Proceedings of the 18th International Conference on Nuclear Engineering
ICONE18
May 17-21, 2010, Xian, China

ICONE18-30262

MODELING IMPROVEMENTS FOR SYSTEM CODE EVALUATION OF INLET PLENUM MIXING UNDER SEVERE ACCIDENT CONDITIONS USING CFD PREDICTIONS

Christopher Boyd
U.S. Nuclear Regulatory Commission
Rockville, MD, USA

Kenneth Armstrong
U.S. Nuclear Regulatory Commission
Rockville, MD, USA

ABSTRACT

An updated mixing model is developed for application to system codes used for predicting severe accident-induced failures of steam generator (SG) U-tubes in a pressurized-water reactor. Computational fluid dynamics is used to predict the natural circulation flows between a simplified reactor vessel and the primary side of an SG during a hypothesized severe accident scenario. The results from this analysis are used to extend earlier experimental results and predictions. These new predictions benefit from the inclusion of the entire natural circulation loop between the reactor vessel upper plenum and the SG. Tube leakage and mass flow into the pressurizer surge line also are considered. The predictions are utilized as a numerical experiment to improve the basis for simplified models applied in one-dimensional system codes that are used during the prediction of severe accident natural circulation flows. An updated inlet plenum mixing model is proposed that accounts for mixing in the hot leg as well as the inlet plenum region. The new model is consistent with the predicted behavior and can account for flow into a side-mounted pressurizer surge line if present. Sensitivity studies demonstrate the applicability of the approach over a range of conditions. The predictions are most sensitive to changes in the SG secondary side temperatures or heat-transfer rates at the SG tubes. Grid independence is demonstrated through comparisons with previous models and by increasing the number of cells in the model. This work supports the U.S. Nuclear Regulatory Commission (NRC) studies of SG tube integrity under severe accident conditions.

INTRODUCTION

SG tubes are an important component of the primary system pressure boundary in a pressurized-water reactor (PWR). During certain hypothesized severe accident scenarios, the primary reactor coolant system (RCS) pressure boundary can be challenged by the combination of high pressure and temperature conditions. These conditions can lead to a thermally induced creep rupture failure of RCS components. If an SG tube fails during this type of scenario, a potential exists for fission products to bypass the containment system through the SG secondary side with a possible release of radioactive material into the environment. For this reason, this low probability event is studied.

One scenario of interest involves a station blackout (loss of all AC power) with a subsequent loss of secondary side cooling. This can result in a boiloff of water from the primary RCS that may remain at high pressure. As the core is uncovered, the steam in the upper parts of the primary system begins to superheat, and natural circulation flows transfer heat from the core region to the metal mass of the primary coolant system loops. A countercurrent natural circulation flow pattern is expected under these conditions, and this has been experimentally observed as illustrated in Figure 1. The superheated steam from the vessel flows along the top of the hot leg (HL) to the SG inlet plenum and then rises up into a portion of the SG tubes. In this scenario, the reactor coolant loop seal region is filled with water and prevents the steam from flowing back to the reactor vessel in the normal forward path through the cold leg. The flow is forced to turn around in the outlet plenum region of the SG and flow back to the inlet plenum through a portion of the SG tubes. This return flow mixes with the hot forward flow in the SG inlet plenum, and a portion of it flows back to the reactor vessel along the bottom of the HL to complete the counter-current flow loop. This type of flow pattern exists during periods of time when the pressure in the system is slowly changing and unaffected by rapid pressure drops caused by relief valve cycling. The relief valve cycles occur when the system pressure slowly rises to the valve set points and the valve opening interrupts the natural circulation flows. The flows have been predicted to resume very quickly after the relief valve is closed. The counter-current flows transfer heat from the core region out into the reactor loops and result in the heatup and potential induced failures of RCS components. Prediction of these flows is a challenge for typical one-dimensional system code models such as NRC's TRACE or MELCOR codes since turbulent mixing and multidimensional effects are not explicitly modeled. For instance, countercurrent flow in an HL is precluded by one-dimensional pipe assumptions that limit a single-phase flow to one direction in a pipe volume. The typical solution applied for system code models is to split the HL into two separate pipes. This allows a separate flow path for the forward and reverse directions. Mixing parameters are established to mix these separate streams in the SG inlet plenum to account for the experimentally observed mixing. An example of the application of system codes for this type of application and an assessment of the risk of induced failures is found in a report published[1] by NRC.

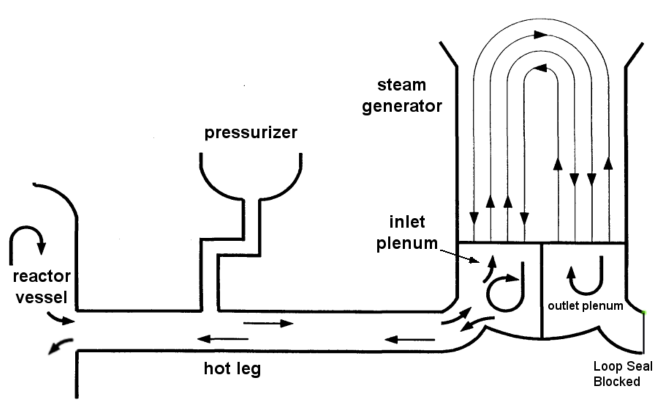

Figure 1. Natural Circulation Flows in Reactor Loop

The effort outlined in this report is one step in NRC's efforts to reduce uncertainty and improve the basis for regulatory decisions related to severe accident induced SG tube failures. NRC's Office of Nuclear Regulatory Research is studying the the flows illustrated in Figure 1 using computational fluid dynamics (CFD). A CFD modeling approach has been benchmarked[2] (NUREG-1781) for these flows using some of the available 1/7th scale data, and this effort demonstrated the ability of the technique to predict key mixing parameters and temperatures in the SG inlet plenum and tube entrance region. The approach was subsequently extended to full-scale[3] (NUREG-1788) conditions in support of system code model development for NRC's research on the potential for severe accident-induced SG tube failures. The current effort builds upon these earlier reports with an improved computational model and assumptions. Most importantly, the basic assumptions of the mixing model are reconsidered to more accurately account for the predicted mixing and entrainment behavior. One challenge is implementing the results from a three-dimensional CFD code into the one-dimensional framework of a system code approach. This study suggests some modifications to the existing approach that make the application more consistent and repeatable.

NOMENCLATURE

- r recirculation ratio (m_t / m)
- m_t mass flow in tube bundle
- m mass flow in HL
- m_s mass flow in surge line
- T_c lower HL temperature
- T_m inlet plenum mixed temperature
- T_h upper HL temperature
- T_{ht} hot tube temperature in tube bundle
- T_{ct} cold tube temperature in tube bundle
- T_s surge line flow temperature
- f mixing fraction

INLET PLENUM MIXING MODEL

The earlier NRC CFD work (References 2 and 3) focused on demonstrating that the CFD method can accurately predict the inlet plenum mixing at 1/7th scale and extended the results to full-scale conditions. The inlet plenum mixing is evident from the experimental and computational results that indicate a significant drop in temperature between the end of the HL and the tube entrance region. Figure 2 illustrates a predicted temperature profile on the vertical HL centerline for a simplified model of a vessel, HL, and the primary side of an SG. Temperatures in the upper HL are observed to slowly drop as the flow travels toward the SG inlet plenum. A significant drop in temperature is observed as the flow exits the HL nozzle and flows upwards toward the tube sheet (entrance to the SG tubes). This temperature result is attributed to the inlet plenum mixing.

The mixing models[4] that have been developed for one-dimensional pipe code models have focused on the inlet plenum mixing and are derived to account for the temperature drop that occurs between the end of the HL (T_h – HL end, Figure 2) and the tube entrance (T_{ht}, Figure 2). The experimental results at 1/7th scale provided temperatures at the end of the HL for the upper hot flow and at the entrance to the hot tube region. These data were utilized to quantify inlet plenum mixing. Subsequent

NRC CFD evaluations (References 2 and 3) used temperature predictions at the end of the HL as the basis for the estimation of inlet plenum mixing to maintain consistency with the existing mixing model approach and the experimental results.

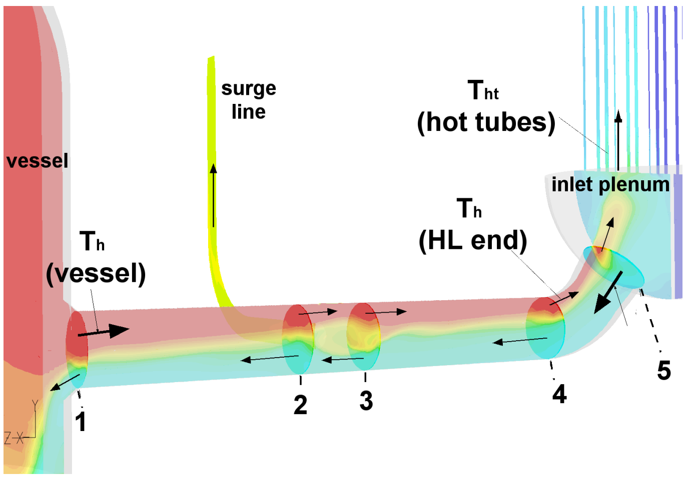

Figure 2. Temperature Contours from CFD Predictions of Natural Circulation Flow (red – hot, blue – cold)

Figure 3 illustrates the nodalization from the end of the split HL to the tube entrance for a system code (one-dimensional pipe components) model setup to apply the inlet plenum mixing as outlined in Reference 4. The approach splits the HL and SG tube bundle into two pipes each. An upper HL carries the hot flow from the vessel to the SG, and a lower HL carries the cooler return flow back to the reactor vessel. The SG tube bundle also is split into two tube regions. The hot tube (a single pipe representing all of the hot flowing tubes) carries the hot flow from the inlet plenum to the outlet plenum, and the cold tube component carries this flow back. The inlet plenum comprises three separate volumes consistent with the assumptions in the inlet plenum mixing model (see Reference 4). This mixing model allows the relatively cool flow returning to the inlet plenum from the cold tubes to mix with a fraction (the mixing fraction, f) of the hot flow from the upper HL before it enters the hot tube. This mixing significantly reduces the temperature of the flow entering the hot tubes and delays the timing of any potential induced failures of the SG tubing.

The inlet plenum mixing model can be derived by applying the first law of thermodynamics and conservation of mass under steady-state-steady-flow conditions to the central inlet plenum control volume (T_m) of Figure 3. It is assumed that the HL and SG tube bundle mass flow rates and temperatures are experimentally determined or predicted with a CFD code. The mixing fraction (f) and the mixed inlet plenum temperature (T_m) are the unknowns that are determined from the mixing model. The mixing fraction represents the fraction of the incoming upper HL flow that mixes completely in the inlet plenum. A recirculation ratio is defined as the ratio of the mass flow in the tubes to the mass flow in the HL (m_t/m). The equations derived for the mixing model formulation are provided below:

$$r = m_t / m$$

$$T_m = (T_h + r\,T_{ct}) / (r + 1)$$

$$f = 1 - r\,(T_{ht} - T_m) / (T_h - T_m)$$

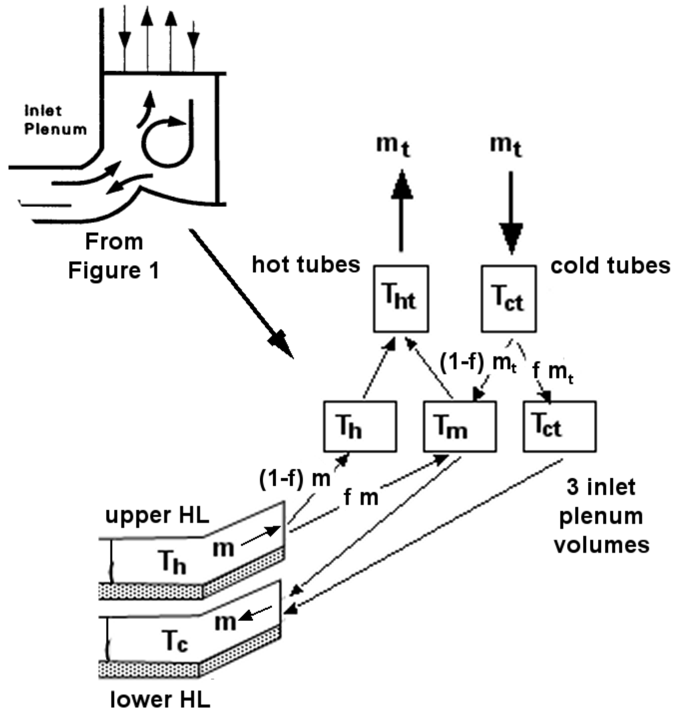

Figure 3. System Code Nodalization of Inlet Plenum Region

Once the mixing fraction and recirculation ratio are determined from experimental or three-dimensional CFD methods, the values are used as a basis to establish the mixing in the inlet plenum volumes of the system code model. Flow coefficients in the system code models are adjusted to ensure that the mass flow rates and flow splits (mixing fraction) are consistent with the predetermined values. This ensures that the tube entrance temperatures and tube bundle mass flow rates are consistent with the experimental observations (or CFD predictions). Because the mixing model formulation only specifies the recirculation ratio and not the absolute value of the flow rates, the HL flow is determined independently. A discharge coefficient[5] approach has been used recently by NRC to establish HL flow rates for these types of scenarios.

CFD MODELING

The most recent NRC CFD predictions[6] of the HL counter current flows represent an update to the NRC's earlier predictions (References 2 and 3). The CFD model utilized represents the primary side of a SG, the HL, a portion of the pressurizer surge line, and a simplified reactor vessel upper plenum along with a small portion of a vessel. The FLUENT 6.3 CFD code is used for the analysis. The predictions qualitatively show all of the flow features observed experimentally in the HL and SG regions. The natural circulation flows are unsteady in nature. To obtain average values, a transient simulation is completed using fixed boundary conditions to produce a series of predictions that are combined to obtain the average behavior. Average mass flows and temperatures are predicted throughout the flow domain and used to find the coefficients for mixing and flow models that are evaluated in this report. Figure 4 shows an overview of the CFD model domain.

The CFD model is based upon the modeling approach that was benchmarked earlier against experimental data (Reference 2). Additions to the model included a significantly improved mesh applied to the tube bundle and inlet plenum regions made possible by the increased computer resources available to NRC at the time this new model was developed. In addition, a grid sensitivity study was completed on the inlet plenum meshing that indicated no grid dependence on the predicted results. The model has been shown to be consistent with the approach that was benchmarked with the 1/7th scale data (Reference 2) and provides further refinement of the results with the significant improvements to the tube bundle model. The earlier work involved square tubes with a porous media formulation to account for the tube heat transfer and pressure drops. This new model combines each 3 x 3 grouping of tubes into a single round tube with the correct flow area. Pressure drops at the inlet are modeled explicitly, and tube bundle pressure drops and heat transfer rates are adjusted to account for the tube groupings through a detailed comparison. Further details of the CFD modeling effort can be obtained from Reference 6 and are beyond the scope of this report.

CFD RESULTS AND OBSERVATIONS

Early on during NRC's CFD evaluations of the 1/7th scale test data and subsequent evaluations under full-scale conditions, it was observed that some mixing and entrainment had occurred in the HL region. This mixing is not accounted for in the system code formulation because the upper and lower HLs are completely separated. The CFD predictions indicate that the hot flow in the upper HL accelerates down the pipe and entrains the lower cooler fluid back into the forward flow direction. This effectively reduces the temperature and increases the flow rate at the end of the HL in a manner that is not accounted for in the one-dimensional pipe model formulation or the mixing model formulation.

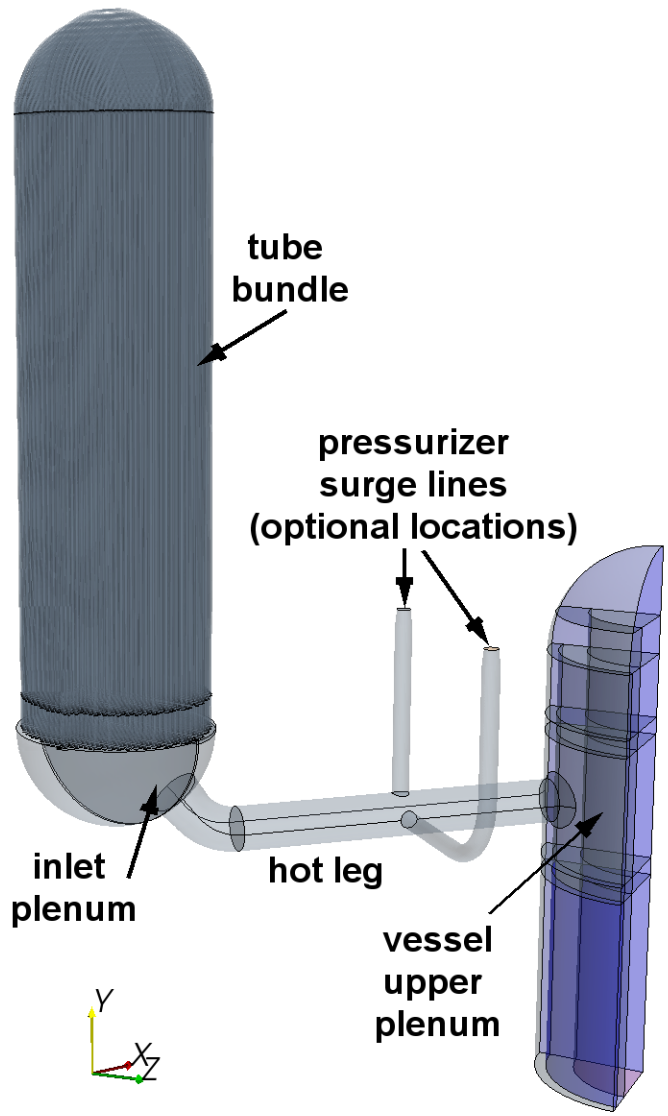

Figure 4. Overview of CFD Model Domain

In the determination of the mixing fraction and recirculation ratio from the CFD predictions, it became clear that the results are sensitive to the location of the measurement or prediction of the HL temperature (T_h) and mass flow. This analysis establishes a fixed location for this measurement and removes this potential inconsistency in the results.

Another consideration for the HL flows is the possible presence of the pressurizer surge line. At the highest pressure conditions, where the system is slowly pressurizing toward the pressurizer relief valve set point, mass from the HL is forced into the pressurizer surge line as the system pressure increases. Because these surge line flows can be a significant fraction of the HL mass flow, they need to be addressed. In addition, RCS components in the reactor loop with the pressurizer have been predicted to fail earlier than the same components in nonpressurizer loops when the pressure is high enough to trigger the relief valves to cycle. If the system is slowly depressurizing, flow in the surge line is from the pressurizer to the HL and, in these cases, it has been predicted that nonpressurizer loop components are the first to fail. For this reason, the presence of the pressurizer is only considered significant during the high pressure cases.

Figure 2 shows temperature contours that illustrate results for a typical CFD prediction with surge line flows from Reference 6. In addition, Figure 2 highlights five cross-sectional slices of the HL pipe. These are used to obtain integrated mass flows and temperatures along the HL for flows in both directions. Section 1 is at the junction between the HL and reactor vessel. Sections 2 and 3 are on either side of a side-mounted surge line junction. Section 4 is at the end of the horizontal HL section where the HL temperature and mass flow are typically referenced in the mixing model formulation. Section 5 is the junction between the HL elbow and the inlet plenum. Table 1 provides results for a prediction of mass flow and mass-averaged temperature for the forward and reverse flows at each of the cross sections. These results help to illustrate the HL mixing and entrainment and represent a single snapshot in time during the unsteady simulation.

Table 1. Prediction of HL Flows and Temperatures

Upper HL Flow – Vessel to Inlet Plenum					
Section	*1*	*2*	*3*	*4*	*5*
m(kg/s)	4.46	5.14	4.42	4.95	5.60
T (K)	1,231.6	1,203.2	1,213.1	1,152.4	1,105.5
Lower HL Flow – Inlet Plenum to Vessel					
Section	*1*	*2*	*3*	*4*	*5*
m(kg/s)	3.19	3.85	4.43	4.98	5.67
T (K)	952.5	949.2	954.7	920.5	903.0

It is important to note that the CFD model assumed adiabatic walls for the HL so that the observed temperature reduction is due to mixing only. The snapshot of data in Table 1 indicates over 100 K of temperature reduction in the upper HL from one end to the other. This is considered significant. Another observation from the CFD predictions in Reference 6 is that the flow into the side-mounted surge line, on average, comes nearly equally from the forward and reverse HL flow paths. This flow split into the surge line is very unsteady, and the snapshot in time from Table 1 is not representative of the long-term average.

The predicted upper HL flow rates and temperatures indicate entrainment of the lower cold leg flows between Sections 1 and 2. The flows at Sections 2 and 3 are used to determine the origin of the flows that enter the pressurizer surge line (which in this case is mounted horizontally to the pipe centerline between Sections 2 and 3). Beyond the surge line junction, the upper HL flow continues to accelerate and entrain the cooler HL flow as illustrated by the reduction in the cross-sectional area of the hot flow region along with the increased mass flow observed in Table 1. These results suggest that focusing only on inlet plenum mixing may not accurately represent the overall loop flows and temperatures. They also highlight the ambiguity of defining the mixing fraction and recirculation ratio from a somewhat arbitrary point near the end of the HL. A more consistent approach is suggested that removes the ambiguity surrounding the measurement location for the HL flow and temperature while accounting for potential mixing in the HL region and the presence, if any, of surge line flows. This approach has been applied successfully in NRC's latest severe accident predictions conducted by the Office of Nuclear Regulatory Research.

UPDATED MIXING MODEL APPROACH

The updated mixing model applies the lessons learned from the CFD predictions resulting in a more consistent approach. The new approach includes both the HL and inlet plenum mixing in the determination of the mixing fraction and the recirculation ratio. The goal is to provide best estimate mixing and entrainment modeling for flows leaving the reactor vessel to ensure realistic predictions of the tube bundle flow rates and temperatures that can potentially lead to induced tube ruptures. The solution that is proposed simply modifies the location where the HL (hot temperature and mass flow) values are referenced. In the past, the mixing has been quantified by comparing the temperatures between Section 4 on Figure 2 and the tube entrance plane. Similarly, the recirculation ratio is based upon the flow rates at these two locations. The new approach obtains the HL temperature and mass flow at the vessel junction (Section 1 in Figure 2). This effectively sets up a larger mixing zone that spans the entire HL and inlet plenum regions. In addition, this approach takes the entrainment in the HL into account when defining the recirculation ratio. Another benefit of this approach is the clear definition of the location for measuring the HL flow and temperature. The HL flow and temperature at location 1 represents the flow conditions that enter the HL from the vessel.

The approach needs to account for the inclusion of the pressurizer surge line flows on the mass balance because failures in the loop with the pressurizer have been predicted to occur prior to failures in nonpressurizer loops in some of the highest pressure scenarios. For a side-mounted surge line, the CFD predictions in Reference 6 indicate that the surge line flows come nearly equally from the upper and lower HL flows.

Figure 5 shows a diagram of the updated mixing model flow paths with the inclusion of a pressurizer surge line.

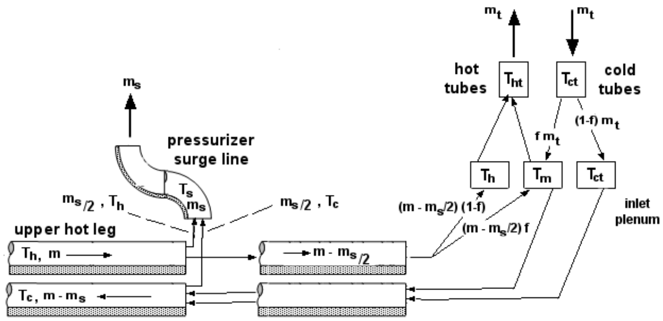

Figure 5. Nodalization Diagram for Updated Mixing Model

The same approach is used for the updated mixing model. A steady-state-steady-flow first law analysis is applied to the inlet plenum mixing volume and the surge line mass flow is taken into account. The following equations are derived for the mixed mean temperature and the mixing fraction:

$$r = m_t / (m - m_s/2)$$

$$T_m = (T_h + r\,T_{ct}) / (r + 1)$$

$$f = 1 - r\,(T_{ht} - T_m) / (T_h - T_m)$$

The equations are identical to the previous mixing model with the exception of the definition of the recirculation ratio, r, which takes into account the potential for surge line flows. For loops without a surge line ($m_s = 0$), the equations are identical to the previous formulation.

Application of this new mixing model to system codes is similar to the previous approach. The principal difference is in the definition of the upper HL temperature and the allowance for the surge line flows. Instead of getting the HL temperature from the SG end of the HL, the temperature is monitored at the junction between the vessel and the upper HL. In the system code, the upper HL mass flow is monitored downstream of the surge line and therefore already has the surge line mass flow removed.

Recent NRC efforts to predict these severe accident natural circulation flows for a Westinghouse plant model with a side-mounted surge line have been successful. However, careful attention to the details is necessary to ensure the modeling is consistent with the experimental or computational observations and the assumptions in the mixing model formulation. The surge line mass flows are one area that required some additional attention. In the recent NRC experience, a horizontal pressurizer surge line connection to the HL was positioned at the mid-level elevation of the HL and connected to both the upper and lower HL components. In this configuration, all of the flow to the pressurizer through this surge line was predicted to come from the upper HL flow. The system code had to be modified slightly to match the CFD predictions in this case. The problem here is the separation between the upper and lower HL pipes. The surge line connection to each of the pipes is actually separated by a large flow distance around the natural circulation loop, and the pressures at these two locations are not the same. Because the lower HL is downstream of the upper HL, it has a lower pressure. The solution to this issue is to force the upper and lower HL flows to be equal going into the surge line and therefore maintain consistency with the CFD predictions of the surge line flows.

Another consideration for the system code model is that the upper and lower HLs are completely separate and no mixing is possible between the two pipes. This does not pose a problem for the overall application of the mixing model, but it is noted that all of the HL and inlet plenum mixing is forced to happen in the inlet plenum volumes with this approach. The mass flow in the upper HL is isolated and does not change (with the exception of the surge line flow) as the hot steam mixture progresses from the vessel to the SG. The NRC approach establishes the HL flow at the vessel end of the HL (as outlined in Reference 5) to ensure that the system code predictions of flows leaving the vessel are consistent with experimental results or appropriate CFD predictions. It is important that the system code prediction of HL flow and temperature at the vessel end of the HL is accurate since this condition affects the core cooling and therefore the overall system response. The HL region near the vessel is also important since this is the most likely region of interest for induced failures of the HL. Temperatures and heat transfer further down the HL in a one-dimensional system code model (closer to the SG) do not see the expected mixing and entrainment due to the separated HL pipe flows, and this impact could be significant depending on the specific application. The proposed modeling approach is appropriate for the purposes of predicting potential induced failures of the HL that are expected to occur near the vessel end of the HL where temperatures are highest.

A key area of interest for induced failure screening is the tube bundle region. This updated mixing model approach applies the full amount of mixing and entrainment to the hot forward flow prior to the flow entering the tube sheet. The average tube bundle mass flow and temperature is consistent with the three-dimensional CFD predictions at this location. The proposed approach provides the best estimate of tube bundle mass flows and temperature and therefore is considered adequate for tube bundle failure modeling in the system code.

The updated mixing model is essentially a reevaluation of the existing approach with recommendations on where to measure the key parameters to account for the mixing and entrainment in the reactor loop under these specific severe accident natural circulation conditions. In the past, the HL mass flow and temperature was defined to be consistent with measurements or predictions near the SG end of the HL. All mixing and entrainment was generally referred to as inlet plenum mixing. For a given experiment or CFD prediction, moving the reference point for the HL mass flow and temperature to the vessel end of the HL lowers the value for the HL mass flow and increases the HL temperature. This results in an increase in the prediction of both the recirculation ratio and mixing fraction.

The proposed mixing model refinements attempt to account for mixing and entrainment in the HL, which appears to be significant based on NRC's most recent set of CFD predictions. The method works well with the approach used to establish the HL flows (Reference 5) that also are referenced at the vessel end of the HL. Referencing the HL flow and temperature at the entrance to the HL from the vessel removes a potential ambiguity in the reference location. A potential bias is eliminated through this clear definition of the HL flow reference point.

The potential for a side-mounted surge line is considered in this model. Flow into the surge line is impacted by the rate of change in the system pressure. In cases where the system pressure is increasing, flow goes from the HL to the pressurizer through the surge line. In these cases, the relief valve set points can be reached and, when the relief valves open for short periods of time, the surge line flows rise quickly and disrupt the natural circulation flows. Recent NRC predictions indicate that components in the pressurizer loop fail earlier than the components in other loops in these high-pressure scenarios. It is noted that the mixing model formulation is only valid for periods of time when the relief valves are closed.

In cases where the pressure is decreasing, the flow is from the pressurizer to the HL. Recent NRC predictions indicate that when flow enters the HL from the pressurizer, the failures in this loop (the pressurizer loop) are delayed relative to the other nonpressurizer loops. This is the result of the cooler gas entering the loop from the pressurizer. In these cases, the focus shifts to loops without the pressurizer where failures are more likely, and the mixing model has not been used for cases where flow enters the HL from the surge line.

The most important aspect of this new approach is the definition of parameter locations. CFD evaluations indicate that HL mixing and entrainment are significant enough to consider. It is recommended that experimental evaluations consider the impact of HL flow and entrainment when developing data for mixing fractions and recirculation ratios under these conditions.

CONCLUSIONS

The NRC's Office of Nuclear Regulatory Research has built upon prior work with an improved CFD model and an updated evaluation of inlet plenum mixing during severe accident natural circulation flows. This study benefits the evaluation of induced primary system failures in the HL and tube bundle during low probability severe accident scenarios. An improved method of application for the mixing model has been used by NRC that incorporates HL mixing and entrainment into the inlet plenum mixing model. The approach is similar to previous efforts and is applied in system code models using a similar approach. A consideration of the potential for HL mixing and entrainment should be included in any future CFD or experimental investigations of these phenomena.

ACKNOWLEDGMENTS

The authors acknowledge Don Fletcher at Information Systems Laboratories, Inc. for his expertise and advice on system code modeling issues related to the prediction of this severe accident behavior.

REFERENCES

1. U.S. Nuclear Regulatory Commission, "Risk Assessment of Severe Accident-Induced Steam Generator Tube Rupture," NUREG-1570, March 1998, ADAMS Accession No. ML003769765.
2. U.S. Nuclear Regulatory Commission, "CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1781, October 2003, ADAMS Accession No. ML033140399.
3. U.S. Nuclear Regulatory Commission, "CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1788, May 2004, ADAMS Accession No. ML041820239.
4. D. B. Ebeling-Koning, et al., "Steam Generator Inlet Plenum Mixing Model for Severe Accident Natural Circulation Conditions," 1990 ASME Winter Meeting, Dallas, Texas.
5. C.F. Boyd, "Prediction of Severe Accident Counter Current Natural Circulation Flows in the Hot Leg of a Pressurized Water Reactor" (U.S. Nuclear Regulatory Commission), Proceedings of ICONE 14, International Conference on Nuclear Engineering, 17-20 July 2006, Miami, Florida.
6. U.S. Nuclear Regulatory Commission, "Computational Fluid Dynamics Analysis of Natural Circulation Flows in a Pressurized-Water Reactor Loop under Severe Accident Conditions," NUREG-1922, February 2010.