Conf-940402--6
ANL/RA/CP--79958

Paper to be presented at the ARS '94 International Meeting on Advanced Reactor Safety, American Nuclear Society, April 17 - 21, 1994, Pittsburgh, PA

DIANA: A Multi-phase, Multi-component Hydrodynamic Model for the Analysis of Severe Accidents in Heavy Water Reactors with Multiple-tube Assemblies*

A. M. Tentner

Reactor Analysis Division
Argonne National Laboratory
9700 South Cass Avenue
Argonne, IL 60439

*Work supported by the U.S. Department of Energy, Nuclear Energy Programs under Contract W-31-109-ENG-38.

DIANA: A Multi-phase, Multi-component Hydrodynamic Model Model for the Analysis of Severe Accidents in Heavy Water Reactors with Multiple-tube Assemblies*

A. M. Tentner
Argonne National Laboratory
9700 South Cass Avenue
Argonne, Illinois 60439

ABSTRACT

A detailed hydrodynamic fuel relocation model has been developed for the analysis of severe accidents in Heavy Water Reactors with multiple-tube Assemblies. This model describes the Fuel Disruption and Relocation inside a nuclear fuel assembly and is designated by the acronym DIANA. DIANA solves the transient hydrodynamic equations for all the moving materials in the core and treats all the relevant flow regimes. The numerical solution techniques and some of the physical models included in DIANA have been developed taking advantage of the extensive experience accumulated in the development and validation of the LEVITATE [1] fuel relocation model of SAS4A [2, 3]. The model is designed to handle the fuel and cladding relocation in both voided and partially voided channels. It is able to treat a wide range of thermal/ hydraulic/neutronic conditions and the presence of various flow regimes at different axial locations within the same hydrodynamic channel.

I. INTRODUCTION

A detailed hydrodynamic fuel relocation model has been developed for the analysis of severe accidents in Heavy Water Reactors with multiple tube assemblies. This model describes the Fuel Disruption and Relocation inside an NPR assembly and is designated by the acronym DIANA. The numerical solution techniques and some the physical models included in DIANA have been developed taking advantage of the extensive experience accumulated in the development and validation of the LEVITATE fuel relocation model of SAS4A.

The model calculates the fuel and cladding relocation that occurs in a multiple-tube assembly following the melting and disruption of the fuel elements. It is designed to interact with other phenomenological modules to describe the integrated response of the reactor core during a postulated accident. DIANA is initiated when the first fuel element begins to melt, leading to fuel and/or cladding relocation. It continues to work until accident termination or until the occurrence of significant loss of assembly integrity.

DIANA describes the axial fuel, cladding, target and water relocation in each hydrodynamic channel. A fuel assembly consists of several concentric cylinders. Some of these cylinders contain fuel while others are used for isotopic enrichment and are referred to as target elements. The outside assembly wall is referred to as "sleeve". A hydrodynamic channel is defined as the space bounded by two adjacent cylinders (either fuel, target or sleeve). Axially, this channel is bounded by the bottom and top of the fuel/target region. The gradual melting and disruption of the fuel and target elements are also modeled. As the fuel elements disrupt significant intra-assembly geometry changes occur. The original hydrodynamic channels become connected, allowing material transfer from one channel to the others and pressure equilibration. There is a need for modeling of radial mass and energy transport. Furthermore, as the assembly outer wall begins to fail, fuel may be ejected into the moderator space. The fuel ejection calculation provides the interface with the model describing the moderator and fuel behavior in the moderator space.

The model is designed to handle the fuel and cladding relocation in both voided and partially voided channels.

---

*Work supported by the U.S. Department of Energy under Contract W-31-109-ENG-38.

It is able to treat a wide range of thermal/hydraulic/neutronic conditions and the presence of various flow regimes at different axial locations within the same hydrodynamic channel.

The main result of DIANA calculations are the spatial fuel, cladding and target mass and temperature distributions. These spatial distributions, which change constantly in time, are used by the neutronic models to calculate the associated changes in reactivity and power.

II. MODEL DESCRIPTION

A. Phenomenological Description

DIANA models each intact hydrodynamic channel in a one-dimensional geometry. An intact hydrodynamic channel is defined as the coolant channel space bound by two neighboring cylindrical elements, which provide a solid barrier between the DIANA channel and the neighboring channels. As the accident proceeds, the disruption of the fuel and target cylindrical elements removes the physical barriers between the original channels and two-dimensional effects can become important. DIANA includes a 2-D formulation and the modeling of 2-D effects is currently in the developmental stage.

After initiation DIANA describes all the physical phenomena in the hydrodynamic channel bound axially by the bottom and top of the fuel region. These phenomena include the hydrodynamics of all the components in the coolant channel, the heatup and melting of the solid fuel elements bounding the DIANA channel and the freezing of the molten fuel, cladding and target material leading to crust and blockage formation. DIANA also models the disruption of the solid fuel and target elements, which creates the possibility of inter-channel material relocation. Figure 1 shows two adjacent hydrodynamic channels where the fuel relocation has begun. Some of the DIANA phenomenological models are illustrated.

B. Physical Conditions at the Time of DIANA Initiation

DIANA is initiated in any channel where the possibility of material relocation exists, This means, depending on the accident sequence that either the cladding or the fuel or both melt at a certain axial location. If the fuel and cladding have similar solidus temperatures and the temperature gradients in the fuel element are small, as expected, the relocation of cladding and fuel will be initiated simultaneously.

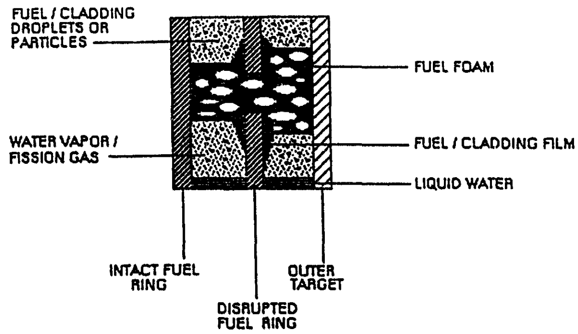

Fig. 1. Two Adjacent Hydrodynamic Channels Illustrating Some of the DIANA Phenomenological Models

The material relocation can be initiated when the cladding and/or fuel are only partially molten, i.e., solid particles or fuel chunks are still present. DIANA has been designed to model the relocation of molten and solid components independently.

C. Hydrodynamics of the Coolant Channel

Before the DIANA initiation the coolant channel contains only water vapor or liquid water. But, as the fuel element begins to melt and disrupt the fuel, cladding and fission gas begin to interact with the original components. A complex situation develops, involving a large number of components that have to be tracked separately. The moving components in the channel are:

a. fuel - solid, liquid, vapor
b. cladding - solid, liquid, vapor
c. target - solid, liquid, vapor
d. coolant - vapor or liquid
e. fission gas - free or associated with the liquid or solid fuel.

The material motion is described by a multi-component, multi-phase, nonequilibrium hydrodynamic model. The region described by this model is bound axially by the liquid water slugs, and is generally referred to as "the interaction region". This region can increase or decrease depending on the dynamics of the liquid water slugs which is described by a simple incompressible model.

The dependent variables in the interaction region are the density, velocity and enthalpy. A separate mass and energy equation is solved for each component, but only three coupled momentum equations for three velocity fields are solved. The components treated together in the velocity fields are: (a) liquid fuel, cladding and target, (b) fission gas, fuel vapor, cladding vapor, target vapor and water vapor, and (c) solid fuel, cladding and target chunks or particles.

The interaction between the different components present in the channel, i.e., mass, energy and momentum transfer, is largely determined by the local configuration which, in turn, is determined by the local flow regime. Due to different conditions along the channel, several flow regimes can exist in the same channel at any given time. Furthermore, as conditions change with time the flow regime configuration in each channel will also change. DIANA models the flow regime dynamics by allowing one of several flow regimes at each axial location of the channel.

D. FLOW REGIMES

The flow regimes modeled by DIANA are a particulate/droplet flow regime, a fuel/target/cladding bubbly flow regime and a fuel/target/cladding partial or total annular flow regime. Various bubbly or annular flow regime configurations may be modeled depending on the dominant molten component, i.e., fuel, cladding or target. In both cases solid particles or droplets can be present. If the continuous molten phase is not present, a pure droplet/particle flow regime is modeled.

D.1 Particulate/Droplet Flow Regime

The particle/droplet field is used to describe a wide range of physical configurations. This field can exist alone or together with a continuous molten component field, offering significant modeling flexibility. The droplets or particles can be formed by entrainment due to the shear forces exerted by the flowing vapor stream on the molten or nearly molten cladding or bare fuel elements. Solid particles or chunks can also be generated by disruption of nearly molten fuel elements, breakup of frozen crusts or bulk freezing of flowing molten components. The particle/droplet field is characterized by size and composition at each axial location. A cylindrical geometry of the particles is used in order to simplify the calculations. Each particle/droplet can contain fuel, cladding and target in various proportions, depending on the sequence modeled. The proportions are determined by the mass of each individual component, which is in turn calculated by solving the mass conservation equation in each cell. The particles/droplets exchange heat and momentum with the surrounding component, and can melt or freeze, respectively. The particle/droplet field can coexist with a moving molten component, which can be in the form of either foam or films. In the case of foams, only particles will be maintained as a separate field. In the case of molten films, molten droplets can continue to exist in the vapor stream. They can be deentrained, joining the molten film on the cladding. Depending on the local conditions, additional droplets can be entrained from the molten films, joining the molten droplets field.

D.2 Foamy or Bubbly Flow Regime

The melting and disruption of irradiated fuel elements leads to the formation of fuel foams which can cover the entire cross section of the hydrodynamic channel at a certain axial location. This situation is modeled by the "foamy" or "bubbly" flow regime. In this case the continuous component is the fuel meat, which contains imbedded large fission gas bubbles. In addition to fission gas, these bubbles can also contain water vapor, entrapped during the foaming process. The fission gas/vapor bubbles can move at a speed different from that of the fuel meat, and thus separation or penetration of the foams can occur. However, the velocity of the imbedded bubbles generally remains close to the fuel meat velocity due to the inertial forces acting on the bubbles. Molten cladding and target can also be imbedded in the foam, exchanging heat with the surrounding elements. Similarly, the foam can contain solid particles or chunks, which exchange both energy and momentum with the molten foam.

D.3 Film Regime

The film flow regime is used to describe the flow of molten cladding or fuel along the solid cladding or bare fuel elements. Two separate films are modeled, one on each side of the hydrodynamic channel. The films have different temperatures but in the first version of DIANA they are assumed to have the same velocity. An additional velocity field will be introduced in later versions to allow the independent movement of the films. Each film exchanges energy and momentum with the underlying structure and with the vapor stream flowing in the channel. The film contains molten cladding, fuel and target in various amounts, determined by the previous history of the film. Film freezing leading to crust formation, and droplet entrainment from the film are also modeled.

E. Freezing and Melting Phenomena

At the axial locations where the fuel elements are still intact, the coolant channel is bounded by the solid cladding or fuel surface. The temperature field in the cladding and fuel is calculated by a transient heat-transfer model, using the temperatures in the DIANA channel and the temperatures in the neighboring channel as boundary conditions. When a continuous molten component is present - i.e., annular or bubbly fuel flow - the temperature of the molten/solid interface can fall below the freezing temperature of the molten component or above the melting temperature of the solid component, leading to phase changes and geometry changes. If the interface temperature drops below the freezing temperature of the molten component, DIANA allows the formation of an annular or partial annular crust which can grow in time. If, on the other hand, the interface temperature of the solid cladding or fuel is above the corresponding melting temperature, melting of the solid component is modeled. This can result in melting and ablation of the initial cladding or fuel. It can also lead to the remelting of the previously formed crusts. The temperature of the frozen crust, at any given location, is calculated by the heat transfer model. Depending on its temperature and other local conditions which will be described in detail later, the frozen crust can continue to grow, can start to remelt or can break up when the underlying solid cladding begins to melt.

Freezing of the molten droplets due to heat loss to the surrounding components does not affect the channel geometry. The frozen particles continue to move as part of the particle/droplet field. However, the molten droplets can lead to changes in the channel geometry if they are first de-entrained. The de-entrained droplets are assumed to form a local film, which can freeze, leading to the formation of a frozen crust, as described previously. Melting of the solid particles has no immediate effect on the other components if the continuous molten component is not present or it is in the film flow regime. However, if the continuous molten component is in the foam flow regime, the molten component of the particle/droplet field is assumed to merge immediately with the molten component and only the solid particles continue to be tracked separately.

F. Fuel Element Disruption

As the accident proceeds the fuel and cladding begin to melt. When a large fraction of the fuel has become molten a certain axial location and the cladding is no longer effective in restricting the radial motion the fuel element is totally disrupted. The area previously occupied by the fuel element becomes part of the coolant channels and only two fuel ring segments remain. The fuel element disruption opens a path of communication between two neighboring channels.

If the channels on either side of a disrupting fuel element are not DIANA channels, then the disruption event will initiate the DIANA model in these channels.

III. SOLUTION ALGORITHMS AND GENERAL NUMERICAL CONSIDERATIONS

The independent variables used in DIANA are the radial coordinate z, the radial coordinate r, and the time t.

Only one spatial coordinate is necessary prior to the disruption of fuel or target rings, as DIANA models the coolant channel in a one-dimensional geometry. After the occurrence of ring disruption the radial transport between channels will be modeled. The discretization in the radial direction is determined by the assembly geometry, i.e., each hydrodynamic channel is represented by a radial cell.

The dependent variables calculated by the hydrodynamics model for each cell are the generalized density ρ', the enthalpy h (or temperature T), and the velocity u.

The generalized density, for component k is defined as:

$$\rho'_{k,i} = \rho_{k,i} \cdot \frac{A_{k,i}}{AXMX} = \rho_{k,i} \cdot \theta_{k,i} \qquad (1)$$

where

$\rho_{k,i}$	=	physical density of component k at location i,
$A_{k,i}$	=	cross sectional area occupied by component k at location i,
AXMX	=	reference area, and
$\theta_{k,i}$	=	generalized area fraction of component k at location i.

The use of the generalized smeared densities in DIANA has been prompted by the many moving and stationary components in this problem. Its use simplifies the finite difference equations for variable cross section flow.

If the total area of the assembly is AXMX, the generalized volume fraction of component k is:

$$\theta_{k,i} = \frac{A_{k,i}}{AXMX} \qquad (2)$$

where $A_{k,i}$ has been defined above. The reference area AXMX, which is an input quantity, can be arbitrarily chosen. This is because the code is invariant to the choice of AXMX (i.e., as long as it is not varied by several orders of magnitude which can lead to differences due to truncation errors). However, the recommended value of AXMX is the cross sectional area of an assembly (including the housing) because the volume fractions of the different components that appear in the DIANA output can be better understood in this case.

The mass, energy and momentum partial differential conservation equations are solved using an Eulerian finite difference semi-explicit formulation, as explained below. The mesh grid used for the finite difference formulation is presented in Fig. 2. As indicated in this figure, the densities and enthalpies are defined at the center of each cell, while the velocities are defined at the boundaries. Because of the highly irregular geometry treated by DIANA, special attention was necessary for the treatment of abrupt area changes [5-2]. Fuel velocities are defined at each cell boundary, with u_i' being the velocity just before boundary i, and u_i'' the velocity just after that boundary. The terms "before" and "after" are used in relation to the positive direction of the axial coordinate z.

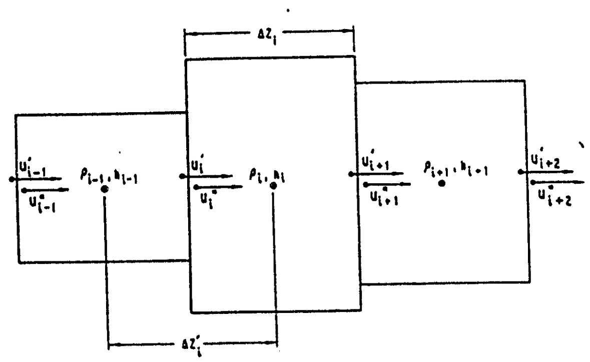

Fig. 2. Mesh Grid Used in the Channel Hydrodynamic Model

In order to reduce the numerical diffusion characteristic of Eulerian numerical schemes, the boundaries of each region containing a certain component are tracked separately as they move through the Eulerian cells.

IV. FUEL RELOCATION CALCULATIONS IN A NPR FUEL ASSEMBLY USING THE DIANA MODEL OF SAS-HWR

This section describes the results of fuel relocation calculations performed with the DIANA model. They illustrate the current capabilities of the DIANA model and provide initial information about the dynamics of fuel and cladding particles/droplets in a voided multiple-tube assembly following the disruption of a fuel cylinder.

The accident analyzed is a Reactivity Insertion Accident with a reactivity ramp of 0.01$/s up to total of 0.2$.

The increase in power causes the heatup of the fuel and coolant, leading to coolant boiling in channels 4 and 5 of the assembly. The DIANA fuel relocation model is initiated in both channels 4 and 5 at 27.004 s. The initiation of DIANA is triggered when the cladding reaches the relocation temperature at any axial location in a coolant channel. The cladding relocation temperature is an input value. In this calculation it was 922 K, equal to the cladding solidus temperature. The conditions in the fuel assembly at the time of DIANA initiation are illustrated in Fig. 3. The cladding of the outermost fuel cylinder (No. 4) has exceeded the relocation temperature in the upper part of the fuel cylinder, between 3.51-3.70 m. from the bottom end. The fuel has also exceeded the fuel relocation temperature (919 K) between 2.93 - 4.27 m. from the bottom of the cylinder.

DIANA allows the disruption of the fuel ring No. 4 between 3.51 - 3.70 m., where both the cladding and the fuel meat exceed their respective relocation temperatures. The disruption leads to the formation of fuel and cladding particles with a fixed diameter of 1 mm. The radius of the particles is an input that can be changed by the user. Although some fission gas was assumed to be present in the fuel, it does not play any role in this calculation, because the disrupted fuel temperature remains at all times below the melting temperature and the fission gas can be released only after fuel melting. The fuel and cladding particles move under the influence of pressure gradients, vapor drag, wall friction, and gravity. They exchange heat with the streaming water vapor and the cladding surface of the cylinders bounding the coolant channel. Internal heating of the fuel particles is also included.

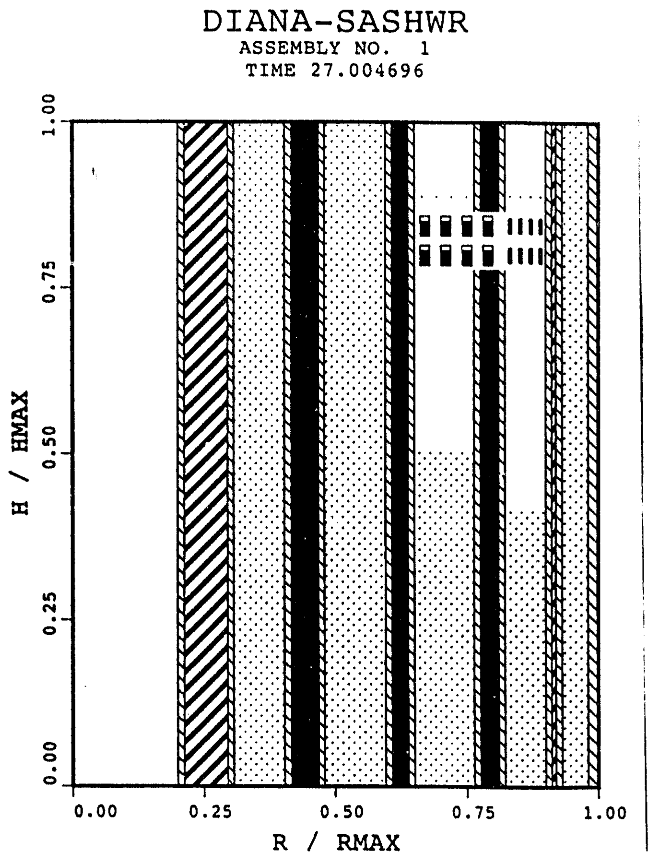

Fig. 3.

The results of the DIANA calculation show that the fuel particles are entrained upwards by the streaming water vapor, which plays a significant role in the fuel relocation. DIANA provides a tight coupling between the dynamics of water region at the bottom of the channel and the fuel/vapor hydrodynamic calculation. It describes the heating and boiling of the liquid water slug as well as the water ejection or reentry under the influence of the pressure difference between the DIANA compressible region and the lower assembly plenum.

Two more axial nodes of the fuel cylinder No. 4 are disrupted about 100 ms after the DIANA initiation. No fuel particles have entered the upper assembly plenum, but the first particles are approaching the top of the fuel cylinder. The amount of moving fuel in the coolant channel is 1.88 kg. These results are illustrated in Fig. 4.

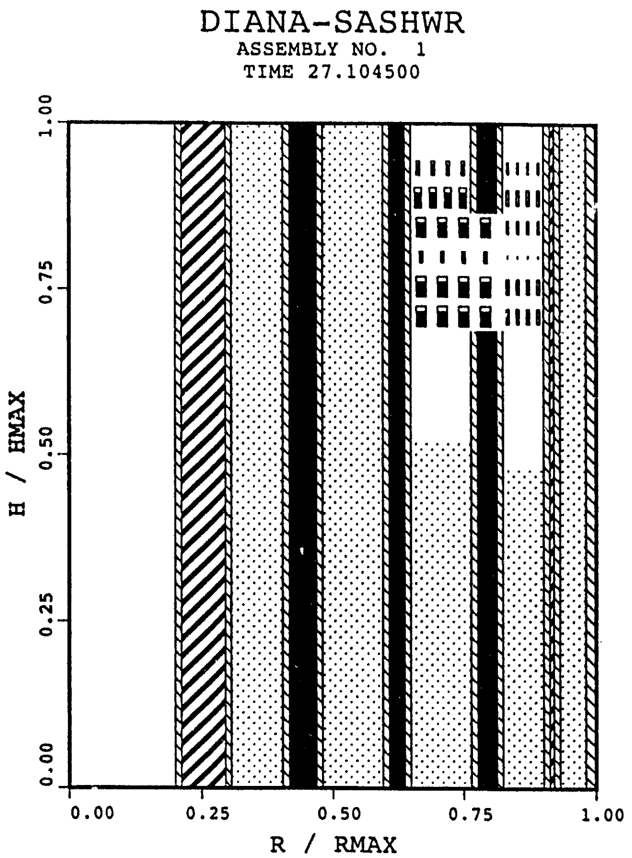

Fig. 4

The first fuel particles enter the upper assembly plenum at about 200 ms after the initial fuel element disruption. At this time five axial fuel nodes have disrupted and the amount of moving fuel in the channel is 2.28 kg.

The calculation was continued until 255 ms after the DIANA initiation. Fuel particles with a total mass of 1.88 kg are moving upwards with velocities of about 6 m/s and 0.4 kg of fuel have already reached the upper assembly plenum. The results of DIANA calculations at 250 ms after the initial fuel disruption are illustrated in Fig. 5.

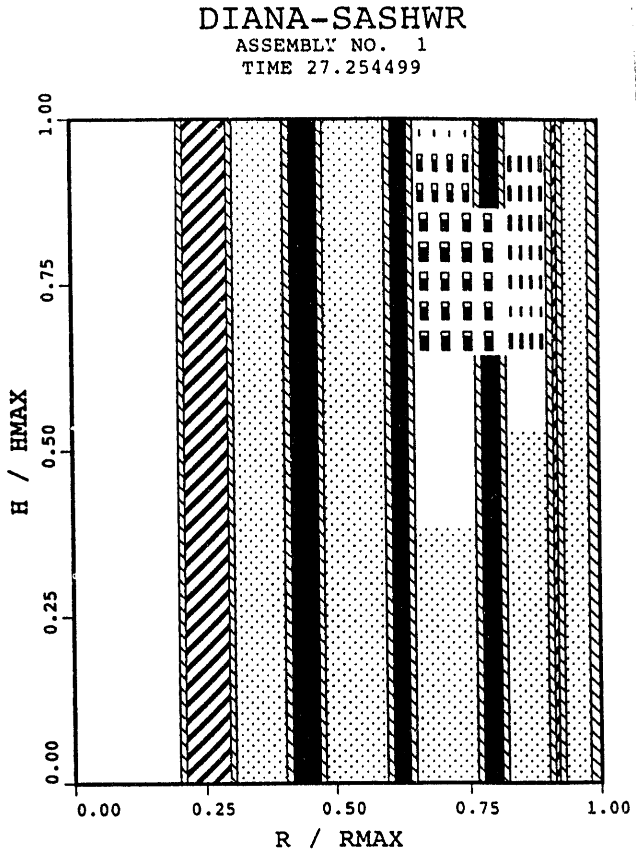

Fig. 5

CONCLUSIONS

A detailed hydrodynamic fuel relocation model, DIANA, has been developed for the analysis of severe accidents in Heavy Water Reactors with multiple tube assemblies. DIANA describes the fuel and cladding relocation that occurs in a multiple-tube assembly following the melting and disruption of the fuel elements.

The results of the DIANA calculations indicate that, for the RIA examined, if a particulate/droplet fuel flow regime is present the fuel and cladding particles are entrained upwards by the streaming vapor and can reach the top of the fuel cylinder in about 200 ms after the initial fuel disruption. The fuel dynamics is strongly affected by the vapor generation at the lower water slug interface. This phenomenon is modeled by DIANA, which provides a tight coupling between the fuel and water vapor dynamics. However, the results of these calculations indicate that water boiling can also occur below the slug interface. This phenomenon is not yet modeled by DIANA. Boiling below the water slug interface can provide additional vapor, which can accelerate the fuel particle entrainment.

The target heatup is under-estimated in this calculation by the fact that all the disrupted fuel is allocated to the coolant channel No. 4 as the material exchange between the coolant channels was not yet modeled in DIANA. However, independent heat-transfer calculations indicate that the time for target melting due to molten fuel contact at power levels 1-2 times nominal power might be of the order of several seconds which is considerably longer than the time needed for the fuel particles to reach the upper assembly plenum.

REFERENCES

1. A. M. Tentner and H. U. Wider, "LEVITATE-A Mechanistic Model for the Analysis of Fuel and Cladding Dynamics Under LOF Conditions," Intl. Mtg. on Fast Reactor Safety Technology, Seattle, WA (Aug. 1979).

2. J. E. Cahalan and A. M. Tentner, "Advanced LMR Safety Analysis Capabilities in the SASSYS and SAS4A Computer Codes," these proceedings.

3. A. M. Tentner et al., "The SAS4A LMFBR Whole Core Accident Analysis Code," Proc. Intl. Mtg. on Fast Reactor Safety, 2:989, Knoxville, TN (April 1985).

END

DATE
FILMED
5/16/94