ENHANCEMENTS TO THE MODULAR MODELING SYSTEM TO BUILD A DYNAMIC SIMULATION OF THE NEW PRODUCTION REACTOR

by

Norman S. Yee and John R. Burris

April 1991

B&W Nuclear Service Company
3315 Old Forest Road
P.O. Box 10935
Lynchburg, VA 24506-0935

ENHANCEMENTS TO THE MODULAR MODELING SYSTEM TO BUILD A DYNAMIC SIMULATION OF THE NEW PRODUCTION REACTOR

ABSTRACT

The New Production Reactor is a 2500-MWt Heavy Water Reactor Facility (HWRF) being designed by B&W Nuclear Technologies as a team member with Ebasco Services to provide the production capacity for supplying tritium and other nuclear materials. The Modular Modeling System (MMS) methodology was selected to build a dynamic simulation because it has been demonstrated to be versatile and flexible in simulations for both nuclear and fossil power plants.

The dynamic simulation of the HWRF requires modeling the flow of subcooled light and heavy water for conditions including full flow, low flow, reverse flow, and natural circulation. New modules are being developed to more economically compute the pressure and flow distribution for this range of flow conditions by using the approximation that the subcooled water is incompressible, but thermally expandable. These new modules require the solution of simultaneous algebraic equations to calculate the pressure and flow distribution. The mathematics of the solution method is straightforward, but the programming of the method is complicated by the MMS requirements for modularity and compatibility with variable time-step integration algorithms.

Development of enough component modules has been completed to simulate multiple flow loops involving a reactor, heat exchangers, pumps, and piping. Testing has demonstrated that the algebraic solution method has been programmed correctly, satisfies the requirement for modularity, and is compatible with variable time-step integration algorithms. Supplemental coding written to perform initialization achieves a solid steady state beginning at time zero. When the simulated system is manually perturbed, the time-step size is manually reduced momentarily and then the integration algorithm automatically increases the time-step size as the transient effects of the perturbation dissipate.

Development and testing of the dynamic simulation are ongoing. Testing completed thus far indicates that reverse flow and natural circulation are properly simulated. Detailed comparisons of input data and output results with other simulations or experimental data are in progress. Modules are being developed to simulate additional components such as air-water heat exchangers, cooling towers, pump motors, control rod drives, and various transducer and control components.

ENHANCEMENTS TO THE MODULAR MODELING SYSTEM TO BUILD A DYNAMIC SIMULATION OF THE NEW PRODUCTION REACTOR

INTRODUCTION

The New Production Reactor (NPR) is a 2500-MWt Heavy Water Reactor Facility (HWRF) being designed by B&W Nuclear Technologies as a team member with Ebasco Services to provide the production capacity for supplying tritium and other nuclear materials. A dynamic simulation of the HWRF is being developed to support component and control system design. This simulation may eventually become the basis for a plant simulator.

As the design of the HWRF evolves, different configurations are expected to be conceived and require evaluation. Therefore it is important for the simulation methodology to be adaptable. The Modular Modeling System (MMS) methodology was chosen because it has been demonstrated to be versatile and flexible in simulations for both nuclear and fossil power plants.

The dynamic simulation of the HWRF requires modeling the flow of subcooled light and heavy water for conditions including full flow, low flow, reverse flow, and natural circulation. New modules are being developed to more economically compute the pressure and flow distribution for this range of flow conditions by using the approximation that the subcooled water is incompressible, but thermally expandable. These new modules require the solution of simultaneous algebraic equations to calculate the pressure and flow distribution. The mathematics of the solution method is straightforward, but the programming of the method is complicated by the MMS requirements for modularity and compatibility with variable time-step integration algorithms.

This paper describes the HWRF plant model, the MMS methodology, and the solution method used for incompressible, thermally expandable flow.

OVERVIEW OF THE HWRF PLANT MODEL

This overview presents an abbreviated description of the HWRF, describing only those components and systems selected for simulation.

The HWRF uses a 2500-MWt low-temperature heavy-water moderated and cooled reactor with a fuel/target system similar to the existing Savannah River Site reactor.

Reactor Core

The reactor core contains hundreds of vertical fuel/target assemblies. Each fuel assembly contains an inner target tube, three fuel tubes, an outer target tube, and the flow tube. Coolant flows between each tube and around the flow tube. Each flow tube is a right circular cylinder open at each end, separating the coolant and moderator regions from the bottom plenum to the upper plenum of the reactor pressure vessel. Ribs are located on the flow tube, and on the outside of the inner fuel tube and target tube to maintain the proper spacing between the tubes. The channels

between the flow tube and the outer target and the inside of the inner target are orificed for minimum flow because little heat is transferred to those channels.

Cooling Water Loops

The schematic in Figure 1 illustrates the major cooling water systems that removes heat from the reactor core. The HWRF consists of a 2500-MWt reactor cooled by six primary loops (making up the Reactor Coolant System or RCS). Each primary loop transfers heat to a closed secondary loop (Secondary Coolant System or SCS), which in turn is cooled by an open tertiary loop (Circulating Water System or CWS) containing a Cooling Tower System (CTS) as the ultimate heat sink.

In addition to the six primary loops, four Residual Heat Removal (or RHR) loops can also remove heat from the reactor. The RHR primary loops are cooled by a closed secondary loop (Component Cooling Water System or CCWS) and an open tertiary loop (Essential Services Cooling Water System or ESCWS) which uses a large cooling basin as the ultimate heat sink. The RHR System provides the active long-term capability to transfer reactor decay heat to the ultimate heat sink under all modes of operation.

Each loop contains pumps, heat exchangers, and piping. Piping contains sensors for temperature, flow, and pressure measurements. Heavy water flows in the RCS and RHR loops, light water in the other loops. Gas pressurizers control pressure in the closed loops.

The RCS is arranged above the reactor core to provide for natural circulation of the reactor coolant to remove core decay heat should power to the reactor coolant pumps be lost. The SCS contains water-to-air heat exchangers capable of removing decay heat from the RCS by natural circulation.

THE MMS METHODOLOGY

During the design process, alternative configurations of the HWRF are likely to be conceived and require evaluation. To accommodate the simulation of these alternative configurations, the dynamic simulation is programmed using a methodology that has been used successfully in the Modular Modeling System (MMS). The MMS is a software system that eases the development of dynamic simulations by providing pre-engineered software modules that can be interconnected like plant components to simulate plant systems and subsystems. The central feature of the MMS methodology, illustrated in Figure 2, is that the models of individual components are prepared in the form of software modules that can be selected and interconnected in as components are interconnected in the plant. This modularity provides greater flexibility than a simulation with a fixed configuration.

The increased flexibility provided by the MMS methodology allows the simulation to accommodate the inevitable changes in plant configuration that occur during the design process. When particular subsystems can be identified to be less important, MMS methodology allows the model for that subsystem to be eliminated or replaced with a simpler model to produce a faster running simulation. On the other hand, when additional subsystems are required to be modeled, the MMS methodology readily supports extension of the simulation to include additional subsystems. Adhering to conventions to achieve modularity increases programming difficulty, but the increased modeling flexibility compensates for the additional programming effort.

Simulation Language

The high-level simulation language used in the MMS is ACSL (Advanced Continuous Simulation Language). A simulation language provides features to facilitate the development of simulations. Features especially important to the development of the MMS are

- Macro capability
- Automatic sorting of modeling equations
- Integration algorithms

The most important feature of ACSL with respect to modularity is its capability to define and use macros. MMS modules are ACSL macros following MMS naming conventions. An MMS module expresses the modeling equations for a single type of component. Since a simulation may use the same type of component several times, all modules must follow naming conventions to avoid name conflicts and to provide communication between modules. The macro expansion features of ACSL generate variable names by performing text substitution using arguments passed to each module by the user.

The name of each variable consists of a base name and a suffix. The suffixes are arguments passed to the macro when it is invoked. One suffix, called the "Module ID," is different for each macro invocation. The module ID is appended to base names to generate unique variable names for each macro, thus avoiding name conflicts.

Often one module requires a variable that is calculated by another separate module. To provide communication between these modules, other suffixes, called "Stream IDs," are used to form variable names in common among communicating modules. In MMS a stream ID is associated with each stream connecting two modules. When a name is to be common between two modules, each module uses the same stream ID. For example the outlet enthalpy calculated by one module is the inlet enthalpy needed by another module. Each module must use "H" as the base name for enthalpy. The ID used by the inlet stream of one module is specified to be the same as the ID used by the outlet stream of the other module.

The automatic sorting feature of ACSL arranges the equations introduced by each module so that the equation defining any given variable is evaluated before any equations using that variable. Thus, the user may interconnect the modules without concern for the order in which the macros are invoked.

The integration algorithms offered by ACSL relieve the analyst from having to code a numerical solution method to integrate the differential modeling equations. The analyst may choose from the seven integration algorithms offered by ACSL or he may include coding to define an algorithm of his own. The algorithm recommended for MMS is the Gear algorithm, a variable order, variable time-step size method that is both efficient and suitable for the stiff equations that often occur in power plant modeling.

Building and Using a Simulation

The major steps in building a simulation using the MMS methodology are:

- Select modules
- Prepare a block diagram interconnecting the selected modules
- Assign IDs to each block and each interconnecting "stream"
- Collect physical and operating point data for each module
- Calculate parameters with the guidance of parameterization instructions and complete the worksheets for each module

- Enter the ID assignments and the component data into an ACSL model file
- Process the model file through translation, compilation, linking, and loading to produce an executable program.

After generating the executable program by the above steps, ACSL runtime commands allow the analyst to start, stop, and continue a simulation. The runtime commands allow displaying, printing, plotting, and redefining almost any variable in the simulation. A long simulation can be interrupted and its state variables be saved for later restart and for the study of different scenarios.

DESCRIPTION OF THE PRSIM SIMULATION SOFTWARE

PRSIM (PRoduction Reactor SIMulation) software is made up of a library of ACSL macros and supporting FORTRAN subroutines that model the HWRF using the MMS methodology. The PRSIM simulation software does not represent any particular plant configuration, but is a collection of building blocks whose interconnections are specified by user input. Table 1 summarizes the major features simulated by PRSIM.

The dynamic simulation of the HWRF requires modeling the flow of subcooled light and heavy water for conditions including full flow, low flow, reverse flow, and natural circulation. The modules being developed for PRSIM computes the pressure and flow distribution for this range of flow conditions by using the approximation that the subcooled water is incompressible, but thermally expandable. These new modules are different from currently available MMS modules because they solve simultaneous algebraic as well as differential equations to calculate the pressure and flow distribution. The mathematics of the solution method is straightforward but the programming of the method is complicated by the requirement for modularity and compatibility with variable time-step integration algorithms.

Overview of the Modeling Equations

The state of a one-dimensional single-phase single-component thermal-hydraulic flow system is known, when the pressures, flows, and enthalpies are known everywhere. The mass and energy conservation equations for each control volume together with the state equations, the heat transfer equations, and the pressure loss equations are solved simultaneously to determine the pressures, flows, and enthalpies everywhere. Some of those equations are ordinary differential equations and some are algebraic equations.

The ACSL simulation language provides integration algorithms to collect, sort, and solve simultaneous differential equations. ACSL does not perform those actions for algebraic equations. However, by using the ACSL features to control the sorting, (and at the expense of complicating the programming) it was possible to develop for PRSIM a method to collect and solve the algebraic equations used by PRSIM. Simultaneous non-linear algebraic equations (also known as algebraic loops) in a simulation are difficult to solve generally. For the equations it uses, PRSIM has been able to find solutions according to testing performed thus far.

Equations for Enthalpies and Temperatures

Given flows into and out of each control volume and given the pressures within each control volume and each metal node, enthalpies in each control volume and the temperatures in each metal node can be found by integrating the equations derived below.

The equation used to find the average enthalpy of water in each control volume is a differential equation derived from the conservation of mass and energy for one dimensional flow in and out of a fixed control volume. The assumption of perfect mixing within a control volume means that the exit enthalpy is taken to be equal to the average enthalpy.

The heat transfer equations calculate the rate of heat flow into a control volume as an algebraic function of the temperatures of the fluid and of the adjacent metal, which is divided into nodes in the same way as the fluid is divided into control volumes. The temperature of water is in general a function of the pressure and the enthalpy of the fluid, though the dependence on pressure for subcooled water is weak and can often be neglected. The temperature of water is found by using algebraic curve fits, different ones for light and heavy water.

The conservation of energy equation for each metal node is expressed in terms of temperatures and is the differential equation used to find the temperature of metal nodes given the rate of heat flow.

Conservation of Mass for Water. For a fixed control volume in a fluid system,

$$\frac{d\,\bar{\rho}}{dt} = \frac{(w_{in} - w_{out})}{V_t} \tag{1}$$

where

$$\bar{\rho} \doteq \frac{\int_V \rho\, dV}{V_t} \,. \tag{2}$$

Conservation of Energy for Water. For a fixed control volume in a fluid system,

$$\frac{d\,\overline{\rho u}}{dt} = \frac{w_{in} h_{in} - w_{out} h_{out}}{V_t} \tag{3}$$

where

$$\overline{\rho u} \doteq \frac{\int_V \rho u\, dV}{V_t} \,. \tag{4}$$

Rate of Change of Average Enthalpy and Exit Enthalpy for Water. Enthalpy is defined as follows:

$$h \doteq u + \frac{P}{\rho} \,. \tag{5}$$

For each control volume, it is assumed that the entering fluid is perfectly mixed with the fluid in the control volume. This assumption leads to the following equation for average enthalpy.

$$\frac{d\,\bar{h}}{dt} = \frac{w_{in}\left(h_{in} - \bar{h}\right) + q}{\bar{\rho} V_t} + \frac{1}{\bar{\rho}} \frac{d\,P}{dt} \tag{6}$$

where

$$\bar{h} = \frac{\int_V h\, dV}{V_t} \,. \tag{7}$$

The perfect mixing assumption implies that the enthalpy of the fluid leaving the control volume is equal to the average enthalpy:

$$h_{out} = \bar{h} \,. \tag{8}$$

Heat Transfer Equations. Heat flow q from a metal node (e.g., fuel) with temperature T_m into a control volume with average temperature T is calculated using the following equation:

$$q = UA \cdot (T_m - T) \tag{9}$$

where the overall heat transfer conductance UA is given by:

$$UA = \frac{1}{k_1 + \frac{1}{|W|^{0.8} k_2}} \quad . \tag{10}$$

The heat transfer conductance UA represents heat transfer through a fixed thermal resistance in series with a conductance proportional to fluid mass flow rate raised to the 0.8 power. The constant k_1 represents the fixed thermal resistance and includes the heat transfer surface area. The constant k_2 is the proportionality constant for the mass flow dependent thermal conductance.

Conservation of Energy for Metal. For metal nodes, the conservation of energy equation is expressed in terms of the metal temperature. The term q_w represents heat flow from the metal to surrounding fluid and is defined to be positive for flow out of the metal node. Metal nodes usually separate two fluids. Heat transfer from each fluid would then contribute to q, and q would be the sum of those contributions. The term q_s represents a heat source internal to the node. In fuel for example, fission and decay heat contribute to q_s,

$$\frac{d T_m}{dt} = \frac{-q_w + q_s}{(Mc_p)_m} \quad . \tag{11}$$

Nomenclature. In the above equations, a bar over a variable is used to indicate volume-averaged quantities. The distinction was made to highlight the difference between the value of a property at a particular point in space and the value averaged over a volume. The modeling equations used by PRSIM assume perfect mixing within a control volume, so the point value and volume average value are the same. In the remainder of this paper, the bar will be omitted and no distinction will be made between point values and volume averages.

Symbol	Description
h	Specific enthalpy, (J/kg)
$\bar{h}$	Volume-averaged specific enthalpy, (J/kg)
h_{in}, h_{out}	Specific enthalpy of water flowing in (out), (J/kg)
k_1	Proportionality constant for flow independent thermal resistance, (K/W)
k_2	Proportionality constant for flow-dependent thermal conductance, $(W/(K\text{-}(kg/s)^{0.8}))$
$(Mc_p)_m$	Thermal capacity of a metal node, (J/K)
P	Pressure, (Pa)
P_{in}, P_{out}	Pressure at inlet (outlet) port, (Pa)
T_m	Metal temperature, (C)
q_w	Rate of heat addition to water, (W)
q_s	Rate of heat generation within a metal node, (W)
u	Specific internal energy, (J/kg)
UA	Heat conductance, (W/K)
V_t	Total volume, (m^3)
w_{in}, w_{out}	Mass flow rate of water into (out of) control volume, (kg/s)
$\int_v dV$	Volume integral, (m^3)

ρ	Density, (kg/m^3)
$\overline{\rho}$	Volume-averaged density, (kg/m^3)
$\overline{\rho u}$	Volume-averaged energy per unit volume, (J/m^3)

Pressure and Flow Calculations

All modules except the module for the reactor allow both forward and reverse flow. The pressure and flow distributions are found by solving a set of simultaneous differential and algebraic equations relating pressure and flow.

The hydraulic system is modeled as a network of nodes and branches. Branches are series connections of pipe modules. The pipe modules are characterized by terms for flow resistance, elevation pressure rise, flow inertia, compressibility, and thermal surge flow. Each pipe module works with neighboring pipe modules to calculate branch-wide equivalent terms for flow resistance, elevation pressure rise, etc.

Nodes correspond to locations where the pressure is given, such as the boundaries of an open system or the pressurizer of a closed system. Nodes also correspond to locations where the flow splits or joins. At those locations the pressure is an unknown to be found by solving the pressure-flow equations (the equations relating pressure and flow to each other). The branch-wide equivalent terms collected by the pipe modules are used by the node modules to form the pressure-flow equations.

Pipe modules. In this discussion the term "pipe module" refers to any module that has one inlet, one outlet, and constant volume. Pipe modules naturally include modules which model sections of pipes, but also include all modules which model a component as having only one flow channel with one inlet and one outlet. The user characterizes a pipe module by specifying volume, elevation rise, flow length/area ratio, and flow conductance which is defined as follows:

$$k_{fc} = \frac{|w|}{\sqrt{\rho_{avg}\,|\Delta P - \Delta Z\,\rho_{avg}\,g\,|}}\;sign(\Delta P - \Delta Z\,\rho_{avg}\,g) \qquad (12)$$

where

k_{fc}	flow conductance, (m^2)
w	mass flow rate, positive for flow from the inlet to the outlet, (kg/s)
ρ_{avg}	average density, (kg/m^3)
g	acceleration of gravity, (m/s^2)
ΔP	pressure drop (inlet pressure) - (outlet pressure), (Pa)
ΔZ	elevation rise (outlet elevation) - (inlet elevation), (m)

and where sign() is a function defined as follows:

$$sign(x) = -1.0 \quad \text{if } x < 0.0$$
$$= 1.0 \quad \text{if } x \geq 0.0\ .$$

The pipe modules calculate the following additional variables, which are used to model series connections of pipes.

Flow resistance:

$$k_{fr} \doteq \frac{1}{k_{fc}^2\,\rho_{avg}} \qquad (13)$$

Elevation pressure rise:

$$\Delta P_e = -\Delta Z\,\rho_{avg}\,g \qquad (14)$$

Flow inertia:

k_{la} = (length of flow channel) / (cross-sectional flow area) (15)

Compressibility:

$$\frac{dM}{dP} = \frac{\partial \rho}{\partial P} V \tag{16}$$

Thermal surge flow:

$$\delta w = \frac{\partial \rho}{\partial h} \frac{dh}{dt} V \ . \tag{17}$$

The flow-resistance term accounts for pressure changes that are proportional to the square of mass flow rate, including recoverable as well as non-recoverable pressure changes. Because recoverable pressure changes are also included, the flow resistance term may be negative in cases where the flow area expands. The elevation pressure rise term accounts for pressure changes that are independent of flow rate such as the pressure changes caused by changes in elevation. The flow inertia accounts for the inertia of the fluid when the pressure changes. The compressibility accounts for the changes of mass in the volume of the pipe component. The thermal surge flow accounts for the mass flow out of the volume as the mass within the volume expands or contracts with temperature changes.

Branches. A branch is a series connection of pipe modules, where the outlet of one module is connected to the inlet of the next. The equations in the previous section show how flow resistance, elevation pressure rise, compressibility, and thermal surge in individual pipe module are calculated using the average fluid properties in each module. The equivalent variables for a branch are calculated by summing the values for each pipe module in the branch as follows:

$$k_{eq} = \sum_i k_{fr,i} \tag{18}$$

$$\Delta P_{e,eq} = \sum_i \Delta P_{e,i} \tag{19}$$

$$k_{la,eq} = \sum_i k_{la,i} \tag{20}$$

$$\left[\frac{dM}{dP}\right]_{eq} = \sum_i \left[\frac{dM}{dP}\right]_{e,i} \tag{21}$$

$$(\delta w)_{eq} = \sum_i (\delta w)_{e,i} \tag{22}$$

where the subscripts are used as follows:

- $_{eq}$ equivalent value for all the pipe modules connected in series in the same branch
- $_{i}$ value for individual pipe modules in the same branch.

A branch does not exist as a separate module. The calculation of branch-wide equivalent terms are performed within the pipe modules and relies on the sorting that is performed by the ACSL translator.

Flow Through a Branch. PRSIM considers the flow into and out of a branch to have two components: (1) flow caused by the pressure difference across the branch and (2) surge flow caused by the expansion or contraction of the fluid in the branch.

The flow caused by the pressure difference is found by solving the following differential equation:

$$\frac{dw}{dt} = \frac{1}{k_{la,eq}}\left(p_i - p_o - k_{fr,eq}w^2 + \Delta P_{e,eq}\right) . \quad (23)$$

The surge component is associated with changes in density of the fluid in the branch and is calculated by the following equation:

$$w_{surge} = (\delta w)_{eq} - \left(\frac{dM}{dP}\right)_{eq} \frac{dP}{dt} . \quad (24)$$

The first term on the right hand side of the above equation is the surge caused by the effect of pressure changes on the density. The second term is the surge caused by the effect of temperature changes on the density.

PRSIM assumes half of the surge flow occurs at each end of the branch. That means that the flow at the outlet of the branch is

$$w_{out} = w + \frac{w_{surge}}{2} \quad (25)$$

and the flow at the inlet of the branch is

$$w_{in} = w - \frac{w_{surge}}{2} . \quad (26)$$

Nodes. Nodes are points with no volume, at which the pressure is either given or is to be found by solving the pressure-flow equations. For each node where the pressure is to be found, the continuity equation is written in terms of the pressures at that node and at the neighboring nodes. By collecting and solving the resulting system of equations for the unknown pressures, the pressure will be known for all nodes and then the flow can be found for every branch.

The end of an open pipe, which is modeled as a pressure source or sink, is an example of a node whose pressure is specified. A pipe tee, where flow from several branches combine and divide, is an example of a node where the pressure is unknown. The module for the pipe tee uses the continuity equation to formulate a pressure-flow equation to be used to solve for the unknown pressure.

Summary of Solution Techniques

A discussion of the programming methods used in PRSIM to implement the algebraic solver requires a detailed understanding of the sorting algorithm used by the ACSL translator and is too long and specialized to be included in this paper. Brief summaries are given here of the techniques used by PRSIM to deal with anticipated and unanticipated programming and mathematical difficulties.

To allow for reverse flow, average enthalpies are passed along both the outlet and inlet streams of each module. The enthalpy of the volumes on either side of each control volume is available. The energy equation is coded to use the appropriate donor enthalpy depending on the direction of flow.

For convenience and uniformity, average density and temperature are passed with enthalpy along all inlet and outlet streams.

To track the density during natural circulation, many modules allow the specification of a variable number of subvolumes. A pipeline module uses a "bucket brigade" to model a transport delay and avoid introducing additional differential equations.

Supplementary coding was developed to parameterize and initialize each module. When given consistent input data, the initialization procedure produces a solid steady state at time zero without running a null transient or trimming.

While the fits for the water properties currently used contain no dependence on pressure, the pressure-flow equations in PRSIM are formulated to include terms to model a pressure dependence, should different fits be used in the future.

The algebraic solver uses the Newton-Raphson method to solve simultaneous non-linear equations. The Gauss-Jordan elimination method using the maximal pivot strategy was used to solve the set of linear equations produced by the Newton-Raphson method.

To be compatible with variable time-step integration algorithms and avoid non-repeatable derivative evaluations, initial pressure and flow guesses for the algebraic solver had to be saved for restoration at the beginning of every derivative evaluation.

The modeling of surge flow together with flow inertia complicates the flow calculations. Surge flows refer to the impact on the flow distribution caused by density changes in control volumes. Density changes cause small but rapid changes in flow to redistribute the mass. To look at it another way, density changes cause large but short-lived pressure changes. The problem is to calculate to what extent the flow in each branch is affected by changes in density in any control volume. The approach used was to represent pressures with two components: (1) a component that varies slowly when compared with the time step size and (2) a component consisting that decays off rapidly when compared with the time step size.

The rapidly decaying component of pressure is made up of impulses of short duration which can be treated like Dirac delta functions. PRSIM uses a separate pressure-flow equation to relate the surge flow with the rapidly decaying pressure component. This approach leads to the result that surge flows are distributed along the paths of least flow inertia. This result is reasonable since the surge flows are rapid flow changes that would be most impeded by flow inertia.

CONCLUSIONS

Development of enough component modules has been completed to simulate multiple flow loops involving a reactor, heat exchangers, pumps, and piping. Testing has demonstrated that the algebraic solution method has been programmed correctly, satisfies the requirement for modularity, and is compatible with variable time-step integration algorithms. Supplemental coding written to perform initialization achieves a solid steady state beginning at time zero. When the simulated system is manually perturbed, the time-step size is manually reduced momentarily and then the integration algorithm automatically increases the time-step size as the transient effects of the perturbation dissipate.

Development and testing of PRSIM are ongoing. Testing completed thus far indicates that reverse flow and natural circulation are properly simulated. Detailed comparisons of input data and output results with other simulations or experimental data are in progress. Modules are being developed to simulate additional components such as air-water heat exchangers, cooling towers, pump motors, control rod drives, and various transducer and control components.

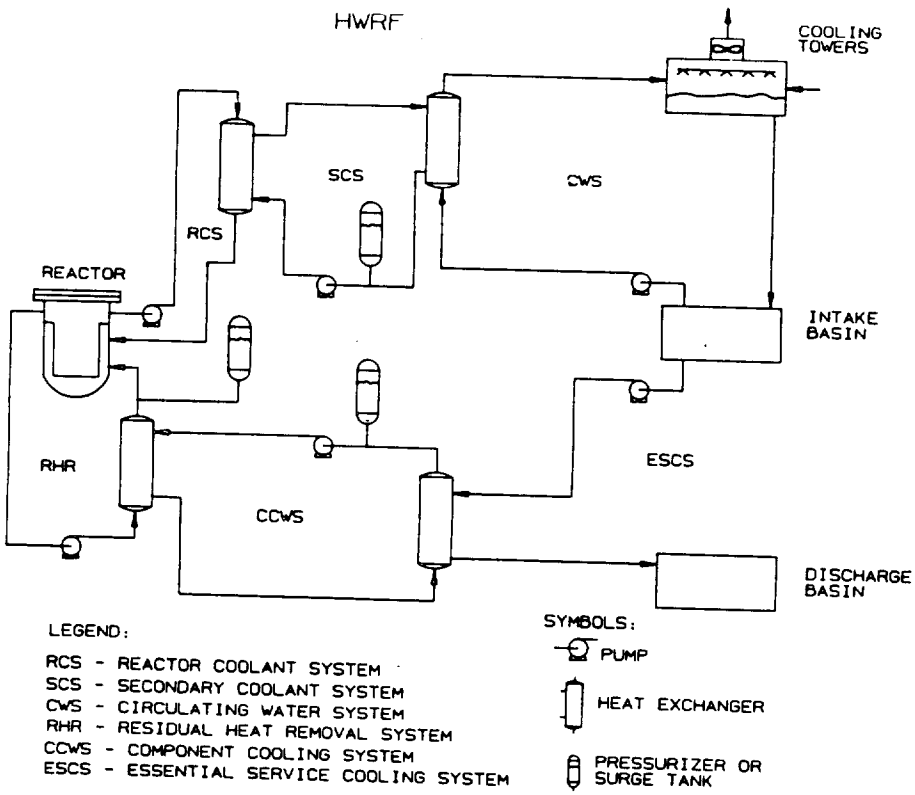

Figure 1. HWRF Cooling Water Loops

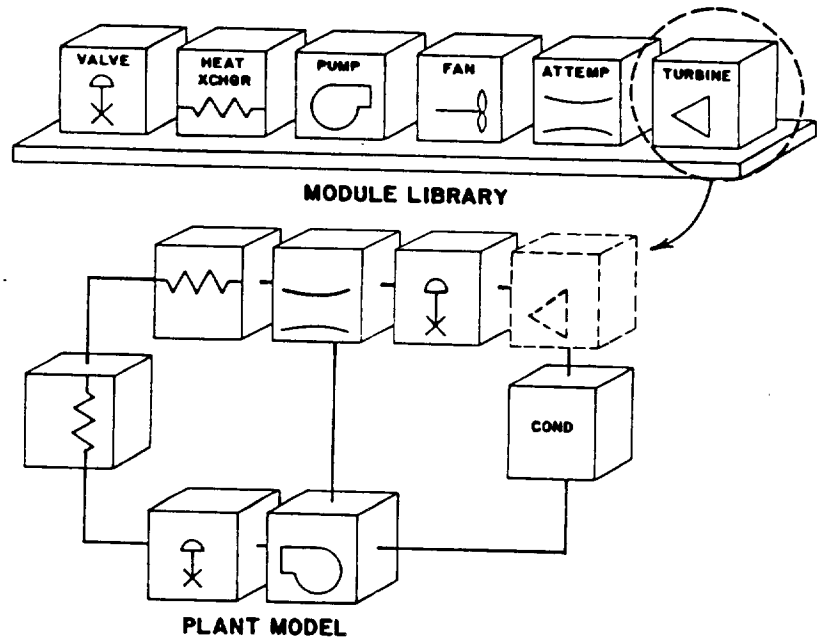

Figure 2. The Modular Modeling Concept

Table 1

MAJOR FEATURES SIMULATED BY PRSIM

Plant Simulation

- Reactor Coolant System -- Use two loops to represent combinations of the six loops
- Residual Heat Removal System -- Use two loops to represent combinations of the four loops
- Reverse flow allowed in all components except in reactor
- Natural circulation
- Single-phase flow
- Range of operation 100% to 1% power
- Operational transients and anticipated operational occurrences
- Reactor protection systems
- Instrumentation and control system
- Secondary Coolant System
- Circulating Water System
- Cooling Tower System
- Light and heavy water properties based on fits developed at Savannah River Laboratory, giving density and enthalpy of subcooled liquid as a function of temperature.

Reactor Model

- Point reactor kinetics
- Prompt jump approximation
- N groups of delayed neutron, including a gamma-D_2O delayed neutron source
- Three group representation for decay heat
- Reactivity terms
  --Safety rods vs. time following trip
  --Control rods
  --Temperature feedback coefficients for fuel, target, coolant, and moderator.
- Model core using an "average" fuel assembly multiplied by a user specified factor to represent any number of fuel elements operating under identical conditions.
- Each fuel assembly module models a user specified number of fuel/target/housing annuli with coolant/moderator annular flow channels around and within each fuel/target/housing annulus. The inclusion of center coolant channel will be selectable by the user.
- Energy storage is modeled with one axial node and one radial heat transfer through fuel modeled with one radial node per annulus and one node per coolant channel.
- Heat transfer to fluid modeled as a fixed thermal resistance in series with a film conductance proportional to coolant flow rate to the 0.8 power.
- Allow for heat generation in moderator and in target
- Hydraulics - A separate flow resistance is associated with each coolant channel.

Table 1 (CONTINUED)

MAJOR FEATURES SIMULATED BY PRSIM

Gas Pressurizer Model

- Calculation of thermal surge flow
- Heat transfer between liquid, gas, walls, and ambient
- Feed and bleed of liquid
- Feed and bleed of gas
- Use ideal gas relations

Piping Model

- Transport delay
- Reverse flow allowed

Pump Model

- Four-quadrant curves
- Reverse flow allowed
- Coastdown and locked rotor

Heat Exchanger Model

- Single-phase fluid conditions
- Heavy or light water on tube side
- Heavy or light water on shell side
- Reverse flow
- User selectable number of axial nodes