UNITED STATES
NUCLEAR REGULATORY COMMISSION
ADVISORY COMMITTEE ON REACTOR SAFEGUARDS
WASHINGTON, D. C. 20555

September 2, 1999

MEMORANDUM TO: ACRS Members

FROM: P. Boehnert, Senior Staff Engineer

SUBJECT: SUPPORTING DOCUMENTS - G. WALLIS'S DISCUSSION ON STATUS OF EPRI RETRAN-3D CODE REVIEW - 465th ACRS MEETING, SEPTEMBER 1, 1999

During the subject discussion, Dr. Wallis made reference to two documents: a critique of the two-phase flow models in the RETRAN-3D code by former NRC-RES staff member J. Kelly, and an excerpt from NUREG-75/056, "WREM: Water Reactor Evaluation Model, [Revision 1]", dealing with a description of momentum equation used in the WREM code. Subsequent to conclusion of the Committee's discussion of this matter, several requests were made to obtain copies of the above two documents. Copies are attached for your perusal.

Attachments: As Stated

cc: R. Savio

cc w/o attach (via E-mail):
J. Larkins
S. Duraiswamy
ACRS Technical Staff & Fellows

August 26, 1999

**MEMO TO:** Farouk Eltawila

**FROM:** Joseph M. Kelly

**SUBJECT:** **RETRAN-3D REVIEW**

As part of NRR's review of the RETRAN-3D thermal-hydraulic analysis code, I was asked to review the two-phase flow models with specific emphasis on the Chexal-Lellouche drift flux model. This memo serves to document the results of my review.

I have misgivings about granting some form of "generic approval" for the usage of the RETRAN-3D code (or any code for that matter) for "all transients except LOCA". I feel very strongly that a well structured process such as that of the CSAU is necessary to ensure that all of the important phenomena are accurately modeled over the full range of parameters for which they are important (see Section I below). In the case of RETRAN, I have even more reservations due to the inordinate number of modeling options available in the code and the consequent potential for a large user effect (see Section II). Perhaps a quote from the RETRAN-3D Applications Manual will say this better than I can:

> "Qualification (of the code) is an additional step that lies beyond both verification and validation. Qualification is the process of demonstrating that the code and a specific plant model are adequate for a given application, e.g., analysis of a boiling water reactor response to a turbine trip event for support of reload fuel licensing. Although the code developer can perform generic demonstration analyses to support qualification, completing the qualification is ultimately the responsibility of each individual code user. This statement is particularly appropriate for RETRAN because of the flexibility of the code and because much of the modeling is established by user input."

Similarly, the RETRAN-02 Modeling Guidelines Manual says:

> "In many cases, the data on which the correlations are based do not cover the entire range of conditions encountered during transient analysis of NSSS. Each calculational application of RETRAN should be checked to see that the correlations are being applied with conditions that are appropriate."

I appreciate the candor evidenced by the above quotes. However, with such a large burden placed upon the user (and also the reviewer) for each individual application of the code, I can only restate my reservations about some form of generic approval.

First, it was necessary to gain some overall familiarity with the RETRAN-3D code. This was accomplished largely through perusing the code manuals and the associated documentation. During this process, a number of issues were raised that are not directly

associated with the drift flux model. Nevertheless, I felt it was important to document them (see below). Similarly, because of the questions concerning the RETRAN momentum equation raised by Prof. Wallis, I was asked to take a look at the momentum equation as well. "Taking a look" at the derivation of the momentum equation in RETRAN turned out to be a rather long and frustrating process which adversely impacted the level-of-effort available for the review of the drift flux model.

## I. Code Review Process

I have reservations about any kind of "generic approval" for the use of any thermal-hydraulic system analysis code. To me, the approach laid down in the CSAU methodology makes a lot of sense as a framework within which to examine a code's applicability to a given transient even when the further step of uncertainty quantification is not pursued. The basic steps of this process are given below along with comments regarding the RETRAN review:

- Scenario Specification: this step, together with steps 2 & 3, allows the dominant phenomena to be identified so that the review can be focussed on what is really important. Obviously, this is not the case with the RETRAN review where "all transients except LOCA" must be considered. By not following this process, the burden is shifted onto the reviewer who cannot possibly expend enough resources to review everything in sufficient detail. Consequently, one must rely on one's experience and judgment to try to focus the review in lieu of a scrutable well-ordered process such as that afforded by the CSAU methodology. The result is that the review process becomes more subjective and more dependent on the reviewer's experience.

- Nuclear Power Plant Selection: differences in plant design (e.g., a deep loop seal, vent valves, OTSG or UTSG) can have a pronounced effect on its behavior during a transient. Consequently, the selection of the plant type (e.g., a 4-loop Westinghouse design) is a critical step in determining the dominant phenomena. RETRAN is being reviewed for application to all PWR's and BWR's of US design.

- Phenomena Identification and Ranking Table: with steps 1 and 2, this determines the importance of the phenomena. I have heard the applicant claim that if this were to be done for RETRAN the result would be "the whole code". I am not sure that this is the case. For chapter 15 transients, it is probable that only a handful are dominant for any given transient/plant combination and that this handful would likely be the same for most transients. Having this clearly and defensibly stated would greatly expedite the review process.

- Frozen Code Version Selection: it goes without saying that one cannot review "a work in progress" though it seems that this is being asked in the case of RETRAN 3-D. It is my understanding that an NRR analyst, through using the code, discovered that code development was still underway on the 5-equation thermal non-equilibrium model (and perhaps others as well). Also, in the code assessment manual (volume 4), code calculations are

simply labeled as "RETRAN 3-D" and are not identified with any specific version number from the code's configuration control system. Thus, there is no guarantee that the documented assessment results actually are representative of the current code version (much less some future version after development is completed).

- Provision of Complete Code Documentation: producing comprehensive yet readable documentation for a system's thermal-hydraulic code is a daunting task. To be fair, some of the RETRAN documentation is excellent but other sections are almost unreadable. As regards the provision of "complete" documentation, the RETRAN submittal is remiss in four areas: the manuals are being revised to include current code development efforts and (eventually) assessment results, the topic of scalablity of the models and correlations is not treated explicitly, a user's guide does not exist for the new models included with RETRAN 3-D, and in several cases (and most notably with respect to the drift flux model assessment & applicability) the best documentation is in other references and is not part of the code documentation.

- Determination of Code Applicability: involves the conservation equations, the closure relations, the numerics, and the nodalization. For RETRAN, the code applicability "statement" is dispersed throughout the various manuals and must be pieced together by the reviewer based on their experience. It would greatly enhance the process to have a clear statement identifying the important phenomena for a particular scenario, the relevant physical models, the assessment demonstrating applicability of the models, etc.

- Establishment of Assessment Matrix: from the PIRT results, one determines the range of parameters (pressure, mass flux, quality...) over which a given highly ranked phenomena is considered to be important, then a test matrix is devised to assess the model over this range of parameters. Without this, the reviewer cannot be certain that the model performs acceptably over the required range.

- NPP Nodalization Definition: this step would have to be part of a submittal for an individual plant and is not explicitly part of a code review. However, in my opinion, not enough guidance is given in the RETRAN manual. This is especially the case considering the large numbers of modeling options and "dials" available to the RETRAN user (see discussion on user effects below).

- Definition of Code and Experimental Accuracy: this goes with step 7 above. While some assessment results are available for RETRAN 3-D, the bulk are plant calculations for which detailed measurements are not available so that the assessment is somewhat qualitative.

- Determination of Effect of Scale: when many experiments are small-scale, this is a necessary step to ensure their applicability and I have found little evidence of it in my perusal of the RETRAN documentation. However, the relatively large number of plant transients does give some credibility albeit in a qualitative manner.

The remainder of the steps in the CSAU methodology focus on the quantification of uncertainty and are not relevant here. Although step 12 (Performance of NPP Sensitivity Calculations) would be very useful with respect to informing the reviewer of the needed accuracy for a given model.

In summary, almost none of the criteria used in the CSAU study are satisfied by the RETRAN submittal. Some of these, for example code development being pursued during the code review process and assessment not being up-to-date with the latest code version, are serious enough that I would recommend putting the review on hold. Similarly, unless there is some demonstrated significant impact on the industry due to delaying this review, I would recommend that all T/H code reviews be postponed until the revised accident analysis Regulatory Guide and Standard Review Plan have been issued.

## II. User Effects

Much has been written about the so-called "user effect". Briefly, when multiple users have been given the assignment of modeling one well-documented experiment in an integral test facility (e.g., the International Standard Problems) with the same version of the same code, and with the same information on geometry, test procedures and boundary conditions provided to all participants, the resulting calculations are often startlingly different. This has prompted the international community to consider the formulation of a "code user training and certification program". Indeed, some have maintained that a concentrated training program of 8 to 12 years would be necessary to certify a code user.

Given that the calculated code results can be so dependent upon the user for a code such as RELAP5, what would one expect of RETRAN? I would expect the user effect for RETRAN to be considerably larger than that for RELAP due to the overwhelming number of user options available in RETRAN. For example, there are (at least) three different ways to model a volume: thermodynamic equilibrium, partial non-equilibrium (liquid phase can be subcooled), or the two-region non-equilibrium model. Add to that the options for determining the temperature profile within a volume - the temperature transport delay model, the enthalpy transport model, and the method-of-characteristics – and the options for phase separation within a volume and one can begin to get an idea of the nearly infinite number of combinations of models that the user can select from.

The situation is even more complicated for junctions where there are seven (depending on how one counts) slip options. Furthermore, in at least two of these options, the user can adjust the model's coefficients. In other places, a couple of examples are noted in the sections below, the modeling guidelines acknowledge a modeling deficiency in the code and suggest that the user overcome it by using the control system to adjust a model. With such a large potential user effect, and with the plentiful opportunities to misuse the code to get any desired answer, any RETRAN submittal should be the subject of a very careful review.

## III. RETRAN Validation

I have not performed a review of the entire assessment package and will only make a few specific comments:

- Experiments vs. Plant Transients: there are relatively few separate effects tests and very few integral effects tests in the RETRAN validation manual. Conversely, there are a large number of plant transients. While having extensive validation against plant transients is good, it is equally (or perhaps more) important to qualify the code against a comprehensive set of well instrumented experiments so that the validation becomes more quantitative rather than qualitative.

- Code Version: all validation efforts should specify the exact version of the code used to perform the analysis.

- Uniform Modeling Guidelines: it is my understanding that many of the presented plant transients were performed by RETRAN user group members. Has any systematic review of these calculations been performed to ensure that the code is being used in a consistent manner, i.e., the same modeling options chosen for similar situations?

- SBLOCA Analysis: there were only two SBLOCA integral effects tests simulated. Both were essentially the same LOFT transient and were relatively benign (no core heatup). There were also no separate effects tests involving loop seal clearance, reflux condensation, or the effect of tee off-take on critical flow . Due to the paucity of presented assessment for SBLOCA, I would recommend that RETRAN use be limited to transients that are far removed from the possibility of core dryout and heatup.

- Main Steam Line Break: for an MSLB, steam generator heat transfer and lower plenum mixing (to determine the degree of mixing between the colder loop and the others to produce the core inlet temperature distribution) are important. I did not see any quantitative assessment of these phenomena. Any use of RETRAN for this type of transient would have to justify the modeling approach.

Again, while plant transients demonstrate that RETRAN qualitatively reproduces the major trends, the relative lack of quantitative assessment against well-instrumented integral and separate effects tests leads one to question the code's accuracy.

## IV. Vector Momentum Equation

In his presentation to the ACRS, Prof. Wallis raised a number of questions concerning the derivation of the so-called "vector momentum equation" in RETRAN. Unfortunately, rather than giving a coherent point-by-point defense of their formulation, the representatives from EPRI were only able to say something like: "we had it reviewed and we believe the formulation is correct" and "it has been used in a large number of plant calculations without any problems". Needless to say, this type of response is somewhat unsatisfying.

Due to the high profile given this issue, I have spent an inordinate amount of time looking at the derivation of the RETRAN momentum equation. The reason such a level-of-effort was required is the RETRAN documentation itself. While no one expects a code manual to be as leisurely a read as a novel, I found the RETRAN documentation for the momentum equation to be particularly exasperating. I won't try to document what I found deficient in their approach (that would significantly lengthen this report), but a few of my pet peeves are:

- Pretension to rigor: in an apparent effort to show a rigorous approach, much paper and ink are devoted to forms of the equation containing (for example) three-dimensional fluid shear stress terms and terms accounting for moving solid surfaces, neither of which are present in RETRAN. Besides wasting paper (and time), this approach significantly complicates and obscures the exposition being attempted.

- Mixed, confusing and non-standard notation: the momentum equation derivation begins using the indicial notation common to CFD texts and then transitions without much warning to a very non-standard RETRAN specific notation. While some degree of code specificity is expected, I was surprised at the difficulty I had trying to understand this description despite my experience level with a number of other T/H codes. In particular, the notation changes considerably as one goes through the description so that an equation one understood on one page is almost unrecognizable on another.

- Typos: the numerous sections dealing with the momentum equations are rife with typos, one is often not sure whether there has been a typo or whether or not some new notation has been introduced.

- Distributed description: the description of the momentum equations is strewn over a large number of sections making it very difficult to comprehend.

- Terms missing from nomenclature: a number of terms used in the momentum equations are not to be found in the nomenclature causing a lot of time to be wasted trying to find their definition in the text.

- Missing steps: despite the incredible detail lavished on the initial stages of the derivation, later on, large gaps appear as the interaction terms are defined.

What is needed is a clear and concise statement of the PDE's being solved and the implicit assumptions involved, the process used to volume average the equations, how the resulting volume averaged conservation equations are differenced (both with respect to time and the control volume formulation), and the specification of the interaction terms.

Indeed, I found the RETRAN discussion of the momentum equation to be so confusing that I had great difficulty in trying to ascertain whether or not the deficiencies Prof. Wallis identified were real or whether they were the result of the poor description.

## IV.1 Momentum Flux Terms and Volume Averaged Flowrate

One of the deficiencies noted by Prof. Wallis concerns the definition of the "volume averaged flowrate" in a specified direction that is used in the momentum flux terms of the momentum equation. The definition of this quantity for a generalized 2-D volume with multiple junctions (as illustrated in Figure II.3-6) is given in equation II.3-31 as

$$\overline{W}_{k,\Psi} = \sum_{i \in k} \zeta_{ik} W_i \left|\cos(\Phi_i - \Phi_j)\right| \cos(N_{ik} - \Phi_i) S_{gn}\left[\cos(N_{ik} - \Phi_j)\right]$$

This rather grotesque looking formula defines the mass flowrate at the center of volume "k" that is in the direction $\Psi$. I will not attempt to define all the various terms. Fortunately, almost all of them are simply related to getting the "sign" of the mass flow correct and can be ignored for the discussion to follow. The important variables for this discussion are

$W_i$ = the flow through junction "i"

$\cos(\Phi_i - \Phi_j)$ = the fraction of $W_i$ lying in direction $\Phi_i$ that is the component in direction $\Phi_j$

where $\Phi_j$ is the angle of the junction that is in direction $\Psi$. The generalized form of the momentum equation is given in equation II.3-30 for the ith junction connecting volumes "k" and "l". In this equation, the momentum flux term is

$$\frac{\overline{W}_{k,\Psi}^2}{\rho_k A_k^2} - \frac{\overline{W}_{l,\Psi}^2}{\rho_l A_l^2}.$$

Here we see that the volume average flowrate is squared. This means that the direction cosine term is also squared and this is the error that Prof. Wallis noted.

Going backwards in the RETRAN momentum equation derivation to equation II.2-106, we see that the momentum flux term is given by

$$\sum_j [\rho \tilde{v} v A]$$

where

$\tilde{v}$ - is the velocity vector

$v$ - is the component of $\tilde{v}$ that is perpendicular to the junction area $A$

In other words, we get the product of a mass flowrate times a vector velocity which is correct. Decomposing this for the ith junction into its component of the momentum flux in the $\Phi_j$ direction, we get

$$W_i \cdot |v| \cdot \cos\left(\Phi_i - \Phi_j\right)$$

which is the product of the junction flowrate, the magnitude of the velocity and the direction cosine. Note that the cosine term is not squared as it is in the above equation.

So, a real error does exist in the definition of the momentum flux term in the vector form of the RETRAN momentum equation. How important is it? If we consider a momentum flux coming into a volume at an angle, then some fraction of it will be lost resulting in an artificial pressure recovery. However, because of the conservative formulation used for the mixture momentum equation, this pressure rise will be exactly offset by a subsequent pressure decrease as the flow is accelerated in the new direction.

This is a major advantage of using a conservation formulation for the momentum equation (note that a "transportive" or "non-conservative" form is used for the velocity difference equation). The advantage is that, despite errors in the magnitude of particular momentum flux terms, no artificial "pumps" are created because the momentum flux that is convected out of one junction control volume is exactly (within machine accuracy) convected into another junction control volume. While I cannot guarantee that this is the case in RETRAN solely from looking at the manual, it would appear to be so although the formulation is not "strictly conservative" because the momentum equation has been divided by an area.

I do feel the need to make a comment regarding the usage of a vector momentum equation for a generalized volume with multiple junctions. My interpretation of the RETRAN documentation is that it tries to impress upon the reader that the momentum flux terms are rigorously accounted for no matter how complex the geometry and flow pattern. Of course this cannot be the case. The details of a two-phase flow within a large control volume are not resolved and to believe that one can correctly predict the transfer of momentum from one junction to another is simply wishful thinking. I think that the best that any system T/H code aspire to would be for momentum to be rigorously conserved (in a global sense) so that no artificial "pumps" are induced in the calculation.

## IV.2 Elbow Example and Wall Forces

One of the points Prof. Wallis made concerned the neglect of wall forces and the implicit substitution of pressure forces to make up for it. The elbow example (see section 3.1.2.4 of Volume 1) provides a good illustration of this. Note: the notation of Figure II.3-8 is used in the subsequent discussion.

From the text, the point of the example is to demonstrate that the RETRAN vector momentum equation reduces to equation II.3-39 for a constant area elbow. That is, that the pressure drop across the elbow is due solely to the irreversible pressure losses that are *modeled* by the user specification of a loss coefficient. Another way of putting this would be to say that the discretized form of the momentum equation used in RETRAN does not induce any artificial pressure drop due solely to the difference formulation. Indeed, working through the example, one can see that this is the case.

However, if one examines the state of the fluid as it passes through the elbow - and not just looking at the difference between upstream and downstream conditions – the

discrepancy Prof. Wallis noted emerges. Using the 3 volume example of Figure II.3-8, where volume #2 is upstream of the elbow, volume #3 is the elbow, and volume #4 is downstream, we see that (in the absence of irreversible losses and with constant density) there is a pressure recovery between volumes 2 and 3 followed by an off-setting pressure loss between volumes 3 and 4. Specifically,

$$\Delta P_{2,3} = \frac{3}{4}\cdot\frac{W_1^2}{\rho\cdot A_1^2} = -\Delta P_{3,4}$$

At first glance, this behavior appears reasonable as the x-component of the flow must be decelerated, resulting in a pressure rise, to be followed by an equal pressure drop needed to accelerate the fluid in the y-direction. But is this really what happens? And, if not, how important is it?

Making a number of simplifying assumptions, such as frictionless flow in a channel with no secondary flows etc., the angular momentum equation can be solved for the velocity and pressure distribution at the bend. The result is that on the inside surface of the bend a high-velocity low-pressure region exists whereas on the outside of the bend a low-velocity high pressure region is found. So, how does this compare to the volume averaged pressure of RETRAN?

Basically, it doesn't. With the above assumptions, the channel is a stream tube for which the Bernoulli equation can be applied and one can show that the pressure at the bend is the same as that at the inlet (i.e., no head loss has occurred). What has happened in the RETRAN example, as Prof. Wallis rightly concluded, is that the wall force on the fluid has been ignored, where from the linear momentum equation, for this example, the x- and y-components of the wall force on the fluid are

$$F_{w,x} = -\frac{W_1^2}{\rho\cdot A_1} = F_{w,y}$$

If these forces had been included and the differencing of the momentum flux terms introduced no error, then the pressure at the bend should equal that at the inlet and outlet.

In retrospect, it is curious that this discrepancy arose. In other sections of the momentum equation derivation, see the discussion on flow area changes, much use is made of the mechanical energy equation (from which the Bernoulli equation derives) to check the differencing of the momentum flux terms. If the same approach had been applied here, the omission of the wall force would have become obvious.

One question is where in this long and tedious derivation does the error occur. Equations II.2-37 and II.2-70 give

$$P_i^{wa} = \int_{S_{wa}}\left[p_a n_i^a - \sigma_{ik}^a n_k^a\right] dS$$

for the "contribution due to the interaction between the walls of the control volume and the phase or fluid 'a'." This integral is over the surface area of the walls that is in contact

with fluid 'a'. The variable $n_i^a$ represents the ith component of the unit normal vector to the wall surface. The first term of the integrand yields the wall force discussed above and the second term yields the viscous stresses. At this point in the derivation the wall force is still present.

However, in equation II.2-70, this interaction term is divided into two components

$$P_i^{wa} = \int_{S_{wa}} \left[ p_a n_i^a - \sigma_{ik}^a n_k^a \right] dS = F_{i,f}^{wa} + F_{i,G}^{wa}$$

where,

$F_{i,f}^{wa}$ = ith component of the wall pressure and viscous forces for friction

$F_{i,G}^{wa}$ = ith component of steady-state and transient forces for the flow channel geometry

This is still ok, but in subsequent equations the term $F_{i,f}^{wa}$ is treated solely as a wall friction term and $F_{i,G}^{wa}$ is the force due to "local" loss coefficients. So, the contribution due to the pressure force normal to the wall is simply forgotten about.

The next question is then whether or not this "discrepancy" is important. That a non-physical pressure rise is predicted to occur at the bend is real but what are its effects? For example, for a compressible two-phase flow, this pressure rise might well cause some condensation, changing the quality and decelerating the mixture. Is this important? For most classes of problems, probably not, and one could argue that this error is of the same magnitude as that incurred by the usage of a 1-D formulation that ignores the pressure distribution across the pipe. But as with other items I have observed in the course of this review, this lack of consistency and apparent sloppiness in the derivation raises the question of credibility and of what else might be wrong.

Of course, RETRAN (like RELAP5) does not really resolve a 3-D spatial representation of a piping network where every bend and twist must be specified. Instead, most of the calculations depend solely on the pipe length and elevation change. Thus, the user has a choice of whether or not to input an angle for a junction that (in reality) represents an elbow. Indeed, it is noted in section 3.6 of the "Modeling Guidelines" volume for the RETRAN-02 code that "for elbows, the momentum losses are generally modeled through the junction from loss coefficients instead of using the junction angle." Should the user then let the code default of zero degrees be used, this artificial pressure recovery/loss will not occur. This is not because the wall force terms have somehow reappeared but rather that the turning of the fluid momentum is not calculated. Instead a loss coefficient is input to handle the irreversible pressure losses.

This is exactly what happens in the RELAP5 code (and I believe in TRAC as well, though I haven't checked). A user will input a junction angle with respect to the vertical so as to conserve the elevation gain. However, the resultant change of direction for the fluid is not treated in the momentum flux terms so this "discrepancy" does not occur.

## IV.3 The Tee and the Jet Pump

There is no special component model for a "tee" in RETRAN. Instead, a fluid computational volume with multiple junctions is used and reliance is placed on the vector momentum equation. Consequently, there is no section in the RETRAN-02 modeling guidelines to describe how a "tee" should be modeled. In section 3.6 of this volume it is stated that: "The junction angle can be used to model the vector nature of the momentum flux terms in the momentum equation. It can be used to represent converging or diverging streams to or from a single control volume such as tees, manifolds, or multi-dimensional regions." From this, and the tee example given in the theory and numerics volume (section 3.1.2.4) one is led to conclude that RETRAN can accurately model the momentum mixing that occurs in a tee.

In light of the discrepancies uncovered in the vector momentum equation described above, one must then question whether or not this is indeed the case. One piece of evidence that a tee is not correctly modeled solely from the usage of the junction angles can be discerned from a perusal of the documentation on the jet pump model.

First, let me digress a little and give some background on the jet pump model development in the TRAC code. The BWR version of TRAC (TRAC-BF1) was a direct offshoot of the PWR version (TRAC-PF1/mod1). Early in its development, all of the models were the same. Then, special BWR component models were added and some of the constitutive models were changed to better represent BWR conditions. The jet pump model was built off of the TRAC-PF1 tee model. However, it was discovered that the TRAC-PF1 tee model did not account for proper mixing of the momentum for two converging streams. So, "tee momentum source" terms were added to the jet pump model to overcome this deficiency. The current version of TRAC-P, however, does model these momentum fluxes properly so that when the BWR jet pump model was "consolidated" into the TRAC-M code, these special "tee momentum source" terms were not needed.

So, what is the case with the RETRAN jet pump model? As was the case for TRAC-B, special terms are added to properly account for the mixing of the momentum for the two converging streams (see section VI.1.2). Indeed, it is noted that: "The momentum mixing (jet pump) model can be applied to any instance in which two streams are mixed, and is not limited strictly to a jet pump." To me, this sounds (in a back-handed kind of way) like the "momentum mixing model" should always be used for tees. Although, I am not sure whether or not there is a built-in assumption about the angle of the drive line.

While perusing the RETRAN-02 modeling guidelines looking for how to model a tee, I did discover some guidelines concerning the tee formed by a PWR hot leg and the pressurizer surge line. There is no guidance given regarding momentum flux modeling however there is a paragraph concerning the specification of loss coefficients. The assertion is made that the loss coefficients are functions of the flow split and can't simply be input as constant values (as is normally done). It is suggested that the user can model this dependence using the control system so that the loss coefficients can change as a function of time. Needless to say, I find it somewhat alarming that the user is expected to "program" this functional dependence for loss coefficients via the control system. This is an item that would need close review for every input deck.

## IV.4 Spatial Differencing of the Momentum Flux Terms

In equation II.3-31, the definition of the "volume average flowrate", I did not bother to define $\zeta_{ik}$. This term is a weighting factor that determines the nature of the differencing for the momentum flux terms. See the discussion on page II-86 of Volume 1 for its definition. The RETRAN-3D default is to use the "arithmetic average" formulation which is essentially a second order accurate central difference scheme. The "donor cell" formulation, which can be selected as a user option, is a first order accurate scheme. It is noted in the RETRAN manual that "for some types of problems, notably where the momentum flux is important and friction is not, the solution technique in RETRAN-02 can be unstable for an arithmetic average, and using a donor cell has provided a stable solution."

It is not stated whether or not this is the case with RETRAN-3D but one imagines that it is as no specific effort was made to stabilize this formulation (at least that I noted). If so, I believe that the usage of a central difference in RETRAN is a bad idea. In effect, one is giving the user a code that can be unstable and then telling the user to watch out for it and change the momentum flux differencing option (on a junction by junction basis) if it occurs. I would imagine that there are RETRAN calculations out there that begin using the arithmetic average formulation and then during the transient, the user restarts the calculation with the donor cell option invoked. For such a scenario, how can a regulator be expected to perform an effective review?

## IV.5 Temporal Differencing of the Momentum Flux Terms

I have not reviewed RETRAN with respect to its capability to perform stability analyses (the reason that the method of characteristics[1] technique was implemented for the junction enthalpy). Consequently, the following is more a question than a finding. The solution scheme (as I understand it) is implicit even if the time step is reduced below the Courant limit. Implicit schemes are well known for being diffusive. I would expect that if a user selects the "donor cell" formulation for the momentum flux terms that the implicit differencing would introduce an artificial damping on the solution. Has this been observed when looking at stability analyses? Is there any guideline that forces a user to always use the second order arithmetic average scheme when performing a stability analysis? If we are considering approving the usage of RETRAN for stability analyses, this should probably be looked at.

## IV.6 Momentum Transfer Due to Phase Change

In the phasic momentum equations there is an interaction term to account for the momentum transfer associated with phase change. A general definition of this term is given in equations II.2-38 and II.2-39. When the phasic momentum equations are added to yield the mixture momentum equation, the vapor and liquid components of this

[1] The method of characteristics technique is explicit and requires that the time step be less than the Courant limit. Is this enforced via the RETRAN time step control? Or does the user have to impose a maximum time step small enough that the Courant limit is never exceeded?

momentum flux cancel out each other. For the velocity difference equation, also known as the dynamic slip equation, the magnitude of this term should be doubled.

However, in the RETRAN formulation (see equation II.2-113 and the paragraph preceding it) this term is neglected without any justification being given. How serious is this omission? What are the potential effects?

If one uses a conservative formulation for the momentum equations, then the effects of neglecting this term are readily apparent. Specifically, one allows interphase mass transfer but does not account for the associated momentum transfer. Thus the donor phase (the phase that is losing mass), will maintain its momentum while losing mass so that it will accelerate. Conversely, the phase that is receiving the mass transfer will artificially slow down because its mass has increased without any increase in its momentum. Normally, this is not a significant error and is masked by all the other modeling uncertainties such as the interfacial drag. However, as a phase is being depleted, that is its mass is going to zero, small momentum imbalances can result in large velocity changes and I would expect that RETRAN occasionally suffers from this behavior. In my experience, it is omissions such as this that cause code developers to go to great lengths to devise "fixes" to problems they need not have faced.

## V. Bubble Diameter

In the discussion on interphase friction, see Section 5.1.2.3, the maximum particle size is given as a function of a critical Weber number as

$$We_{crit} = \frac{2 \cdot r_{max} \cdot \rho_c \cdot (V_c - V_d)^2}{\sigma}$$

For bubbles, the critical value of the Weber no. is given as 1.24 (eq. III.5-16b) and the paper of Sevik & Park (1973) is given as the reference for this value. This is clearly incorrect.

When one reads the paper of Sevik & Park, one discovers that the value of 1.24 for the critical Weber no. is indeed given, however, **the definition of the Weber no. is different.** In RETRAN-3D, the form of the Weber no. uses the phasic relative velocity difference, whereas Sevik & Park define

$$We_{crit} = \frac{r_B \cdot \rho_c \cdot \bar{v}^2}{\sigma}$$

with

$$\bar{v}^2 = 2.0 \cdot (\varepsilon \cdot D_B)^{\frac{2}{3}}$$

where ε is the turbulent energy dissipation per unit mass and time. The value of 1.24 is then deduced by equating the natural frequency of the lowest fixed volume oscillation of a bubble with a characteristic frequency of turbulent flow . Hence, the physical phenomena controlling the bubble diameter is different between the two cases.

In this simple equation for bubble diameter, there are then two errors. First, and most importantly, the definition of the Weber no. is misconstrued as the relative velocity in the RETRAN formulation has nothing to do with the turbulent energy dissipation used by Sevik & Park. Second, there is also a factor of 2 discrepancy between the two formulations as the Sevik & Park value of 1.24 was based on the bubble radius rather than the diameter.

It is unclear how important this estimation of the bubble diameter is to the prediction of void fraction (via interphase drag) by RETRAN when using the "dynamic slip model" with the junction slip flag ISFLAG = 1. However, errors of this type undermine the credibility of the technical base upon which the code is built. That is, if the code authors could have such a fundamental misunderstanding in this one case, what other errors are waiting to be uncovered?

Furthermore, the interfacial area is based on an "average" particle size that is given by

$$\frac{r_p}{r_{max}} = 0.06147$$

according to a formula reported by Moeck (1970) for droplet size distributions. This value is considerably smaller than what one expects if the Sauter mean were to be used. Indeed, later in the RETRAN manual when interfacial heat transfer is being described, the interfacial area is calculated using the ratio

$$\frac{r_p}{r_{max}} = \frac{3.6}{6} = 0.6$$

which gives an order of magnitude difference. Also, for the estimation of bubble diameter for interfacial heat transfer, a value of 10 is used for the critical Weber no. Thus, for the same phasic relative velocity difference, the bubble diameter used for the calculation of interfacial friction is almost two orders of magnitude smaller than that used for the calculation of interfacial heat transfer. Somewhat curiously, this is just the opposite trend from what one expects as the interfacial heat transfer is primarily governed by small bubbles (large interfacial area) and the void propagation is primarily affected by large bubbles (large relative velocity).

The net result of all this is that one would expect the interfacial shear for bubbly flow (when ISFLAG = 1) to be over-predicted and the resulting void fraction to also be over-predicted. How important is this? How often even is this slip option selected by RETRAN users[2] ? As stated above, fundamental misconceptions and inconsistencies of this type, if nothing else, undermine the code's credibility.

## VI. Pressure Drop – Form Losses

The irreversible losses associated with flow area changes (expansion/contraction), orifices, bends, etc. are accounted for by the addition of a form loss term to the mixture momentum equation. This irreversible "local" pressure drop is given by

[2] Note that in the separate effects assessment for pressure drop at flow area expansions/contractions

$$\Delta P_{loc} = \frac{1}{2} \cdot K \cdot \frac{G^2}{\rho}$$

where the density is that of the mixture for two-phase conditions. It should be noted that the form loss coefficient, $K$, is either:

- a user input constant,
- a constant value determined by the Borda-Carnot formula for sharp-edged expansions/contractions,
- a constant value determined by the steady state initialization algorithm, or
- a value determined by the control system during the transient,

and that no provision is made for either a two-phase multiplier or for Reynolds number dependence. Both of these omissions bring into question the accuracy of this formulation when applied to off-normal conditions as discussed below.

Considering two-phase conditions first, one can rewrite the above equation as

$$\Delta P_{loc} = \frac{1}{2} \cdot K \cdot \frac{G^2}{\rho_f} \cdot \Phi_{fo}^2$$

where a two-phase multiplier relative to the entire fluid (liquid + vapor) flowing as liquid has been introduced. Comparing these two equations, it is evident that the RETRAN formulation has an implicit two-phase multiplier equal to

$$\Phi_{fo}^2 = \frac{\rho_f}{\rho} .$$

If the value of the junction density is calculated with the no-slip option, the above reproduces the two-phase multiplier of the HEM model. While, one could find a more accurate formulation for a particular flow combination of geometry and flow regime, it is common practice to use the HEM multiplier. So, what is the problem here?

The problem, as I see it, is that the effective two-phase multiplier is dependent upon the user's choice of two-phase flow model from amidst the myriad of options available in RETRAN. If any of the various slip options (algebraic or dynamic) is selected, then the effective two-phase multiplier will lower than that given by the HEM model without the user having made a conscious choice that this is appropriate. Furthermore, any efforts to validate the pressure drop model through SET assessment would then need to examine the effects of using the various different two-phase flow models.

Let's take a look at the SET assessment results for two-phase pressure losses presented in the RETRAN manual (volume 4). A total of four cases for losses at sharp-edged expansions/contractions are presented. At first glance, the results for contractions (see Figures IV.1-1 & IV.1-2) look pretty good. For Test 7D-4 however, RETRAN over-predicts the pressure drop but the reader is assured that "the maximum error was less than 1.5 psia." At this point, I would like to comment that while in general

I find the RETRAN Applications Manual to be fairly frank, this is simply not the case here. If we focus on the pressure drop across the contraction only (subtracting out the contribution of wall drag for the regions immediately upstream and downstream), we see that RETRAN predicts a value of ~1.96 psi versus a value of 1.23 psi for the data, an over-prediction of ~60%. Note that this pressure drop includes both the recoverable and irrecoverable losses so that the prediction of the irreversible losses is even worse than this value implies. Similarly, one notes that the pressure drop due to wall friction is over-predicted by about 50%.

I will not repeat this exercise for the case of an abrupt expansion, see Figures IV.1-3 and IV.1-4, except to say that for this case, RETRAN significantly over-predicts the pressure recovery. Despite this, the total pressure drop for the test section is reasonably well predicted due to the compensating error of over-predicting the pressure drop due to wall friction. So, what are the implications? It is clear that the two-phase pressure losses calculated by RETRAN are not as accurate as implied in the assessment manual, but can one at least say that they are always somewhat conservative with respect to calculating the flow through the core as the flow transitions from a single-phase forced to a two-phase natural circulation mode?

It is impossible to answer this question because the two-phase form loss model is dependent upon the user selected two-phase flow model as noted above. Thus, it is fair to ask what conditions were covered in the experiments used in the assessment and what code modeling options were chosen. The RETRAN applications manual is good about specifying what modeling options were chosen (at least for simple SET cases like this one) but does not list the experimental conditions for the tests actually simulated (instead the ranges for the entire experimental program are given). I will not digress into a lengthy discussion of whether or not the assessment is sufficient to demonstrate model applicability (see the section on Code Qualification above) but instead will focus on the model selection for this particular assessment effort. However, it should be noted that the pipe diameters considered here range from 0.49 to 0.59 inches, which of course raises the question of scalability.

Returning to the effects of two-phase model selection upon the results, the RETRAN manual states that this assessment used the dynamic slip model with the interfacial friction coefficient calculated from the "vertical bubble flow regime". This RETRAN option calculates the interfacial shear as the drag on a population of bubbles (the diameter of which is calculated using the flawed critical Weber no. criterion discussed above) irrespective of the actual flow conditions. For all four of the cases chosen for this assessment, the measured void fraction upstream of the area change was greater than 85%. Clearly, the actual flow regime would be annular-dispersed rather than bubbly and the resultant slip would be considerably greater. Why would the analyst force the flow regime to be bubbly for these high void fraction conditions? How much better or worse would the results be is another modeling option had been chosen? How should a user specify the loss coefficient for a flow area change in a real plant? If the user lets the steady state initialization algorithm select a loss coefficient so that a measured pressure drop at operating conditions is matched, is this loss coefficient then relevant when the flow is coasting down and has become two-phase? The point of all this is that the SET assessment for two-phase pressure drop is irrelevant and insufficient.

The other potential modeling deficiency noted above for the form loss model is the lack of ability to input a form factor with a Reynolds number dependency. I will first note that

the ACRS subcommittee on T/H considered this to be a very important issue during the assessment of the applicability of the RELAP5 code to the AP600 design. Next, consider the one natural circulation SET presented in the RETRAN manual (see section 6.1 of Volume 4), a single-phase case from the Purdue thermosyphon test facility. When the default wall drag model was used, the natural circulation rate was significantly over-predicted. Indeed, to bring the RETRAN predictions in line with the data the wall drag for the laminar flow regime was increased from the default value of

$$f_{lam} = \frac{16}{\mathrm{Re}}$$

to

$$f_{lam} = \frac{151}{\mathrm{Re}^{1.17}}$$

If a Reynolds number dependency were included in the form loss specification, would such a dramatic change in the wall friction be necessary? More importantly, what are the implications of this for a plant analysis?

## VII. Pressurizer Modeling

RETRAN has a special "two-region non-equilibrium" pressurizer model. I am in favor of developing component specific models that more accurately represent the physics occurring within a component rather than using the same numerics and models for every volume/junction no matter what type of component they represent. I did not review the RETRAN pressurizer model, although I am aware that much criticism has been raised against it in past reviews. The point to be raised here has to do with modeling guidelines and the user effect.

When perusing the applications manual, I looked at the comparisons of the Shippingport pressurizer tests. The first thing to note is that a significant difference exists between the RETRAN results obtained with implicit numerics and a curve labeled "standard" which matches the RETRAN-02 results better. No discussion of this is given. Should a user not use implicit numerics if they use the pressurizer model?

The real surprise came, however, when I turned to the next pressurizer assessment case: the MIT pressurizer tests. Here, instead of modeling the pressurizer with a single non-equilibrium volume (as was done for Shippinport), a 1-D stack of 10 equal axial cylindrical volumes was used. What is going on here? To compare better with a small-scale SET, one use 10 volumes, but then only one for a full scale plant?

The answer was found when I looked in the RETRAN-02 modeling guidelines. The non-equilibrium model is portrayed as necessary when the transient involves an insurge (especially of subcooled liquid), or when modeling sprays or the pressurizer heaters. However, "during insurge transients, the lower (mixture) region will tend to subcool as a result of the incoming hot leg fluid. The RETRAN model of the mixture region assumes complete instantaneous mixing of the fluid in the region. This assumption may be questionable in a transient where the insurge is of a fairly short duration followed

immediately by an outsurge. This response is fairly typical of loss of load or turbine trip types of transients."

For these types of transients, a multi-volume modeling approach employing the temperature transport delay model is suggested. However, the caveat is given that "care should be exercised in selecting the size of the volume that uses the temperature transport since if the pressurizer level dropped to the top of this volume, the intent of the pressurizer model itself would be defeated. Experience with this modeling approach is limited...". So, for different transients, different noding of the pressurizer is needed?

The point of the above is that RETRAN results are very dependent upon the user. Good modeling guidelines are needed and each individual use of the code must be carefully reviewed.

## VIII. Critical Flow

I will not go into an extensive review of the critical flow models (there are three!) available in RETRAN as Prof. Schrock has already done so. However, I do want to point out that, as is noted in the RETRAN manual, none of these models is appropriate for conditions where non-condensable gases are present. According to the manual: "when non-condensables are present ... the critical flow model is bypassed, regardless of the choking options specified."

For the majority of transients for which RETRAN is used this does not cause a problem. However, this could limit the code's applicability to exclude shutdown transients where air can be present in the system. Also, I would consider it bad practice for a code to bypass a model calculation automatically unless it "raises a flag" to warn the user that this is the case. Does RETRAN have such a user warning when the critical flow model is bypassed?

## IX. Drift Flux Model – Chexal-Lellouche

This was originally supposed to be the main focus of this review, however, as stated above, I ended up spending a disproportionate amount of time on the derivation of the RETRAN momentum equation. Consequently, some of my planned activities, such as performing some independent assessment of the Chexal-Lellouche drift flux model were curtailed. Therefore, the following review is based entirely on the documentation provided and not upon an independent investigation.

One could say that there is "good news" and "bad news" about the Chexal-Lellouche model. The bad news is that it is not a mechanistic model but rather an elaborate curve fit (see discussion below). The good news is that this curve fit is based on a large and fairly comprehensive data base. I will first comment on the model itself with respect to fluid scaling, annular flow and horizontal flow, then discuss CCFL and the model's validation.

## IX.1 Fluid Scaling

The first real surprise in this review came when I discovered that the Chexal-Lellouche model has a so-called "fluid parameter" that directly affects the value of the distribution coefficient, $C_0$. If the model were mechanistic in nature, or even based on the appropriate property groups, fluid scaling would be implicit in the model. That is not the case. The fluid parameter is a set of empirical relations, see equations III.5-91d through III.5-91l in Volume 1 of the RETRAN documentation. I have also attached pages 4-12 through 4-15 of the "Void Fraction Technology for Design and Analysis" brochure[3]. These pages detail the fluid parameter and also show its dependence as a function of void fraction. Note the dramatically different behavior for "steam-water" versus "air-water" and recall from equations III.5-91a & III.5-91b that the distribution coefficient is directly proportional to the fluid parameter.

The RETRAN manual is up-front about referring to the Chexal-Lellouche model as "an empirical model", so what is the difficulty about having empirical "fluid parameters"? There are three potential concerns:

- Range of Applicability: for steam-water, the fluid parameter is an explicit function of pressure, not a function of fluid properties. Due to its empirical nature, it cannot be extrapolated beyond its database. However, this is not much of a problem as the diabatic steam-water data base went up to pressures of 150 bar, and the adiabatic up to 180 bar.

- Applicability of Air-Water Validation: a fair amount of the validation, especially in the case of CCFL, was performed for air-water. Because of the fluid parameter, one has to consider this as the validation of a separate and distinct model. That is, validation performed for air-water is not relevant to the steam-water case used in most RETRAN calculations.

- Slip with Noncondensables: if different fluid parameters are used for steam-water and air-water cases, what is used in the case when the gas phase is a mixture of steam and noncondensable gases? If the steam-water parameter is used, how is this justified?

I recognize the need for a high degree of empiricism in the constitutive models used in a reactor system thermal-hydraulic analysis code. The hardware and the expected flow conditions are simply too complex for everything to be derived from "first principles". However, I do feel that correlations should be based on the correct fluid property groups and non-dimensional parameters, to do otherwise is, in my opinion, not the best engineering practice.

## IX.2 Applicability to Annular Flow Regime

Normally, the drift flux model is used for vertical flow where the two phases are tightly coupled. Indeed, this is the fundamental premise behind the drift flux model, namely that

[3] Note: most of the detailed information on this model, that is the parameter range of its data base, validation cases etc. are in this reference and not in the code manuals.

the buoyancy and interfacial forces balance each other as is usually expressed by using the formula for bubble terminal velocity as the drift velocity

$$V_{gj} = V_{B,\infty} = k \cdot \left[ \frac{g \cdot \Delta\rho \cdot \sigma}{\rho_l^2} \right]^{\frac{1}{4}}$$

In my experience with other codes, viz. RELAP5 and CATHARE, a drift flux model may be used for the bubbly-slug flow regime but a different model is used for the annular flow regime. The reason for this is that in annular flow, the buoyancy force becomes progressively less important and it is the wall shear that begins to offset the interfacial friction. There are drift flux models, such as that of Ishii, where the drift velocity is a function of flow regime. For example, for annular flow, Ishii gives

$$V_{gj,ann} = \frac{1-\langle\alpha\rangle}{\langle\alpha\rangle + \left( \frac{1+75\cdot(1-\langle\alpha\rangle)}{\sqrt{\langle\alpha\rangle}} \cdot \frac{\rho_g}{\rho_f} \right)^{\frac{1}{2}}} \cdot \left\{ \langle j\rangle + \left[ \frac{g \cdot \Delta\rho \cdot D_H \cdot (1-\langle\alpha\rangle)}{0.015 \cdot \rho_f} \right]^{\frac{1}{2}} \right\}$$

In the denominator of the first term on the right hand side, the Wallis interfacial friction factor for annular flow is clearly visible.

The case is not so clear cut for the Chexal-Lellouche model. Here the bubble rise velocity is retained as the cornerstone of the drift velocity, that is,

$$V_{gj} = 1.41 \cdot \left[ \frac{g \cdot \Delta\rho \cdot \sigma}{\rho_l^2} \right]^{\frac{1}{4}} \cdot C_1 \cdot C_2 \cdot C_3 \cdot C_4$$

The added multipliers adjust the drift velocity for all the various other cases, annular flow, horizontal flow etc. Unfortunately, the formulas for these multipliers are once again elaborate curve fits and do not represent (at least not clearly) the governing physical phenomena.

Retaining the bubble rise velocity as a principle component of the drift velocity for flow regimes where it is clearly not relevant cannot help but raise a question as to the applicability of the model itself. One supposes, because of the large database behind the Chexal-Lellouche model, that one or more of the multiplying factors must compensate for this error, but how? and how accurately?

The code application manual (Volume 4) gives only a few cases for which the measured void fraction is above 80% and the flow regime may be annular (see Figures IV.4-9 through IV.4-11 attached). From these figures, one can conclude that RETRAN does a good to excellent job of predicting the void fraction for nominal BWR conditions for the bubbly-slug (or churn turbulent) flow regime. However, at high values of the void fraction, a consistent under-prediction is evident. If these plots were of the liquid fraction (or film thickness) then the error would be strikingly evident. This is only a small sample, and perhaps other assessment studies show a better comparison at high void fractions.

As with all of the above code deficiencies, it is difficult to quantify how important this is. Overall, the Chexal-Lellouche model is probably more accurate than most. However, until I see more assessment results in the annular and annular/mist flow regimes, I have to question its applicability for these regimes.

## IX.3 Applicability to Horizontal Flow

The concern as to the applicability of the Chexal-Lellouche model with respect to horizontal flow has the same origin as the discussion on annular flow above. Namely, that the balancing forces are now wall drag and interfacial friction, so that using the bubble terminal velocity as the foundation of the drift velocity is incorrect. Again, one must assume that there is a compensating factor in one of the multipliers but it is not immediately obvious. For horizontal two-phase flow, a further complication is the propensity for the flow to stratify, especially for large diameter pipes.

Given that the Chexal-Lellouche model is completely empirical, does its two-phase horizontal flow data base cover the regimes of interest for LWR applications? The answer appears to be no. First, recall that the "fluid parameter" and hence the distribution coefficient is different for air-water and steam-water systems. Thus, any validation or model development based on air-water data is questionable when applied to a steam-water system. Also, the majority of the Chexal-Lellouche model's horizontal flow data base is air-water. Indeed, all of the inclined flow data are air-water.

Looking at the steam-water data base (see the attached page 5-7 from the "Void Fraction Technology..." booklet), one sees that three tube diameters are included covering the range from 22 to 75 mm, so that the largest is less than 3 inches in diameter. This clearly is too small to represent the large diameter pipes found in reactor systems, as one would see a type of slug flow in the small tubes at conditions where the flow would be stratified in a large pipe. If one were to include the air-water data base (see page 5-9), ignoring the disqualifying effect of the fluid parameter, the largest diameter is 127 mm which is probably large enough, however, for this test the mass flux range is so high (3600-4700 kg/s-$m^2$) that the regime would be dispersed. So, in my opinion, the data base used in the Chexal-Lellouche model is insufficient for an empirical model for horizontal two-phase flow in reactor scale piping.

## IX.4 Applicability to Steam Generator Tubes

It is well known that there are large differences between tubes and rod bundles of the same hydraulic diameter for the bubbly-slug two-phase flow regime. Specifically, in small diameter tubes, slug flow occurs limited by the tube diameter, whereas in a rod bundle large vapor structures form that span numerous subchannels. Indeed, early drift flux models for rod bundles used the diameter of the bundle housing as a correlating parameter instead of the hydraulic diameter. The Chexal-Lellouche model uses the hydraulic diameter and makes no provision for differences between tubes and rod bundles.

If the model well represents the vertical two-phase flow in rod bundles, how is it also able to represent the conditions in a relatively small diameter tube such as a steam

generator tube? Note that there are several tubes of appropriate size in the model's steam-water data base.

## IX.5 Applicability to the Counter-Current Flooding Limit

It is claimed that the Chexal-Lellouche model automatically reproduces the CCFL condition. Although there is a significant amount of CCFL data in the model's data base (see page 5-10), I have several concerns:

- Geometry Effects: CCFL at complicated geometries such as tie plates requires a highly empirical correlation specific to that particular geometry. The Chexal-Lellouche model makes no distinction between a tube or a tie plate. The only input geometrical parameter is the hydraulic diameter. How can the model sufficiently resolve complicated geometries?

- Air-Water Data Applicability: due to the fluid parameter, one must exclude all air-water data from the model's validation.

- RETRAN Validation: there are no data comparisons for CCFL given in the RETRAN applications manual (Volume 4). All of the comparisons are given in the Void Fraction Technology booklet. It is unclear if or how CCFL is implemented in RETRAN. It is my understanding that the data comparisons given in the booklet were obtained using a driver code and not with RETRAN. Has any CCFL validation been performed with RETRAN?

- Pressure Dependency: based on the data comparisons given in the Void Fraction Technology booklet, see attached Figures 5-6 and 5-10, the pressure dependency of the predicted CCFL seems to be incorrect. In Figure 5-6, which is plotted using the Kutateladze no., the pressure effect is greater over-stated in the calculation, and in Figure 5-10, plotted using the Wallis scaling, the predicted pressure trend is opposite that of the data.

For all of the above reasons, I recommend that RETRAN usage be restricted temperature situations where CCFL is not important unless validation for the precise geometry and expected flow conditions has been performed satisfactorily.

## IX.6 Validation of the Chexal-Lellouche Model

In the RETRAN Applications manual (Volume 4) there is some assessment for void fraction in rod bundles presented. Most of the comparisons are at nominal BWR operating conditions and one case is at SBLOCA boil-off conditions. In addition there are a few plots from one GE 1-foot and one GE 4-foot level swell tests (note: the level swell comparisons don't look very good).

The bulk of the validation for the Chexal-Lellouche model is given in the Void Fraction Technology booklet. The data base used in the development of the model is quite extensive (except as noted above) and well documented. Furthermore, I was very

pleased to see the "statistical comparison", see for example the attached Table 5-8. I have only two concerns:

- RETRAN Validation: as I understand it, the statistical comparisons were done using a driver code outside of RETRAN, how can we be assured that the drift flux model implementation within RETRAN yields the same results?

- Statistics and Trends: while I applaud the statistical comparisons giving mean error and standard deviation for each data set, I feel that statistics can hide some deficiencies and overall trends. For example, suppose that for one data set the bulk of the measurements is at low void fraction and is well predicted, however at high void fraction there are few data points and the comparisons are terrible. The statistics will mask this deficiency and obscure the trend of degrading accuracy as void fraction increases. What is needed are scatter plots of error versus the parameters of pressure, mass flux, heat flux, and void fraction.

In summary, within its data base, the Chexal-Lellouche model is probably accurate enough for most applications. However, due to its empirical nature, care must be taken to avoid extrapolation. Also, for some applications (see above) such as annular flow or stratified flow in large pipes, more validation needs to be performed.

## X. Phase Separation at Tees

For some transients phase separation at tees might be important (though without a PIRT I don't know if that is the case for RETRAN). RETRAN has no specific model for phase separation at tees or for "horizontal stratification and entrainment" as in RELAP5. Instead, there is a "bubble rise" model (also referred to as a "phase separation" model) that can be applied to any volume to affect the junction void fraction. Originally, in RELAP2, this model was developed to account for the separation of a flashing two-phase mixture which occurs during the depressurization of a vertically oriented vessel.

When invoked by the user, this model uses two input parameters – the bubble rise velocity and a "bubble-gradient parameter" – to determine the location of a separated interface and the void fraction distribution below this interface. For the bubble rise velocity, the user can select either the Wilson bubble rise model, specify a constant value, or use a control block to vary its value. Similarly, for the bubble-gradient parameter, the user can input a constant value (no default value is provided) or use a control block to vary its value.

With such freedom to define the "physics" that the code is using, a user can generate practically any desired result. There is no way this can be reviewed external to the context of a particular simulation. Also, because this model was developed for a "large stagnant vertical volume", its usage for any other situation must be treated with caution.

In summary, for any transient where phase separation at tees is important (e.g., the vapor pull-through or liquid entrainment at a small break) the RETRAN model is deficient and bounding calculations would be needed.

## XI. Steady State Initialization

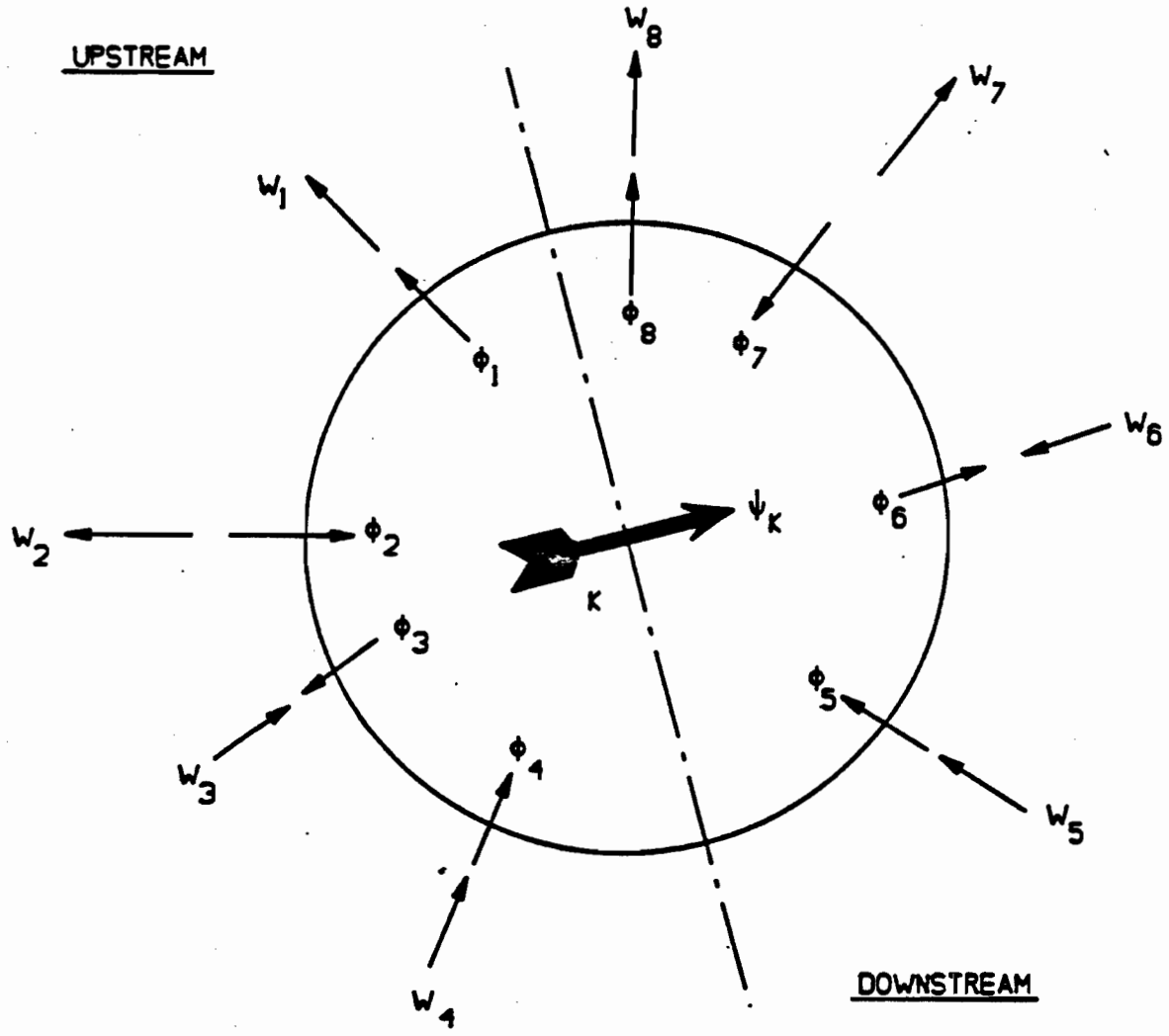
I have not reviewed this model but want to reiterate one caveat. As part of the steady state initialization option, the user can specify a pressure drop (e.g., across the core) and have the initialization adjust the loss coefficient to match the specified pressure drop. This is a very powerful user convenience. However, as stated in the RETRAN-02 user guidelines: "Computed loss coefficients should be inspected to determine that the initialization has produced reasonable values." I would like to place this caution in large bold type. If the user (or reviewer) does not check these values, erroneous results can be generated.

## XII. Boron Transport

There are several models in RETRAN to minimize numerical diffusion or provide front tracking for fluid temperature fronts: the method-of-characteristics, the transport delay model, and the enthalpy transport model. Each of which is used in a particular circumstance as a user option. Boron transport is handled as a passive contaminant by the "general transport model", see Section VII-5.0 of Volume 1. This model uses a first order accurate upwind difference scheme with an implicit temporal differencing. This approach is well known for being highly diffusive especially if the Courant limit is exceeded. I found no reference to any special model to overcome this large amount of numerical diffusion.

In the RETRAN-02 Modeling Guidelines, it is stated that: "Injecting a material into the coolant can cause material front, e.g., injection of boron for reactivity control. Unlike the pressure wave and thermal fronts, RETRAN does not contain a hydrodynamic model for material fronts. These processes can be modeled with RETRAN's control system components." Apparently this "guideline" has been superseded by the general transport model, and while this is a step forward, I was surprised that, considering the effort to implement 3-D kinetics, that such a highly diffusive scheme was used for boron transport (or conversely, the transport of boron dilution).

So, how important is numerical diffusion with respect to boron (or boron dilution) transport? Very. The timing of a front arrival can be spread out over a long period and its amplitude reduced to about half that of the peak. Unless, my understanding of the RETRAN general transport model is incorrect, I would advise against using RETRAN for any case where boron transport (or boron dilution) is important.




$\phi_i$ = Angle of junction i. $0 \leq \phi < 2\pi$

$\psi_K$ = Angle along which vector momentum equation is solved.

**Figure II.3-6. Generalized Two-Dimensional Volume with Several Flow Paths**

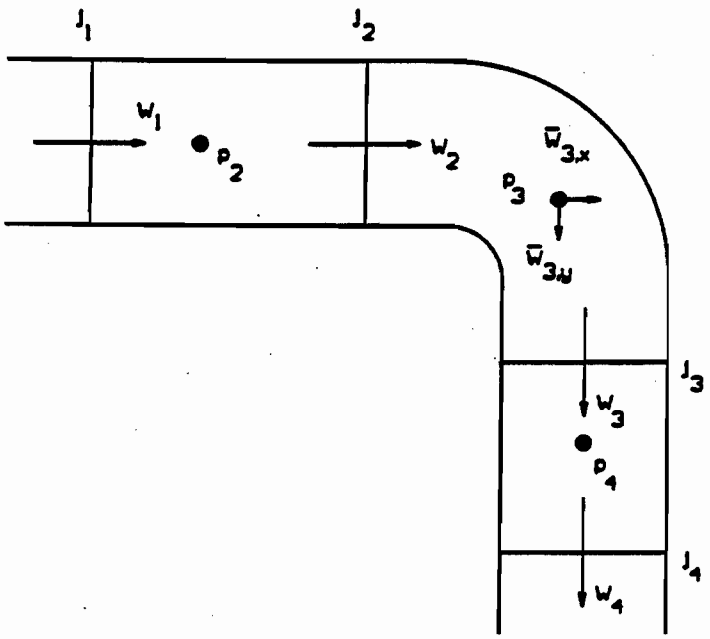



**Figure II.3-8. Control Volumes for an Elbow**

With

$$\overline{W}_{2,x} = \frac{1}{2}\,(W_1 + W_2) \quad , \tag{II.3-38a}$$

$$\overline{W}_{3,x} = \frac{1}{2}\,(W_2) \quad , \tag{II.3-38b}$$

$$\overline{W}_{3,y} = \frac{1}{2}\,(W_3) \quad , \tag{II.3-38c}$$

and

$$\overline{W}_{4,y} = \frac{1}{2}\,(W_3 + W_4) \quad , \tag{II.3-38d}$$

and Eq. II.3-37a, for the constant-flow-area case, the pressure change across the elbow is

$$P_4 - P_2 = -\left(e_2^* + e_3^*\right)\frac{W\,|W|}{2\rho A^2} = -\,e^*\,\frac{W\,|W|}{2\rho A^2} \quad . \tag{II.3-39}$$

while for air-water and refrigerants, it is a function only of void fraction. Since much of the air-water data is at low pressure, there is no basis for introducing a pressure variation. With appropriate adjustments to this parameter, the general form should be applicable to any fluid.

### Steam-Water

For steam-water mixtures the form of $L$ is selected to ensure proper behavior as the pressure approaches the critical pressure.

**Table 4-2 Chexal-Lellouche Fluid Parameter $L$ (used in $C_0$) for Steam-Water**

| | $L_v$ (Vertical) | $L_h$ (Horizontal)[1] |
|---|---|---|
| Cocurrent Up | $\frac{1-\exp(-C_p\langle\alpha\rangle)}{1-\exp(-C_p)}$ | $\frac{1-\exp(-C_p\langle\alpha\rangle)}{1-\exp(-C_p)}\left[1+\langle\alpha\rangle^{0.05}(1-\langle\alpha\rangle)^2\right]$ |
| Countercurrent | same as above | same as above |
| Cocurrent Down | same as above | same as above |

(1) See Table 4-1 note 2.

where:

$$C_p = \frac{4p_{crit}^2}{\left[p(p_{crit}-p)\right]} \qquad \text{(eq. 4-19)}$$

### Air-Water

For air-water mixtures, the pressure range of available data is not large and a form of $L$ is used that is dependent only on the void fraction and flow orientation.

**Table 4-3 Chexal-Lellouche Fluid Parameter $L$ (used in $C_o$) for Air-Water**

| | $L_v$ (Vertical) | $L_h$ (Horizontal)[1] |
|---|---|---|
| Cocurrent Up | $\min\left[1.15\langle\alpha\rangle^{0.45}, 1.0\right]$ | $\min\left[1.125\langle\alpha\rangle^{0.6}, 1.0\right] * \left[1+\langle\alpha\rangle^{0.05}\left(1-\langle\alpha\rangle\right)^2\right]$ |
| Countercurrent | same as above | same as above |
| Cocurrent Down | $\min\left[1.05\langle\alpha\rangle^{0.25}, 1.0\right]$ | same as above |

(1) See Table 4-1 note 2.

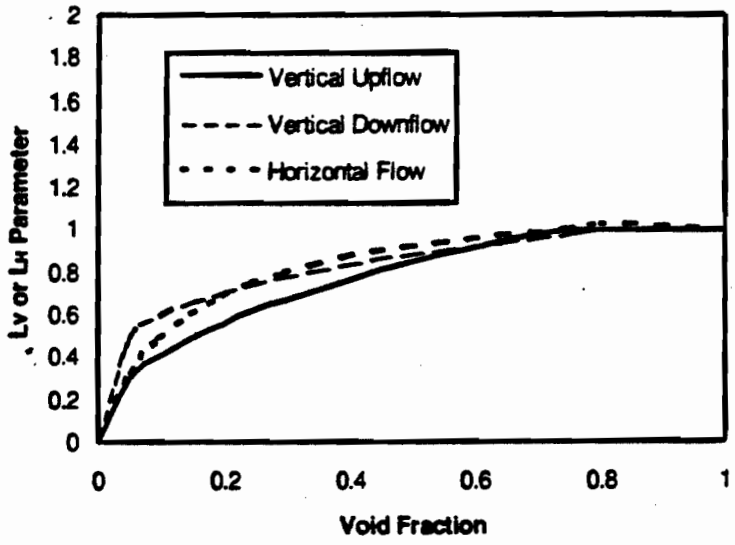



**Figure 4-6 Chexal-Lellouche Fluid Parameter for Air-Water**

### Refrigerants

For refrigerants as well, the pressure range in the available data is not large and $L$ is correlated in terms of the void fraction and flow orientation.

**Table 4-4 Chexal-Lellouche Fluid Parameter $L$ (used in $C_o$) for Refrigerants**

| | $L_v$ (Vertical) | $L_h$ (Horizontal)[1] |
|---|---|---|
| Cocurrent Up | $\langle\alpha\rangle^{0.025(1+10\langle\alpha\rangle)} \exp[0.5(1-\langle\alpha\rangle)]$ | $\langle\alpha\rangle\left[1.375-1.5(\langle\alpha\rangle-0.5)^2\right]*$ $\left[1+\langle\alpha\rangle^{0.05}(1-\langle\alpha\rangle)^2\right]$ |
| Countercurrent | same as above | same as above |
| Cocurrent Down | $\left[1-\exp(-100\langle\alpha\rangle)\right]*$ $\left[0.2(\langle\alpha+4\rangle)-0.74\langle\alpha\rangle^2(1-\langle\alpha\rangle)\right]$ | same as above |

(1) See Table 4-1 note 2.

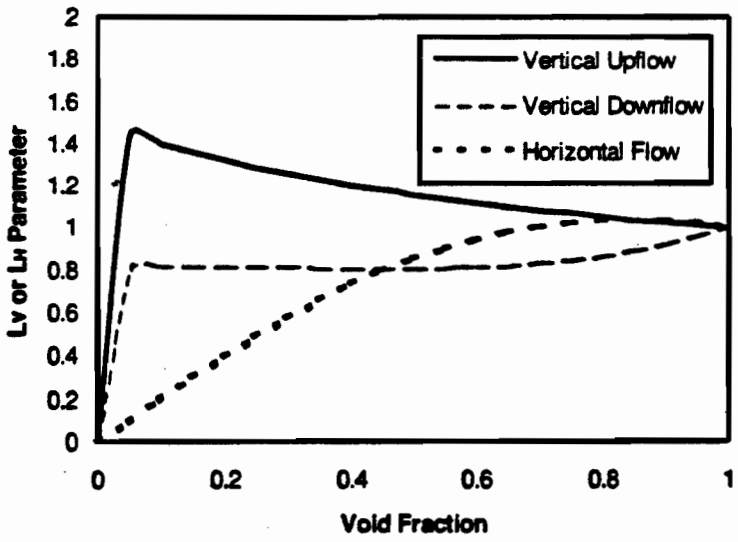



**Figure 4-7 Chexal-Lellouche Fluid Parameter for Refrigerants**

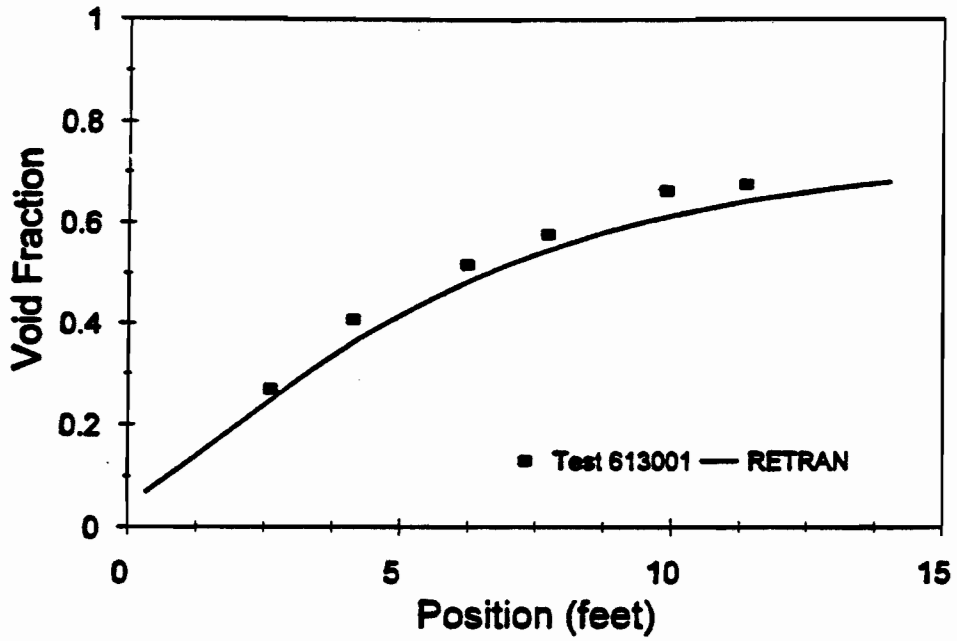



**Figure IV.4-8. FRIGG-4 Test 613001 Analysis**

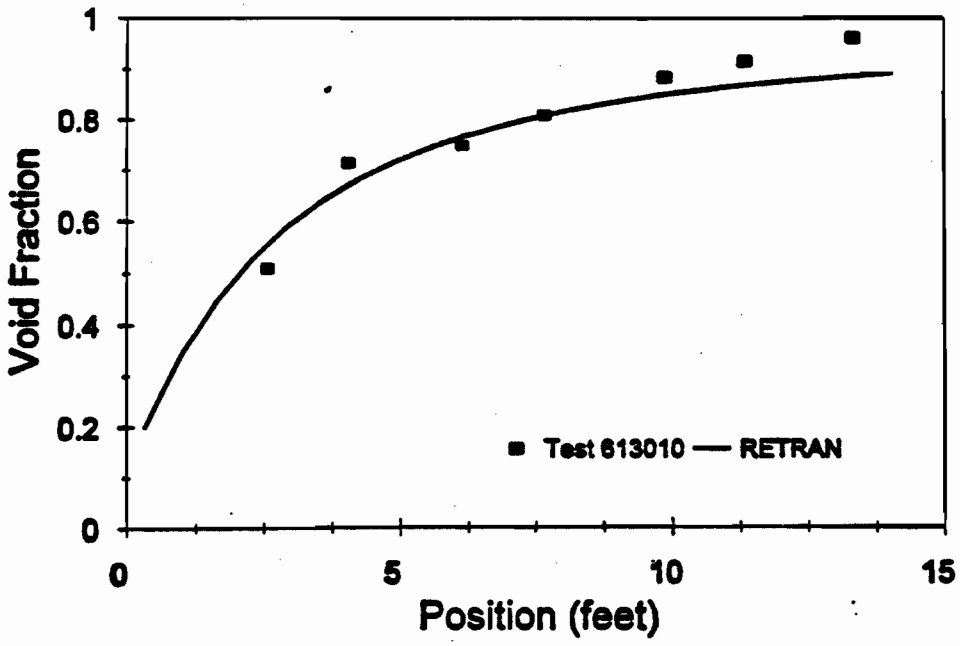



**Figure IV.4-9. FRIGG-4 Test 613010 Analysis**

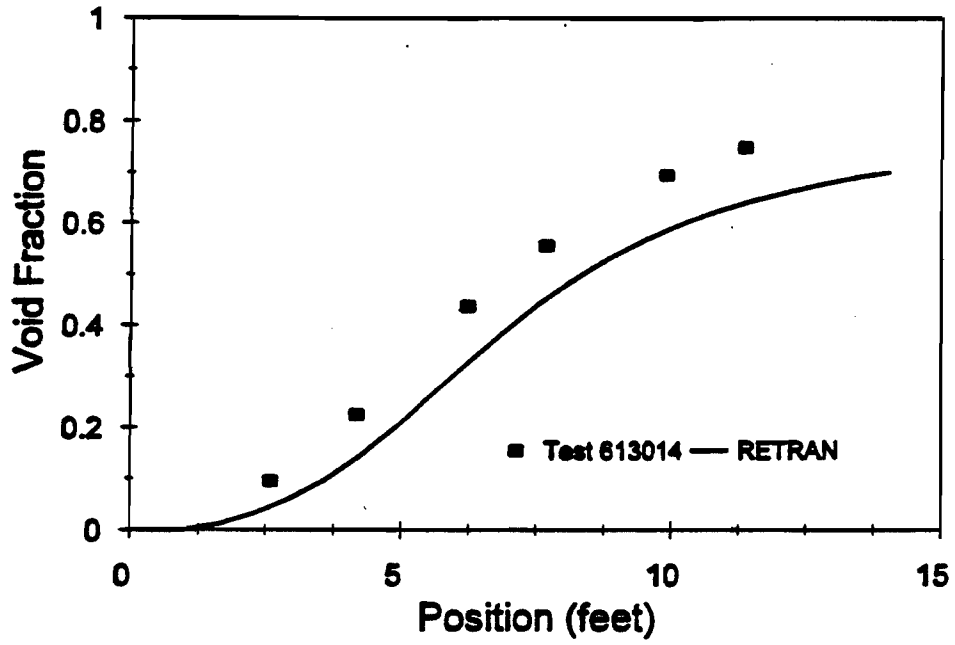



**Figure IV.4-10. FRIGG-4 Test 613014 Analysis**

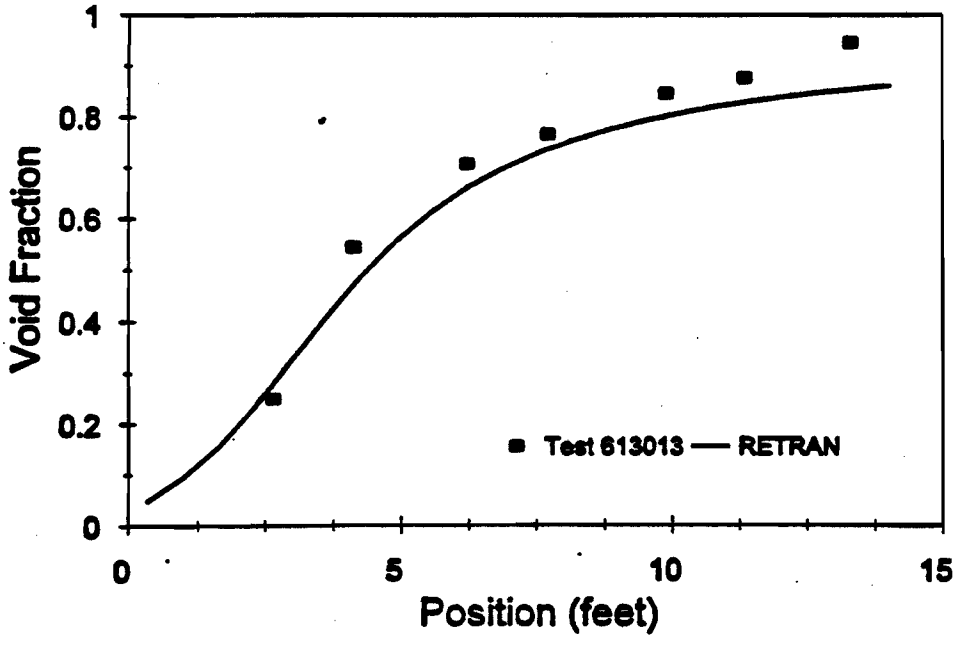



**Figure IV.4-11. FRIGG-4 Test 613013 Analysis**

## Table 5-3 Adiabatic Steam-Water Void Fraction Data

| Experiment [ref.] | Test Geometry | Hyd. Dia. mm (ft) | Pressure bar (psia) | Mass Flux kg/s-m$^2$ (Mlb/hr-ft$^2$) | Quality Fraction | Number of Data Points |
|---|---|---|---|---|---|---|
| **Upflow Data** | | | | | | |
| Kasai (1985) [5.11] | Tube | 4.6 (0.0151) | 70 (1015) | 1450 (1.07) | 0 - 0.2 | 35 |
| Beattie-Sugawara (1986) [5.12] | Tube | 73.9 (0.225) | 2, 10, 70 (29, 145, 1015) | 130 - 2600 (0.01 - 1.9) | 0.03 - 0.56 | 54 |
| Carrier (1963) [5.13] | Tube, steam bubbling through water | 456 (1.5) | 41 - 138 (600 - 2000) | 3.9 - 47.8 (0.003 - 0.035) | 1[(1)] | 60 |
| Hughes (1958) [5.14] | Tube | 168 (0.552) | 83 - 166 (1200 - 2400) | 119 - 363 (0.08 - 0.27) | 0.05 - 0.22 | 55 |
| Turnage-Davis (1979) [5.15] | Tube | 89 (0.292) | 7.25 (105) | 72 - 295 (0.05 - 0.22) | 0.004 - 0.48 | 17 |
| Hall-Ardron (1978) [5.7] | Tube, steam bubbling through water | 105 (0.345) | 1 - 4 (14.5 - 58) | 0.03 - 0.98 (0.00002 - 0.0007) | 1[(1)] | 19 |
| **Horizontal Flow Data** | | | | | | |
| Pye (1984) [5.16] | Tube | 22 (0.072) | 40 - 180 (580 - 2610) | 97 - 1914 (0.07 - 1.41) | 0.04 - 0.97 | 60 |
| Rajan-Daymond (1979) [5.17] | Tube | 50, 75 (0.164, 0.2461) | 54.8 (780) | 176 - 2550 (0.13 - 1.88) | 0.01 - 0.76 | 77 |
| **Cocurrent Downflow Data** | | | | | | |
| Petrick (1962) [5.18] | Tube | 49.3 (0.162) | 41, 69, 103 (600, 1000, 1500) | 163 - 1125 (0.12 - 0.83) | 0 - 0.11 | 144 |

(1) Zero liquid flow at the exit of the channel.

(1) Zero liquid flow at the exit of the channel.

## Table 5-4 Air-Water Void Fraction Data, continued

| Experiment [ref.] | Test Geometry | Hyd. Dia. mm (ft) | Pressure bar (psia) | Temp °C (°F) | Mass Flux kg/s-m$^2$ (Mlb/hr-ft$^2$) | Air Mass Fraction | Number of Data Points |
|---|---|---|---|---|---|---|---|
| Horizontal Flow Data | | | | | | | |
| Fukano (1987) [5.26] | Pipe | 26 (0.125) | 1.03 - 1.38 (15 - 20) | 20 (68) | 54 - 460 (0.04 - 0.34) | 0.035 - 0.356 | 79 |
| Simpson (1981) [5.27] | Pipe | 127 (0.417) | 1.24 - 1.38 (18 - 20) | 11 - 23 (52 - 73) | 3600 - 4700 (2.63 - 3.46) | 0.001 - 0.0006 | 13 |
| Tapucu (1980) [5.28] | Rectangular Channel | 12.7 (0.0415) | 1.08 - 1.91 (15.7 - 27.7) | 20 (68) | 760 - 4180 (0.56 - 3.08) | 0.0001 - 0.011 | 62 |
| Kowalski (1986) [5.29] | 7-Rod Bundle | 10 (0.03409) | 2.25 (32 - 63) | 20 (68) | 54 - 3200 (0.04 - 2.36) | 0.0002 - 0.447 | 80 |
| Beggs (1972) [5.25] | Pipe | 25.4, 38.1 (0.083, 0.125) | 3.6 - 6.8 (52.2 - 98.6) | 3 - 36 (38 - 96) | 60 - 2620 (0.04 - 1.932) | 0.0008 - 0.833 | 57 |
| Downflow Data | | | | | | | |
| Oshinowo (1974) [5.30] | Pipe | 25 (0.0819) | 1.72 (25) | 18 (65) | 150 - 2000 (0.11 - 1.46) | 0.0005 - 0.279 | 70 |
| Sokolov (1969) [5.31] | Pipe | 50 (0.164) | 1.01 (14.7) | 27 (80) | 780 - 2000 (0.59 - 1.47) | 0.00007 - 0.001 | 34 |
| Beggs (1972) [5.25] | Pipe (Various Angles) | 25.4, 38.1 (0.083, 0.125) | 2.4 - 6.8 (34.8 - 98.6) | 3 - 36 (38 - 96) | 60 - 5200 (0.04 - 3.83) | 0.0008 - 0.833 | 247 |

**Table 5-5 Countercurrent Flow Void Fraction Data**

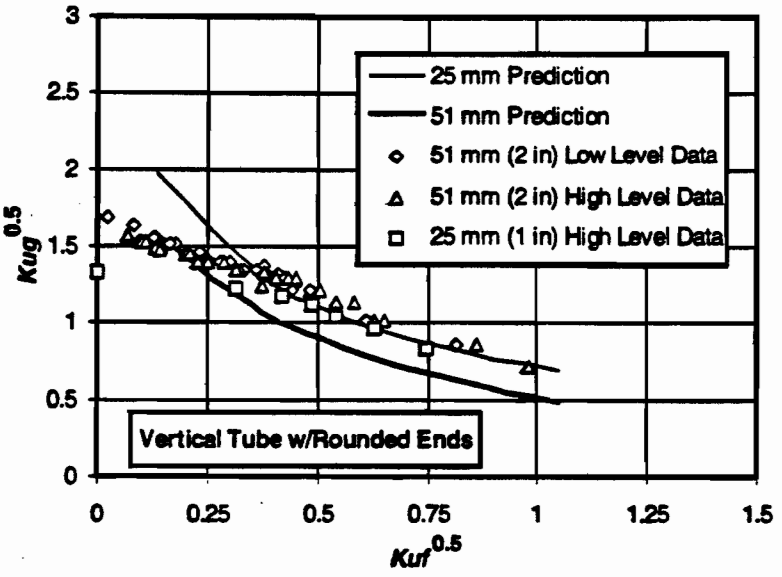
| Experiment [ref.] | Geometry | Hyd. Dia. mm (ft) | Pressure bar (psia) | Vapor Superficial Velocity m/s (ft/s) | Liquid Superficial Velocity m/s (ft/s) |
|---|---|---|---|---|---|
| Air-Water Data | | | | | |
| Ghiaasiaan (1994) [5.32] | Tube 0°, 8°, 28°, 45°, and 60° from vertical | 19 (0.0623) | 1.6 (23.2) | 0.01 to 1.5 (0.03 to 4.92) | -0.01 to -0.15 (-0.03 to -0.492) |

**Table 5-6 Countercurrent Flow Limit Data**

| Experiment [ref.] | Geometry | Hyd. Dia. mm (ft) | Pressure bar (psia) | Vapor Superficial Velocity m/s (ft/s) | Liquid Superficial Velocity m/s (ft/s) |
|---|---|---|---|---|---|
| Steam-Water Data | | | | | |
| Jones (1977) [5.33] | Vertical Orifice | 61.8, 37.6, 31.9 (0.203, 0.123, 0.105) | 1.03 (15) | 6 to 24 (20 to 80) | -0.003 to -0.19 (-0.01 to -0.63) |
| Thomas-Combs (1983) [5.34] | Bundle Upper Tie Plate | 10.5 (0.034) | 6.9, 4.5, 2.1 (100, 65, 35) | 5 to 18 (15 to 60) | -0.003 to -0.14 (-0.01 to -0.47) |
| Weiss (1992) [5.35] | Horizontal Pipe | 639 (2.096) | 3, 15 (43.5, 217.6) | 18 to 30 (59 to 98) | 0 to -0.08 (0 to -0.26) |
| Air-Water Data | | | | | |
| Thomas-Combs (1983) [5.34] | Bundle Upper Tie Plate | 10.5 (0.034) | 1.1 (16) | 3.0 to 5.6 (10.0 to 18.3) | -0.01 to -0.09 (-0.4 to 0.30) |
| Bharathan (1979) [5.36] | Tube | 25, 51 (0.08, 0.17) | 1.01 (14.7) | 2.00 to 8.25 (6.6 to 27.1) | -0 to -0.18 (-0 to -0.6) |

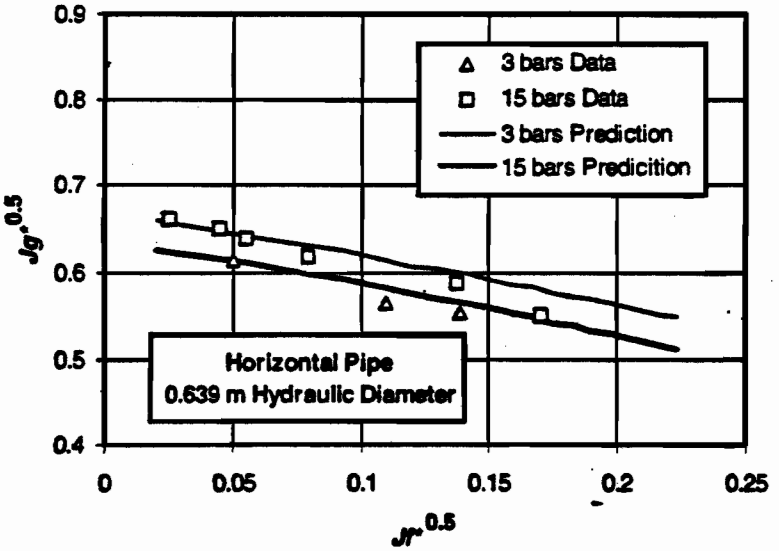
**Table 5-7 Refrigerant Void Fraction Data (Adiabatic)**

| Experiment | Test Geometry | Hyd. Dia. mm | Pressure bar | Mass Flux kg/m²s | Vapor Quality | Number of Data Points |
|---|---|---|---|---|---|---|




FL Data

**Figure 5-9 Chexal-Lellouche (1996) Prediction vs. Bharathan Air-Water CCFL Data**




r CCFL

**Figure 5-10 Chexal-Lellouche (1996) Prediction vs. Weiss et al. UPTF CCFL Data**

Some comparisons may also use the square root of the Wallis number ($j_k^{*0.5}$) [5.47]. The Wallis number for each phase is defined as follows:

$$j_f^* = \langle j_f \rangle \left[ \frac{\rho_f}{gD(\rho_f - \rho_g)} \right]^{0.5} \qquad j_g^* = \langle j_g \rangle \left[ \frac{\rho_g}{gD(\rho_f - \rho_g)} \right]^{0.5} \qquad \text{(eq. 5-2)}$$

where:

$D$ is the diameter of the flow path, and
$g$ is the acceleration of gravity.

The following five charts illustrate the comparison of the Chexal-Lellouche (1996) void model with the CCFL data.

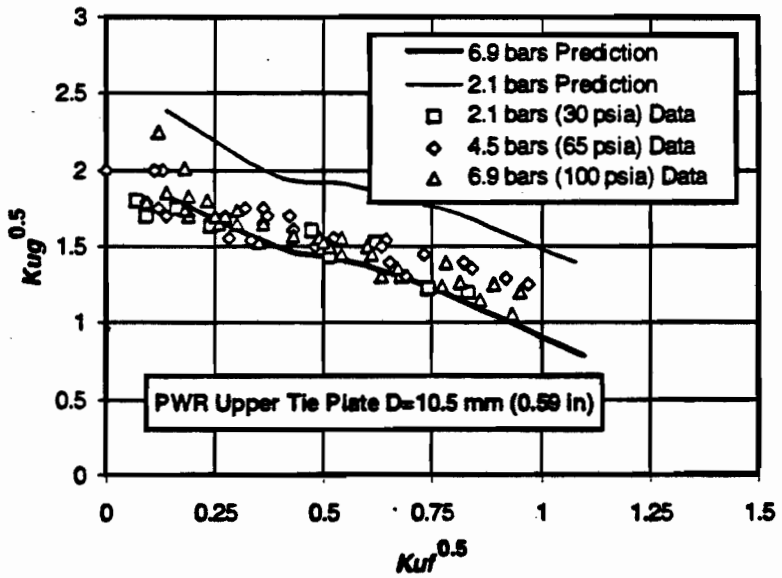



**Figure 5-6 Chexal-Lellouche (1996) Prediction vs. Thomas-Combs Steam-Water CCFL Data**

**Table 5-8 Statistical Comparison for Upflow Data**

| Data Set | Source | Number of Points | Chexal-Lellouche (1996) Mean Error | Chexal-Lellouche (1996) Standard Deviation |
|---|---|---|---|---|
| Steam-Water Adiabatic Upflow | Kasai | 35 | -0.006 | 0.032 |
| | Beattie-Sugawara | 54 | -0.062 | 0.050 |
| | Carrier | 60 | -0.031 | 0.072 |
| | Hughes | 55 | -0.001 | 0.026 |
| | Turnage-Davis | 17 | -0.010 | 0.099 |
| | Hall-Ardron | 19 | 0.016 | 0.037 |
| Steam-Water Diabatic Upflow | Nylund | 765 | 0.002 | 0.031 |
| | Seedy-Muralidhoran | 37 | -0.041 | 0.038 |
| | Anklam | 53 | -0.009 | 0.074 |
| | Hall-Ardron | 18 | 0.045 | 0.032 |
| | Wong-Hochreiter | 14 | 0.009 | 0.071 |
| | Jowitt | 117 | -0.004 | 0.063 |
| | Bartolomei | 423 | 0.026 | 0.038 |
| Air-Water Upflow | Hall-Ardron | 34 | -0.039 | 0.047 |
| | Smissaert | 161 | 0.019 | 0.050 |
| | Tapucu | 99 | 0.021 | 0.024 |
| | Hashemi | 31 | 0.042 | 0.077 |
| | Schleappi | 32 | -0.054 | 0.030 |
| | Borishansky | 46 | -0.005 | 0.040 |
| | Turnage | 16 | -0.044 | 0.046 |
| | deMello | 23 | 0.063 | 0.031 |
| | Beggs(1) | 260 | -0.016 | 0.031 |

(1) Includes angles from 0° to 85°.

## Horizontal Flow

Table 5-9 illustrates the statistics of the model predictions for all of the horizontal data. The degree of variations in the statistics of the horizontal data sets is greater than that of the vertical data. Kowalski [5.29] predictions could be improved by variations in model parameters, but at the expense of the comparisons with the other test data.

nuch of

uld be

the

ater

q. 4-19)

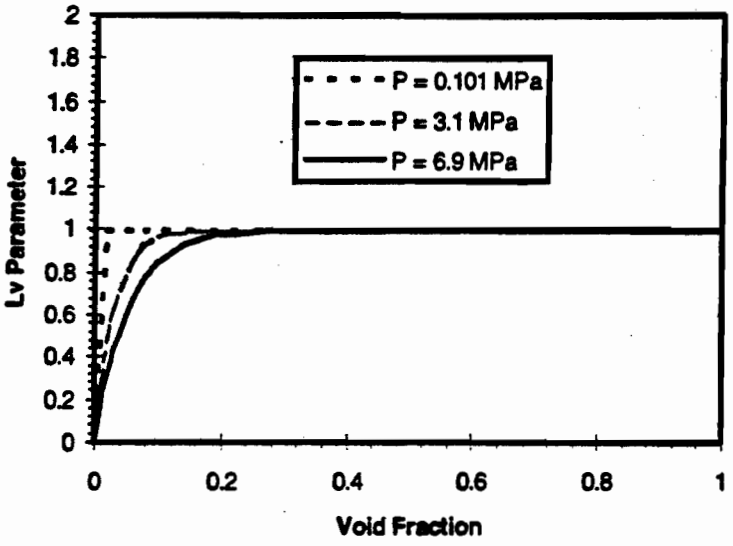



**Figure 4-4 Chexal-Lellouche Vertical Fluid Parameter for Steam-Water**

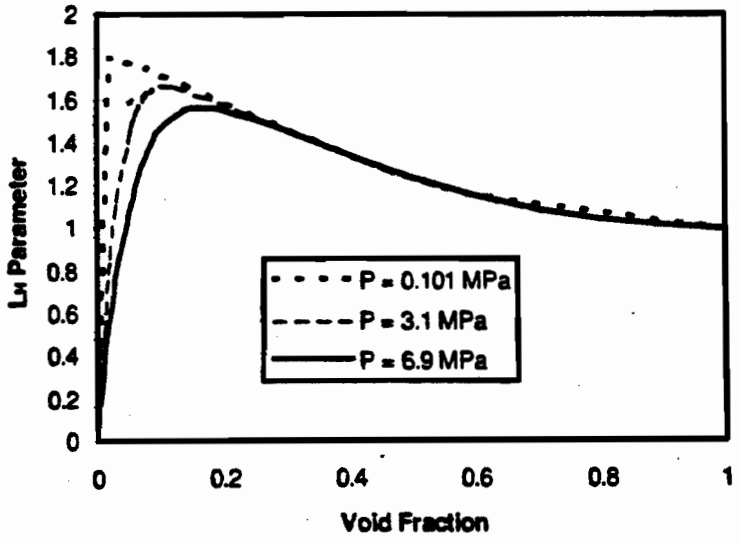



**Figure 4-5 Chexal-Lellouche Horizontal Fluid Parameter for Steam-Water**

NUREG-75/056

# WREM:

# Water Reactor Evaluation Model

[Revision 1]

MAY 1975

DIVISION OF TECHNICAL REVIEW
NUCLEAR REGULATORY COMMISSION

## 3.7 Momentum Equation

The conservation of momentum equation to be used by the Evaluation Model computer programs should account for certain effects as specified by the acceptance criteria. The effects to be accounted for are given in the following paragraph from the Commission acceptance criteria:

*"3. Momentum Equation. The following effects shall be taken into account in the conservation of momentum equation: (1) temporal change of momentum, (2) momentum convection, (3) area change momentum flux, (4) momentum change due to compressibility, (5) pressure loss resulting from wall friction, (6) pressure loss resulting from area change, and (7) gravitational acceleration. Any omission of one or more of these terms under stated circumstances shall be justified by comparative analyses or by experimental data."*

Several options are available to the user* to select the form of the momentum equation that is desired. These equations are selected by the assigned value of the input variable MVMIX on the junction data cards (card 08XXXY). This selection may be modified by the input value of the input variables JCHOKE and ICHOKE also on the junction data cards. The input variables JCHOKE and ICHOKE select the critical flow option as discussed in the critical flow section

*Presently (May 1975) the Regulatory staff uses the incompressible equation for "simple" nodes and no momentum flux equation for complex nodes for base-case PWR calculations (refer to the input Section 3.10). For further discussion of this point, the reader should refer to the staff report on momentum equation sensitivity studies[34].

(Section 3.2). In addition to the six options available through the selections of MVMIX, the user must choose a momentum equation based on the complexity of the junctions that connect the volumes in question.

As stated earlier, the form of the momentum equation to be used in RELAP4-EM is selected by the value used for MVMIX. Momentum equations that can be used are presented in the form used for sudden expansions or sudden contractions. The geometry for the sudden expansion is presented in Figure 3.8.

1. Compressible Mechanical Energy Balance Equation: MVMIX = 0

$$I_j \frac{dW_j}{dt} = \left(P_K + \frac{\bar{v}_K \bar{W}_K}{A_K} + P_{Kgj}\right) - \left(P_L + \frac{\bar{v}_L \bar{W}_L}{A_L} + P_{Lgj}\right) \qquad (72)$$

$$+ \rho_{Li} \left(v_{Li}^2 + C_j^2\right) - \rho_{Ko} \left(v_{Ko}^2 + C_j^2\right)$$

$$- F_{fK} - F_{fL} - F_{K,j,L}$$

where:

$\bar{W}$ = volume average mass flow as previously defined for the energy equation

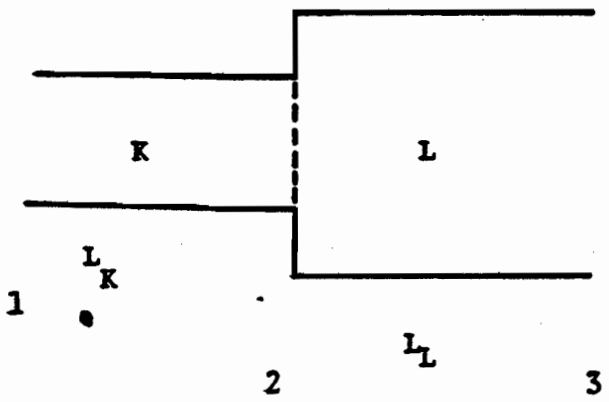
$\bar{v}$ = volume average velocity

$\rho_{Li}$, $\rho_{Ko}$ = fluid density at the inlet side of Volume L and the outlet side of Volume K, respectively

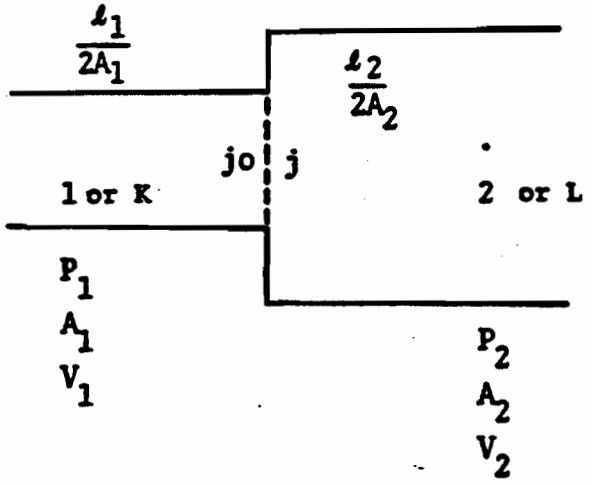
$v_{Li}$, $v_{Ko}$ = fluid velocity at the inlet side of Volume L and the outlet side of Volume K, respectively

Figure 3.8

MOMENTUM EQUATION FOR SUDDEN EXPANSION




COMPRESSIBLE INTEGRAL MOMENTUM EQUATION




MECHANICAL ENERGY BALANCE EQUATION

$C_j$ = fluid sonic velocity = $\sqrt{(\frac{\partial P}{\partial \rho})_S}$

$F_{fK}$, $F_{fL}$ = Fanning friction as previously defined

$F_{K,j,L}$ = frictional pressure loss from the outlet side of Volume K through Junction j into the inlet side of Volume L

S = subscript referring to constant entropy

$I_j$ = geometric "inertia" for Junction j

$W_j$ = mass flow at Junction j

$P_1$, $P_2$ = thermodynamic pressure in control volumes 1 and 2

$v_1$, $v_2$ = velocity in control volumes 1 and 2

$A_1$, $A_2$ = area in control volumes 1 and 2

$P_{1g}$, $P_{2g}$ = gravitational pressure drops in control volumes 1 and 2

$\ell$ = length of control volume

$D_h$ = diameter

K = loss coefficient

2. Mixed two stream compressible flow in the volume "from" side:

MVMIX = 1 (reference Figure 3.6 for definition of subscripts and model figure):

$$I_{j1} \frac{dW_{j1}}{dt} = -I'_{j2} \frac{dW_{j2}}{dt} + (P_K + \frac{\bar{v}_K \bar{W}_K}{A_K} + P_{Kgj1}) \qquad (73)$$

$$- (P_L + \frac{\bar{v}_L \bar{W}_L}{A_L} + P_{Lgj1}) - F_{fK} - F_{fL} - F_{Kj1,j1,Lj1} + \Delta P_{j1}$$

where:

$W_{j1}$ = flow from Volume K through Junction j1 to Volume L

$W_{j2}$ = flow through Junction j2 that is mixed with $W_{j1}$

$I_{j1}$ = inertia for flow $W_{j1}$

$I_{j2}$ = half-volume inertia for flow $W_{j2}$ in the volume common to flow $W_{j1}$

$F_{Kj1,j1,Lj1}$ = friction loss for the flow through Junction j1

$\Delta P_{j1}$ = Fanning friction, Volumes K and L

$P_{Kgj1}, P_{Lgj1}$ = elevation pressures for flow $W_{j1}$ in Volumes K and L, respectively.

More details as to the form of the equation may be obtained from Section 2.32, "Compressible Two-Stream Flow with One-Dimensional Momentum Mixing," p. 24, of the RELAP4 manual.

3. Mixed two-stream compressible flow in the volume "to" side: MVMIX = 2 The flow solution for the "to" side is obtained by appropriate changes in the subscripts of the above equation. More detail may be obtained from Section 2.32 of the RELAP4 manual.

4. Incompressible Mechanical Energy Balance Equation with no momentum flux (no Bernoulli effects): MVMIX = 3

$$I_j \frac{dW_j}{dt} = (P_K + P_{Kgj}) - (P_L + P_{Lgj}) - K_f \frac{\rho_j v_j |v_j|}{2} \phi_{2P} \qquad (74)$$

5. Compressible Integral Momentum Equation, Single stream: MVMIX = 4

$$I_j \frac{dW_j}{dt} = (P_k + \frac{\bar{V}_k \bar{W}_k}{A_k} + P_{kgj}) - (P_L + \frac{\bar{V}_L \bar{W}_L}{A_L} + P_{Lgj}) - F_{fk} - F_{fL} - F_{k,j,L} \quad (75)$$

6. Special case of a fill with negative flow used to represent the outlet: A volume with no normal outlet junction, but with a negative fill position (flow is negative) as the actual volume outlet, constitutes a special case. In order for the correct volume flow to be calculated, the user must use MVMIX = -2 for the negative fill junction.

Through the selection of JCHOKE and ICHOKE, a seventh form of the momentum equation can be selected. If JCHOKE = 0 and ICHOKE = 11, the Incompressible Mechanical Energy balance momentum equation is used, this equation is given by:

$$I_j \frac{dW_j}{dt} = (P_1 + \frac{v_1 W_1}{A_1} + P_{1g}) - (P_2 + \frac{v_2 W_2}{A_2} + P_{2g})$$

$$- \frac{W_j^2}{2} \left(\frac{1}{\rho_{j1} A_1^2} - \frac{1}{\rho_{j2} A_2^2}\right)$$

$$- S_1 \left(\frac{4f\ell}{2D_h}\right)_1 \frac{W_1^2}{2\rho_1 A_1^2} - S_2 \left(\frac{4f\ell}{2D_h}\right)_2 \frac{W_2^2}{2\rho_2 A_2^2} - \frac{S_j K_j W_j^2}{2\rho_j A_j^2} \quad (76)*$$

*Note that for this case $\rho_{j1} = \rho_{j2}$ and the middle term can be written as:

$$- \frac{W_j^2}{2\rho_j} \left(\frac{1}{A_1^2} - \frac{1}{A_2^2}\right)$$

where: f = friction factor

S = 1 for forward flow, -1 for reverse flow

This equation represents the incompressible form of the equation shown under item 1. Selection of JCHOKE=11 calls the evaluation model choking option using the MOODY model for the two-phase region. Therefore, the incompressible mechanical energy balance momentum equation represents the momentum equation used for all Evaluation Model calculations.

## 3.8 Liquid Level Calculation

The liquid level calculation option is not required by the Commission acceptance criteria but has been added to help define the beginning of core reflooding. An equivalent liquid level is defined such that an effective water level in the lower plenum can be calculated as a function of time. A conglomerate volume combining up to twenty RELAP-EM volumes may be described. The order of the RELAP4-EM volumes used to describe the conglomerate volume is arbitrary except that the first RELAP4-EM volume becomes the reference volume. The total liquid mass in the conglomerate volume is obtained by summing the liquid mass in each of the RELAP4-EM volumes. A total liquid volume is obtained by dividing the total liquid mass by the density of saturated liquid in the reference volume. The liquid level in the conglomerate volume is then calculated from the total liquid volume in all the RELAP4-EM volumes. Finally, the liquid level

## 7.0 REFERENCES


1. 10 CFR Part 50, "Acceptance Criteria for Emergency Core Cooling System for Light Water-Cooled Nuclear Power Plants," Federal Register, Vol. 39, No. 3, January 4, 1974.

2. K. V. Moore, W. H. Rettig, "RELAP4- A Computer Program for Transient Thermal-Hydraulic Analysis," ANCR-1127, December 1973.

3. J. A. McClure, "TOODEE - A Two-Dimensional, Time-Dependent Heat Conduction Program," IDO-17227, April 1967.

4. D. R. Evans, "MOXY: A Digital Computer Code for Core Heat Transfer Analysis," IN-1392, August 1970.

5. C. R. Hann, C. E. Beyer, L. J. Parchen, "GAPCON-THERMAL-1: A Computer Program for Calculating the Gap Conductance in Oxide Fuel Pins," BNWL-1778 UC-78, September 1973.

6. Unpublished report concerning the FRAP Code.

7. J. R. Weir, Jr., et al, "Certain Aspects of LWR Fuel Pins During a LOCA," ORNL-CF-72-11-1.

8. M. F. Lyons, et al, "$UO_2$ Pellet Thermal Conductivity from Irradiations with Central Melting," GEAP-4624, July 1964.

9. J. C. Maxwell, "A Treatise on Electricity and Magnetism," Vol. I, 3rd Edition, Oxford University Press (1891), reprinted by Dover, New York, 1954.

10. A. Euken, "The Heat Conductivities of Ceramic Refractory, Calculations of Heat Conductivity from Constituents, Forsch, Gebiete, Ingenieurw, B3 Forschungsheft," No. 353, 1932.

11. H. C. Brassfield, et al, "Recommended Property and Reaction Kinetics Data for Use in Evaluating a Light-Water Cooled Reactor Loss-of-Coolant Incident Involving Zircaloy-4 on 304-SS-Clad $UO_2$," GE, -482, April 1968.

12. D. G. Hardy, "High Temperature Expansion and Rupture Behaviour of Zircaloy Tubing," CONF-730304, pp. 254-273, March 1973.

13. J. B. Conway, et al, "Thermal Expansion and Heat Capacity of $UO_2$ to 2200°C," Trans. ANS, Vol. 6, No. 1, p. 153, 1963.

14. J. S. Gellerstedt et al, "Correlation of Critical Heat Flux in a Bundle Cooled by Pressurized Water," pp. 63-71 of "Two-Phase Flow and Heat Transfer in Rod Bundles," Symposium presented at the Winter Annual Meeting of the American Society of Mechanical Engineers, Los Angeles, California, November 1969.

15. P. G. Barnett, "A Correlation of Burnout Data for Uniformly Heated Annuli and Its Use for Predicting Burnout in Uniformly Heated Rod Bundles," AEEW-R 463, 1966.

16. G. C. Slifer, J. E. Hench, "Loss-of-Coolant Accident and Emergency Core Cooling Models for General Electric Boiling Water Reactors," NEDO-10329, General Electric Company, April 1971, (Equation C-32).

17. F. W. Dittus, L. M. K. Boelter, University of California Publs. Eng., Vol. 2, p. 443, 1930.

18. J. R. S. Thom et al, "Boiling in Subcooled Water During Flow Up Heated Tubes or Annuli," Proc. Instn. Mech. Engrs., Vol. 180, Part 3C, pp. 226-246, 1966.

19. V. E. Schrock and L. M. Grossman, "Forced Convection Boiling Studies, Final Report on Forced Convection Vaporization Project," TID-14632, 1959.

20. J. B. McDonough, W. Millich, E. C. King, "Partial Film Boiling With Water at 2000 psig in a Round Vertical Tube," MSA Research Corp., Technical Report 62, 1958, (NP-6976).

21. D. C. Groeneveld, "An Investigation of Heat Transfer in the Liquid Deficient Regime," AECL-3281 (Rev.) December 1968, revised August 1969.

22. P. J. Berenson, "Film-Boiling Heat Transfer from a Horizontal Surface," J. of Heat Transfer, Vol. 83, pp. 351-358, August 1961.

23. R. S. Dougall and W. M. Rohsenow, "Film-Boiling on the Inside of Vertical Tubes with Upward Flow of the Fluid at Low Qualities," MIT-TR-9079-26, 1963.

24. Proposed ANS Standard, "Decay Energy Release Rates Following Shutdown of Uranium-Fueled Thermal Reactors," ANS-5.1, October 1971.

25. K. Shure, "Fission Product Decay Energy," Bettis Technical Review, Reactor Technology, WAPD-BT-24, December 1961, pp 1-17.

26. F. F. Cadek et al, "PWR FLECHT (Full Length Emergency Cooling Heat Transfer) Final Report," WCAP-7665, April 1971.

27. F. F. Cadek et al, "PWR FLECHT Final Report Supplement," WCAP-7931, October 1972.

28. Response to letter to B&W Company from D. Knuth, AEC, June 1972.

29. A. I. Brown and S. M. Marco, "Introduction to Heat Transfer," McGraw-Hill Book Company, New York, 1958.

30. D. B. Collins et al, "Pump Operation with Cavitation and Two-Phase Flow," paper presented at the Fourth Western Canadian Heat Transfer Conf., Winnipeg, Manitoba, May 29, 1972.

31. E. D. Hughes, "An Examination of Rod Bundle Critical Heat Flux Data and Correlations and Their Applicability to Loss-of-Coolant Accident Analyses," Quarterly Technical Report Nuclear Safety Program Division, K. A. Dietz (ed.), ID-1319, pp. 16-40, May 1970.

32. R. E. Henry, "A Correlation for the Minimum Film Boiling Temperature," American Institute of Chemical Engineers, Preprint 14, Presented at the 14th National Heat Transfer Conference, Atlanta, GA, August 1973.

33. E. A. Eldridge, H. W. Deem, "Specific Heats and Heat of Transformation of Zircaloy -2 and Low-Nickel Zircaloy -2," BMI-1903, May 31, 1967.

34. "Evaluation of LOCA Hydrodymamics," Regulatory Staff: Technical Review, USAEC, Nov. 1974.

35. Lauben, G. N., "TOODEE2, A Two Dimensional Time Dependent Fuel Element Thermal Analysis Program," NRC, May 1975.