6th Conference on Nuclear and Particle Physics
17-21 Nov. 2007 Luxor, Egypt

CODENSATION DURING NUCLEAR REACTOR LOCA

Y. Rihan*, M. Teamah, M. Sorour, S. Soliman*

** Atomic Energy Authority, Hot Lab. Center, Egypt*

Abstract

Two-phase channel flow with condensation is a common phenomenon occurs in a number of nuclear reactor accident scenarios. It also plays an important role during the operation of the safety coolant injection systems in advanced nuclear reactors. Semi-empirical correlations and simple models based on the analogy between heat and mass transfer processes have been previously applied. Rigorous models, compatible with the state-of-the-art numerical algorithms used in thermal-hydraulic computer codes, are scare, and are of great interest. The objective of this research is to develop a method for modeling condensation, with noncondensable gases, compatible with the state-of-the-art numerical methods for the solution of multi-phase field equations. A methodology for modeling condensation, based on the stagnant film theory, and compatible with the reviewed numerical algorithms, is developed. The model treats the coupling between the heat and mass transfer processes, and allows for an implicit treatment of the mass and momentum exchange terms as the gas-liquid interphase, without iterations. The developed model was used in the application of loss of coolant in pressurized water reactor accidents.

Keywords: *Condensation, Noncondensable Gas, LOCA, Modeling.*

INTRODUCTION

Condensation of vapor in the presence of non-condensable gases continues to receive extensive analytical and experimental attention. One of the most significant recent applications is in the design of advanced passive nuclear reactor containments, where steam from the reactor vessel must be condensed on containment walls or in vertical tubes after mixing with nitrogen or air in the containment volume. To study these systems, both separate effect and integral system experiments are understudy.

Small experimental test models that have similar characteristics to the actual reactor are of interest in simulating and studying problems. Institute of Nuclear Research Integral

System Test (IIST) Facility has been established for safety studies of a Westinghouse three-loop pressurized water reactor (PWR). The research goal of the IIST facility is to validate the emergency operating procedure during PWR accidents. The maximum operating pressure of the IIST facility is 2.1 MPa, and the scaling factors for the height and volume in the reactor coolant system (RCS) are ~ ¼ and 1/400, respectively. The IIST has three loops as well as all the systems that are pertinent to studying Westinghouse plant system transients.

An experiment [1] was conducted at the IIST facility to simulate a 2% cold-leg-break loss-of-coolant accident (LOCA) with total high-pressure injection (HPI) failure. The break was located in loop2, which is one of the two loops that does not have a pressurizer. This test explored RCS thermal-hydraulic behavior, such as coolant inventory distribution during the top-down drain, asymmetric coolant holdup in the U-tubes and inlet plenum of all three steam generators (SGs), stratified two-phase flow in the horizontal legs, and core inventory boiloff associated with hot-leg flooding phenomena.

IIST FACILITY

IIST is a reduced-height and reduced-pressure integral system test facility that simulates the thermal hydraulics of a Westinghouse three-loop PWR at the Maanshan nuclear power reactor plant under abnormal conditions as well as during small-break LOCAs. The IIST facility consists of a pressure vessel and three loops-each of which has an active steam generator and a coolant pump. A comparison of major parameters between IIST and the Maanshan NPP is given in Table 1.

The data acquisition system of the IIST facility records from more than 200 instruments, including K-type thermocouples; venturi flowmeters; pressure transducers; and differential pressure transducers used to measure temperature, flow rate, pressure, and differential pressure, respectively. In the IIST facility, 50 ports allow flow visualization of thermal-hydraulic phenomena and thus enhance understanding of two-phase phenomena in the pressure vessel, hot legs, steam generator inlet and outlet, steam generator secondary sides, crossover legs, cold legs, and pressurizer. A total of 13 video cameras are mounted at selected view ports to record the key thermal-hydraulic phenomena during the experiments.

IIST SBLOCA EXPERIMENT

The experiment was performed to simulate a 2% cold-leg break (i.e. a cold-leg break with a break area of 2% of the scaled cold-leg cross-section area) with total HPI failure. A horizontal break nozzle was installed in the cold leg of loop 2 (the loop without a pressurizer). Core power decay and pump coastdown during the initial phase of the LOCA were not simulated in the test. The test started from the initial conditions: primary-and secondary-side pressures of 0.96 and 0.3 MPa, respectively; and hot-leg and core differential temperatures of 450 and 41 K, respectively, with a constant core power of 126 kW to simulate 1.82% of the 1/400 scaled PWR nominal power and natural-circulation core flow rate of 0.66 kg/s.

Table 1. Comparison of the Major Parameters of IIST and Maanshan PWR.

Parameter	IIST	Maanshan PWR
Design pressure (MPa)	2.1	15.6
Maximum core power (MW)	0.45	2775
Primary system volume (m^3)	0.537	215
Number of loops	3	3
Core		
Height (m)	1	3.6
Hydraulic diameter (m)	0.108	0.0122
Bypass area (m^2)	$7.2 * 10^{-5}$	$1.54 * 10^{-2}$
Hot leg		
Inner diameter, D (m)	0.0525	0.735
Length, L (m)	2	7.28
U-tube in one SG		
number	30	5626
Average length (m)	4.08	16.85
Inner diameter (mm)	15.4	15.4
Volume (m^3)	0.0228	18.44
Cold leg		
Inner diameter (mm)	0.0525	0.787
Length (m)	5	15.7
Downcomer		
Flow area (m^2)	0.0185	2.63
Hydraulic diameter (m)	0.0412	0.48
Pressurizer		
Volume (m^3)	0.0932	39.64
Surge-line flow area (m^2)	$3.44 * 10^{-4}$	0.0638

After the break occurred at time zero, the primary pressure dropped until it became only a little higher than the secondary-side pressure. Thereafter, the primary pressure decreased slowly because the energy content of the liquid discharged through the break was only slightly larger than the core energy input. The flow rates in the three loops were asymmetric during the single- and two-phase natural circulation (from 0 to 146 s and 146 to 400 s, respectively). Then, reflux condensation phenomena became more obvious in loops 2 and 3 than in loop 1; i.e., the fluid temperature difference between the U-tube and the secondary side for SG-1 is smaller than that for SG-2 and SG-3.

The air, originally filling the liquid-free space of the pressurizer, flowed through the hot leg into the SG-1 U-tubes after emptying of the pressurizer at 128 s. After 164 s of the break, the loop 1 flow rate suddenly dropped to near zero, which is associated with the decrease of the heat removal capability of SG-1. The effects of noncondensable air caused the suddenly decrease of the natural-circulation flow rate in loop 1 and obviously slowed the ascending temperatures in both the primary and secondary sides of SG-1.

The core collapsed liquid level (calculated from the measured core differential pressure and coolant density) decreased sharply after the break because of the sub-cooled liquid discharge in the time period between 0 to 146 s. Then, the core collapsed liquid level decreased slowly, when the break flow became a two-phase mixture from 146 to 400 s. The core was uncovered with heatup at 1734 s because of the continuous boiloff of the vessel coolant inventory because no coolant makeup is available.

MODELING DESCRIPTION

The modified RELAP5/MOD3.2 code nodalization, including 172 volumes connected by 175 junctions and 141 heat structures, has been developed to simulate the IIST facility, as shown in Figure 1. The code is essentially based on a nonhomogeneous, nonequilibrium, two-fluid model. It solves six basic field equations for six dependent variables, namely, pressure, void fraction, velocities, and specific internal energies of both phases. These governing equations are derived by the conservation of mass, momentum, and energy. The transfer phenomena between the liquid and vapor phases, which are involved in all field equations, are modeled through use of appropriate constitutive equations. The constitutive relations of the code include models for defining flow regimes and flow-regime-related models for interphase drag and shear, interphase wall friction and coefficient of virtual mass [2].

Of particular importance with respect to the LOCA scenario is the core uncovery and heatup. The decided parameter in avoiding core uncovery and heatup, according to the IIST experimental results, is not only the amount of coolant inventory remaining in the primary system but also the part of coolant left in the pressure vessel. This portion of coolant within the pressure vessel is responsible for effectively removing the core decay power.

METHOD OF SOLUTION

The present method is based on the manipulation of the stagnant film model. The numerical solution scheme in this model uses a staggered spatial mesh where the momentum control volumes are centered at the boundaries of the control volumes representing scalar state variables as shown in Figure 2.

Condensation in the presence of noncondensables in a downward channel flow was successfully modeled based on the application of the two-fluid formulation and using the classical stagnant film model [3]. The stagnant film model has previously been incorporated in the RELAP5/MOD3 code [4]. This model is based on a one dimensional, two-fluid formulation of the transient two-phase conservation equations. The numerical solution scheme is based on the semi-implicit finite difference technique, where in the difference conservation equations the terms responsible for the sonic wave propagation, as well as terms representing phenomena with small time constants, are evaluated implicitly. The implicitly evaluated terms include those representing the interphase mass and momentum exchange. The finite difference conservation equations are linearized with respect to the implicit terms and, by algebraic elimination of the unknowns, are then reduced to a single difference equation per control volume in terms of the new step pressure. An important step in this process is the elimination of the implicitly treated parameters affecting the interphase mass and momentum exchange, including the interphase temperature (assumed equal to the saturation temperature

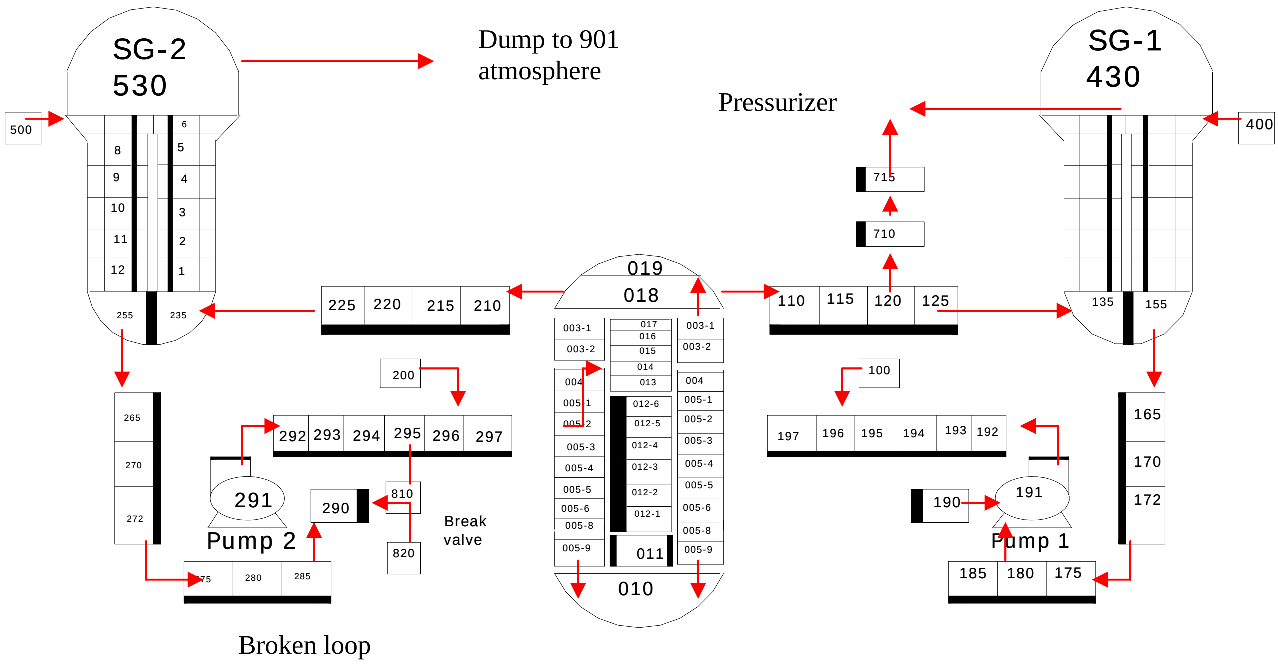

Figure 1. IIST Nodalization for the Model.

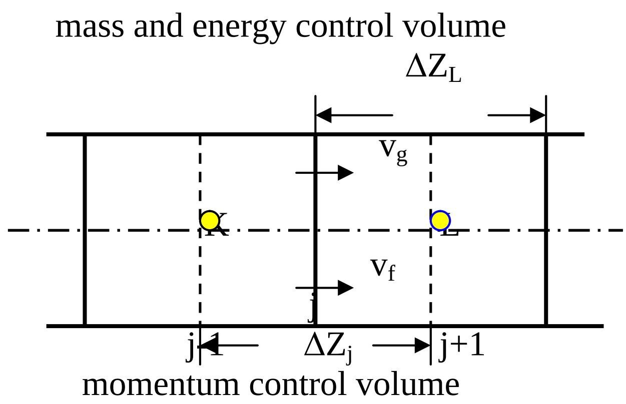

Figure 2. Scalar and Vector Nodes in the Model.

[4]), in favor of the state variables pressure, phasic temperatures and noncondensable mass fraction in the gas phase. This is done by using Taylor expansion, truncated after the first term in the expansion series. The resulting system of linear equations is then solved for the new time step pressures, followed by the calculation of the new time step values of other state variables by back substitution.

In each time step the state variables are updated in essentially two stages. First, using sum and difference mass, energy and momentum equations and defining intermediate step values for several parameters based on Taylor expansion, the new pressures are calculated for all the nodes. The procedure in this stage leads to a system of only N linear equations, N being the number of fluid cells, with N unknown pressures. These linear equations are then solved numerically. In the second stage, using the momentum equations, the new velocities are first calculated, where after other state variables are updated by back substitution into the mass and energy conservation equations.

In this model a quasi-steady, stagnant gaseous film is assumed to separate the liquid-gas interphase from the bulk gas. Heat and mass transfer are assumed to take place through this film by diffusion [2]. Energy balance applied to the interphase.

The interphase drag in RELAP5 code is calculated on the basis of the flow regime. The flow regime can occur with various flow patterns, which are categorized into three cases: the inverted annular flow, the inverted slug flow, and the dispersed flow. The interphase drag in inverted annular flow is evaluated from the drag of bubbly flow and that of annular flow while the drag term of inverted slug flow and dispersed flow is calculated from the evaluation of the drop size that can exist in the flow regimes. Therefore, the evaluation of drop size in code is directly related to the drag coefficient.

The size of drops is evaluated from the Weber number criterion and limitation on the minimum drop size as a function of pressure:

$$D_{We} = \frac{\sigma We}{\rho_g V_{fg}^2}, \qquad We = 12$$

and

$$\begin{aligned} D_{\min} &= 0.0025 \qquad \mathit{for} \quad p^* \le 0.025, \\ &= 0.0025 + (0.0002 - .0025).(4.4444) * (p^* - 0.025) \qquad \mathit{for} \quad 0.025 < p^* < 0.25, \\ &= 0.0002 \qquad \mathit{for} \quad p^* \ge 0.25 \end{aligned}$$

where $p^* = p/p_c$ = reduced pressure. The drop size is determined by the following relation, which is implemented in RELAP5/MOD3.2:

$$D_{drop} = \min\{D_H, \max(D_{we}, D_{\min})\},$$

where D_H is the hydraulic diameter of the flow path.

With this method, D_{drop} is not allowed to be less than 0.0025 m for $p^* \leq 0.025$, which is evidently false when it is compared with the experimental results of different authors [5]. After correction it is apparent that the presented model predicts the experimental data reasonably [6]. The average droplet diameters measured in these experimental results are less than 0.0025 m.

To correct the droplet size at low pressure the following method is suggested based on Weber number criterion and the limitation on the minimum drop size:

$$
\begin{aligned}
D_{\min} &= 0.0008 \qquad for \quad p^* \leq 0.025, \qquad We = 2 \\
&= 0.0008 + (0.0002 - 0.0008).(4.4444) * (p^* - 0.025) \qquad for \quad 0.025 < p^* < 0.25 \\
&= 0.0002 \qquad for \quad p^* \geq 0.25
\end{aligned}
$$

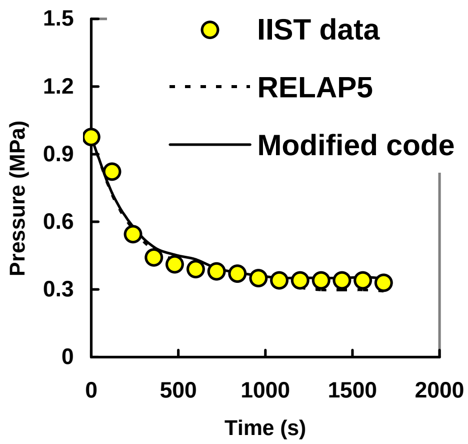

Fig. 3: Primary system pressure.

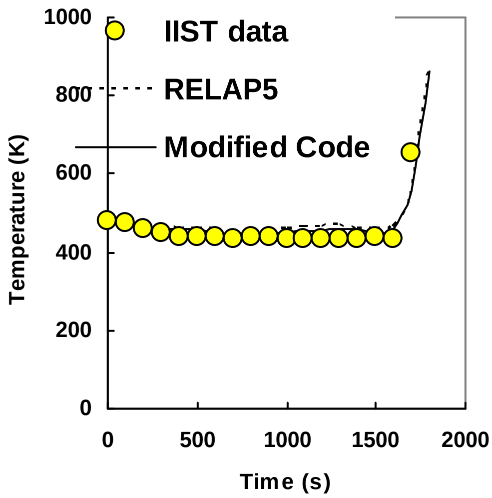

Fig. 4: Core cladding temperature.

APPLICATIONS OF THE MODEL ON LOCA

The modified model [5] was implemented in the code of RELAP5/MOD3.2 to simulate and study the actual reactor facility. It is used to study the pressure and temperature during Loss Of Coolant Accident (LOCA). The experimental data of the IIST [1] was compared with the modified code results. Figure 3 shows the result of comparison of the present investigation results computed at their base case parameters with the IIST measured data. Figure 3 shows that the calculated primary pressure trend is in good agreement with the test results. However, the model can well predict the time to reach core uncover (sharp increase in temperature) associated with heatup of the cladding at 1752 s (compared to 1734 s in the experiment), as shown in Fig. 4. As shown the code is able to qualitatively and quantitatively predict the pressure and temperature transients of the reactor coolant system. The modified code gives better comparison with the experimental data than the original code of RELAP5.

CONCLUSION

A mathematical model was used to study the phenomena of condensation in the presence of noncondensable gases. It was shown that the model can correctly predict all the important data trends. This model was applied to degraded steam condensation in a vertical tube due to the presence of a noncondensable gas. Such a phenomenon could occur in a loss of residual heat removal accident in a pressurized water reactor. The modified version of the code shows better accuracy when comparing with the available experiment data. The model is able to qualitatively and quantitatively predict the pressure and temperature transients of the reactor coolant system.

REFERENCES

[1] Lee, C.H., Liu, T.J., Hong, W.T., Huang, I.M., Chan, Y.K., and Chang, C.J., Nucl. Sci. J., **33**, 251 (1996).
[2] Carlson, K.E., et al, RELAP5/MOD3 Code Manual, Volumes 1, 2, 3, NUREG/CR-5535, EGG-2596, (1990).
[3] Ghiaasiaan, S.M., Kamboj, B.K., and Abdel-Khalik, S.I., Nuclear Science and Engineering, **119**, 1-17 (1995).
[4] Yao, G.F., Ghiaasiaan S.M., and Eghbali, D.A., Nuclear Engineering and Design**, 166**, 277-291 (1996).
[5] Rihan, Y., "Condensation with Noncondensable Gas in Vertical Tubes", Ph. D. Thesis, Alexandria University, Faculty of Engineering, Egypt, (2004).
[6] Sorour, M., Teamah, M., Soliman, S., and Rihan, Y., International Journal of Heat Exchangers, **V**, 51-78 (2004).