Conf-821101--21

SASSYS ANALYSIS OF DEGRADED SHUT-DOWN HEAT-REMOVAL PERFORMANCE IN LMFBRs

Floyd E. Dunn and Frederick G. Prohammer
Argonne National Laboratory[1]
Reactor Analysis and Safety Division
Argonne, Illinois 60439

CONF-821101--21

DE83 008874

Center within this bracketed area on page 1: Title, Author, and Affiliation

Start page 1, beginning with abstract

ABSTRACT

The SASSYS LMFBR systems analysis code was developed to analyze the consequences of failures in the shutdown heat removal system and to determine whether this system can perform its mission adequately even with some of its components inoperable. SASSYS provides a detailed thermal-hydraulic analysis of the reactor core, inlet and outlet coolant plenums, primary and intermediate heat transport systems, steam generators, and emergency heat removal systems for any LMFBR design. One key feature of the code is the sodium boiling model, which can be especially significant in cases where pump power is lost and normal natural circulation heads are insufficient to prevent temporary flow stagnation in part or all of the core. In such cases, boiling in part of the core should provide the driving head to re-establish flow, while at the same time removing enough heat to prevent melting of fuel and clad.

NOMENCLATURE

a_0, a_1, a_2	coefficents in the linearized equation for liquid flow rate changes
A_k	coolant flow area in element k
b_0, b_1, b_2	coefficients in the linearized equation for changes in compressible volume pressure
C_0, C_1, C_2, C_3	coefficients used to estimate core channel flows
c_{ji}	coefficients in the matrix equation for changes in compressible volume pressures
CRBR	Clinch River Breeder Reactor
d_j	coefficients in the matrix equation for changes in compressible volume pressures
DNB	departure from nucleate boiling
EBR-II	Experimental Breeder Reactor-II
FFTF	Fast Flux Test Facility
i	liquid segment number
ic	core channel number
j	compressible volume number
ji	compressible volume at the inlet of a liquid segment
jo	compressible volume at the outlet of a liquid segment
k	liquid element numbers
L	set to 1 for the inlet to a channel, 2 for the outlet from a channel
LMFBR	liquid metal cooled fast breeder reactor
L_1	length of the sub-cooled region in a steam generator
L_2	position where DNB occurs
L_3	position where the superheated region starts
m	number of heat transfer nodes in a pipe
p	pressure
p_{in}	inlet pressure
p_{out}	outlet pressure
t	time
T_{in}	inlet temperature
T_{out}	outlet temperature
W	coolant flow rate
W_c	channel coolant flow rate
$\overline{W}_{in}$	inlet flow rate, averaged over a time step
$\overline{W}_{out}$	outlet flow rate, averaged over a time step
Δp	pressure change for a time step
Δp_{fr}	friction pressure drop
Δp_{w2}	pressure drop due to orifices or other pressure drops proportional to W^2
Δp_v	pressure drop in valves
Δp_{gr}	gravity head
Δp_p	pump head

INTRODUCTION

The SASSYS code is the latest in a series of LMFBR accident analysis codes which includes SAS1A (1), SAS2A (2), SAS3A (3), SAS3D (4), and SAS4A (5).

[1] Work performed under the auspices of the U.S. Department of Energy.

page number

These earlier codes were developed to analyze severe hypothetical core disruptive accidents involving extensive melting of fuel pins, high power levels and short time scales ranging from milliseconds to a few seconds. They treat the reactor core in great detail, but the only analysis of phenomena outside the reactor vessel is a simple treatment of primary loop hydraulics to drive the subassembly coolant dynamics calculations. In contrast, SASSYS is oriented toward the analysis of shut-down heat removal performance, especially under natural circulation conditions, and it is also capable of analyzing operational transients. SASSYS uses a detailed core treatment taken mainly from SAS4A, but it also contains a detailed thermal-hydraulic treatment of the primary and intermediate heat transport systems, as well as the steam generators. SASSYS also treats the control system in considerably more detail than the earlier codes.

A number of other LMFBR systems analysis codes currently exist. These codes include the DEMO code (6), developed to analyze the CRBR reactor, the LANUS code (7), developed to analyze the FFTF reactor, and the NATDEMO code (8) developed for the EBR-II reactor. The main differences between SASSYS and these other codes are in the amount of detail in the core treatment, in the applicability to more than one reactor design, and in the numerical algorithms used.

Many systems codes use a simple core treatment, lumping all of the core driver assemblies into a single average assembly and all of the radial blanket assemblies into a second average assembly. Also, many do not account for flow redistribution between different driver assemblies, or between driver assemblies and blanket assemblies at low flow rates. Most systems codes can not handle boiling of the sodium in the core. In contrast, SASSYS can analyze a number of different core and radial blanket assemblies in detail, accounting for transient flow redistribution, subassembly-to-subassembly heat transfer, and boiling of the coolant.

Typically, a systems code is written for only one reactor design and is not applicable to other designs. The SASSYS code is intended for use with any LMFBR, using either a loop or a pool design, a once-through steam generator or an evaporator-superheater combination, and either a homogeneous core or a heterogeneous core with internal blanket assemblies. In addition to SASSYS, another systems analysis code, the SSC code (9), has the flexibility to handle different reactor designs. SSC also can provide a core treatment that is more detailed than that of most other LMFBR systems analysis codes.

Shut-down heat removal transients are often long, slow transients lasting hundreds or thousands of seconds. In order to reduce the computer time required to run such a case, it is usually desirable to take large time steps for the calculation of flows, pressures, and temperatures. Many systems codes use explicit forward time differencing algorithms which become numerically unstable for large time step sizes. Many of the algorithms used in previous SAS codes are also limited to short time steps. Most of the numerical algorithms in SASSYS use semi-implicit or fully-implicit time differencing schemes that are stable for large time steps.

CORE TREATMENT

The SASSYS core treatment uses a multi-channel treatment, where each "channel" represents a fuel pin, its associated coolant, and a fraction of the subassembly duct wall. Usually a channel is used to represent an average pin in a fuel subassembly or a group of subassemblies. A channel can also be used to represent a blanket assembly or a control rod channel, and the hottest pin in an assembly can be represented instead of the average pin. A channel represents the whole length of a subassembly, including the core, the axial blankets, the gas plenum region, and the spaces above and below the pins. Different channels can be used to account for radial and azimuthal power variations within the core, as well as variations in coolant flow orificing and fuel burn-up. Usually from five to twenty channels are used to represent a reactor.

Fuel Pin Heat Transfer

In general, finite differencing in both space and time is used in SASSYS. Up to 36 axial nodes are used to represent a channel. Up to 24 of these nodes can be in the core and axial blankets, with the rest used for the gas plenum region and the spaces above and below the pins. The sizes of the axial nodes can be arbitrarily set by the user, and it is not necessary to use a uniform node size to obtain accurate temperatures. For the fuel pin heat transfer calculations in the core and axial blankets, each axial node is divided into up to eleven radial fuel nodes, three clad nodes, one coolant node, and two duct wall nodes. Outside the core and axial blankets, temperatures are also calculated, but fewer radial nodes are used. Either an upper or a lower fission gas plenum can be handled.

For the pre-boiling temperature calculations, the fuel, clad, coolant, and duct wall temperatures for an axial node are solved for simultaneously, using a semi-implicit or fully implicit time differencing scheme that is numerically stable for large time steps. Temperature-dependent thermal properties are used. After boiling starts, the fuel pin heat transfer calculations stop at the clad surface, and the clad surface heat flux is used to couple with the boiling calculation.

Coolant Dynamics and Boiling

In general, LMFBRs are designed to provide cooling by natural circulation flow in the primary and intermediate sodium loops if all pump power is lost, but in many designs some sequences of pump power losses or other component failures can lead to temporary flow stagnation and coolant boiling in the core. If the reactor power is down to decay heat levels before flow stagnation occurs, then boiling in part of the core should provide the driving head to re-establish flow while at the same time removing enough heat to prevent melting of the fuel or clad. For this reason, one of the key features of the SASSYS code is the inclusion of a detailed boiling model. This boiling model, which is similar to that used in the SAS3D code, couples directly with the fuel pin heat transfer calculations and with the primary loop hydraulics.

The coolant flow rate calculations for a channel are driven by the inlet and outlet coolant plenum pressures, both before and after the start of boiling. All channels use the same inlet and outlet plenum pressures, so flow redistribution between channels as flow rates and temperatures change is automatically accounted for. Also, the coupling of all channels to common inlet and outlet plenums provides for hydraulic coupling between channels. The onset of boiling and inlet flow reversal in one channel can lead to a temporary rise in the

inlet plenum pressure and an increase in flow in other channels.

The pre-boiling calculations use incompressible flow. Friction, inertia, orifice pressure drops, and grid-spacer pressure drops are accounted for. Also, the gravity head is re-calculated for every time step using the current coolant temperatures.

The boiling model is the multiple-bubble slug-ejection model from SAS4A. This is a detailed model that accounts for vaporization of a liquid film left on the clad after voiding occurs, for condensation of the vapor on cooler clad and duct walls, and for pressure gradients within vapor bubbles due to streaming vapor. This model calculates the cooling of the clad due to vaporization of the liquid film until film dry-out occurs. It also accounts for the bouyancy effects of the vapor bubbles.

Reactor Power and Neutron Kinetics

SASSYS uses a point kinetics treatment for the neutron flux and fission power level. Reactivity changes are computed for control rod scram, the Doppler effect i the fuel, sodium voiding or density changes, and fuel thermal expansion. Also, decay-heat power levels are computed. The time-dependent decay heat power level either can be supplied by the user or it can be computed internally by the code as a function of burn-up and power history.

PRIMARY AND INTERMEDIATE HEAT TRANSPORT SYSTEMS

For the primary and intermediate loop thermal hydraulics calculations, SASSYS uses a generalized geometry as indicated in Fig. 1. A number of compressible volumes are connected by liquid or gas segments,

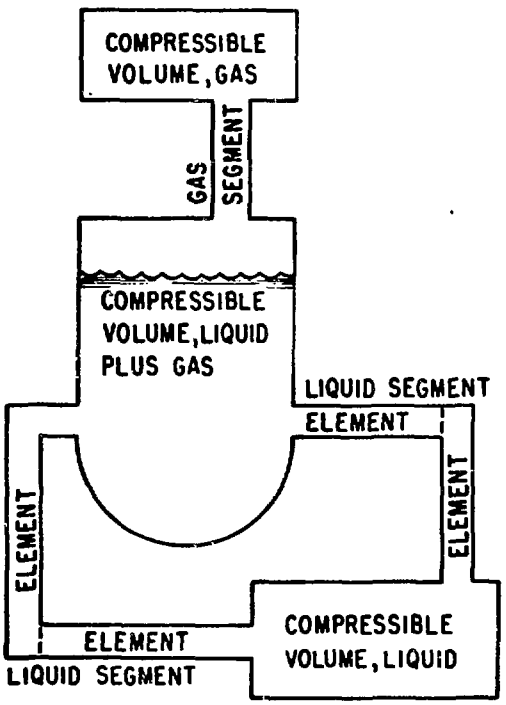

Fig. 1. General Heat Transport System Geometry Used in SASSYS

and each liquid segment can contain one or more elements. This treatment allows SASSYS to be used for an arbitrary arrangement of components, since compressible volumes and segments can be connected in an arbitrary manner.

Table 1 lists the types of elements that can be used to make up a liquid segment. Liquid segments are characterized by incompressible flow, with the possible exception of the core element. The reactor core is a special element that is handled by the core channel treatment. Before the onset of boiling an incompressible flow treatment is used for the core, but after the onset of boiling the flow into the bottom of the core may not match the flow out of the top.

Compressible volumes are characterized by pressures which drive the flows through the liquid and gas segments. If a compressible volume does not contain a cover gas, then the liquid is treated as compressible. Table 2 lists the types of compressible volumes that can be used. All gas segments are treated as pipes. The gas flow through a pipe is calculated using an isothermal treatment by Shapiro (10).

Hydraulics

For solving the hydraulic equations for the primary and intermediate heat transport loops, the use of semi-implicit or fully implicit time differencing is more difficult to implement than explicit forward differencing, especially when using a generalized geometry with an arbitrary number of compressible volumes,

Table 1
Liquid Flow Element Types

Type Number	Description
1	core subassemblies, SAS channels
2	core bypass assemblies
3	pipe
4	check valve
5	pump impeller
6	IHX, shell side
7	IHX, tube side
8	steam generator, sodium side
9	DRACS heat exchanger, tube side
10	DRACS heat exchanger, shell side
11	valve
12	air dump heat exchanger, sodium side

segments, and elements. With an implicit scheme, the pressures and flows for all connected compressible volumes and segments must be solved for simultaneously. By linearizing the hydraulic equations for each time step, SASSYS obtains a semi-implicit or fully implicit solution for the hydraulics equations without iterating.

Linearized semi-implicit or fully implicit methods are most useful for long transients in which temperatures and flows are changing slowly, since in such cases accurate results can be obtained with large time steps as long as the step sizes are small enough that changes during a step are small. For more rapid transients, the step size must be reduced, even with a fully implicit method; and for fast transients accuracy considerations may require comparable step sizes for any type of time differencing.

The finite difference equation used for the pressure change due to liquid flow into and out of a compressible volume during a time step is

Table 2
Compressible Volume Types

Type Number	Description
1	inlet plenum
2	compressible liquid volume, no cover gas
3	closed outlet plenum, no cover gas
4	almost incompressible liquid junction, no cover gas
5	pipe rupture source
6	pipe rupture sink, guard vessel
7	outlet plenum with cover gas
8	pool
9	pump bowl and cover gas
10	expansion tank with cover gas
11	compressible gas volume, no liquid

$$\Delta p(j) = b_0(j) + b_1(j)\ [\sum \bar{W}_{in}(j) - \sum \bar{W}_{out}(j)]$$
$$+ b_2(j)\ [\sum \bar{W}_{in}\ T_{in}(j) - \sum \bar{W}_{out}\ T_{out}(j)] \quad (1)$$

Pressure changes due to gas flow between cover gas volumes are handled separately with a similar equation. The coefficients b_0, b_1, and b_2 depend on the properties of the component and the size of the time step. Usually, b_0 is zero, b_1 is related to the compressibility of the cover gas or the liquid, and b_2 is related to effects of the temperature changes. Since b_0, b_1, and b_2 can change due to changes in temperature and cover gas volume, they are re-computed for each time step.

The basic equation for the flow in liquid flow segment i is

$$\sum_k \frac{L_k}{A_k}\frac{dw(i)}{dt} = p_{in}(i) - p_{out}(i) - \Delta p_{fr}(i)$$
$$- \Delta p_{w2}(i) - \Delta p_v(i) - \Delta p_{gr}(i) + \Delta p_p(i) \quad (2)$$

After finite differencing and linearizing, this equation has the form

$$\Delta w(i) = \frac{a_1(i) + \theta_2(i)\ [a_2(i) + \Delta t\ [\Delta p(ji) - \Delta p(jo)]]}{a_0(i) - \theta_2(i)\ a_3(i)} \quad (3)$$

In general, the a's are sums of contributions from each element, k, in the segment. The term a_0 is related to inertia, a_1 is the pressure drop at the beginning of the time step, a_2 is related to partial time derivatives of the pump head and pressure drop terms, and a_3 is related to the partial derivatives of the pressure drops with respect to flow rate.

Equations 1 and 3 can be combined to give a matrix equation of the form

$$\sum_j c_{ji}\ \Delta p(j) = d_j \quad (4)$$

where each segment i connected to compressible volume j contributes to c_{ji}, c_{jj}, and d_j. This matrix equation is solved by Gaussian elimination.

Coupling Between Primary Loop Hydraulics and Core Flow Calculations. In principle, the coolant flows for all core channels could be calculated simultaneously, along with the primary loop hydraulics; but after the onset of boiling this would unduly complicate the boiling model. Instead, a somewhat relaxed coupling scheme is used to couple the core flow calculations with the primary loop hydraulics at the inlet and outlet coolant plenums. First, the primary loop hydraulics calculations for a time step are made before the channel flow calculations. For this calculation, the core channel flows are estimated using

$$\frac{dW_c(L,ic)}{dt} = C_0(L,ic) + C_1(L,ic)p_{in}$$
$$+ C_2(L,ic)p_x + C_3(L,ic)|W_c(L,ic)\ W_c(L,ic)| \quad (5)$$

The coefficients C_0, C_1, C_2 and C_3 are supplied every time step by the core channel coolant dynamics routines. Equation 5 is used to calculate contributions to c_{jj} and d_j for the inlet and outlet plenums, so the estimated core flow calculations are directly coupled with the primary loop hydraulics calculation. Second, after the primary and intermediate loop hydraulics calculations are complete, the core channel coolant dynamics routines compute the actual channel flows for each channel independently, using the newly calculated inlet and outlet coolant plenum pressures as boundary conditions. Third, the difference between the estimated core flow and the actual computed core flow for a time step is used to adjust the coolant masses in the inlet and outlet coolant plenums before the start of the calculations for the next time step.

Before the onset of boiling, the differences between estimated and actual core flows are very small, since equation 5 is equivalent to the equation used by the pre-boiling core channel coolant dynamics routines, except that the coefficients in equation 5 do not account for the effects of coolant temperature changes during the current time step. After the start of boiling, rapid changes in vapor pressures cause rapid changes in inlet plenum pressure, as well as making accurate estimates of core flow changes difficult for large time steps. Therefore, the time step size must be limited to about .01 second during boiling, whereas time steps of 1 second or more can be used before the onset of boiling.

1. Adjust the inlet and outlet plenum pressures for any errors between estimted and calculated core flows in the last step.
2. Calculate b_0, b_1, and b_2 for compressible volumes in the primary system.
3. Calculate contributions to a_0, a_1, and a_2 for each element in a liquid segment and sum them for all liquid segments in the primary system.
4. Calculate contributions to c_{ji} and d_j for all segments in the primary system.
5. Add contributions to c_{ji} and d_j for the estimated core flow.
6. Solve for Δp
7. Calculate ΔW
8. Repeat steps 2,3,4,6, and 7 for the intermediate loops, if present
9. Repeat steps 2,3,4,6, and 7 for the DRACS loops, if present
10. Calculate liquid temperatures
11. Re-calculate compressible volume pressures with new liquid temperatures
12. Calculate gas flows, adjust cover gas pressures

Figure 2. Computational Sequence for a Time Step

Computational Sequence. Figure 2 shows the computational sequence used for a time step for the primary and intermediate heat transport systems. The liquid flow hydraulics calculations are done first, followed by temperature calculations and gas flow calculations. In order to reduce the sizes of the matrix equations that must be solved, the hydraulics calculations are done for the primary loop first, follwed by the intermediate loops and then the DRACS loops, if any. Adiabatic compression of the cover gasses is accounted for during the initial hydraulics calculations, but heat transfer to the gas and gas flows through connecting pipes are not accounted for until the gas calculations at the end of the computational sequence for a time step.

Temperature Calculations

A number of different temperature calculation algorithms are used for different components. The pipe temperature model is a slug flow model with heat transfer to the pipe walls, as indicated in Fig. 3. The coolant in a pipe is divided into a number of moving nodes or slugs. The node boundaries move with the coolant flow. All nodes in a pipe have equal volumes except for the first and last nodes. The inlet node size starts at zero and grows as the flow continues until it reaches the size of the other nodes. At that point a new node is started at the inlet. Similarly, the outlet node shrinks and eventually is removed when its volume reaches zero. The temperature in a coolant node changes only due to heat transfer to the pipe wall. There is one wall node for each coolant node. One radial node is used in the pipe wall, and an adiabatic boundary is assumed on the outside of the pipe. Wall nodes do not move, so the wall node in contact with a given coolant node changes periodically as the coolant node boundaries pass wall nodes.

STEAM GENERATOR

SASSYS contains two steam generator options. One is a very simple option in which the user specifies the sodium-side temeprature drop as a function of time. The other option is a moderately detailed, but fast running, model. The transients of interest for SASSYS usually do not involve rapid steam generator transients that would require a very detailed model that would consume large amounts of computer time.

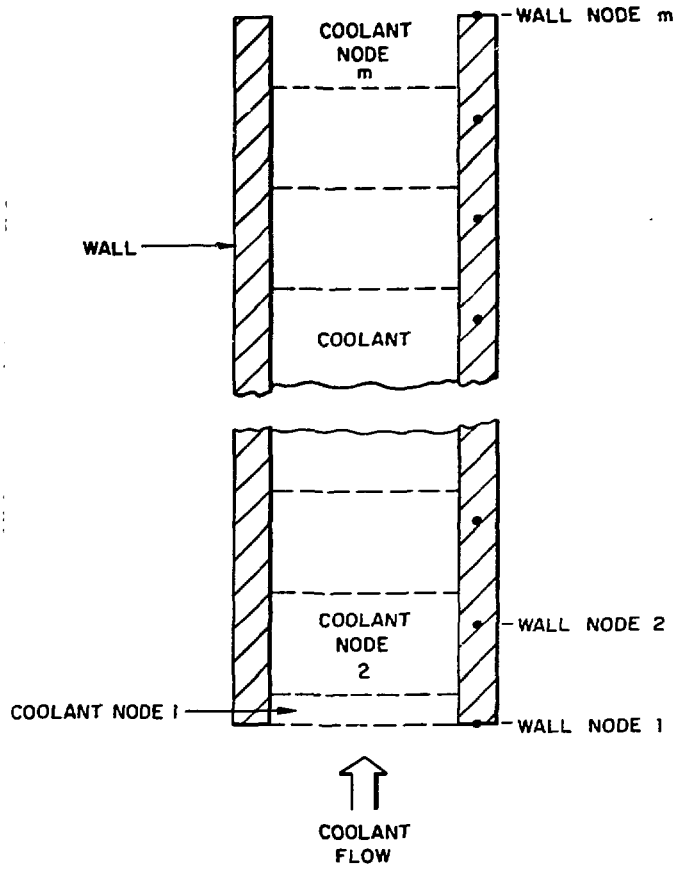

Fig. 3. Pipe Temperature Calculations

The moderately detailed model for an evaporator or a once-through steam generator was developed by R. May and B. Singer (11), using an approach similar to that of Bein and Yahalom (12). This model uses moving nodal boundries, as indicated in Fig. 4. Each axial node represents a well defined physical region with smooth, slowly varying water properties within the region.

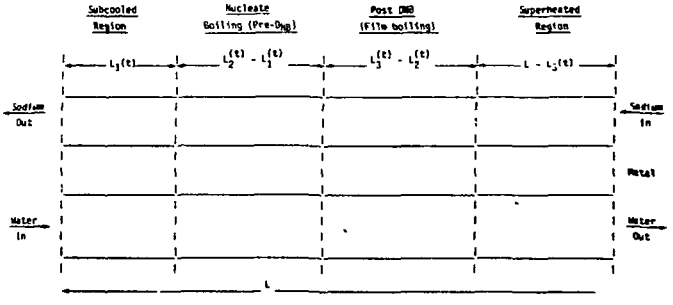

Fig. 4. Evaporator Model

For each of these four regions in the evaporator, an energy equation and a continuity equation is written for the water. These equations include terms for the moving boundaries. For each region, an average heat

Table 3. Channels Used to Represent the Core and Radial Blankets

Channel	Number of Subassemblies	Pins per Subassembly	Relative Power	Initial Flow per Subassembly	Description
1	6	217	1.46	.112	highest power drivers
2	114	217	1.27	.111	average power drivers
3	42	217	.97	.094	lowest power drivers
4	76	61	.59	.176	radial blankets

transfer coefficient is evaluated in order to obtain the heat flux to the water. On the sodium side, single phase incompressible flow is assumed, and only an energy equation is used for each region. In addition, an over-all loop momentum equation is used for the sodium in the evaporator plus any pipes attached to it.

A separate model is used for the superheater. This model is simpler and faster running, since it does not use moving boundaries or deal with phase changes. A quasi-static approximation used for the steam side energy equation provides a very stable solution algorithm.

MODULAR APPROACH

The coding of SASSYS uses a modular approach. In general, each component is treated with a separate routine or group of routines, so a new component treatment can be added or an existing treatment can be modified without affecting the rest of the code.

Another aspect of the modular approach is that in many cases the code has two options for treating a component. A simple, fast running option and a more detailed but somewhat slower model. Thus, for moderately fast transients that are over before the steam generator can have any impact on the core behavior, a very simple steam generator option can be used. Also, a simple IHX treatment can be used to terminate the calculation at the IHX and eliminate the intermediate loop and the steam generators entirely. For the simple IHX treatment, the user specifies the primary side temperature drop and thermal center as a function of time. For analysing the initial core behavior resulting from a rapid transient, such as the complete break of a primary loop pipe, the transient thermal behavior of the IHX is of little importance, and the behavior of the intermediate loops and steam generators is of even less importance. For such a case, the simple IHX treatment can eliminate much unnecessary computation. For slower transients in which IHX and intermediate loop behavior are important, the simple IHX model is not appropriate unless the user happens to know the IHX thermal behavior from other sources.

SASSYS EXAMPLE: STATION BLACKOUT

As an example of the capabilities of SASSYS, a case involving loss of all pump power and loss of feed water to the steam generators was run. For this case, a 1000 MWT loop-type reactor was used. The reactor was initially running at normal power and flow. At time zero, the power to the main sodium pumps and the feed water pumps for the steam generators was lost. The control rods also scrammed at this time. After the primary pumps coasted down to about ten per cent of normal flow, the primary pump pony motors held the flow at this level for five minutes. The pony motors then quit. The intermediate loop pony motors did not run at all. This case is a hypothetical case that was run only to demonstrate the capabilities of the code. It may not be a realistic case for any current reactor design, although it is similar to cases that have been considered by Additon (13) for the FFTF reactor, except that Additon assumed that the air dump heat exchangers in one of the three FFTF loops remained operational.

Four channels were used to represent the core and radial blankets, as indicated in Table 3. Also, an unheated bypass flow around the core was modelled. During normal operation, the unheated bypass flow amounted to about 8.3 per cent of the total primary flow. Outside the vessel, the primary and intermediate sodium pumps, the intermediate heat exchangers, the steam generators, the intermediate loop expansion tanks, and all of the connecting piping were modelled.

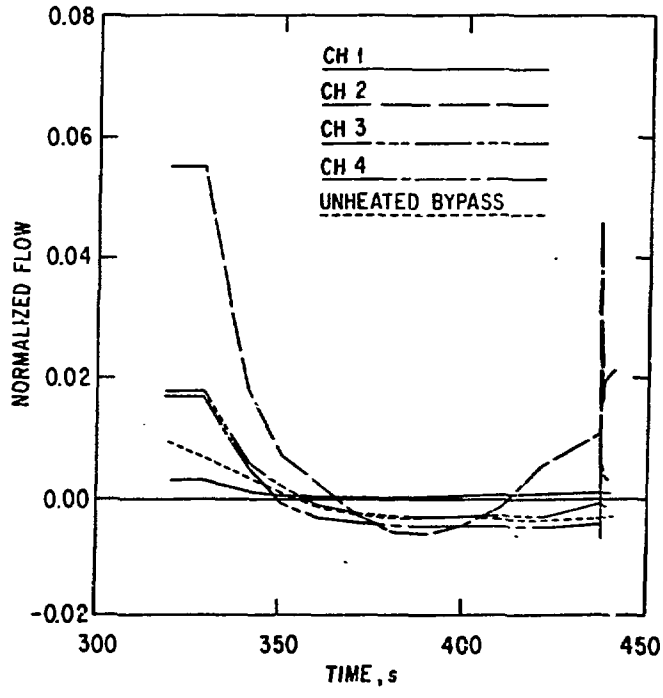

Fig. 5. Flows From the Coolant Inlet Plenum After Loss of Pony Motors

Figure 5 shows the calculated flows out of the coolant inlet plenum after the pony motors stopped. These flows were all normalized to the steady-state total vessel flow. Initially Channel 2, which represented the average power driver subassemblies, had the highest flow, since it contained the largest number of subassemblies. Channel 1, which represented only a few higher power subassemblies, had the highest flow per subassembly before the pony motors stopped.

In the early part of the transient, the reactor power dropped faster than the coolant flow because the control rods scrammed faster than the pumps coasted down. This led to a drop in temperature in the outlet plenum. The five minutes of pony motor operation pushed this cooler outlet plenum sodium into the hot leg of the primary loop, partly destroying the natural circulation gravity head. Also, the operation of the pony motors in the primary loop but not in the intermediate loop resulted in primary loop flows that were higher than the intermediate loop flows, leading to an increase in temperatures in the lower parts of the IHX and a higher IHX primary outlet temperature. This effect also reduced the thermal head in the primary sytem.

The coolant in Channel 1 was hot enough to have enough buoyancy that the flow in this channel never reversed, although the flow almost stopped for a while. The peak coolant temperature reached 57 K below the saturation temperature before the flow rate increases enough to reduce the peak temperature.

The first channel in which flow reversal occurred is Channel 4, representing the radial blankets. The unheated bypass and Channel 3, representing the lower power driver subassemblies reversed flow soon after. These were the coolest channels when the pony motors stopped, and they did not have enough buoyancy to sustain positive flow.

Channel 2 was the only channel that boiled in this case, and it only boiled for a short time. Before boiling the flow reversed for a while in this channel. The flow went positive again before boiling started. At 437 seconds, which is when boiling started, the sodium which boiled initially had spent a long time in the core. This sodium had first gone upward through the core; then it went back down through the core when flow reversed, and upward through the core again when flow went positive.

A superheat of 10 K was assumed before the start of boiling. This superheat caused enough vapor pressure to reverse the inlet flow momentarily in this channel after the start of boiling. The buoyancy of the vapor bubbles tended to increase the flow through the channel, and the channel was not hot enough to sustain the vapor pressure for long; so boiling stopped after about .25 second.

The core flow recovered after the boiling stopped, providing enough flow to cool the core. Eventually though, some heat removal from the sodium must occur, either by re-starting the feed water pumps or by operating an emergency heat removal system, to prevent overheating of the whole system.

This case required 125 seconds of central processor time on an IBM 370 model 195 computer to run the first 437 seconds of the transient, up to the start of boiling. The 0.25 seconds of boiling required an additional 10 seconds of computer time. A time step size of 1 second was used up to the onset of boiling. Then the time step size was cut to .01 second, and the boiling model used an internal time step that averaged .003 seconds.

SUMMARY

The SASSYS LMFBR systems analysis code was developed to provide a flexible and efficient tool for analyzing the consequences of failures in the shut-down heat removal system of an LMFBR. The key aspects of shut-down heat removal transients are addressed using numerical algorithms that provide for rapid computation. The reactor core treatment in SASSYS can be much more detailed than the core treatments usually found in LMFBR systems codes. The code has the flexibility to handle any LMFBR design: loop or pool, hot leg pump or cold leg pump, once-through steam generator or evaporator-superheater combination, and homogeneous core or heterogeneous core with internal blankets. The code uses a modular approach that makes it easy to replace or modify component treatments. Many components can be treated with either a simple, fast running model or a more detailed but slower running model.

REFERENCES

1. MacFarlane, D. R., et al., "SAS1A, A Computer Code for the Analysis of Fast-Reactor Power and Flow Transients," ANL 7607, October 1970.

2. Dunn, F. E., et al., "The SAS2A LMFBR Accident Analysis Computer Code," ANL-8138, October 1974.

3. Stevenson, M. G., et al., "Current Status and Experimental Basis of the SAS LMFBR Accident Analysis Code System," Proc. Fast Reactor Safety Meeting, CONF-740401-P3, 1974, pp. 1303-1321.

4. Dunn, F. E., and Kyser, J. M., "Impact of Advanced Systems on LMFBR Accident Analysis Code Development," Scientific Computer Information Exchange Meeting, CONF-790902, 1979, pp. 71-80 .

5. Cahalan, J. E., et al., "The Status and Experimental Basis of the SAS4A Accident Analysis Code System," Proc. International Meeting on Fast Reactor Safety Technology, Seattle, Washington, 1979, pp. 603-614.

6. Alliston, W. H., et al., "Clinch River Breeder Reactor Plant, LMFBR Demo Plant Simulation Model (DEMO)," Westinghouse Electric Corporation Advanced Reactors Division, CRBRP-ARD-0005, February 1973.

7. Additon, S. L., McCall, T. B., and Wolfe, C. F., "Simulation of the Overall FFTF Plant Performance," Westinghouse Hanford Engineering Development Laboratory, HEDL-TC-556, March 1976.

8. Mohr, D. and Feldman, E. E., "A Dynamic Simulation of the EBR-II Plant During Natural Convection with the NATDEMO Code," Decay Heat Removal and Natural Convection in Fast Breeder Reactors, A. K. Agrawal and J. G. Guppy editors, Hemisphere Publishing Co., 1981, pp. 207-223.

9. Agrawal, A. K., et al., "An Advanced Simulation code for Transients in LMFBRs (SSC-L Code)," Brookhaven National Laboratory, BNL-NUREG-50773 NRC-7, February 1978.

10. Shapiro, A. H., The Dynamics and Thermodynamics of Compressible Fluid Flow, Vol. 1., Ch. 6, The Ronald Press Co., New York, 1953.

11. May, R. D., and Singer, B. S., personal communications, July 6, 1981.

12. Bein, M. and Yahalom, R. "Dynamic Simulation of an LMFBR Steam Generator," Proceedings of the Second Power Plant Dynamics, Control and Testing Symposium, Knoxville, Tenn., Sept., 1975.

13. Additon, S. L., Parziale, E. and Wolfe, C. F., "FFTF Decay Heat Removal Analysis: Effects of Natural Phenomena," Westinghouse Hanford Engineering Development Laboratory, HEDL-TI-75222, 1976.