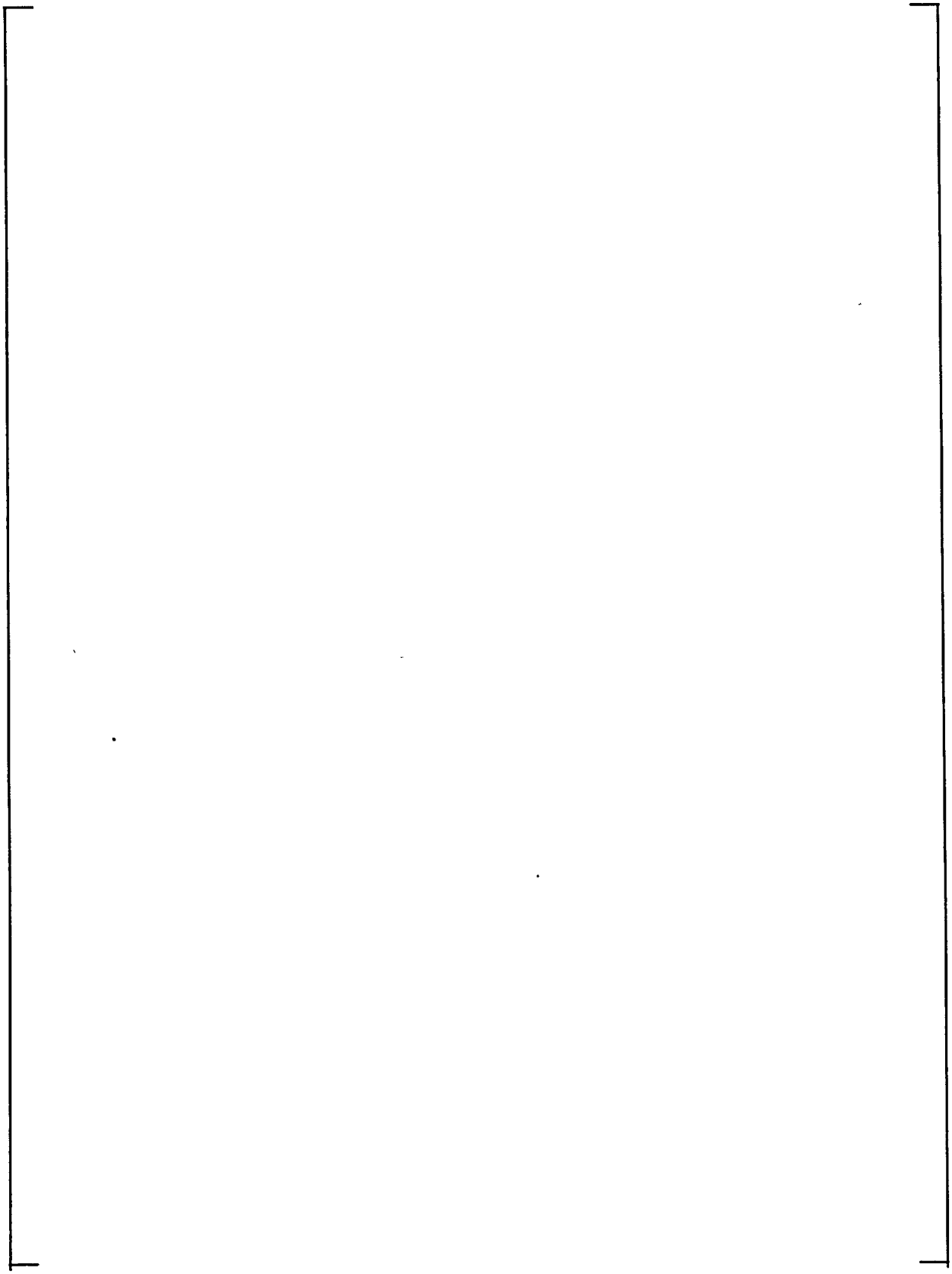
$W_{G,k}$ = Rate at which gas species k enters (or leaves) the free steam/gas space of the node due to any one of the available mechanisms.

Σ = Summation over all available sources and sinks.

The possible source/sink mechanisms for gas transport are described below.







[

]

4.12.3 *Partial Pressures of Gases and Steam*

When non-condensible gases are present, the total pressure is the sum of the partial pressures of the gases and steam. The gas temperature is assumed to equal the temperature of the steam. The partial pressure of each gas species is found using the ideal gas law:

$$P'_{G,k} = \frac{\Re}{144} \frac{\frac{M_{Gk}}{m_{G,k}}(T + 459.7)}{V_{free}}$$

where

$P'_{G,k}$ = partial pressure of gas species k (PPRES_NONC)

$M_{G,k}$ = Mass of gas species k (MASS_NONC_SEP)

T = Temperature of the steam space, or temperature of the liquid if no steam is present; °F (TEMP_NONC)

$m_{G,k}$ = Molecular weight of gas species k (NONC_MW)

$\Re$ = Universal gas constant = 1545.33 ft-lbf / lbmole-°R

V_{free} = Free steam/gas volume of the node (PVOL_NONC_TOT).

The total gas pressure in the node (PPRES_NONC_TOT) is then :

$$P_{TOT}^{\cdot} = \sum_{k} P_{G,k}^{\cdot}$$

[

]

4.13 Pressurizer Heater Temperature

The pressurizer heater temperature for each bank of heaters is obtained by solving the heat conduction equation

$$MC_p \frac{dT}{dt} = Q - hA\,(T - T_{cool})$$

where

T	= heater temperature
MC_p	= total heat capacity
hA	= overall heat transfer
T_{cool}	= coolant temperature
Q	= Vol^2 $Admit_{Heat}$/1054.87 (Heater heat generation)
Vol	= voltage
$Admit_{Heat}$	= $1/Res_{heat}$ (admittance)
Res_{heat}	= heaters resistance.

The above equation is solved analytically.

4.14 Flow Calculation of RCS Subsystems

4.14.1 Main Spray System

A typical main spray network is shown in Figure 4.8. The network consists of a spray delivery line to the pressurizer, two bleed lines that take flow from the cold legs, main control valves in the bleed lines, and bypass valves (for the minimum, continuous flow). The flow in the bleed lines, $W_{bleed,1}$ and $W_{bleed,2}$ (P_FLOW_BLEED), is calculated by solving steady state[

]

where

W	= mass flowrate
P_u, P_d	= upstream and downstream pressure
k	= adjustable constant (PRZR_SPRAY_MULT)
ρ	= density
A	= path area, determined by the main control valves and bypass valves
ΔP_{head}	= elevation head
g	= gravity.

The flowrate (P_FLOW_SPRAY) and enthalpy (P_ENTH_SPRAY) of the spray delivered to the pressurizer are:

[ ]

4.14.2 Pressurizer Relief System

A typical pressurizer relief system is represented with a valve network like the one shown in Figure 4.9. The mass flowrate is calculated using the choked flow correlations (Section 4.7), accounting for the flow-related pressure drops between the pressurizer and the valve choke plane and between the valve and the tank. The total leak area is the sum of the path areas connected in parallel, as determined by the relief valve positions.

4.14.3 Vessel Upper Head Vent Subsystem

A typical vessel upper head vent system is represented with the valve network shown in Figure 4.10. The leak flowrate is calculated with the choked flow correlation (Section 4.7) using the corresponding valve areas. The leak flow multiplier is RTRV_VENT_MULT.

4.14.4 Vessel O-Ring Seals Subsystem

In a transient in which the pressure rises above supercritical conditions, leakage will occur around the upper head O-rings. An example of such a transient is a total loss of

feedwater without reactor trip - one type of anticipated transient without scram (ATWS).

The O-ring seal leakage area for such a transient is:

[ ]

where

Area = O-ring seal leakage area
A_{min} = minimum head seal leakage area
A_{max} = maximum head seal leakage area
P_{min} = pressure at which head seal leakage starts
P_{max} = pressure for head seal leakage maximum area.

Variable names corresponding to the above quantities appear in Tables 7.21 and 7.22.

4.15 Calculation of Solute Concentrations

The RCS model tracks solutes as they enter and exit each node via the internal (momentum) flowpaths that connect the node to its neighbors, and via the external (non-momentum) flowpaths that connect the node to the interfacing systems (e.g., charging, safety injection). In addition, gases dissolve or separate at the liquid/gas interface.

The model tracks the solutes in each node by solving the conservation of mass equation separately for each species. The following are highlights of the model.

- Tracking of solutes is performed separately for each node and each internal and external flowpath.
- Transport modeling is done individually for each species.
- Different species of solutes may be modeled simultaneously. These include, but are not limited to: boron, fission products (e.g., iodine, cesium, xenon, krypton), nitrogen-16, and dissolved non-condensible gases. With current dimensioning, the model supports up to twenty species of solutes.
- Each solute may be soluble in liquid only (e.g., boron), may be carried by steam only (e.g., xenon), or may appear in both phases (e.g., iodine). The relative solubilities in steam/liquid are defined by the database parameters for each species.
- The solutes are assumed to be homogeneously mixed within their carrier phases.
- Node-to-node solute transport is determined on the basis of the mixture momentum equation of each flowpath.
- The following mechanisms exist for solutes to enter or leave a node:

[

]

The mass of each solute species k in each node obeys the following conservation equation:

$$\frac{d}{dt}(M_T\ C_k) = \sum_{\substack{\text{sinks \&} \\ \text{sources j}}} (W_j\ C_{j,k}) + W_{gen,k} + W_{gas} - \frac{M_T\ C_k}{(\ T_{HL,k}\ /\ln 2\ )} \qquad (4.15.1)$$

where,

C_k = Concentration of solute *k* in the node (RCS_CONC_SOLUTE), based on the total mass of liquid and steam; i.e., C_k = (mass of solute *k*) / M_T

$C_{j,k}$ = Concentration of solute k in flowpath j (P_CONC_SOLU)

M_T = Total mass of liquid and steam in the node (MASS_TOT)

W_j = Mass flowrate in flowpath j (P_FLOW)

$W_{gen,k}$ = Mass rate of generation of solute k within the node:

[ ]

W_{gas} = Rate of dissolution (positive) or separation (negative) of non-condensible gases (Section 4.12.2).

$T_{HL,k}$ = Decay half-life of solute k: HALF_LIFE_SOLUTES(k) for every species.

ln 2 = Natural logarithm of 2.0.

When combined with equation (4.2.2) for the conservation of mixture mass, Equation (4.15.1) becomes:

$$\frac{dC_k}{dt} = \frac{1}{M_T}\left(\sum_{\substack{\text{sinks \&} \\ \text{sources j}}} W_j(C_{j,k} - C_k) + W_{gen,k} + W_{gas}\right) - \frac{C_k}{T_{HL,k}/\ln 2} \qquad (4.15.2)$$

[

]

Equation (4.15.4) gives the concentrations in all internal (momentum) flowpaths, and in external (non-momentum) flowpaths that exit their connecting nodes. For an external path entering a node, the interfacing system model (or the user) provides the concentrations.

Section 5.8.2 provides additional specific detail about solutes transport in the RCS, as part of a larger description of the CENTS dose calculation model. Section 7.2.7 describes the input and output variables for the solutes transport and dose models.

4.16 Quench Tank (Pressure Relief Tank)

The mass and energy in the quench tank are calculated by explicit solution of the equations

$$\frac{dM}{dt} = \Sigma W$$

$$\frac{dM_{NC}}{dt} = \Sigma W_{NC}$$

$$\frac{dU}{dt} = \Sigma W H$$

where

M = fluid mass
W = fluid mass flowrates
W_{NC} = non-condensibles mass flowrates
U = fluid energy

H = fluid enthalpy (sinks and sources).

The sources and sinks for the quench tank are shown in Figure 7.4

The quench tank pressure is calculated from[

] The quench tank level is calculated from geometric formulas for horizontal circular-cylindrical volumes.

The quench tank solute concentrations (boron, iodine, etc.) are calculated by solving concentration equations of the type

$$\frac{d}{dt}(XM) = \Sigma W_i X_i$$

where

X = solute concentration

W_i = flowrate

X_i = flow solute concentration

M = total water mass in quench tank.

4.17 Heat Transfer Between CEA Node and Core Node

The heat transfer between the CEA node and the core node is calculated by the expression

$$Q = f\, hA\, (T_{CEA} - T_{core})$$

where

hA = overall heat transfer coefficient

T_{CEA} = CEA node liquid temperature

T_{core} = Core node liquid temperature

f = fraction of contact area covered with liquid.

4.18 Geometric Losses for CEA Node Paths

The geometric loss k-factors for the paths entering and leaving the CEA node are a function of the average control rod position. They are obtained by[

]

4.19 Outputs to the Instrumentation Models

4.19.1 Instrumentation Pressurizer Level

The pressurizer level indicated by level instrumentation is calculated by unfolding the pressure difference between the pressurizer and the reference leg using calibrated conditions for the pressurizer liquid, pressurizer steam, and reference leg densities (calibrated conditions for hot and cold conditions are used). That is,

$$Z_{liq,inst} = Z_{bot} + \frac{Z_{ref}\left(\rho_{ref,cal} - \rho_{stm,cal}\right) - 144\,DP}{\rho_{liq,cal} - \rho_{stm,cal}} \qquad (4.19.1)$$

where

$Z_{liq,inst}$ = pressurizer level indicated by level instrumentation for given calibrated conditions

DP = head pressure difference between the pressurizer and the reference leg.

Z_{bot} = reference leg tap elevation (bottom)

Z_{ref} = reference leg height

$\rho_{liq,cal}$ = liquid density at calibrated conditions

$\rho_{stm,cal}$ = steam density at calibrated conditions

$\rho_{ref,cal}$ = reference leg density at calibrated conditions.

For an actual collapsed liquid height, Z_{liq}, the pressure difference, DP, is calculated as follows:

$$DP = \{Z_{ref}(\rho_{ref} - \rho_{stm}) - (Z_{liq} - Z_{bot})(\rho_{liq} - \rho_{stm})\} / 144 \qquad (4.19.2)$$

where the density variables are as described above, with the difference that they are the actual transient variables.

The liquid density for the reference leg is obtained from the pressurizer pressure and reference leg temperature. The reference leg temperature T_{ref} is obtained by means of a

[

]

The indicated level will deviate from the actual pressurizer collapsed level during a transient, as conditions differ from the calibration conditions. See Section 6.2.2 for additional information.

4.20 The Matrix Inversion Routine Constants

For the integration of the RCS thermal-hydraulic equations, the code internally renumbers the flowpaths in a manner such that the matrix resulting from the discretization of the conservation equations (Section 4.2) is a block type matrix like the one shown in Figure 4.11. The only nonzero elements for this matrix occur in the marked blocks in the matrix i.e., the entry at the i^{th}, j^{th}, position is nonzero only if the renumbered ith path touches the head or tail of the renumbered jth path.

The renumbering is done by first selecting the "chains" of paths (i.e. the paths associated with the loops). Each of the square blocks in Figure 4.11 is a tridiagonal block associated with a chain of paths. The renumbering is completed by adding to the list the remaining paths ("non-chain" paths, i.e., paths in the vessel). The method of the renumbering is documented in Reference 4.7.

This path renumbering is only done internally in the code, invisible to the user or to the programmer outside the integration routine. This renumbering allows the code to use the block inversion technique documented in Reference 4.7 for the solution of the system of conservation equations associated with the RCS thermal-hydraulic model.

4.21 RCS Line Break Models

CENTS simulates the effect of a full or partial break of the following RCS lines:

(a) the letdown line exiting the Regenerative Heat Exchanger (RHX);

(b) a letdown line feeding the RHX from the RCS piping;

(c) a tributary line attached to the RCS piping; or

(d) the RCS main loop piping.

CENTS determines the critical break flow based on the pressure and enthalpy at the break location. For models (a)-(c), CENTS explicitly calculates frictional losses in the affected line between the RCS piping and the break.

4.21.1 Letdown Line Break

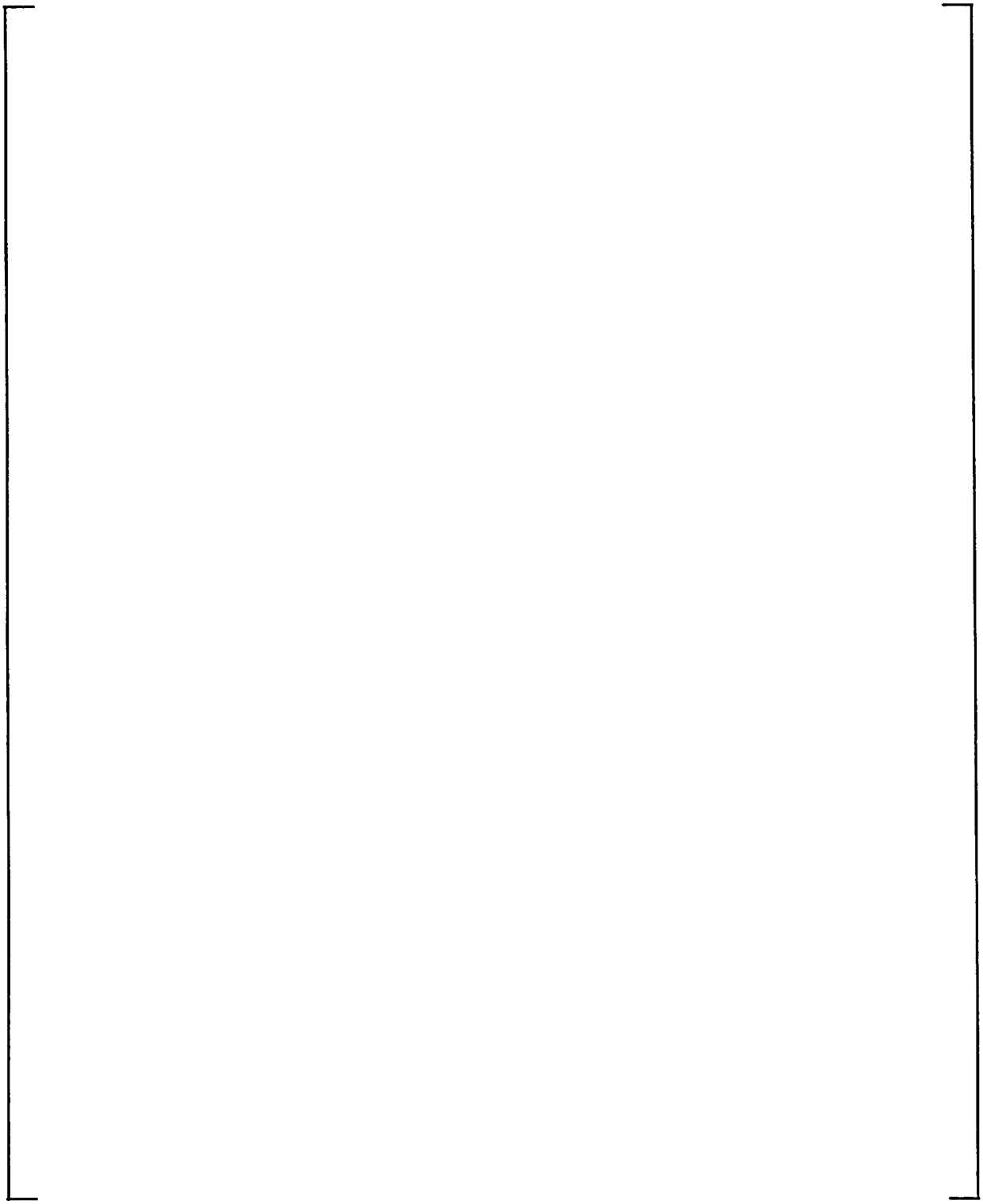
Figure 4.12 presents a typical representation of the letdown and charging lines and the possible break locations. Letdown flow is provided by a dynamic Control System demanded flowrate. In the absence of a break, CENTS assigns this demanded flow to the letdown lines, which are represented by non-momentum flowpaths exiting the RCS. Charging flow is provided by up to three charging pumps, whose operation is controlled by the Control System. The charging and letdown streams exchange energy in the RHX. The charging flow returns to the RCS at a higher thermal state, via non-momentum flowpaths. See Section 6.2.

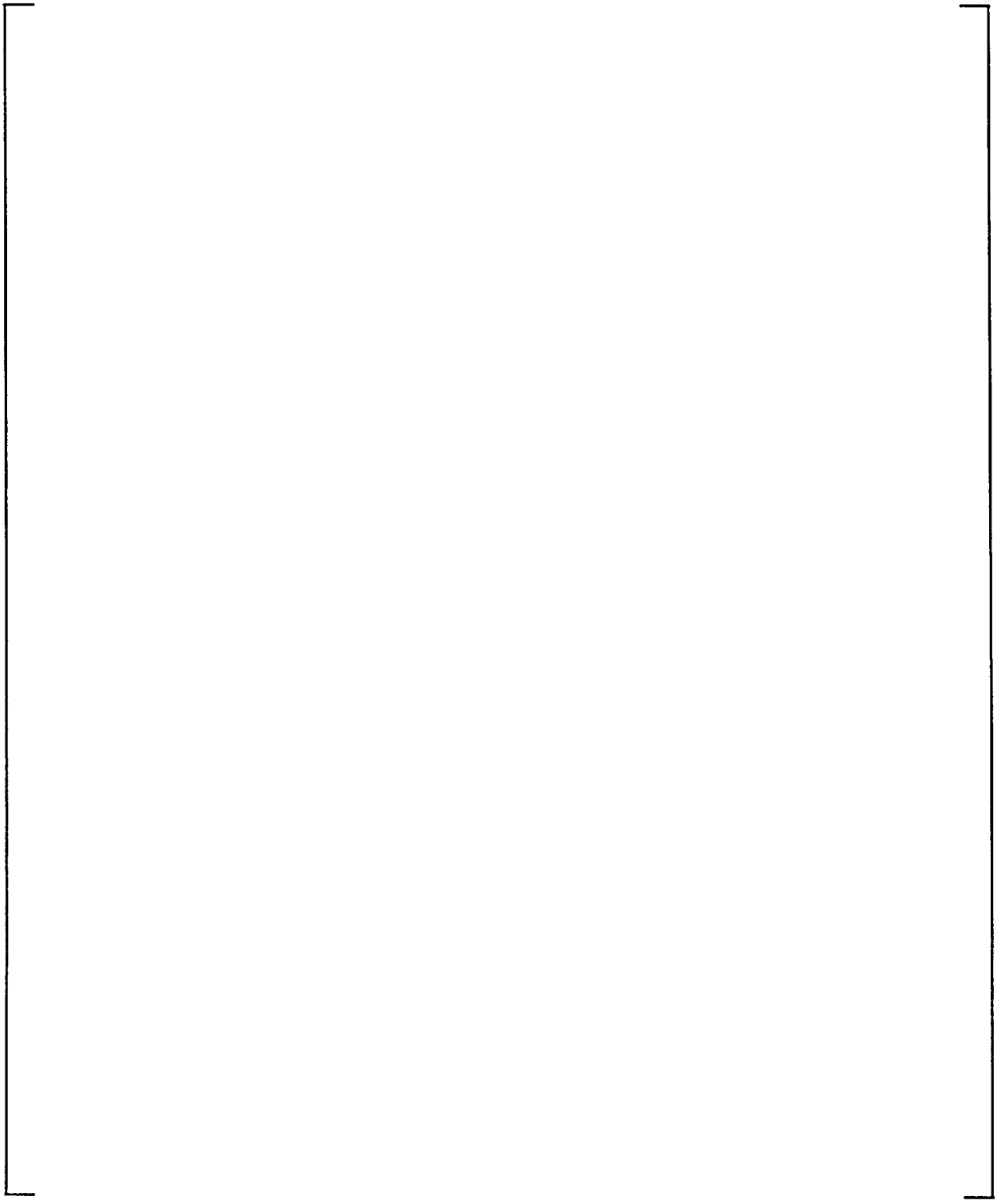
Break Flag and Parameters. A break in the letdown line is user-defined via the malfunction flag MAL_LDN_BREAK, defined as ± the break area (ft2).

- When MAL_LDN_BREAK > 0, the break is located in the letdown line downstream of the RHX.
- When MAL_LDN_BREAK < 0, the break is located in the first letdown line upstream of the RHX. (The number of active letdown lines from the RCS to the RHX is RCS_NUMOUT_LDNS ≤ 4.)
- When MAL_LDN_BREAK = 0, the break model is not active.

Section 7.5.1 identifies the input parameters that describe the letdown line losses.

Pressure Drops. CENTS calculates pressure drops in the letdown line. If there is no break, then the pressure drops are calculated from the RCS piping to the RHX inlet. If there is an upstream break, then the pressure drops are calculated from the RCS piping to the break. If there is a downstream break, then the pressure drops are calculated separately from the RCS piping to the RHX, and from the RHX to the break. The pressure losses between any two points consist of the following component losses:





The break enthalpy depends on the location of the break:

- The letdown line inlet enthalpy is the local enthalpy of the upstream node at the elevation of the letdown line.
- If the break is upstream of the RHX, then h_{break} equals the letdown line inlet enthalpy.
- If the break is downstream of the RHX, then h_{break} equals the enthalpy at the RHX exit.

[

]

4.21.2 Small Break LOCA

Break Flag and Parameters. The Small Break malfunction is initiated via the malfunction flag, MAL_SB_LOCA(1:4), which is defined as the break area (ft^2). These four Small Break flags correspond to four non-momentum flowpaths.

The model supports specification of line pressure losses between the RCS piping and the break, such as when the break is simulated to occur in a tributary line. Section 7.5.1 describes the input necessary to define a break line and its loss parameters. If these parameters are not input, then the break is modeled at the RCS piping, using the local RCS fluid properties for the critical flow calculation.

4.22 References

4.1 Wallis, Graham B., One-Dimensional Two-Phase Flow, McGraw Hill, Inc., 1969.

4.2 Thom, J. R. S., "Prediction of Pressure Drop During Forced Circulation Boiling of Water," International Journal of Heat and Mass Transfer, Volume 7, pp. 709-724, 1964.

4.3 Martinelli, R. C. and Nelson, D. B., "Prediction of Pressure Drop During Forced Circulation Boiling of Water," Transactions of the ASME, August, 1948.

4.4 Bird, R. M., Stewart, W. E., and Lightfoot, E. N., Transport Phenomena, John Wiley and Sons, Inc., 1960.

4.5 Kocomustafaogullari, G., "Thermo-Fluid Dynamics of Separated Two-Phase Flow," Ph.D. Thesis, School of Mechanical Engineering, Georgia Institute of Technology, Atlanta, Georgia (December, 1971).

4.6 Ishii, M., Thermal-Fluid Dynamic Theory of Two-Phase Flow, Collection de la Direction des Etudes et Recherches D'Electricite de France, Eyrolles, Paris (1975).

4.7 Porsching, T. A., et. al., "FLASH-4: A Fully Implicit Fortran IV Program for the Digital Simulation of Transients in a Reactor Plant," WAPD-TM-840, March, 1969.

4.8 RELAP4-MOD7, Idaho National Engineering Laboratory, December, 1980.

4.9 Moody, F. J., "Maximum Discharge Rate of Liquid-Vapor Mixtures from Vessels," ASME Winter Annual Meeting, November, 1975.

4.10 R. E. Henry and H. K. Fauske, "The Two-Phase Critical Flow of One-Component Mixture in Nozzles, Orifices and Short Tubes," Journal of Heat Transfer, May 1971.

4.11 Murdock, J. W., and Bauman, J. M., "The Critical Flow Function for Superheated Steam," Trans. ASME, Series D, Volume 86, 1964.

4.12 Von Glahn, V. H., "An Empirical Relation for Predicting Void Fraction with Two-Phase Steam-Water Flow," NASA Technical Note D-1189, January, 1962.

4.13 Ramsthaler, J. H., "Best Estimate Pump Head and Torque Models from Semiscale Pump Data," DMR-7-74, April 30, 1974.

4.14 "Calculative Methods for the C-E Small Break LOCA Evaluation Model," CENPD-137P, August, 1974 (Proprietary).

4.14A Supplement 1-P, January, 1977 (Proprietary).

4.15 Anlkam, T. M. and White, M. D., "Experimental Investigations of Two-Phase Mixture Level Swell and Axial Void Fraction Distribution Under High Pressure, Low Heat Flux Conditions in Rod Bundle Geometry," pp 4-67 through 4-77, ANS Specialist Meeting on Small Break Loss of Coolant Accident Analyses in LWRs, Monterey, Cal. EPRI WS-81-201, August 25-27, 1981.

4.16 "Small Break LOCA Realistic Evaluation Model. Volume 1, Calculational Models," CEN-420-P, October, 1993.

Figure 4.1

PRIMARY SYSTEM GEOMETRIC MODEL

(TYPICAL 2-LOOP CE PLANT)

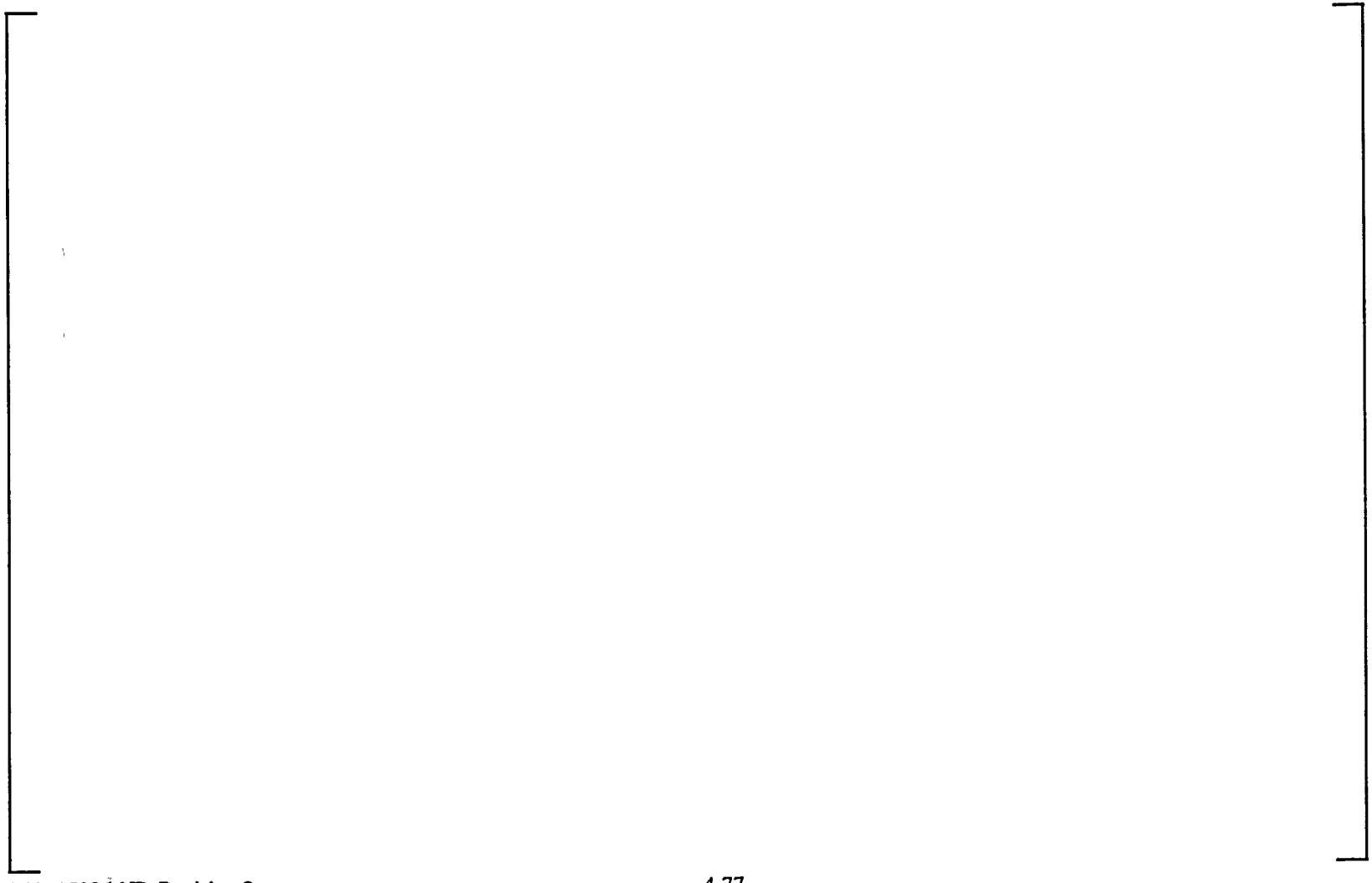


Figure 4.2

PRIMARY SYSTEM GEOMETRIC MODEL

(TYPICAL 3-LOOP W PLANT)

Figure 4.3

PHASE SEPARATION IN LUMPED NODES

Figure 4.4

PHASE SEPARATION IN SECTIONALIZED NODES

Figure 4.5

PHASE SEPARATION IN SEGMENTED NODES

Figure 4.6

STEAM FRACTION – PATH FULLY COVERED

Figure 4.7

STEAM FRACTION – PATH PARTIALLY COVERED

Figure 4.8

MAIN PRESSURIZER SPRAY: TYPICAL FLOW NETWORK

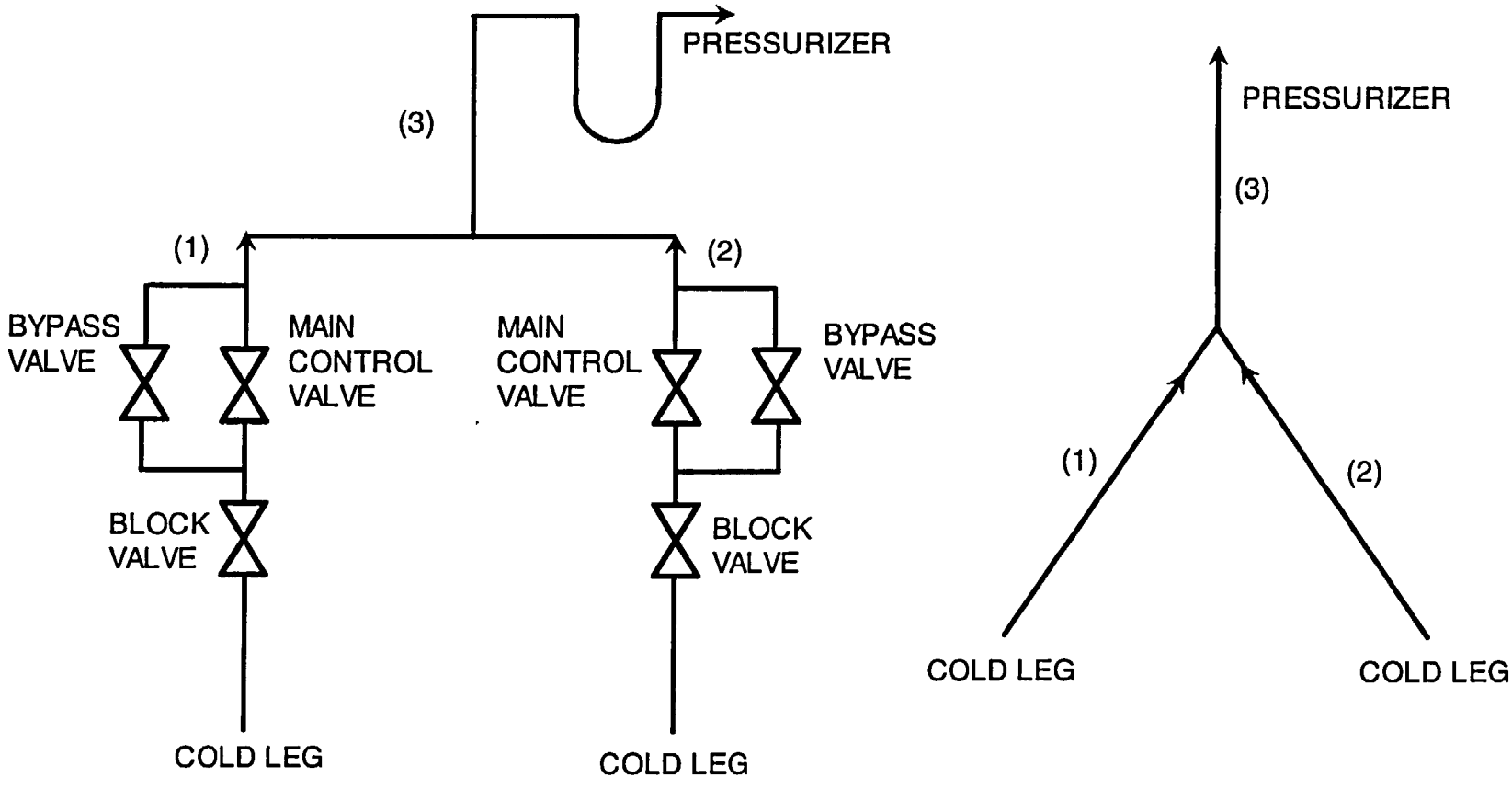


Figure 4.9

PRESSURIZER RELIEF SYSTEM: TYPICAL NETWORK

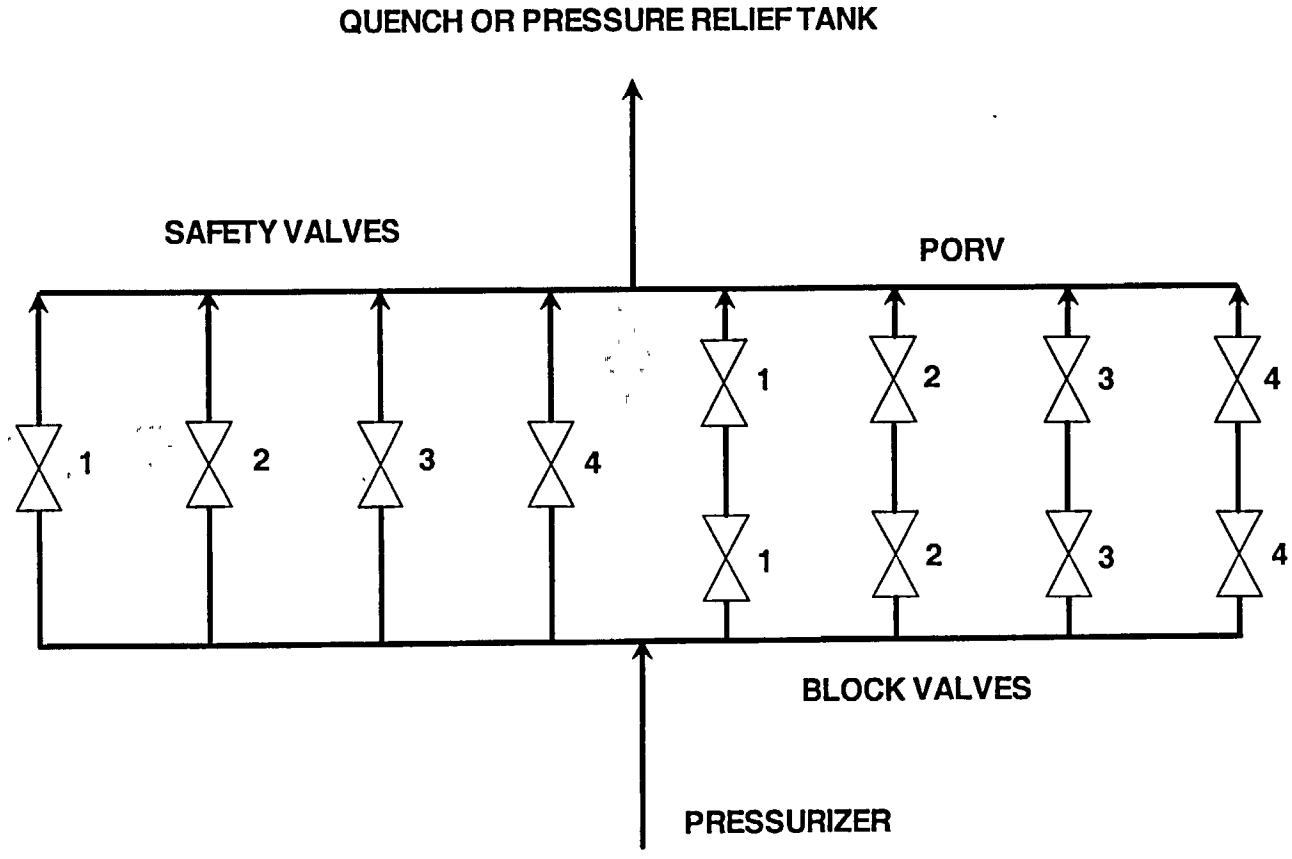


Figure 4.10

REACTOR VESSEL UPPER HEAD VENT SUBSYSTEM: TYPICAL LAYOUT

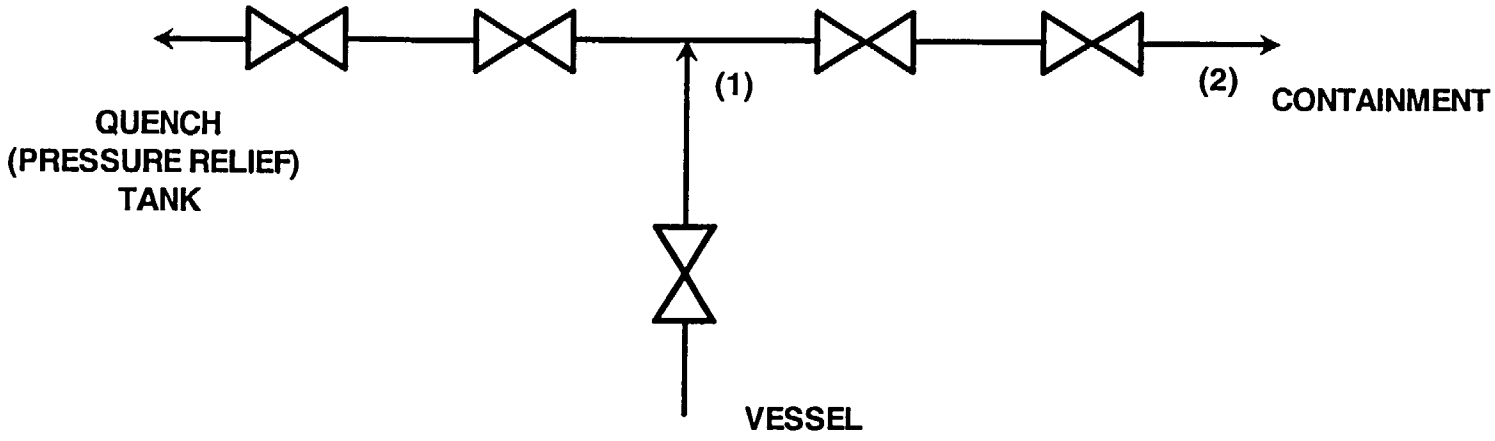
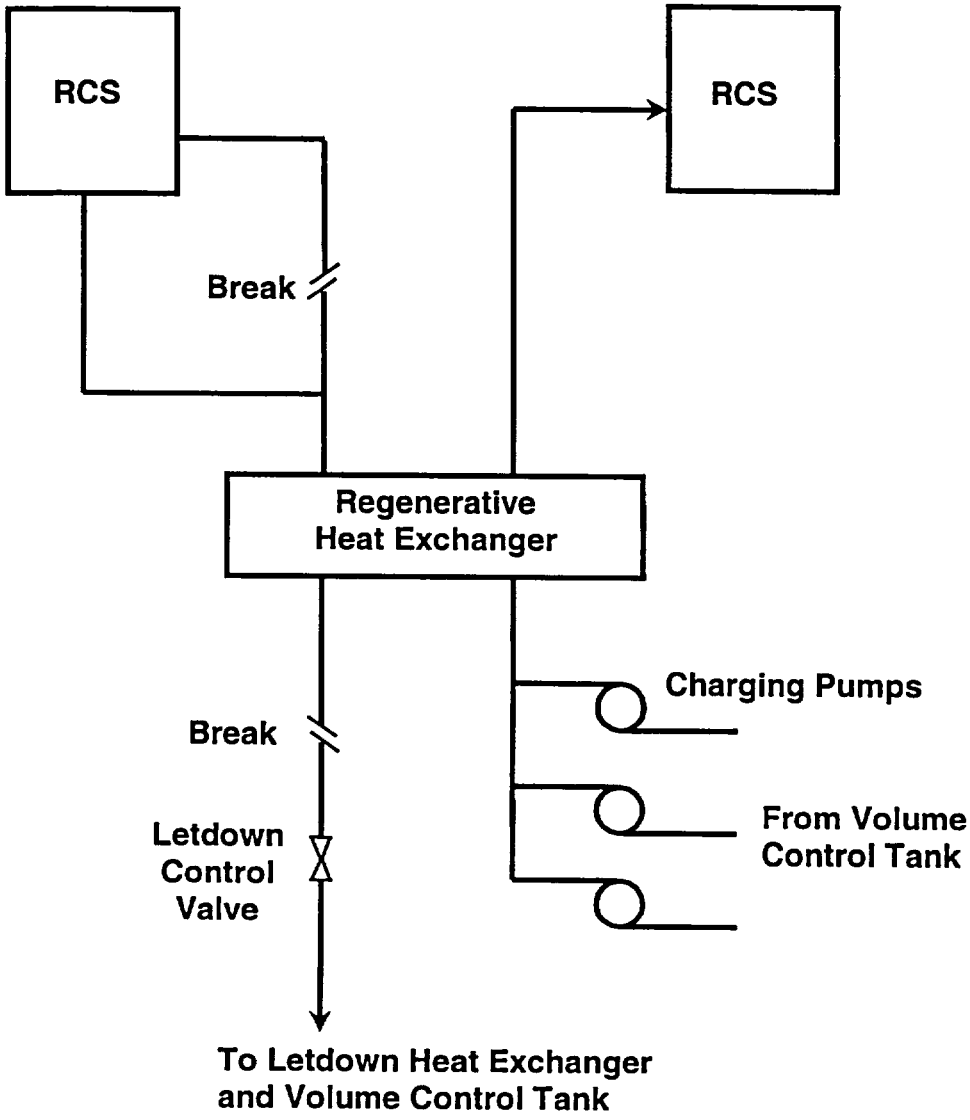


Figure 4.11
BLOCK TYPE MATRIX

Figure 4.12

Charging and Letdown Lines: Typical Network



5.0 SECONDARY SYSTEM MODELS

5.1 Geometric Representation

The typical secondary system for a PWR is represented by the node-flowpath model shown in Figure 2.2. Three nodes represent the secondary side of each steam generator -- a downcomer (saturated or subcooled liquid and saturated steam), an evaporator/riser/economizer region (saturated or subcooled liquid and saturated steam) and a steam dome (saturated or superheated steam). Additional nodes (Section 5.6) represent the common steamline header. This system representation allows accurate modeling of the recirculation phenomena and the downcomer and evaporator water levels. The model represents all major components, including the secondary safety valves, atmospheric dump and turbine bypass valves, main steam isolation valves, steamline and feedline check valves, and a steam generator blowdown system. Control systems are described in Chapter 6.

5.2 Steam Generator Secondary Model

The steam generator secondary model considers three control volumes (nodes) for each steam generator -- downcomer, evaporator/riser and steam dome. The model maintains conservation of mass and energy in each region, and satisfies a momentum balance in a quasi-steady fashion. The model allows for non-equilibrium states, condensation of steam by feedwater when the downcomer water level is below the feedring, heat losses and condensation on the steam generator shell, recirculation flow and steamline drains.

5.2.1 *Node and Path Conservation Equations*

The steam generator secondary model conserves mass and energy in each node by

$$\frac{dM}{dt} = \Sigma W_{in} - \Sigma W_{out} \tag{5.1}$$

$$\frac{dU}{dt} = \Sigma (WH)_{in} - \Sigma (WH)_{out} + \Sigma Q_{in} - \Sigma Q_{out} \tag{5.2}$$

where

M = mass
U = internal energy
W = flowrates
Q = heat transfer rates
H = specific enthalpies
t = time.

Once Equations (5.1) and (5.2) are integrated over a computational time step, the new pressure is found by an iterative solution of the equations,

$$M_f v_f (P) + M_g v_g (P) = V \tag{5.3}$$

$$M_f + M_g = M \tag{5.4}$$

$$M_f u_f (P) + M_g u_g (P) = U \tag{5.5}$$

where

M_f, M_g = liquid and steam masses

v_f, v_g = liquid and steam specific volumes, functions of pressure

u_f, u_g = liquid and steam specific internal energies, functions of pressure

P = pressure

V = total volume

M = total mass

U = total internal energy.

5.2.2 *Internal Flow Calculations*

For the following discussion, refer to the node numbering system shown in Figure 7.6.

Flowrates are computed using various formulations, depending on the type of flow. The inter-region flows are

W_{21} = Steam flow from evaporator/riser to the steam dome.

W_{32} = Circulation flow from downcomer to evaporator/riser at bottom of downcomer.

W_{31} = Steam flow from downcomer to steam dome when the downcomer becomes saturated.

W_{23} = Recirculation flow from the evaporator/riser to the downcomer over the can deck.

W_{13} = Condensate flow from the steam dome to the downcomer along the walls.

Steam flow from the evaporator/riser to the steam dome is determined on the basis of the steam bubble disengagement rate given by the Wilson correlation[5.1]. The flowrate is given by

$$W_{21} = U_{21} A_{int} \rho_{g2} F(\alpha) \quad (5.6)$$

where

U_{21} = steam separation velocity obtained by the Wilson correlation

A_{int} = interface area between the steam and mixture region

ρ_{g2} = steam density from node 2

$F(\alpha)$ = low power adjustment factor for bubble stratification

α = void fraction.

Circulation liquid flow from the downcomer to the evaporator/riser is calculated[

]

[

]

The steam flowrate from the downcomer to the steam dome is calculated[

]

[

]

The recirculation flow from the evaporator/riser to the downcomer is calculated [

]

[

]

The recirculation ratio above is found at time t as

[

]

The condensate flowrate from the steam dome to the downcomer is calculated by adding two condensate source terms: condensation on the walls due to heat loss and condensation due to pressure change.[

]

[

]

5.2.3 *Dry Steam Generator*

When the steam generator secondary side becomes dry [

] The conservation equations solved for this case are

Mass Balance:

[

]

Energy Balance:

[ ]

where

m = steam generator secondary side water mass
W = mass flowrate
U = steam generator secondary side internal energy
h = enthalpy
Q_{in} = heat input
Q_{out} = heat loss

and the subscripts are

G = steam
BD = blowdown
FW = feedwater
R = ruptured tube

5.2.4 *Inter-Region Heat Transfer*

Heat transfer between the regions is calculated by the following equations:

$$Q_{12} = U_{12} A_{12} (T_1 - T_2) \quad (5.17)$$

$$Q_{23} = U_{23} A_{23} (T_2 - T_3) \quad (5.18)$$

where

Q = heat transfer rate

U = heat transfer coefficient

A = contact area between the regions

T = temperature

and the subscripts are

1,2,3 = Steam generator secondary side regions (Figure 7.6).

5.2.5 Steam Generator Heat Loss

In calculating heat loss from the steam generators, the steam generator walls are divided into those in contact with liquid and those in contact with steam. The heat loss is calculated by solving the following transient heat transfer equation:

[ ]

where

MC_p = steam generator shell heat capacitance

T = temperature

hA = area times overall heat transfer coefficient

and the subscripts are

w = steam generator shell wall

c = containment

liq = water

stm = steam.

The heat transfer coefficients are calculated by

[ ]

where

A = wall surface area

h = heat transfer coefficient

R = wall heat transfer resistance

The [ ] heat transfer coefficients are given as

[ ] (5.23)

where

ΔT = fluid/wall temperature difference

L = characteristic length

C = constants (Btu/hr-ft^2-°F)

[ ]

The outside wall heat transfer coefficient is a constant.

5.2.6 *Critical Flow*

Critical flow of steam, where required, is calculated using the modified CRITCO equation[5 2]. The CRITCO equation is given as

$$(W/A)_{crit} = C_o (P - P_b)/(H - 185) \qquad (5.24)$$

where

W/A = mass flux, lbm/sec-ft^2
C_o = 0.53/0.000268
P = upstream pressure, psia
P_b = back pressure, psia
H = specific enthalpy, Btu/lbm.

Critical flow of liquid or two-phase is calculated using one of two correlations, selected by the user via the flag SGS_CRIT_MODEL. They are the homogeneous equilibrium model (HEM) correlation[5 3] and the HenryFauske (H-F) correlation[5 4]. The HEM and H-F tables are given in Appendix B.

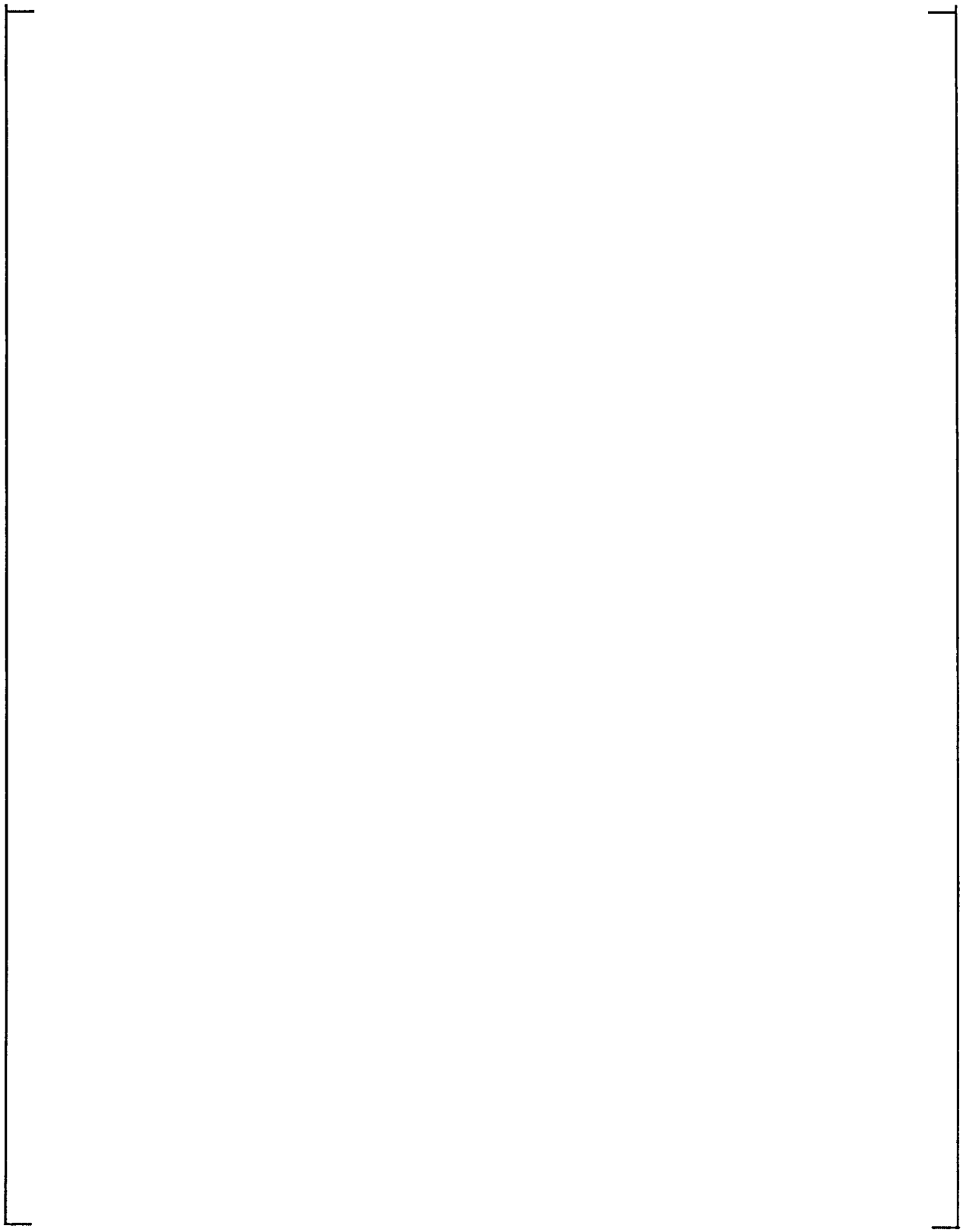
5.3 Steam Generator Tube Heat Transfer

The steam generator heat transfer model is derived from a time dependent conservation of energy equation accounting for the thermal capacitance and conductivity of the steam generator tubes. Heat transfer coefficients are found for the fluid conditions on the primary and secondary side including the effect of voids. Level dependence of heat transfer area is modeled. [ ] is calculated explicitly. Provision is made for forward as well as reverse heat transfer.

The steam generator tubes are sectionalized along their length. On these tube sections, CENTS calculates dynamic distributions of the tube metal temperature, primary side fluid conditions, primary side coolant-to-tube heat transfer rates, and secondary side tube-to-coolant heat transfer rates.

Sections 5.3.1 and 5.3.2 describe the calculations related to the sectionalization of the tubes. These calculations relate mostly to the primary side dynamics. However, they are integral to the steam generator tube heat transfer model, and are described here.

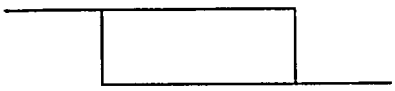
5.3.1 Geometric Model of Sectionalized Tubes



[ ]

5.3.2 *Dynamic Application of Sectionalized Geometry on Primary Side*

5.3.2.1 Dynamic Sectionalization

[ ]

[

]

5.3.2.2 Distribution of Flowrates Through the Sections

Given a primary tube node with N sections, the inlet and exit flowrates of the node are given by the RCS path information. Those are distributed smoothly over the node's subsections:

$$w_0 = W_0$$

$$w_i = w_{i-1} + (W_N - W_0) / N \ , \qquad \text{for } 1 \le i \le N$$

where

w_i = Flowrate at top of section i

W_0 = Inlet flow to node

W_N = Exit flow from node

If there is a Tube Rupture, then the flows are adjusted accordingly.

5.3.2.3 Distribution of Specific Enthalpies Through the Sections

Given a tube node with N sections and the flows determined as above, the specific enthalpies of the node's inlet and exit flows are given by the RCS path information. Those are distributed smoothly over the node's subsections, based primarily on the tube heat transfer rates:

$$h_0 = H_0$$

$$h_i = h_{i-1} - Q_i / W_i \quad , \qquad \text{for } 1 \le i \le N$$

where

h_i = Specific enthalpy at top of section i

H_0 = Specific enthalpy of inlet flow to node

H_N = Specific enthalpy of exit flow from node (used below)

Q_i = Heat transfer rate in section i (positive from the primary fluid to the tubes).

[

]

5.3.2.4 Temperature Profile for Heat Transfer

An input option, RCS_SG_SECT_TOPT, determines the method for calculating the section temperatures, for the purpose of interfacing to the calculation of tube metal temperatures and heat transfer rates.

5.3.3 *Conservation of Tube Metal Energy*

Conservation of energy in the tube metal is applied separately each section (Section 5.3.2) of the tube bundle of each steam generator. For section *i*:

[ ]

where

C_p = heat capacitance of tubes

h = heat transfer coefficient (Sections 5.3.4 and 5.3.5)

A = area

M = tube mass

T = temperature

and the subscripts are

p = primary side fluid conditions

i = tube section number

[ ]

T = steam generator tubes.

The primary heat transfer[ ]is reduced as the tubes are uncovered. The secondary side[ ]are adjusted to account for the[ ] Provision is also made for a user-specified degradation of tube heat transfer area due to plugging of tubes.

The model determines heat transfer rates separately for the primary and secondary sides of the tubes, for each tube section of each node of the hot and cold sides of the tube bundle, for each steam generator.

5.3.4 *Primary Side Heat Transfer*

Both the forward and reverse heat transfer are modeled for the primary side. The forward heat transfer consists of the following regimes:

1. Subcooled forced convection: Dittus Boelter equation[5 5]

2. Two phase flow with condensation: Akers, Deans and Crosser[5 6].

The reverse heat transfer consists of the following regimes:

1. Nucleate boiling: Thom's correlation[5.7]

2. Superheated Steam: Dittus-Boelter equation[5.5].

5.3.5 *Secondary Side Heat Transfer*

Both the forward and reverse heat transfer are modeled for the secondary side. The forward heat transfer considers the following relations:

1. Pool boiling for liquid: Modified Rohsenow[5 8]

2. Steam superheat in free convection: McAdams[5 9]

The reverse heat transfer considered is

1. Free convection: McAdams[5 9].

5.4 Indicated Level

The water level indicated by the steam generator level instrumentation is calculated by computing each reference leg's hydrostatic differential pressure as follows,

$$\Delta P = \rho_r h_r - \{\rho_3 h_3 + \rho_1(h_r - h_3)\}$$

where

ΔP = differential pressure at reference leg taps
ρ = density
h = water column height

and the subscripts are

r = reference leg
1,2,3 = Steam generator secondary side inner regions (Figure 7.6).

The reference leg temperature is normally taken as the containment temperature. CENTS calculates the indicated water level from the differential pressure and the calibration densities, using the same model that determines the indicated pressurizer level, as described in Section 4.19.1. See Section 6.7.4 for additional information. The indicated level will deviate from the actual downcomer level during a transient, as conditions differ from the calibration temperature.

5.5 Feedwater Line Model

The details of the feedwater line model depend on whether the feedwater pumps and piping are modeled explicitly in the CENTS database. The choice is indicated by the CENTS user/modeler via the variable NUM_FWS_PUMPS for the number of feedwater pumps. The two modeling approaches are described in the following subsections.

If NUM_FWS_PUMPS $\geq$ 1, CENTS employs a detailed model of the Main Feedwater and Auxiliary Feedwater systems. The detailed model is described below in Sections 5.5.1 - 5.5.9.

If NUM_FWS_PUMPS = 0, CENTS employs a simplified model, described below in Section 5.5.10.

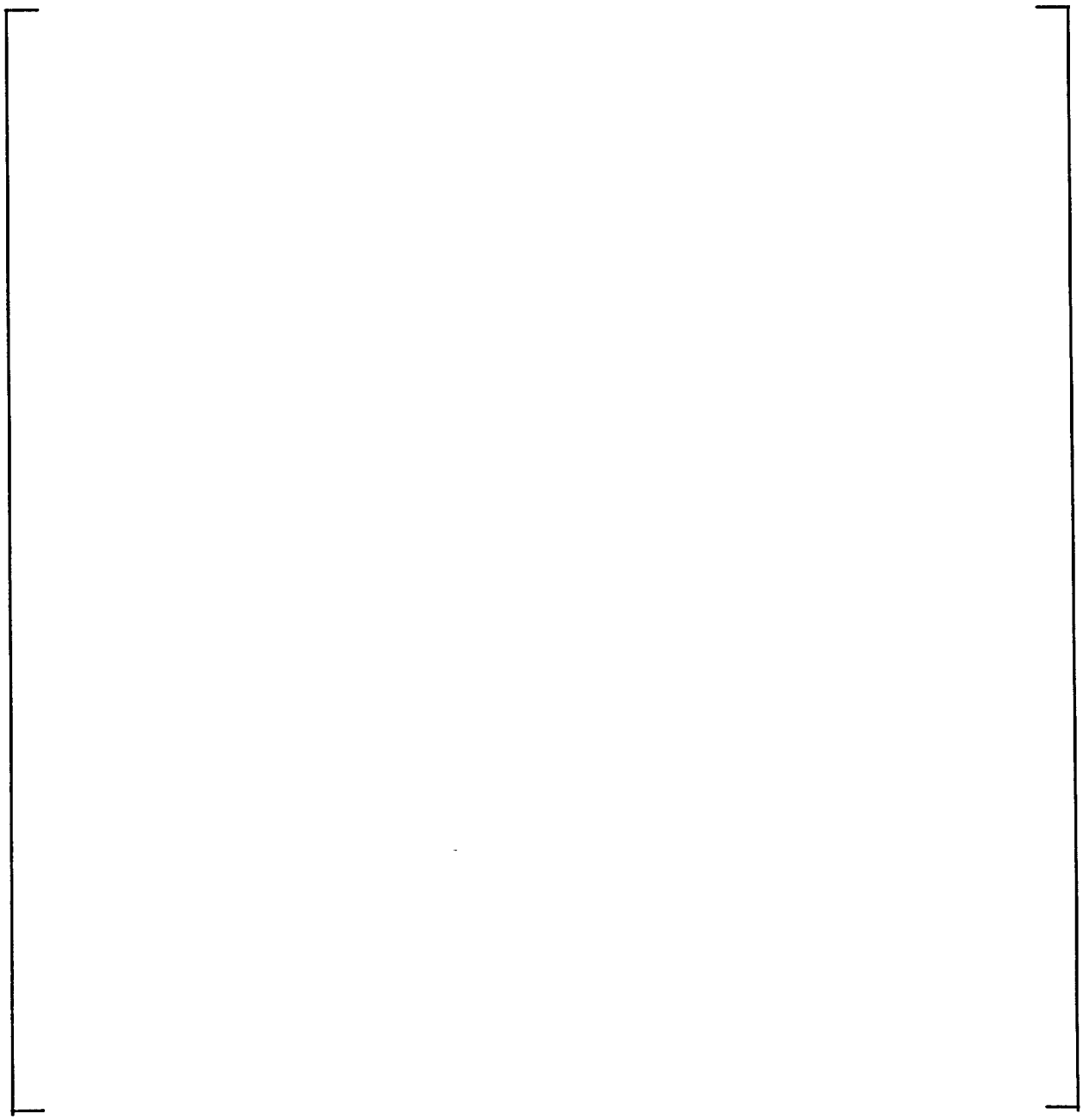
5.5.1 Detailed Model – General Description

This section describes the CENTS discrete Main Feedwater (MFW) and Auxiliary Feedwater (AFW) models. The model enables accurate, time dependent transient simulation for the MFW and AFW systems. This model allows the user to develop a network of discrete MFW and AFW components and piping through input or through the text input data file - *fdtrn.dat*. The system network is adaptable to different plant-specific designs. CENTS control system components are used to control MFW and AFW system valves and pumps.

If the number of feedwater pumps (NUM_FWS_PUMPS) is greater than zero, CENTS employs this detailed feedwater model.

The discrete MFW and AFW model includes control valves, minimum flow recirculation control valves, check valves, isolation valves, condensate pumps, main

feedwater pumps, heater drain pumps, auxiliary/emergency feedwater pumps, feedwater heaters, the condenser, heater drain tanks and the feedwater piping including cross connects. Pump head-flow curves, line and valve losses are modeled.



5.5.2 *Detailed Model – Piping Network*

The feedtrain model is a node and flowpath model which calculates steady-state flow through the various flowpaths necessary to balance pump head rise against head losses due to piping flow losses, valve flow losses, and elevation losses. The piping from the outlet of the condenser to the steam generator inlets is modeled including the heater drains.

The equations are as follows:

For each path:

$$P(in) - P(out) - \rho/62.4*R*Q*|Q| + \rho/144.*\{ht(in) - ht(out)\} + \rho/144.*(hd_{pump}) = 0.0$$

where:

P ≡ node pressure, psia

Q ≡ flow, gpm

= w*448.83/ρ

R ≡ flow resistance (psi/(gpm)^2)

ht ≡ elevation, feet.

hd_{pump} ≡ pump head, feet.

w ≡ path flow, lbm/sec

For each node:

$$\Sigma w(in) - \Sigma w(out) - w_{extf} = 0.0$$

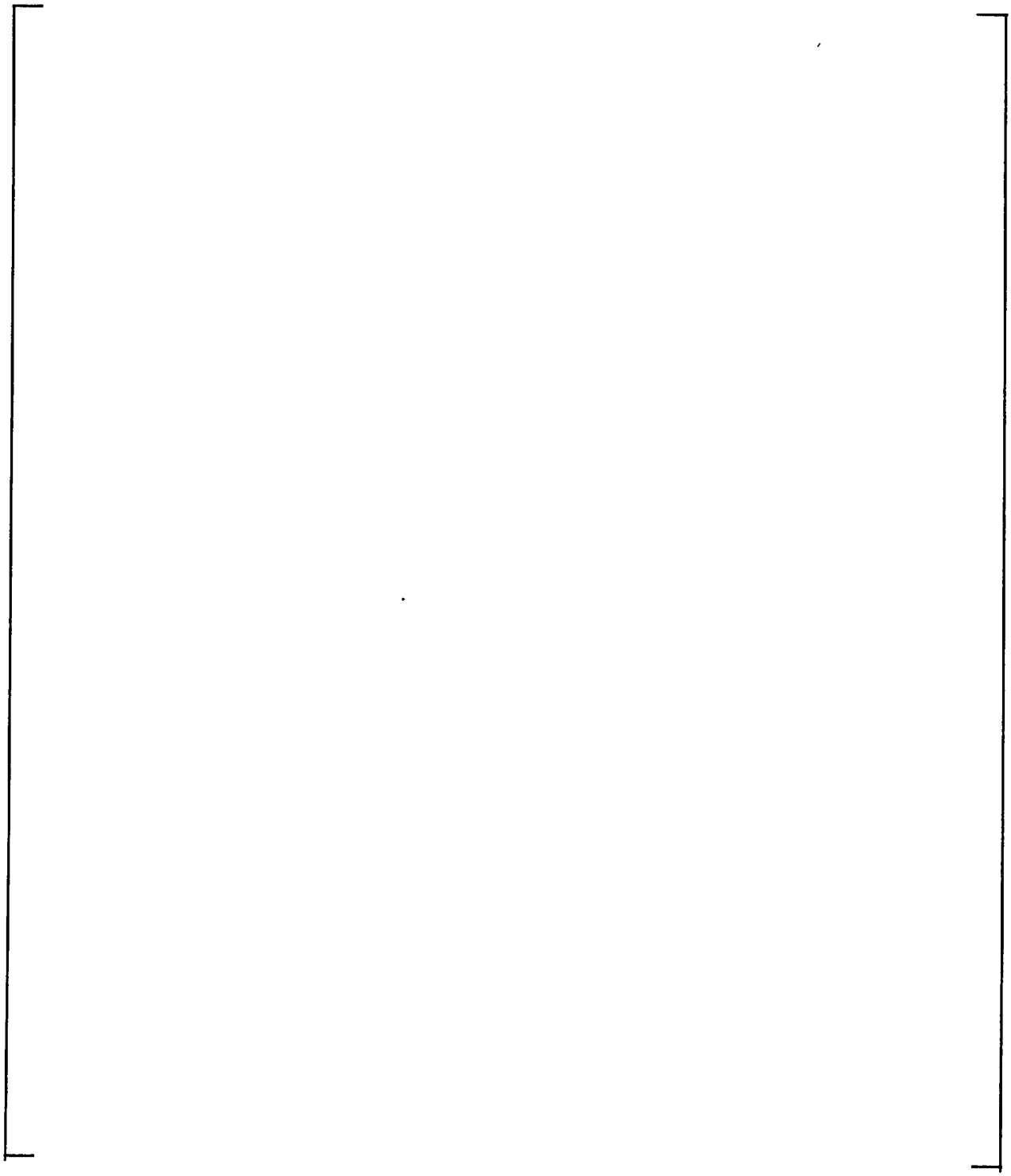
where:

w ≡ path flow (lbm/sec)

w_{extf} ≡ external flow out of node.

[

]



A matrix solution technique is used to simultaneously solve for flow and pressure throughout the feedtrain. The iterative solution is continued until

]converges.

The calculation of reverse flow is prevented in those paths that are defined as including a check valve.

The feedtrain node and flowpath configuration is defined by an input data file or by CENTS data base input which include the definition of paths and nodes, flowpath flow coefficients, external connections (pressure and enthalpy), and initial conditions. Input data files also define the number and type of pumps or valves in a flowpath including pump head-flow or valve C_V vs. position characteristics.

5.5.3 Detailed Model – Feedwater Line Break

The momentum equation (above) is not used for the path defined to be a feedwater line break path. For a feedwater line break (FWLB), it is assumed that the flow is proportional to the upstream pressure. The mass flux is obtained from critical flow (Section 5.2.6) given the upstream node pressure and fluid properties.

$$w = \text{flux} * A_{fwlb}$$
$$\approx \text{flux} * A_{fwlb} * P / P_o$$

where,

flux = f(P, h, hg, v, Pcont) – (lbm/sec/ft^2)

A_{fwlb} = feedline break flow area (ft^2)

P = pressure at break location (psia)

P_o = pressure at break location at beginning of time step (psia)

Therefore, the equation for a FWLB path becomes:

$$P - (P_o / (flux * A_{fwlb})) * w = 0.0$$

This equation allows the feedwater break flow and the pressure at the break to be calculated as part of the simultaneous solution.

5.5.4 *Detailed Model – Feedwater Line Flow Leaks*

The model allows for the calculation of leaks at each node. [

]

[

]

5.5.5 *Detailed Model – Feedwater Pumps*

The pump head rise is calculated for both the variable-speed (turbine driven) pumps and for the constant-speed (motor driven) pumps, using pump head-flow characteristics. A centrifugal pump model calculates the pump head rise using the pump affinity laws to perform a parabolic fit of pump head versus flow about the head axis for a given pump speed.

$$(Q_1 / Q_0) = (N_1 / N_0)$$

$$(H_1 / H_0) = (N_1 / N_0)^2$$

where,

Q = Capacity in gpm

N = Impeller speed in rpm

H = Pump head in feet

$()_0$ = At pump test speed (rated)

$()_1$ = At current pump speed

Tables of flow (gpm) versus pump resistance coefficient (psi/gpm^2) and flow (gpm) versus total head (ft) are calculated from input tables of pump head (ft) versus flow (gpm) during initialization as follows:

$$H_T = H_D + kQ^2$$

$$k_i = -\{H_T(i) - H_T(i-1)\} / (Q_i^2 - Q_{i-1}^2)$$

where,

H_D = Pump head (feet)

H_T = Total Pump head (feet)

k = Pump flow resistance (feet/(gpm)^2)

R_p = Pump flow resistance (psi/(gpm)^2)

The total head and pump flow resistance coefficients are calculated during the transient by interpolating the above calculated tables as a function of flow. The flow resistance coefficient and pump head are calculated as follows:

$$R_p = k_i * \rho / 144.$$

$$hd_{pump} = H_T * (N_1 / N_0)^2$$

Pump speed is calculated using a first order differential equation given pump speed demand and using input first order lag time constants for pump acceleration or deceleration.

$$S(t + \Delta t) = S_{dem} + (S(t) - S_{dem})*e^{-\Delta t/\tau}$$

where,

S = pump speed (fraction) = (N / N_0)

S_{dem} = speed demand (fraction)

τ = time constant (seconds)

t = current time (seconds)

Δt = time step (seconds)

Code input for the feedtrain pumps include rated pump head versus flow curves, pump acceleration and deceleration first order lag time constants, initial pump speed (fraction of rated speed), and the path number in which the pump is located. CENTS control system components can be used to define the pump speed demand signal for the main feedwater and auxiliary feedwater pumps.

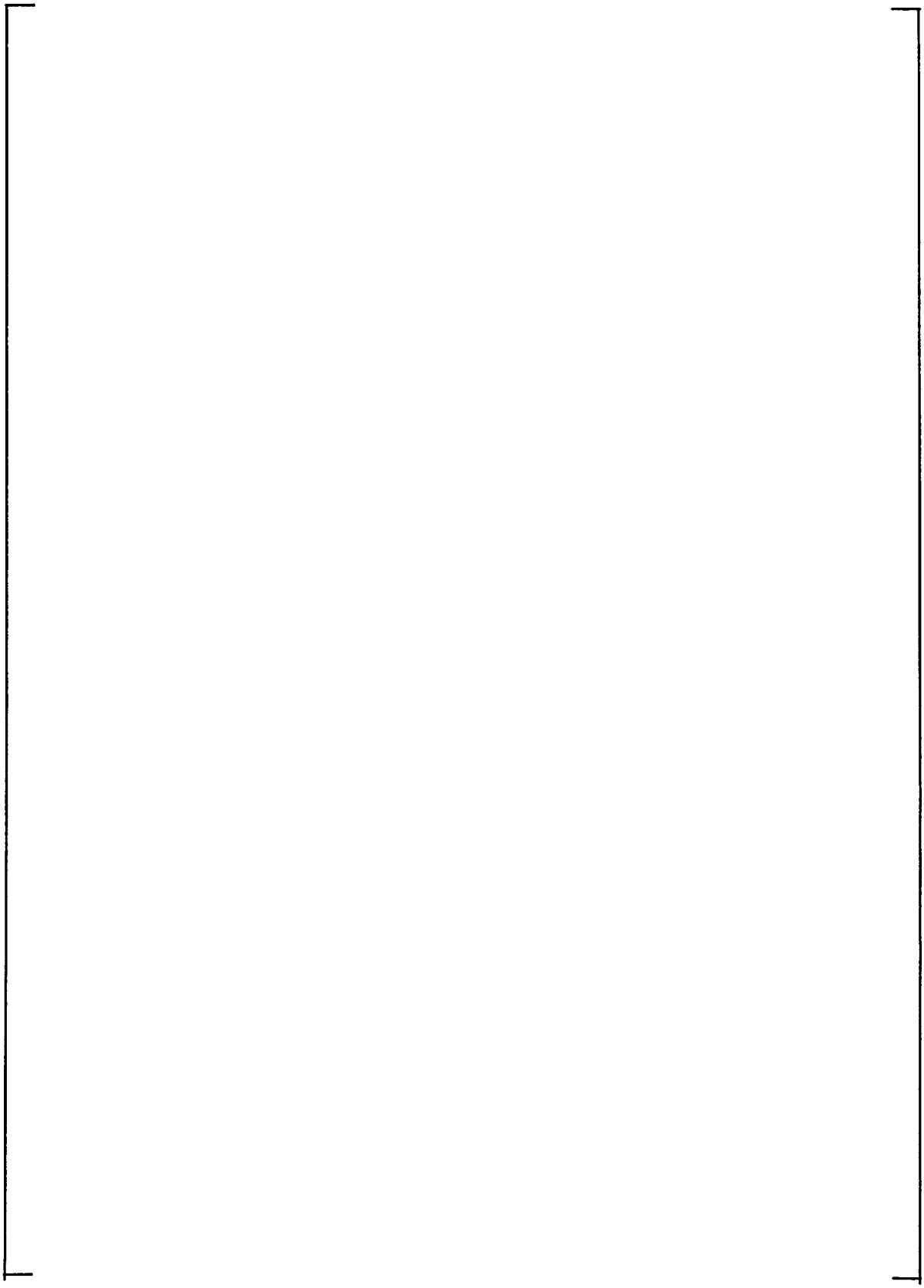
5.5.6 Detailed Model – Feedwater Control Valves

The flow coefficient, C_V , of each control valve is calculated at each time step using a table of valve C_V versus valve position. [

]

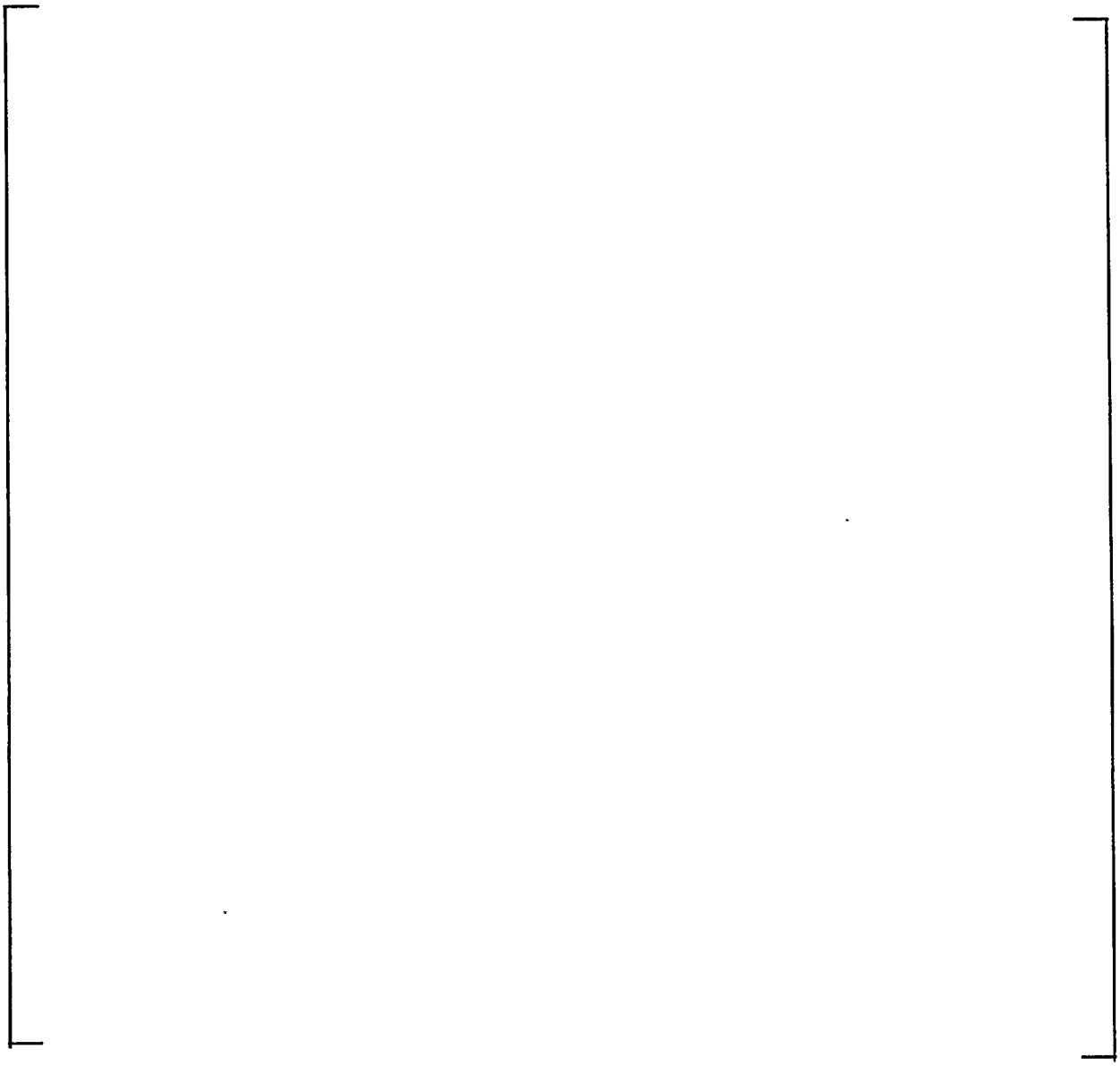
[

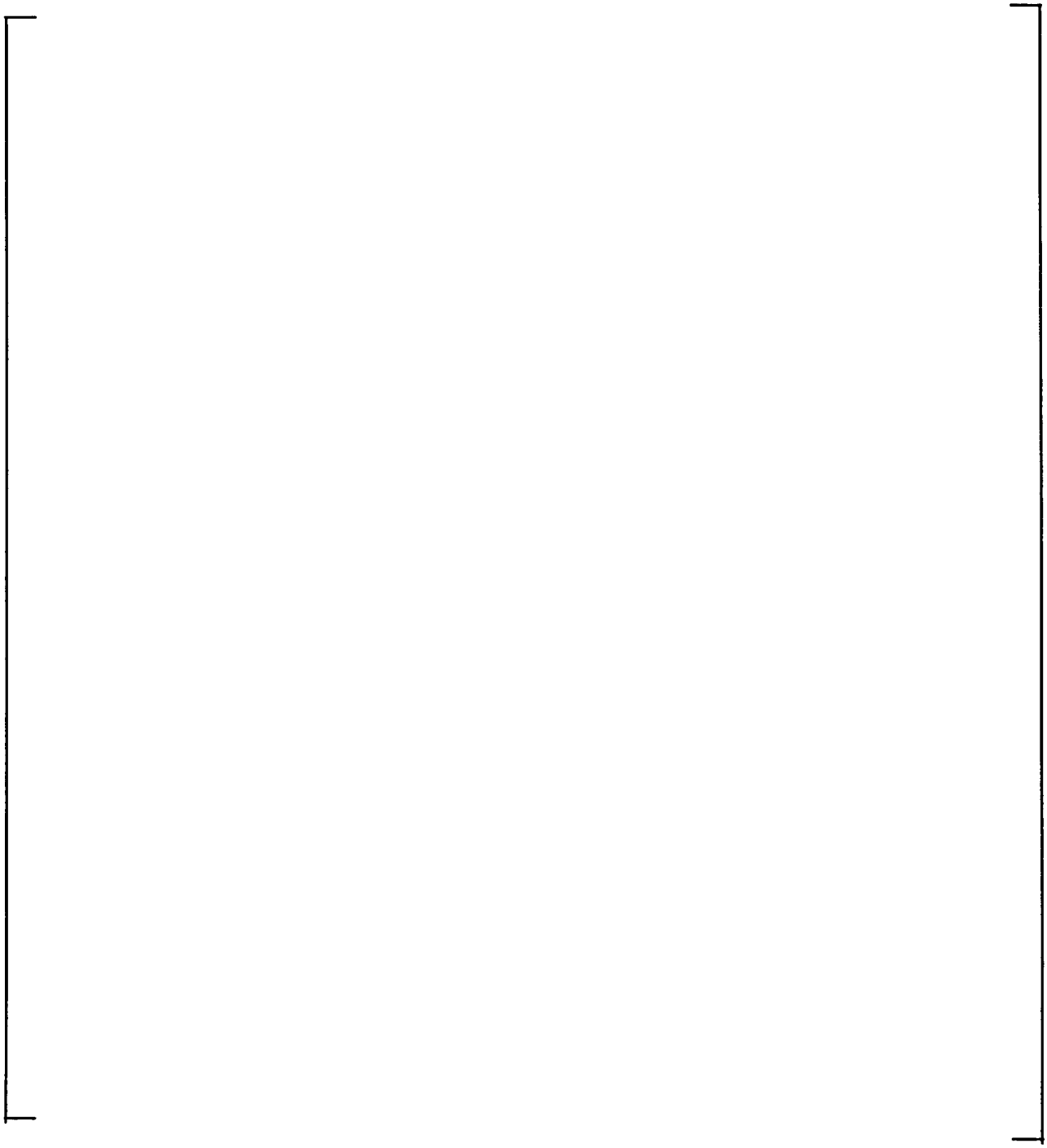
]



5.5.7 *Detailed Model – Feedline Check Valves and Isolation Valves*

Each flowpath can include a check valve, whose presence is defined by code input. The input also includes the fully-open flow resistance of the check valve. If the flow resistance for the check valve is zero, the check valve is assumed to be strictly bi-stable. If the flow resistance is non-zero, a variable flow resistance across the check valve is calculated as the check valve begins to close as flow decreases.





Each flowpath can include an isolation valve. The isolation valve is assumed to either be open or closed. [

]

5.5.8 *Detailed Model – Unisolable Feedline Mass Expansion Flow*

The flow network model assumes that the sum of the masses into and out of a node is zero. However the expansion of feedwater mass in the feedlines can be modeled by

[

]

The mass flowrate to the steam generator because of expansion is the difference in the steady state mass from one time step to the next.

[

]

5.5.9 *Detailed Model – Feedwater Enthalpy and Feedwater Heater*

A secondary heat balance is performed to determine the feedwater enthalpy to the steam generators. The condenser outlet enthalpy is provided as an external node enthalpy through input data.

Given turbine power, the heat addition from the low pressure and high pressure feedwater heaters is calculated by interpolating input tables as a function of turbine power. The enthalpy at the outlet of the heaters is calculated as a function of the heat

added, the inlet enthalpy from the upstream node, and the mass flowrate. The enthalpy calculation includes a representation of the thermal capacitance of the heaters. The heater outlet enthalpy can be limited to the saturation enthalpy corresponding to the extraction steam pressure to the last set of low or high pressure feedwater heaters through input data. Input data for the feedwater heaters includes tables of heat added and maximum outlet enthalpy versus turbine power and thermal heat capacitance.

The heater drain enthalpy is calculated given input tables of heater drain tank enthalpy versus turbine power. The heater drain flowrate is calculated given input tables of heater drain flow versus turbine power. The enthalpy at nodes where flowpaths with different enthalpies combine is calculated using a mass and energy balance given the node inlet flows and enthalpies.

5.5.10 *Simplified Feedwater Line Model*

If the number of pumps is set to NUM_FWS_PUMPS = 0, CENTS employs a simplified model, in which the feedwater flowrate delivered by the pumps for each steam generator is specified directly by the control system (Chapter 6 and Appendix C). The model feeds the indicated flows to the steam generators, unless a feedwater line break has occurred.

The break flow from feedwater lines is calculated by the critical flow model (Section 5.2.6). The feedwater flowrate to the steam generator is given by

$$|W_{SG}(t)| = F_o \sqrt{\rho_{FW} |(P_{FL} - P_{SG})|} \qquad (5.26)$$

$$W_{SG}(t) = W_{FW}(t) - W_{break}(t) \qquad (5.27)$$

where

$$F_o = W_{FW}(0) / \sqrt{\rho_{FW}^{(0)} \left(P_{FL}^{(0)} - P_{SG}^{(0)}\right)}$$

W = Mass flowrate
P = Pressure
ρ = Density

and the subscripts are

FW = feedwater pump
FL = feedwater line
SG = steam generator feedwater nozzle
break = feedline break.

The feedline pressure (P_{FL}), the feedline break flowrate (W_{break}) and the flowrate at the steam generator feedwater nozzle are determined by a simultaneous solution of Equations (5.26) and (5.27) and the applicable critical flow correlation. The solution is subject to the following factors depending on the feedline state:

1. Existence of break,
2. Break location relative to the feedline check valve,
3. Break size,
4. Feedwater pump delivery (W_{FW}), and
5. Break area relative to the critical area, defined as the minimum area required to divert all feedwater to a choked break. Note that if the break area exceeds the critical area, then the flowrate at the feedwater nozzle, W_{SG} in Equation (5.27), is negative.

5.6 Main Steamline and Header Model

5.6.1 General Description

The Main Steamline and Header Model includes the following (see Figure 7.10):

- The model simulates multiple main steamline header (MSLH) nodes and calculates the cross flow between them. The calculation of the cross flow allows for two different flow areas and flow resistances, depending on whether the turbine is tripped or not. This supports the main steamline break (MSLB) analysis by providing accurate calculation of the steam flow from the intact steam generator to the break.

- The model allows a MSLB to occur at any location along the steamline, specified by the input parameter MSLH_FKBRK to specify the flow resistance from the affected steam generator to the break location.

- The model checks for critical flow in all the steamline flowpaths, including the flowpath downstream of the steamline break location, and the cross-connect if the multiple header node model is being used.

NUM_MSLH is the input number of steamline header nodes, which with the current dimensioning may equal 1 or 2. Accordingly, the main steamline model calculates the input number of mass/energy nodes, which include the unisolable steamline volume downstream of the main steam isolation valves (MSIVs). Each steamline is modeled. An intermediate pressure between the steam generator outlet nozzle and the MSLH node is calculated. For a steamline break in the steamline, the intermediate steamline pressure calculation is performed at the point of the break. This allows the model to correctly calculate the split between the steam flow to the break and the steam flow to or from the downstream MSLH node. For all other conditions, the steamline pressure calculation is assumed to be at a point just upstream of the MSIVs. The intermediate steamline pressure is calculated from flow balance equations at the point of the pressure calculation.

Each MSLH node contains a metal wall. The wall metal temperature and heat loss are calculated for each node. Steamline drain flows are modeled.

5.6.2 Metal Temperature and Heat Loss

For each steamline node, the metal wall energy equation is:

$$MCp \ \frac{dT}{dt}w = HA_{in}(T_J - T_w) + HA_{out} \cdot (T_{amb} - T_w) \qquad (5.28)$$

where,

HA_{in} = heat transfer coefficient from steam to wall node (constant)

HA_{out} = heat transfer coefficient from wall node to ambient (constant)

T_J = steam temperature in Main Steam Header Node.

T_{amb} = ambient temperature

T_w = steamline wall metal temperature.

The heat loss from steam header node to wall is:

$$Q_{LOSS} = HA_{in} (T_J - T_w) \tag{5.29}$$

Equation (5.28) is a first order linear differential equation, the solution of which is:

$$T_w(t) = T_{w_{ss}} + (T_w (t-\Delta t) - T_{w_{ss}}) e^{-\Delta t/\tau} \tag{5.30}$$

where,

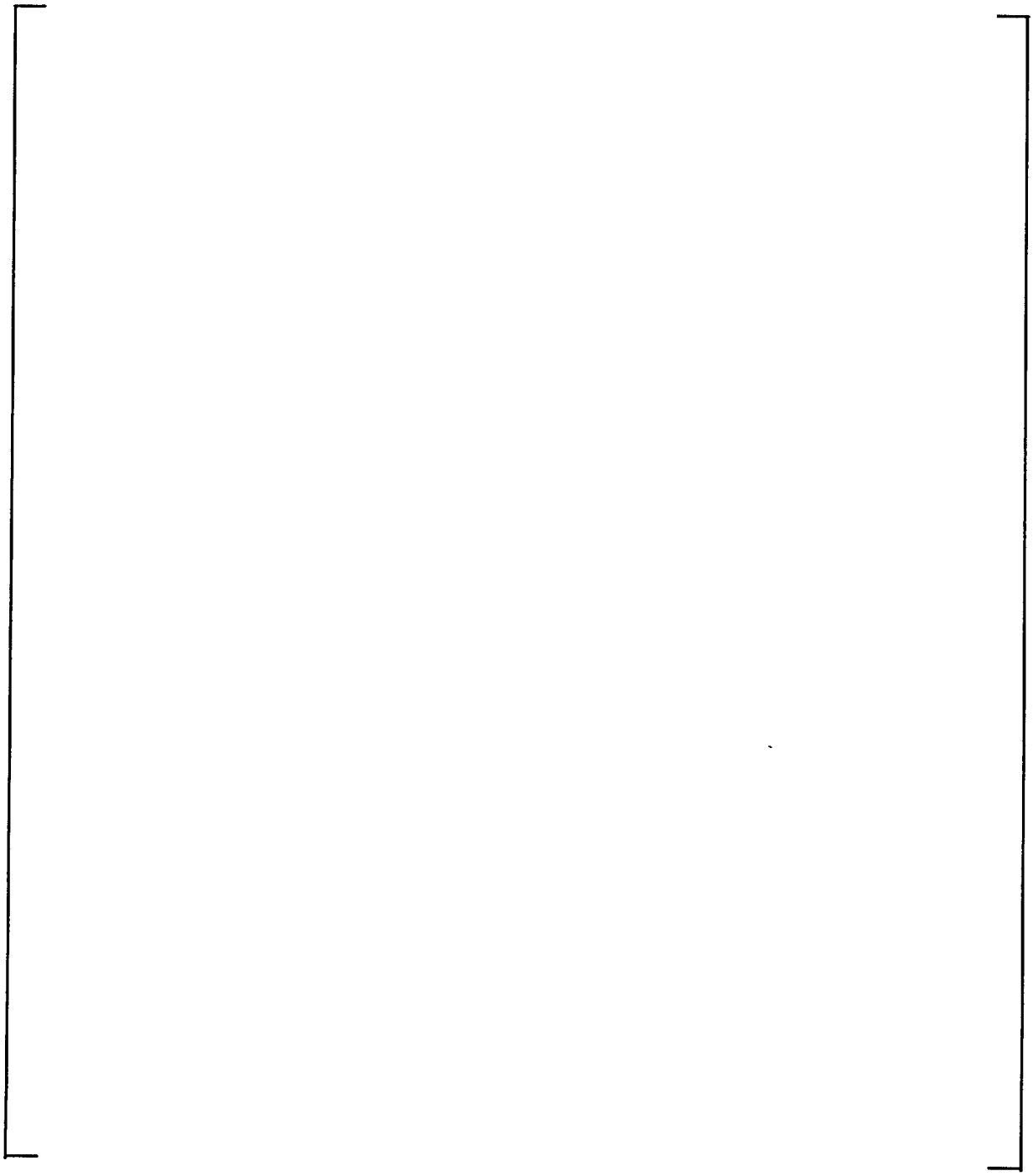
$$\tau = MCp / (HA_{in} + HA_{out}) \tag{5.31}$$

$$T_{w_{ss}} = (HA_{in} \cdot T_J + HA_{out} \cdot T_{amb}) / (HA_{in} + HA_{out}) \tag{5.32}$$

5.6.3 Steamline Drain Flows

The steamline drains remove liquid from the main steamline header nodes. The amount of liquid that goes out the steamline drains is assumed to be a function of the flow velocity and the amount of liquid available in the main steamline header nodes.

[ ]



5.6.4 *Steamline Pressures and Flows*

The main steamline header model calculates the steamline flows by iterating on steamline pressure until there is a flow balance. Steady state flow equations are used. The steam generator pressure and the main steamline header pressure at the beginning of the time step are used in calculating the steamline flows.

The unknowns are:

$P_{SL}(I)$ = intermediate location steamline pressure.

$W_{out_{SG}}(I)$ = steam flow out of steam generator for each steamline.

$W_{BK}(I)$ = steam flow out steamline break for each steamline.

$W_{RLF}(I)$ = steam flow out secondary safety valves and atmospheric dump valves for each steamline.

$W_{in_J}(I)$ = steam flow into steamline node from each steamline.

The knowns are:

$P'_{SG}(ISG)$ = steam generator pressure for each steamline. (ISG = SG ID)

$P'_J(IJ)$ = main steamline header pressure (IJ = mslh node ID)

Primed values denote at the beginning of the time step Δt.

The steamline flow equations are:

The CRITCO Critical Flow Equation (Section 5.2.6) is:

$$W = C_3 \cdot AREA \cdot PRESSURE \tag{5.36}$$

or, as applicable,

$$W = C_3 \cdot AREA \cdot (PRESSURE - BACK\ PRESSURE) \tag{5.37}$$

where

$$C_3 = \frac{C_0}{ENTHALPY - 185.0}$$

$C_0 = 0.53/0.000268$ CONSTANT

Steamline flow equations for each steamline:

Flow balance equation (Conservation of mass):

$$W_{out_{SG}}(I) = W_{BK}(I) + W_{RLF}(I) + W_{in_J}(I) \tag{5.38a}$$

Steamline break flow:

$$W_{BK}(I) = A_{BK}(I) \cdot C_3 \cdot (P_{SL} - P_{CONT}) \tag{5.38b}$$

where A_{BK} is the break area, P_{CONT} is the containment pressure (replaced by the atmospheric pressure P_{ATM} for a break discharging to the atmosphere), and C_3 is defined above for Equations (5.36)-(5.37).

Steamline safety valve or atmospheric dump valve flow with valve open area A_{RLF}:

$$W_{RLF}(I) = A_{RLF}(I) \cdot C_3 \cdot (P_{SL} - P_{ATM}) \tag{5.38c}$$

Steam flow from SG to intermediate steamline location (P_{SL}) :

$$W_{out_{SG}}(I) = A_{line}\{\rho_{SG} \cdot (P'_{SG} - P_{SL}) / (K_{SL_I} / (2 \cdot 144 \cdot g_c))\}^{1/2}$$

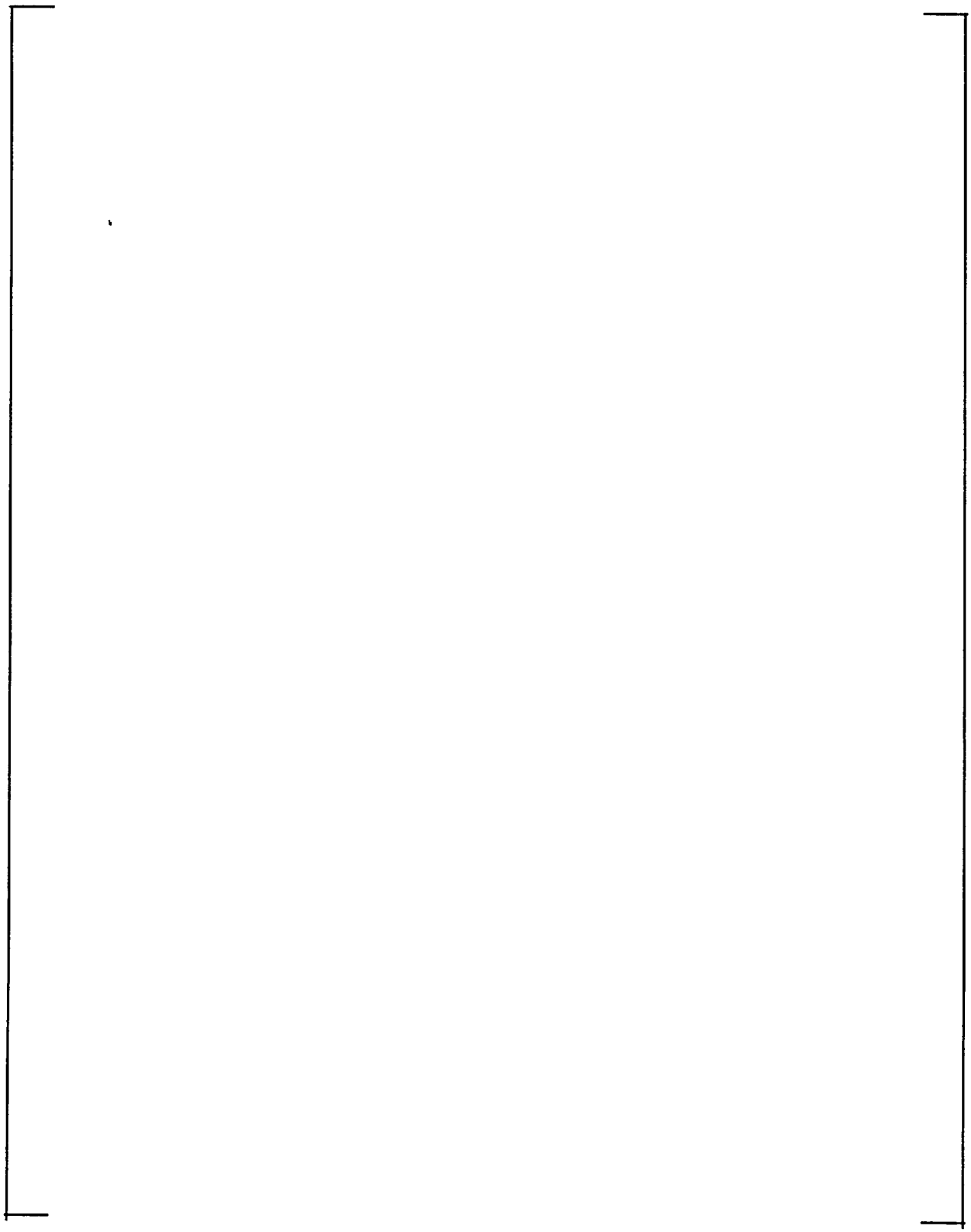
Steam flow from intermediate steamline location (P_{SL}) to main steam header (P_J):

$$W_{in_J}(I) = A_{line}\{\rho_{SG} \cdot (P_{SL} - P'_J) / (K_{SL_O} / (2 \cdot 144 \cdot g_c))\}^{1/2}$$

where A_{line} is the steamline pipe flow area, and K_{SL_I} and K_{SL_O} are pressure loss coefficients for upstream and downstream portions of the steamline.

[

]



[

]

Choked flow in the steamline:

Considering the possibility of choked flow in the steamline, apply[

]

[

]

[ ]

5.6.5 *Main Steamline Header Flow to Turbine and Condenser*

Main Steam Header flow to the turbine.

$$W_{turb} = A_{turb} \cdot C_3 \cdot (P'_J - P'_{turb})$$

C_3 is defined above for Equations (5.36)-(5.37). If NUM_MSLH =2, the value of A_{turb} used for each Main Steamline Header node is one half of the total turbine area demanded.

Main Steam Header flow to the condenser.

$$W_{cond} = A_{cond} \cdot C_3 \cdot (P'_J - P_{cond})$$

Main Steam Header flow to the atmosphere.

$$W_{rlf} = A_{rlf} \cdot C_3 \cdot (P'_J - P_{atm})$$

In the above formulations, A_{turb}, A_{cond} and A_{rlf} are the total flow areas (valves) from the header to the turbine, condenser and atmosphere, respectively; and P'_{turb}, P_{cond} and P_{atm} are the corresponding downstream pressures.

5.6.6 *Main Steamline Header Cross-Connect Flows*

If NUM_MSLH > 1, the model calculates cross flow W_{cross} between the main steamline headers.

[

]

5.6.7 *Main Steamline Header Conservation Equations*

The conservation of mass and energy equations are solved for each main steamline header node. If NUM_MSLH > 1, multiple main steamline header nodes are modeled, and the total number of main steamlines (NLINES) is divided among the header nodes.

Equations:

Constant Volume: $dM_J / dt = 0.0$ (5.47)

$$\text{Conservation of Mass:} \quad dM_J / dt = \sum_{i=1}^{NLINES} W_{in_J}(i) - \Sigma W_{out_J} \qquad (5.48)$$

$$\text{Conservation of Energy:} \quad dU_J / dt = \sum_{i=1}^{NLINES} Wh_{in_J}(i) - Wh_{out_J} - Q_{loss} \qquad (5.49)$$

Q_{loss} $\equiv$ heat loss from steamline.

W_{out_J} $\equiv$ total steam flow leaving steam header node.

If NUM_MSLH = 2, then

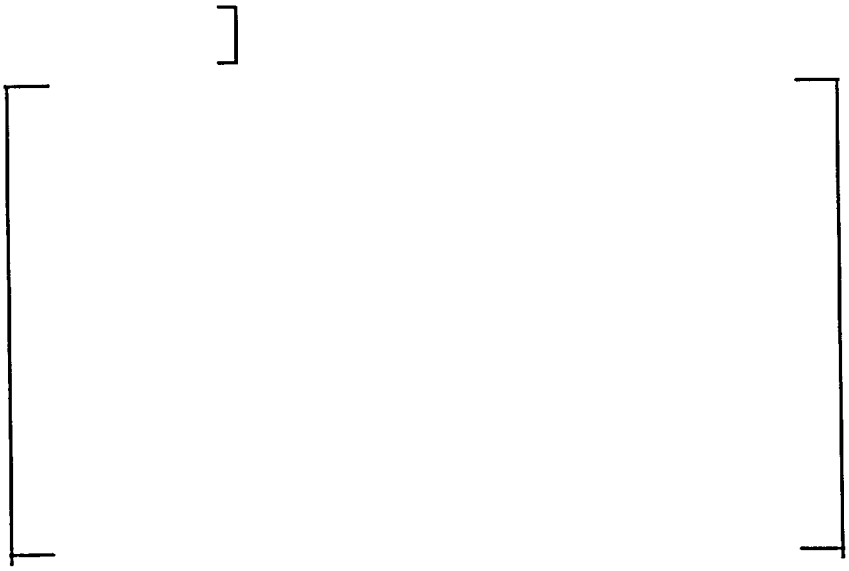
Node 1
Conservation of Mass: $$dM_{J1} / dt = \sum_{i=1}^{NLINES/2} W_{in_{J1}}(i) - \Sigma W_{out_{J1}} - W_{cross}$$

Node 2
Conservation of Mass: $$dM_{J2} / dt = \sum_{i=1+NLINES/2}^{NLINES} W_{in_{J2}}(i) - \Sigma W_{out_{J2}} + W_{cross}$$

Node 1
Conservation of Energy: $$dM_{J1} / dt = \sum_{i=1}^{NLINES/2} Wh_{in_{J1}}(i) - \Sigma Wh_{out_{J1}} - Q_{loss} - Wh_{cross}$$

Node 2
Conservation of Energy: $$dM_{J2} / dt = \sum_{i=1+NLINES/2}^{NLINES} Wh_{in_{J2}}(i) - \Sigma Wh_{out_{J2}} - Q_{loss} + Wh_{cross}$$

The conservation equations for the main steam header node are solved simultaneously for two unknowns; P_J, h_J. [



ENERGY EQUATION:

$$A_{21} \cdot P_J + A_{22} \cdot h_J = A_{23}$$

Steamline Header Conservation Equations

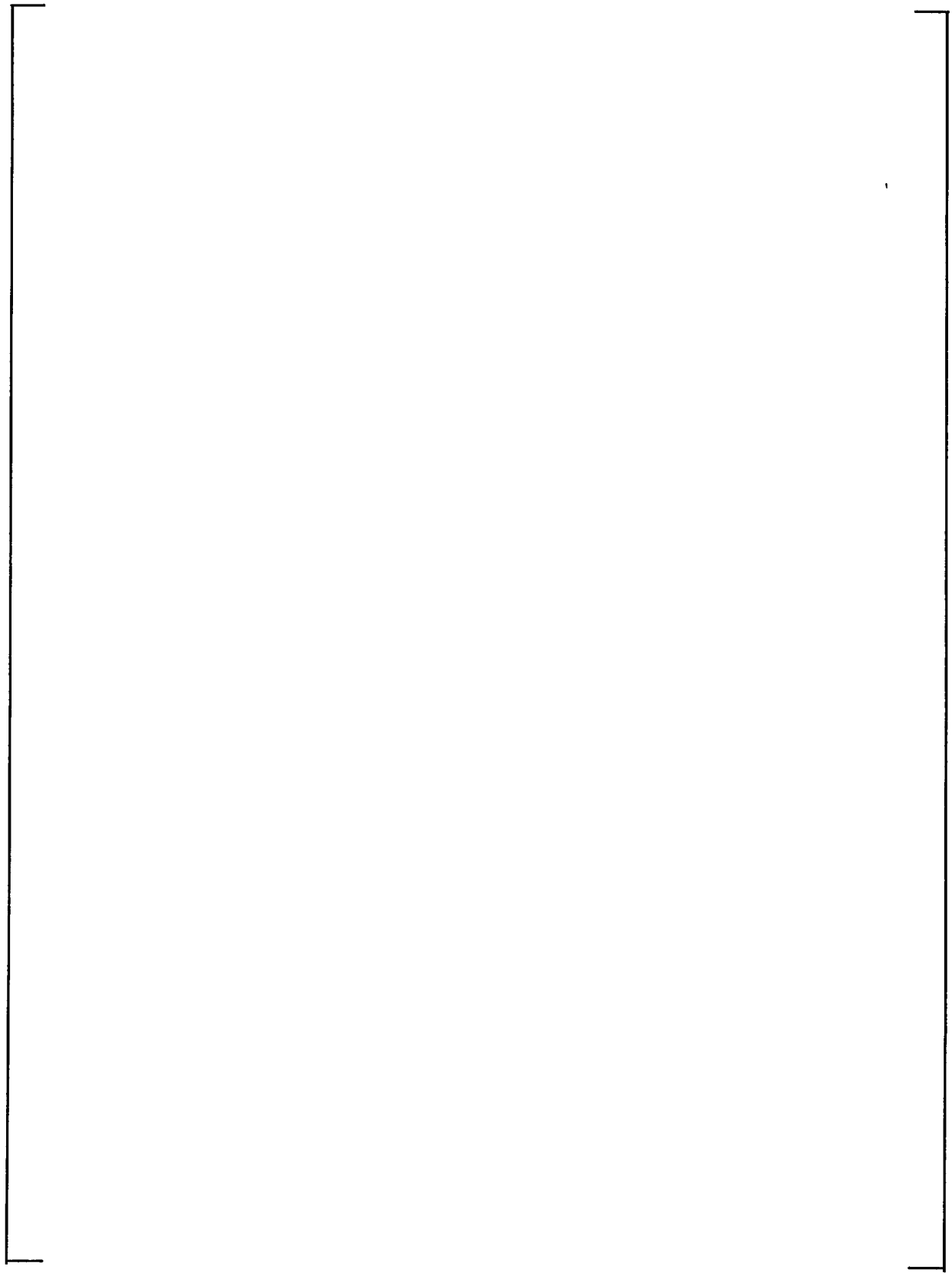
Constant Volume

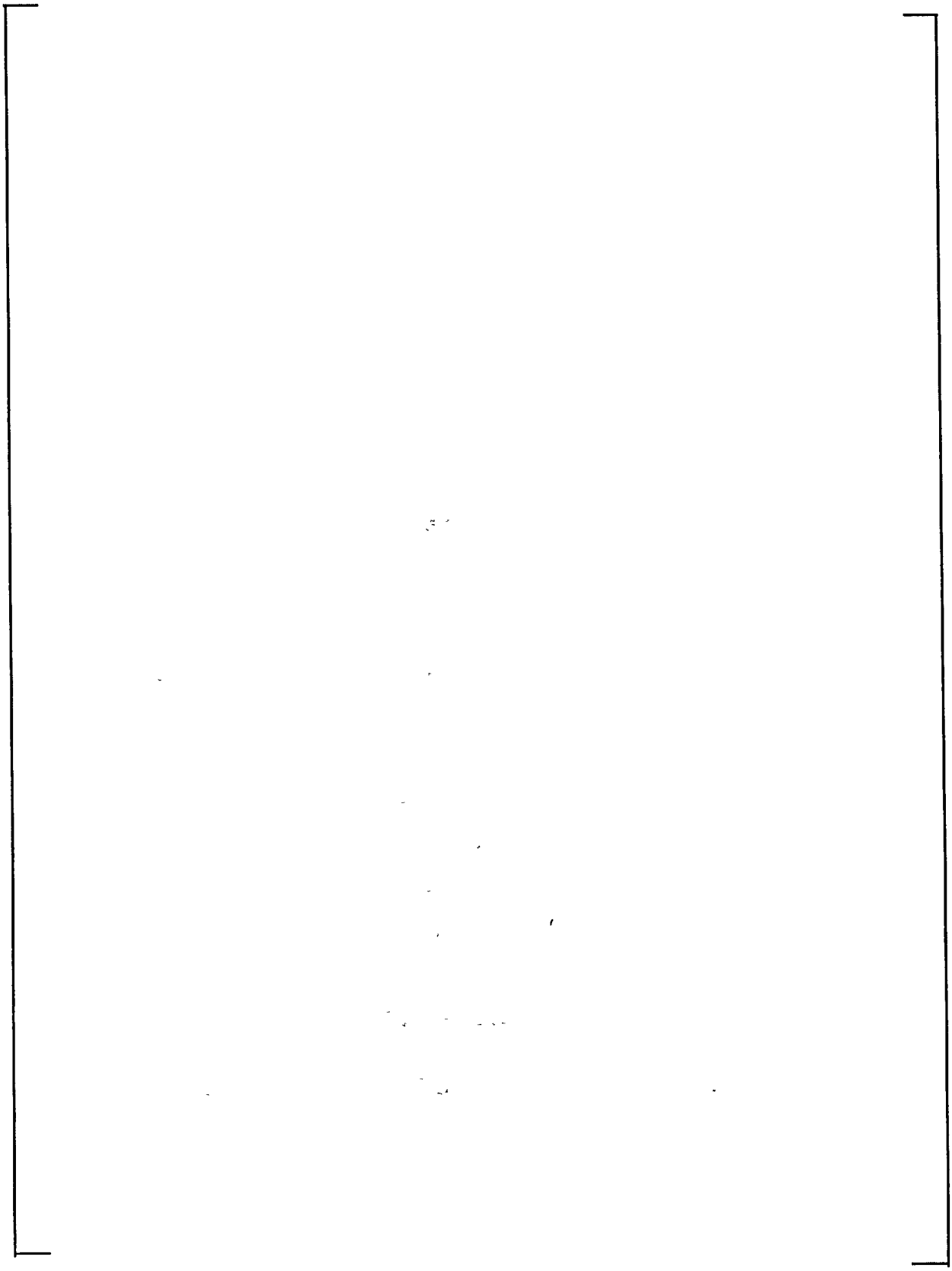
$$\frac{dV_J}{dt} = \frac{d\left(M_J v_J\right)}{dt} = \frac{dM_J}{dt} v_J + M_J \frac{dV_J}{dt} = 0$$

$$\frac{dM_J}{dt} v_J + \left(\frac{\partial v}{\partial P} \frac{dP_J}{dt} + \frac{\partial v}{\partial h} \frac{dh_J}{dt}\right) M_J = 0$$

Solving for $\frac{dM_J}{dt}$:

$$\frac{dV_J}{dt} = \frac{d\left(M_J v_J\right)}{dt} = \frac{dM_J}{dt} v_J + M_J \frac{dV_J}{dt} = 0 \qquad (5.50)$$





5.7 Steam Generator Tube Rupture Model

5.7.1 General Description

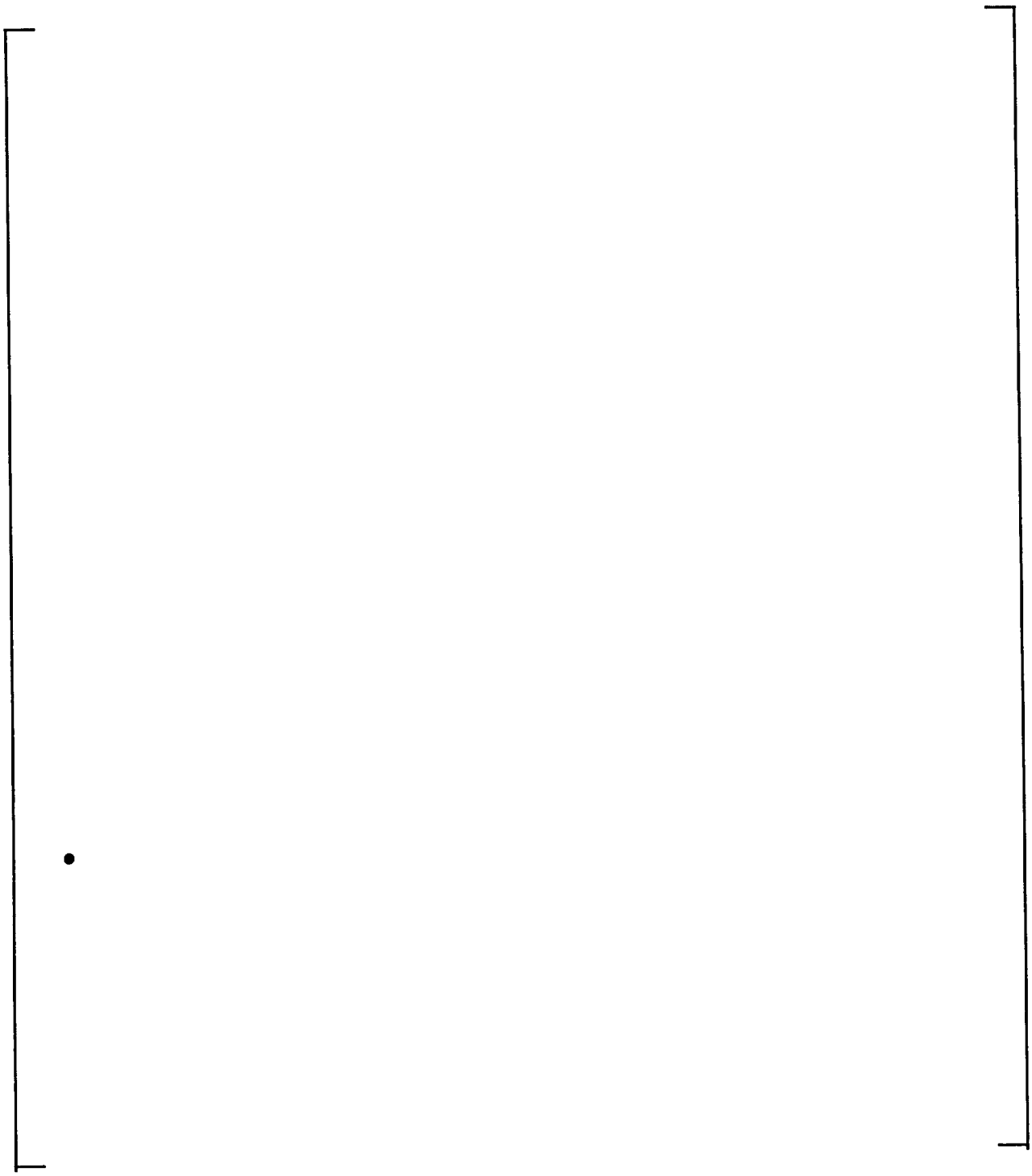
The CENTS Steam Generator Tube Rupture (SGTR) model simulates the effect of breaking one or more steam generator tubes.

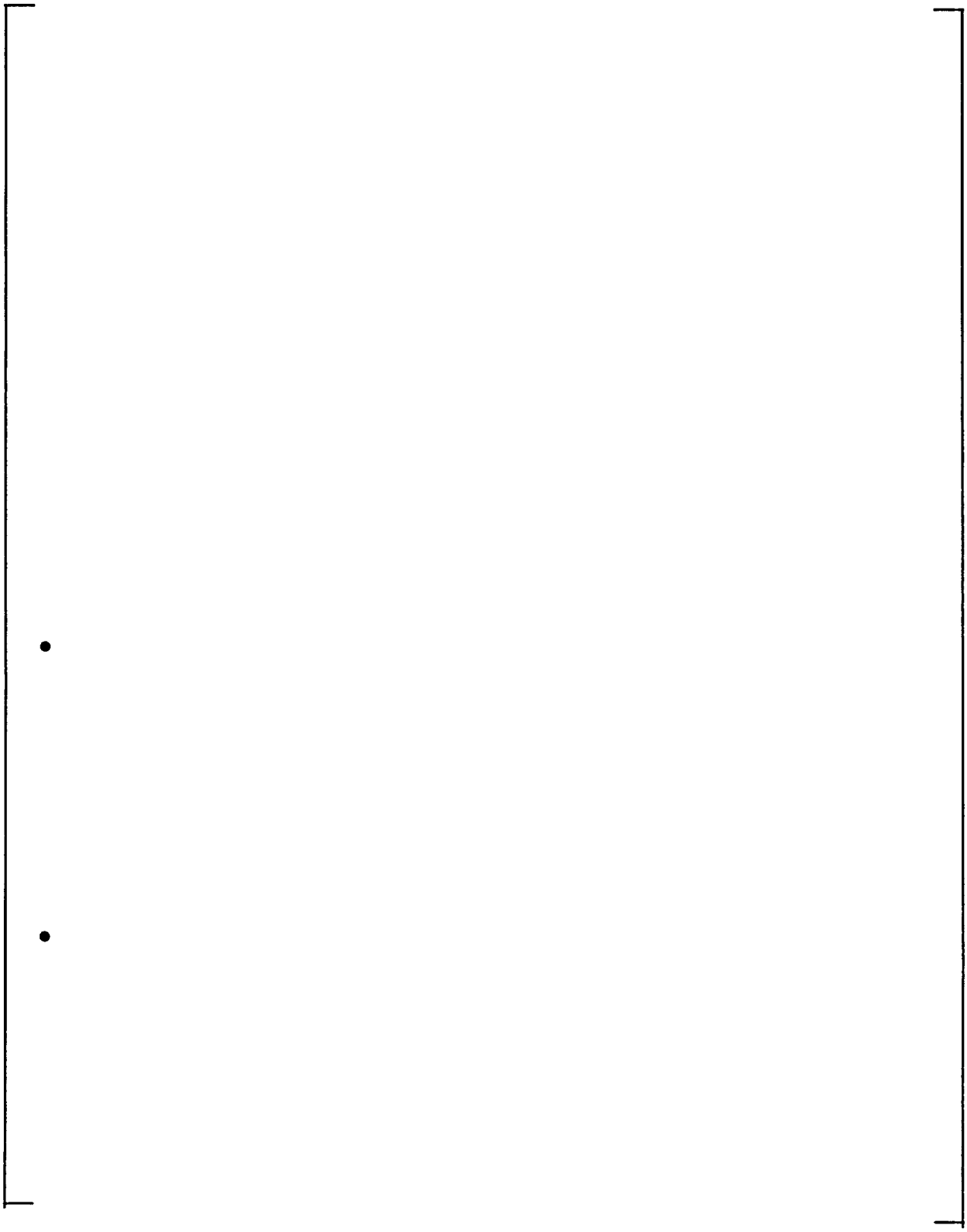
CENTS models the flow from both sections of a ruptured steam generator tube. Frictional losses are explicitly calculated. Critical flow is calculated using the tube pressure at the break location rather than the upstream RCS node pressure. In addition, a slot break may optionally be modeled.

5.7.2 Double Ended Guillotine Break Model

A single malfunction flag is provided for each steam generator. MAL_SGTR(1:4) is defined as the number of ruptured tubes. See Section 7.5.2 and Table 7.27.

For a double-ended guillotine break (DEGB), flow is calculated explicitly for both broken ends of the ruptured tube. The flow area of each end is SG_TUBE_AREA, and the total flow area for each ruptured tube is 2*SG_TUBE_AREA.





5.7.3 Slot Break Model

5.8 Iodine Dose Model

5.8.1 Model Overview

CENTS calculates integrated offsite dose due to Iodine 131. The CENTS dose model tracks iodine concentration throughout the primary and secondary systems, including primary-to-secondary leaks. Iodine releases to atmosphere, to the condenser, to the containment building, and from the containment building to atmosphere are integrated throughout the run. Iodine doses are then determined based on these integral releases and user input χ/Q data, breathing rates and dose conversion factors.

The user may specify how iodine partitioning of the tube leakage flow should be treated. The user can also input the iodine partition factor between the steam generator liquid and steam.

The dose model assumes that all isotopes of iodine are represented by an equivalent concentration of Iodine 131, and that the input is consistent with this assumption.

5.8.2 RCS Solute Concentrations

The primary system solute transport model is described in Section 4.15.

CENTS tracks the concentrations of solute species in the primary side nodes in the array RCS_CONC_SOLUTE(50,20), where the index pair is (node,species). For convenience, the node concentrations of several specific solute species may be read and written via the arrays:

RCS_CONC_BORON(1:50)
RCS_CONC_IOD(1:50)
RCS_CONC_PART(1:50)
RCS_CONC_XEN(1:50)

Boron concentration is important because it is used in the calculation of core power. Iodine concentration is used in the calculation of doses but does not affect the transient calculation.

5.8.2.1 Initial Conditions

The initial primary side solute concentration can be set as follows.

RCS_DOSE_INIT_IOD is a user input that defines the initial iodine concentrations (μCi/lbm) of each RCS node during initialization.

RBINIT is a user input that defines the initial boron concentration (ppm) during initialization.

The concentrations of the other solute species are set to 0.0 by the code during initialization.

The user may load values into any of the arrays listed above, for each node after initialization. For example, setting RCS_CONC_IOD = 17*27216 sets the concentration of nodes 1 through 17 to 27,216 μCuries/lbm, which is 60 μCuries/gm.

5.8.2.2 Primary Side Path Solute Concentrations

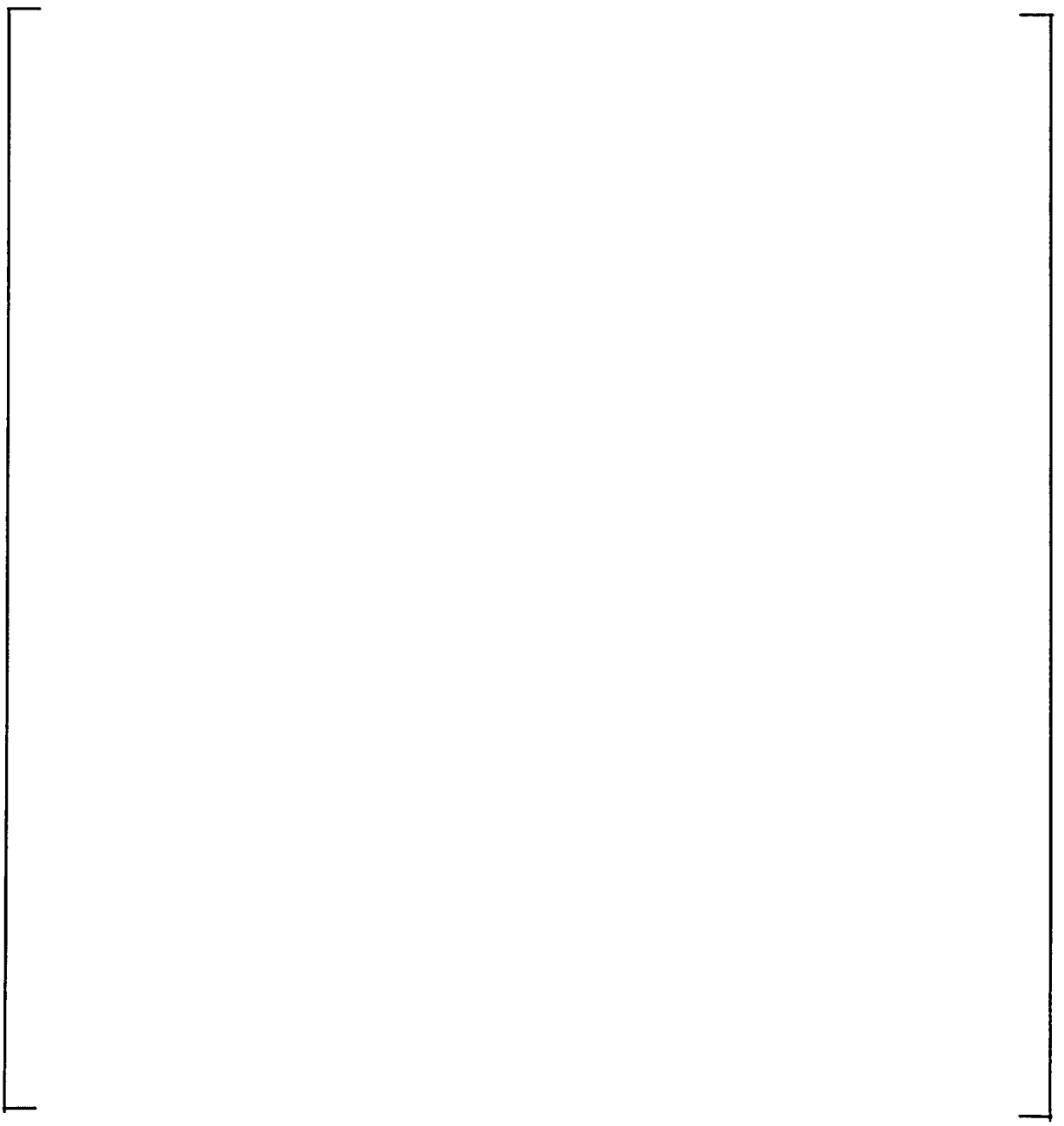
P_CONC_SOLU(150,20) is the concentrations of solute species in the flowpaths, where the index pair is (path,species). For convenience, the non-momentum path concentrations of several specific solute species are available:

RCS_P_BORON(1:50)

RCS_P_IOD(1:50)

RCS_P_PART(1:50)

RCS_P_XEN(1:50)



5.8.2.3 Primary Side Solute Balance

Solute concentrations in CENTS (including Iodine) are explicitly calculated in each of the RCS nodes by solving the concentration equation, as described in Section 4.15. The rate of accumulation of solute in each node is calculated as:

$$RCS\_CUM\_SOLU_{N,K} = \sum_{J} PFLOW_J * (P\_CONC\_SOLU_{J,K} - RCS\_CONC\_SOLUTE_{N,K})$$

where:

The summation is performed for all paths J connected to node N.

$RCS\_CUM\_SOLU_{N,K}$ = rate of accumulation of solute K in node N.

$RCS\_CONC\_SOLUTE_{N,K}$ = concentration of solute K in node N (μCuries/lbm).

$PFLOW_J$ = mass flowrate through path J connected to node N (lbm/sec).

$P\_CONC\_SOLU_{J,K}$ = concentration of solute species K in path J (μCuries/Lbm).

The concentration of solute in each node is then updated at time t+Δt as:

$$RCS\_CONC\_SOLUTE_{N,K} = RCS\_CONC\_SOLUTE_{N,K} + \frac{\Delta t * RCS\_CUM\_SOLU_{N,K}}{MASS\_TOT_N}$$

where:

$MASS\_TOT_N$ = total mass (liquid and steam) of primary node N (lbm)

Δt = time step size (sec).

The iodine concentration is then corrected for decay, as discussed in Section 4.15, where Equation (4.15.3) multiplies the iodine concentration by the decay factor:

$$2^{-\Delta t / HALF\_LIFE\_SOLUTES(K)}$$

where HALF_LIFE_SOLUTES(K) is the input iodine half-life (sec), and K is the solute number of iodine (ID_TYPE_IODINE).

5.8.2.4 Global Checks of RCS Iodine Inventory

A global check is performed on the primary side iodine inventory. At each time step, the total iodine inventory of the primary side (Curies) is calculated as:

$$RCS\_TOTAL\_IODINE=10^{-6} \sum_{N} RCS\_CONC\_SOLUTE_{N,K} * MASS\_TOT_{N}$$

where the summation is over all the RCS nodes, and K = ID_TYPE_IODINE.

The rate of change of total RCS iodine inventory (μCuries/sec) is calculated as:

$$RCS\_IOD\_REL\_RATE = \sum_{J} P\_CONC\_SOLU_{J,K} * P\_FLOW_{J}$$

where the summation is over all the non-momentum paths.

Finally, the integrated change in total RCS iodine inventory (Curies) since the beginning of the run is updated at time t+Δt as:

$$RCS\_IOD\_REL\_TOT\ (t+\Delta t) = RCS\_IOD\_REL\_TOT\ (t) + 10^{-6}\ \Delta t * RCS\_IOD\_REL\_RATE$$

These variables can be used to check the solute concentration calculation. The change in RCS_TOTAL_IODINE should be equal to RCS_IOD_REL_TOT.

5.8.3 Steam Generator Solute Concentrations

CENTS keeps track of the concentrations of solute species on the secondary side (Section 7.3.5, Table 7.17). The concentrations in the steam are tracked separately from the concentrations in the liquid.

Iodine is used in the dose model, as described below.

Boron concentration is important only in the case of back flow from the steam generator to the primary side. In that case the concentration of the RCS could be diluted by the steam generator inventory.

5.8.3.1 Initial Conditions

All solute concentrations are initially set to 0.0. After initialization, the solute concentrations can be set to some other value. For example,

SGS_CON_IO1 = 453.0, 0.0
SGS_CON_IO2 = 45.3, 0.0

sets the iodine concentration of the steam generator #1 steam region to 453 μCuries/lbm (1.0 μCuries/gm). The steam generator #1 downcomer and evaporator region concentration is set to 45.3 μCuries/Lbm (0.1 μCuries/gm). Steam generator #2 iodine concentrations are set to 0.0.

5.8.3.2 Solute Flows to and From the Steam Generator

Solutes in Feedwater Flow

Solutes can enter or leave the steam generator via feedwater. When feed flow is into the steam generator, the feedwater solute concentration arrays are set (Section 7.3.5, Table 7.24):

1. Boron 0.0
2. Iodine FWS_CON_IO
3. Particulates FWS_CON_PT
4. Xenon FWS_CON_XE

When feed flow is negative, which can only occur during a feedline break simulation, the solute concentration of the exiting fluid is set to the concentration of the steam generator.

Solutes in Steam Generator Tube Leak

Solutes can enter or leave the steam generator via a tube leak. This section describes how the concentration of the path fluid is determined.

For primary-to-secondary leaks, as described above, the solute concentration of the path is set to the concentration of the primary side node. [

]

[

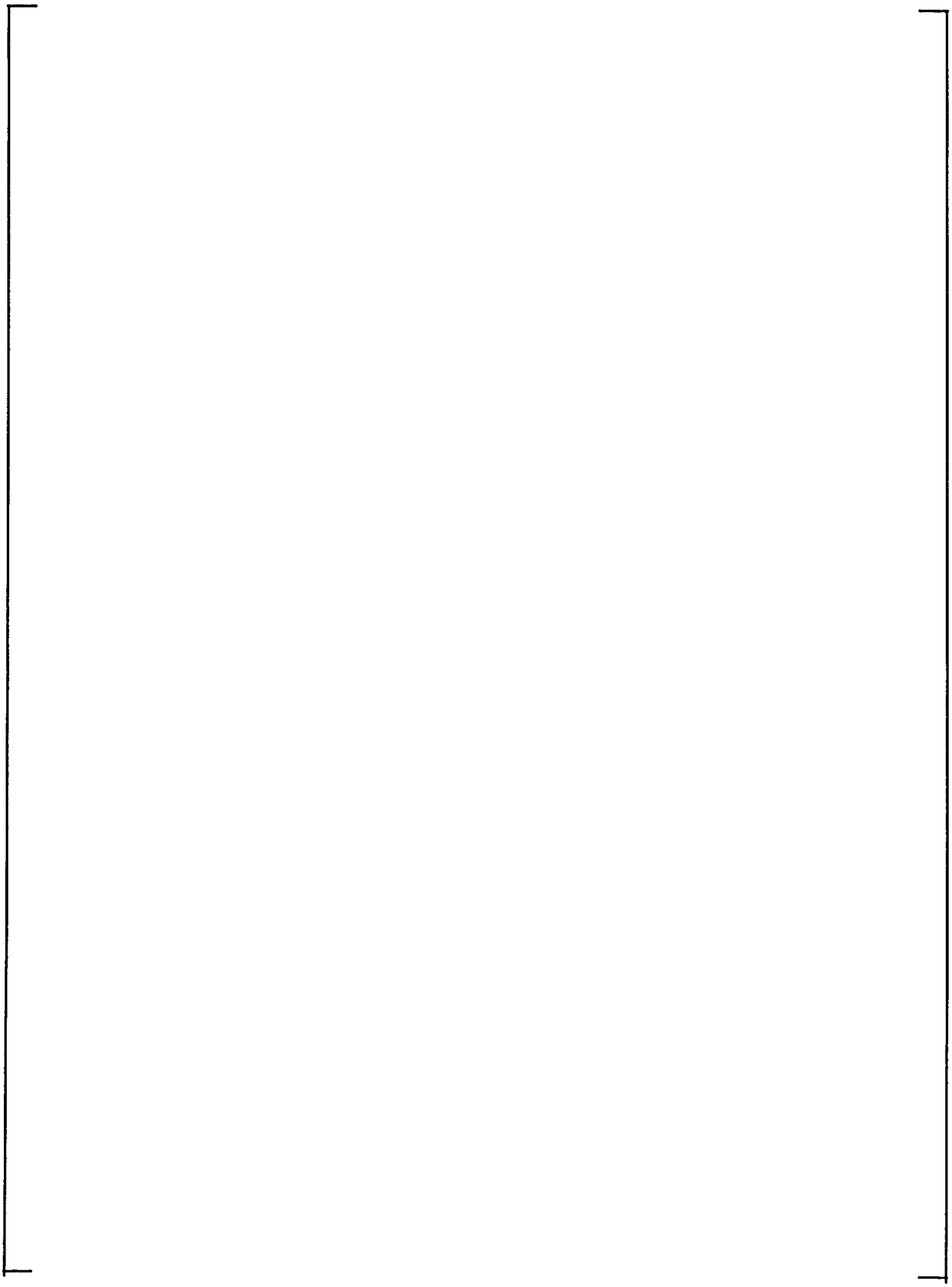
]

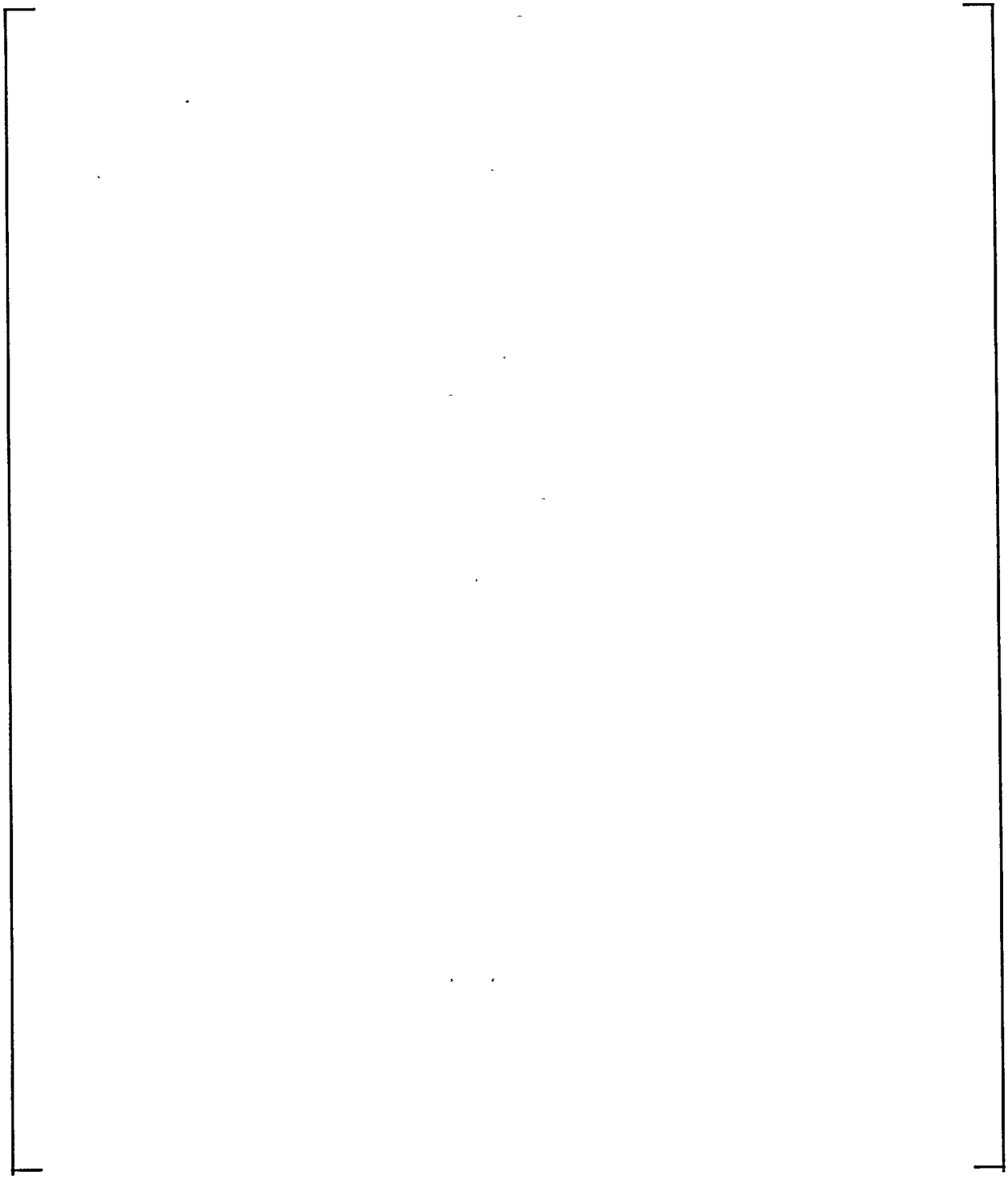
Solutes in Steam Generator Steam Flow

Iodine and noble gases leave the steam generator via the steam flow. For flow out of the steam generator, the concentration of the steam exiting the steam generator is the steam space concentration. For flow into the steam generator, the solute concentration is that of the steamline path (Section 7.3.5, Table 7.24).

5.8.3.3 Steam Generator Solute Balance

An explicit balance of the iodine inventory in the steam space and the liquid region is performed at each time step.





5.8.4 *Steamline Header Solute Flows*

The main steamlines are represented by paths connecting the steam generators to the header nodes. An iodine balance is performed for the header nodes at every time step based on the flows into the header and the current header mass and iodine inventory. The iodine concentration of the steamlines is simply the flow-weighted average of the flows into each steamline.

Steam can leave the steamline header node or the steamlines via any of four exit paths. Some properties of the exiting steam are stored in array MSLH_OUTFLOW(1:5,1:4). Here the first index, J, is the property number identifier:

J=1 => Steam flowrate
J=2 => Steam exit enthalpy
J=3 => Iodine concentration
J=4 => Particulate concentration
J=5 => Xenon concentration

and the second index, I, is the exit path:

I=1 => Atmosphere
I=2 => Condenser
I=3 => Containment
I=4 => Turbine

The iodine flows to each of the exit paths is integrated. The results are stored in four variables. These are:

MSLH_IO_INT_ATM
MSLH_IO_INT_COND
MSLH_IO_INT_CONT
MSLH_IO_INT_TURB

5.8.5 *Release From Containment to Atmosphere*

The flowrate from containment to atmosphere (μCi/sec) is calculated at each time step:

MSLH_IO_CONT_LEAK = (MSLH_IO_INT_CONT -
MSLH_IO_CONT_INT_LEAK) * 10^6 * RCS_DOSE_CONT_LEAK / (24.*3600.)

where:

MSLH_IO_CONT_LEAK = release rate from containment to atmosphere (μCi/sec)

MSLH_IO_INT_CONT = integrated release from header to the containment (Ci)

MSLH_IO_CONT_INT_LEAK = integrated iodine released from containment to atmosphere(Ci)

RCS_DOSE_CONT_LEAK = user input containment leak rate. (fraction / day).

At each time step the integrated leak from containment to atmosphere is updated as :

$$MSLH\_IO\_CONT\_INT\_LEAK(t+\Delta t) = MSLH\_IO\_CONT\_INT\_LEAK(t) + \Delta t * MSLH\_IO\_CONT\_LEAK * 10^6$$

5.8.6 *Total Iodine Release from Secondary Systems*

The sum of the iodine released through all paths is:

RCS_DOSE_TOT_CURIE = MSLH_IO_INT_ATM
+ MSLH_IO_CONT_INT_LEAK
+ MSLH_IO_INT_COND/RCS_DOSE_COND_DF
+ MSLH_IO_INT_TURB/RCS_DOSE_COND_DF

where:

RCS_DOSE_TOT_CURIE = total release of iodine through all sources which contribute to offsite dose.

MSLH_IO_INT_ATM = total iodine release from the main steam system directly to atmosphere. This component includes the contribution from the main steam safeties, the atmospheric dump valves and (possibly) bypass valves which discharge to atmosphere, as well as any contribution from an outside-containment steamline break.

MSLH_IO_CONT_INT_LEAK = total integrated iodine release from containment to atmosphere.

MSLH_IO_INT_COND = total integrated iodine release from the main steam system to the condenser. This component includes discharges from the bypass valves, which dump to the condenser. It does not include discharges through the turbine. The contribution of the iodine discharged to the condenser is reduced by the factor RCS_DOSE_COND_DF, which is the iodine decontamination factor to be applied to leakage through the condenser.

MSLH_IO_INT_TURB = total integrated release from the main steam system to the turbine. The contribution of the iodine discharged to the turbine is reduced by the factor RCS_DOSE_COND_DF, which is the same iodine decontamination factor which is applied to leakage directly to the condenser.

RCS_DOSE_COND_DF = iodine decontamination factor applied to discharges to the turbine and directly to the condenser.

5.8.7 *Global Checks of Secondary Iodine Inventory*

A global check is performed on the secondary side iodine inventory. At each time step, the total iodine inventory of the secondary side is calculated as:

$$\begin{aligned} SGS\_TOTAL\_IODINE = 10^{-6} \{ \sum_I \; & (SGS\_CON\_IO1_I * SGS\_M1_I + \\ & SGS\_CON\_IO2_I * SGS\_M2_I + \\ & SGS\_CON\_IO2_I * SGS\_M3_I + \\ & MSLH\_CON\_IO * MSLH\_M + \\ & MSLH2\_CON\_IO * MSLH2\_M \} \end{aligned}$$

where I is the steam generator number. The last term of the above equation is relevant only if the main steamline header is modeled with multiple nodes, NUM_MSLH = 2 (Section 5.6).

5.8.8 *Dose Calculation*

The CENTS dose calculation converts the total iodine release to a 2-hour and 8-hour offsite dose as follows:

For time ≤ 2 hours:

$$\begin{aligned} RCS\_DOSE\_2HR = & RCS\_DOSE\_TOT\_CURIE * RCS\_DOSE\_CF * \\ & RCS\_DOSE\_BF * RCS\_DOSE\_XOQ2 \end{aligned}$$

For time ≤ 8 hours:

$$\begin{aligned} RCS\_DOSE\_8HR = & RCS\_DOSE\_TOT\_CURIE * RCS\_DOSE\_CF * \\ & RCS\_DOSE\_BF * RCS\_DOSE\_XOQ8 \end{aligned}$$

where:

RCS_DOSE_2HR = "two hour" dose. Iodine releases after 2 hours of simulation time do not contribute to this dose.

RCS_DOSE_8HR = "Eight hour" dose. Iodine releases after 8 hours of simulation time do not contribute to this dose.

RCS_DOSE_TOT_CURIE = total release of iodine through all sources which contribute to offsite dose (Ci).

RCS_DOSE_CF = iodine conversion factor. This factor converts the iodine release from units of curies to units of REM.

RCS_DOSE_BF = effective breathing rate factor (ft^3/sec).

RCS_DOSE_XOQ8 = site dispersion factor (sec/ft^3) which accounts for dispersion of the iodine cloud as it moves from the release point to the site boundary, during the period 0-8 hours after the accident.

RCS_DOSE_XOQ2 = site dispersion factor (sec/ft^3) which accounts for dispersion of the iodine cloud as it moves from the release point to the site boundary, during the period 0-2 hours after the accident.

The information calculated by CENTS as decribed above supports the analysis of pre-existing or generated iodine spike (GIS) scenarios. In a post-analysis calculation, the user can readily determine thyroid and whole-body doses.

5.9 References

5.1 Wilson, J. F., et al., "Primary Separation of Steam from Water by Natural Separation", Joint US/EURATOM R&D Program at "11-1"-1186, Allis-Chalmers Topical Report, ACNP-65002, April 15, 1965.

5.2 Nahavandi, A. N., and M. Rashevsky, "CRITCO Code", CVNA-128, February, 1962.

5.3 Moody, F. J., "Maximum Discharge Rate of Liquid-Vapor Mixtures from Vessels," ASME Winter Annual Meeting, November, 1975.

5.4 R. E. Henry and H. K. Fauske, "The Two-Phase Critical Flow of One-Component Mixture in Nozzles, Orifices and Short Tubes," Journal of Heat Transfer, May 1971.

5.5 Dittus, F. W. and Boelter, L. M. K., "Heat Transfer in Automobile Radiators of the Tubular Type", Pub. in Eng. Vol. 2, n. 13, U. of Calif., 1930.

5.6 Akers, W. W., Deans, H. A. and Crosser, O. K., "Condensing Heat Transfer within Horizontal Tubes", Proc. 2nd Nat. Heat Transfer Conf., ASME/AIChE, August, 1958.

5.7 Thom, J. R. S., et al., "Boiling in Subcooled Water During Flow Up Heated Tubes or Annuli", Proc. Instn. Mech. Engrs., Vol. 180, Pt. 3c, 1965-66.

5.8 Rohsenow, W. M., "A Method of Correlating Heat Transfer Data for Surface Boiling of Liquids," Trans. ASME, Vol. 74, 1952.

5.9 McAdams, W. H., Heat Transmission, 3rd ed., McGraw-Hill Book Col, New York, 1954.

5.10 "Flow of Fluids Through Valves, Fittings, and Pipe," Crane Technical Paper No. 410.

5.11 Martinelli, R. C. and Nelson, D. B., "Prediction of Pressure Drop During Forced Circulation Boiling of Water," Transactions of the ASME, August, 1948.