CFD Predictions of Severe Accident Natural Circulation Flows in a Combustion Engineering PWR

Christopher Boyd
Nuclear Regulatory Commission
Washington DC, 20555
christopher.boyd@nrc.gov

ABSTRACT

Computational fluid dynamics (CFD) is used to predict natural circulation flows between a reactor vessel upper plenum and a steam generator during specific severe accidents in a pressurized-water reactor (PWR). The predictions are based on a model for a simplified Combustion Engineering (CE) design. This work is part of a larger study to investigate the potential for severe accident induced steam generator (SG) tube failures in PWRs being conducted by the U.S. Nuclear Regulatory Commission Office of Nuclear Regulatory Research. The CFD model builds upon the lessons learned from previous studies of a generic Westinghouse PWR under similar conditions. The CE SG is significantly larger than the Westinghouse design and geometric differences are expected to reduce the mixing and increase the potential thermal challenge to the SG tubes. The CFD predictions are utilized to improve the basis for simplified flow and mixing models applied in one-dimensional severe accident codes. Parameters are determined to support system code modeling of hot leg and SG inlet plenum mixing, hot leg and SG tube bundle flow rates, and tube inlet temperatures. This updated modeling uses boundary conditions obtained from system analysis code predictions of severe accident behavior in a CE PWR. Model refinements include an improved tube bundle representation and improvements to the thermal boundary conditions. Results indicate less mixing and higher tube inlet temperatures for the CE design compared to previous Westinghouse studies.

KEYWORDS
Severe Accident, Induced Failures, Steam Generator Tube, Mixing, Natural Circulation

1. INTRODUCTION

The U.S. Nuclear Regulatory Commission (NRC) has spent a number of years studying severe accident induced steam generator tube failures. The scenarios of interest include a flow of superheated steam from the reactor vessel to the steam generators (SGs) that thermally challenges the integrity of the reactor coolant system boundaries. These conditions ultimately lead to a thermally induced failure. Induced failure of a SG tube (or tubes) has the potential for a containment bypass as the primary system gases escape into the SG secondary side. The potential for containment bypass is the reason that this low probability event is studied. Predicting failure location and timing during a severe accident is a difficult task. Details of the flow and heat transfer within the system are needed to assess the thermal challenge. Severe accident system analysis codes such as MELCOR or SCDAP/RELAP5 are used to predict the

overall severe accident behavior. These codes, however, do not model details of the flow and heat transfer that can be significant factors in the determination of induced failures. In cases where three-dimensional behavior is important, the one-dimensional system code models are typically adjusted to be consistent with the expected flow behavior that is determined separately from experiments or suitable multidimensional codes. An early risk assessment for severe accident induced SG tube failure is documented in NUREG-1570 [1]. More recent studies have improved upon that work with updated risk analysis, improved material and structural modeling, and refined thermal-hydraulic analysis. A recent thermal-hydraulic analysis for Westinghouse type PWRs is documented in NUREG/CR-6995 [2].

The severe accident sequences of interest are ones that challenge the integrity of the steam generator (SG) tubes which form an important part of the primary system boundary. In specific severe accident scenarios, the primary system remains pressurized and the systems that deliver emergency water to the steam generators are assumed to fail. The SGs dry out and lose the ability to transfer heat from the primary system. In addition, if significant leakage occurs on the SG secondary side, the secondary side depressurizes resulting in an increased pressure difference (dP) across the SG tubes. The lack of heat transfer away from the primary system causes the primary inventory to boil off leading the system into severe accident conditions. This unlikely series of events leads the plant into a high-dry-low condition. This refers to high primary system pressure combined with dry steam generators and a low SG secondary side pressure. Under high-dry-low conditions, as fuel damage begins, the reactor loops become filled with highly superheated steam and hydrogen. These conditions thermally challenge the primary system components and the details of the natural circulation flows during this period are the subject of these analyses. The flow pattern of interest has been experimentally observed [3] and is illustrated in Fig. 1. This process transfers heat from the reactor core out into the reactor coolant loop structures. Countercurrent flow is expected when the reactor coolant pump loop seal region is filled with water and full loop circulation is blocked. If the loop seal region is cleared along with the lower downcomer, full loop circulation is established and the CFD analyses completed in this report are not applicable.

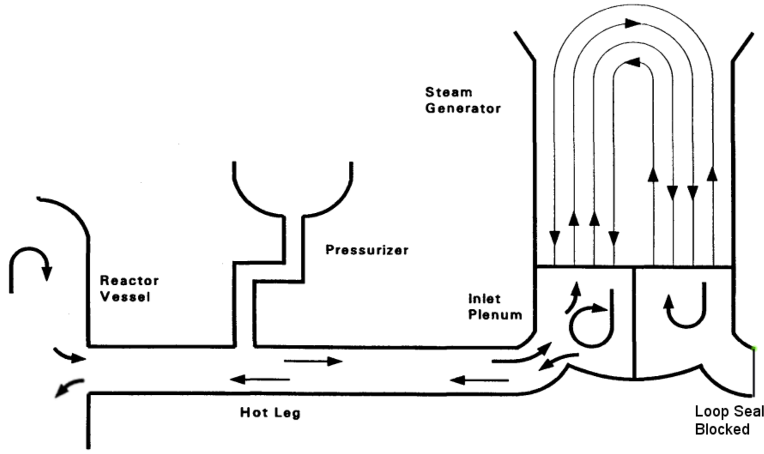

Figure 1. Natural Circulation Flow Pattern

This report represents an extension of a series of efforts to develop and improve the NRC's ability to model the countercurrent natural circulation flows in Fig. 1. Initial efforts focused on benchmarking a CFD model against a set of 1/7th scale test data [4]. This benchmarking exercise demonstrated that CFD methods could be used to predict the inlet plenum mixing, countercurrent natural circulation flow rates, and tube entrance temperatures. A follow on study looked at issues related to prototypical SGs and the scale up from $1/7^{th}$ to full-scale [5]. This work highlighted the significance of inlet plenum design and the scaling of tube bundle heat transfer. It became clear that the 1/7th scale test data included certain scaling distortions that limited their direct application to full-scale plant models. A more recent analysis of the 1/7th scale tests and related scaling issues was completed at the Paul Scherrer Institut in Switzerland [6]. This analysis also determined that CFD methods are adequate for determining the general mixing behavior in the SG inlet plenum and highlighted some of the scaling distortions of the test data.

The NRC built upon the lessons learned from the 1/7th scale benchmark exercise and the follow-on scale up studies to develop an improved CFD model for a Westinghouse type PWR [7]. Refinements included the addition of a reactor vessel upper plenum to facilitate predictions of the full natural circulation flow loop, a surge line connection to the hot leg to study the impact of surge line flows, and a significantly improved tube bundle model. The results have been utilized to determine updated flow and mixing parameters that can be applied in system code models used to predict this type of severe accident behavior. Results included a Froude based correlation for the hot leg flows, an improved mixing model for hot leg and inlet plenum mixing, mixing analyses for both top and side mounted pressurizer surge lines, and an improved understanding of the tube bundle flow and temperature distribution.

Flows and mixing are expected to be different for CE type SGs due to differences in the system design. The generic CE plant SG inlet plenum design considered in this analysis is compared to a generic Westinghouse design in Fig. 2. The CE steam generator is larger and contains over 8000 tubes compared to approximately 3400 tubes in the smaller Westinghouse design. The CE hot leg is larger, closer to the tube sheet, and oriented differently than the Westinghouse design. The most significant difference in the CE design with respect to mixing, is the relative distance from the exit of the hot leg nozzle to tube sheet. The reduced distance results in less mixing and potentially hotter flows entering the tubes. Fig. 3 illustrates a generic steam generator with some basic dimensions identified. Table 1 lists approximate dimensions for the 1/7th scale test facility (Westinghouse, 1993), a generic Westinghouse SG, and the CE steam generator considered in this report.

The CE analysis reported here builds upon the lessons learned from the prior Westinghouse studies [7]. Prior studies included a consideration of grid and time step sensitivities, turbulence modeling studies, and the development of boundary condition and tube bundle modeling approaches. The current effort utilizes a similar modeling approach in order to retain a level of consistency with prior predictions. Any differences found in the predicted mixing parameters for a CE design are therefore attributed mainly to the system design differences and not to variations in the modeling approach.

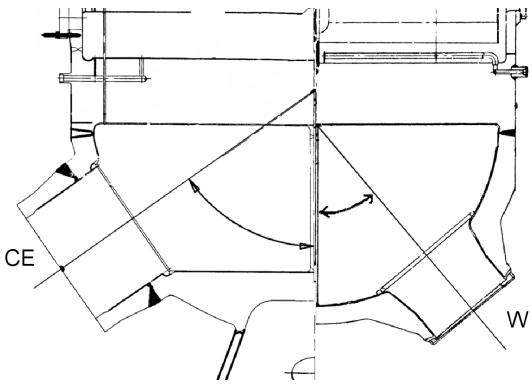

Figure 2. CE vs. Westinghouse SG Plenum

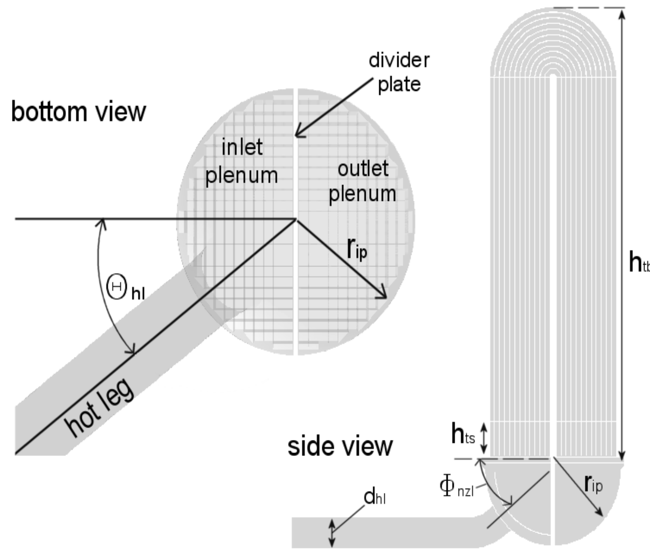

Figure 3. SG Dimensions

Table I. Approximate Dimensions of Steam Generators

Dimension	Westinghouse 1/7th facility	Westinghouse Model 51	CE SG
hot leg diameter, d_{hl}	0.1022 m (4.03”)	0.7366 m (29”)	1.0668 m (42”)
inlet plenum radius, r_{ip}	0.2413 m (9.5”)	1.595 m (62.8”)	1.968 m (77.5”)
hot leg orientation, Θ_{hl} (degrees)	0	36.5	0
nozzle angle, Φ_{nzl} (degrees)	45	50	35
total number of tubes	216	3388	8471
tube inner diameter, d_t	0.00775 m (0.305”)	0.01968 m (0.775”)	0.0169 m (0.666”)
thickness of tube sheet, h_{ts}	0.1143 m (4.5”)	0.5342 m (21”)	0.556 m (21.89”)
height of tube bundle including tube sheet, h_{tb}	1.43 m (56.3”)	11.126 m (438”)	9.964 (381.6”)

2. PHYSCIAL DESCRIPTION OF MODEL

The primary system flow paths between the reactor vessel upper plenum and a simplified steam generator are modeled for a generic CE type plant. Preliminary models included a full 3D representation of the geometry but model size became impractical. A symmetry plane along the hot leg and steam generator is applied to cut the model size in half. Fig. 4 shows the physical

model domain which includes a simplified reactor vessel, a hot leg with the initial section of a top mounted pressurizer surge line, and the primary flow paths in the SG.

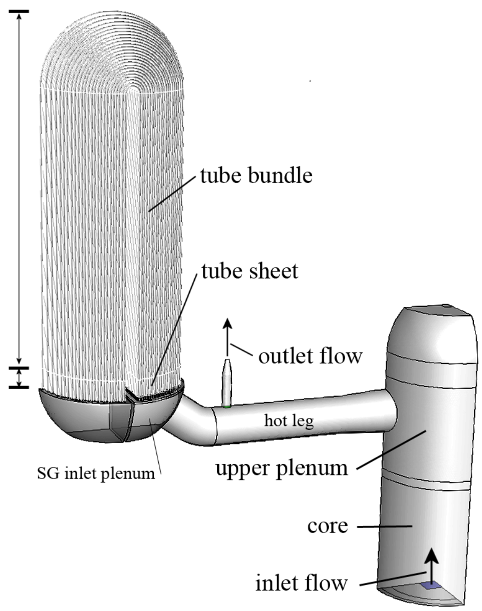

Figure 4. CFD Model Domain

The vessel region represents ¼ of the reactor vessel inside the core barrel extending from near the bottom of active fuel to the upper dome. The simplified core region includes a central region to represent the fuel and a thin outer region to represent the bypass region along the core barrel. Above the core region are surfaces and volumes representing an upper core plate, the upper plenum, the upper support plate, the upper head, and the upper dome. Details of the reactor internal structures are not included. A porous media approach is used to approximate flow losses. Complete blockage, where appropriate, is accomplished using walls. The purpose of the simplified vessel component is to create conditions in the upper plenum region that are consistent with those expected from severe accident code predictions. The vessel portion of the model is not part of the domain of interest in these studies and is treated like a boundary condition attached to the reactor loop where the buoyancy driven flows of interest are established.

The hot leg runs from the vessel upper plenum to the steam generator inlet plenum. There is a 35-degree upward bend at the SG end and the hot leg axis is aligned perpendicular to the divider plate that separates the SG plenums. The centerline distance of the hot leg from the edge of the vessel upper plenum to the entrance of the SG inlet plenum is 6.59 m (259.5 in.). The horizontal section has a length of 4.84 m (190.5 in.) with the remaining length made up by the 35-degree elbow. The diameter of the hot leg is 1.0668 m (42 in.). A small section of the pressurizer surge line is attached to the top of the hot leg. This connection allows surge line flows to be considered as part of the model boundary conditions if needed.

The SG is based on a replacement SG for a CE plant. The bundle includes 8471 individual tubes with inner diameters of 0.0169m and an overall height above the inlet plenum of approximately

10 m. Modeling each tube is impractical with the resources available so the bundle is modeled using fewer individual tubes with larger diameters (D= 0.0501 m). This approach retains total flow area and bundle height while reducing model size and complexity. Each tube in the model represents approximately 9 prototypical tubes. This tube bundle model has a total bundle flow area that is 96.9% of the CE steam generator considered. It is assumed that the remaining 3.1 percent of the tube flow area represents tubes that are plugged. Plugged tubes are not explicitly modeled in this approach. A full three-dimensional representation of the SG model results in 936 individual tubes. Using symmetry, the central row of 20 tubes is cut in half and the resulting model contains 458 full SG tubes along with the 20 half tubes on the symmetry plane. Heat transfer and pressure drop are augmented in the tube bundle using a porous media approach to account for the reduced wetted surface area associated with the larger tubes. Porous media parameters are discussed in the boundary conditions section below.

3. CFD MODELING APPROACH

A overview of the basic CFD solver settings and modeling options is listed below.

- ANSYS/FLUENT version 14.5
- unsteady Reynolds averaged Navier-Stokes (URANS)
- fixed boundary conditions
- 0.05 second time step
- symmetry
- realizable k-epsilon turbulence model
- full buoyancy effects option on turbulence model
- non-equilibrium wall functions
- temperature-dependent thermal properties (steam and hydrogen) at constant pressure
- species tracking model (steam and hydrogen)
- gravity
- segregated solver with 2nd-order differencing on momentum and energy
- porous media model applied to vessel and SG tube bundle region
- 47.3 million computational cells, predominantly hexahedral

The URANS methods are used with fixed boundary conditions in order to find average values over a period of time. The natural circulation flows have some physical variations that require the transient solver. A full transient solution over the duration of the severe accident is not practical so only a snapshot in time is considered. The assumption is that the overall scenario changes relatively slowly compared to the local transient behavior and that a snapshot of the global flow behavior can be obtained using steady boundary conditions. The key part of the scenario is the period of significant core heat up once core damage begins. This process of accelerated core heatup, which drives the system temperatures up and ultimately leads to an induced failure, takes approximately 30 minutes. Local oscillations in the URANS solution are on the order of 30 seconds or less. This separation in time scales makes the quasi-steady approach acceptable for computing a snapshot of the mixing parameters of interest. Solutions are obtained by allowing the URANS solutions to run until long term averages appear steady. The final results are an average of 5 data sets obtained at ten second intervals.

4. BOUNDARY CONDITIONS

The principle boundary conditions that drive the flow include the temperature and gas conditions that are established in the vessel upper plenum along with the tube bundle heat transfer. The hot leg, SG plenums, and vessel walls are adiabatic.

The vessel component is used to establish severe accident conditions in the upper plenum which feed the natural circulation flows. These conditions are determined separately in severe accident analysis codes. Early models included volumetric heat sources in the fuel region to account for the energy coming from the core. This approach made it difficult to get the desired temperatures in the upper plenum since this required a proper balance of the core energy generation, internal flows, heat up of internal structures, and external heat losses. To simplify the model, temperature of all flow through the core and upper core support plate was set to a fixed temperature. A small upward mass flow from the central base of the core moves the gases into the upper plenum. This small mass flow condition is established to replace the gases that are lost through the pressurizer surge line. Mixing and temperature variations are modeled in the upper plenum and upper regions of the vessel but the fixed temperature leaving the core region makes it feasible to quickly establish the desired severe accident conditions in the upper plenum. The simplified vessel model is not expected to accurately produce reactor vessel internal flows and this component is ultimately used only to establish conditions in the upper plenum region that feed the natural circulation flows in the SG loop.

Large temperature variations from top to bottom along the hot leg wall make it difficult to prescribe a suitable thermal boundary condition for a quasi-steady solution. In the final model, adiabatic walls are used for the hot leg and SG plenums. The surface area of these regions is small compared to the tube bundle and the adiabatic assumption is considered a reasonable compromise.

The tube bundle is the most significant region of the model. Tube heat loss affects the temperature along the tube flow length and therefore affects the buoyancy forces that drive the flow. These buoyancy forces are balanced by the pressure drop due to wall shear forces, elevation change, and the abrupt flow area changes at the entrance and exit of the tube bundle. The height of the tube bundle is modeled directly to account for elevation effects and pressure changes at the tube entrance and exit regions are predicted correctly since the tube bundle flow area is preserved. Wall shear force and heat loss, however, need to be adjusted to account for the larger diameter tubes.

Pressure drop along the tubes is adjusted using a porous media approach. The expected pressure drop is predicted using a detailed tube model with the actual tube diameter and best estimate predictions of pressure along the tube flow path are obtained over a wide range of conditions. These predictions are used as the basis to adjust porous media coefficients that are applied in the oversized tube model. This process is outlined in the Appendix of NUREG-1922 [7]. Fig. 5 shows an example of the predictions of pressure along the tube bundle flow path for a prototypical tube, a larger tube of the size used in the CFD model, and a larger tube adjusted with loss coefficients using the porous media formulation. The loss coefficients are adjusted to ensure that the pressure drop for the large tubes is consistent with the prototypical tubes. The

coefficients come from a curve fit that includes conditions from three mass flows each at three temperatures selected to span the severe accident conditions. Equation 1 (written in one-dimension form) is used in the FLUENT codes porous media formulation to add pressure drop to the bundle flows.

$$dP/dx = D \mu V + C \tfrac{1}{2} \rho V |V| \tag{1}$$

In this equation, P is pressure, x is distance along the tube, μ is viscosity, V is velocity, and ρ is the density. The coefficients D (212,717/ m2), and C (1.2/ m) provide the best fit to the data.

Severe accident code predictions of the tube bundle heat transfer at snapshots in time are used as the basis for adjusting the heat transfer. The transient predictions of single phase heat transfer in fully developed pipe flow include the effect of the heat transfer to the secondary side as well as the transient heat up of the tube wall material. A full 3D CFD model of a prototypical CE SG tube is used to predict the temperature drop along the tube with steady flow conditions from a snapshot in time of the severe accident. This CFD model includes flow through the inlet tube sheet, the tube bundle, and the outlet side tube sheet. The inlet and outlet tube sheet regions are modeled with a fixed external wall temperature condition adapted from the transient system code predictions. The remaining portion of the tube has a convective external boundary condition with an external temperature obtained from the transient system code prediction and an external heat transfer coefficient that is adjusted to ensure that heat loss is equivalent to the reference value (predicted by the severe accident code). These CFD predictions for a prototypical tube are then used as a reference case for adjusting the thermal properties of the large diameter tubes.

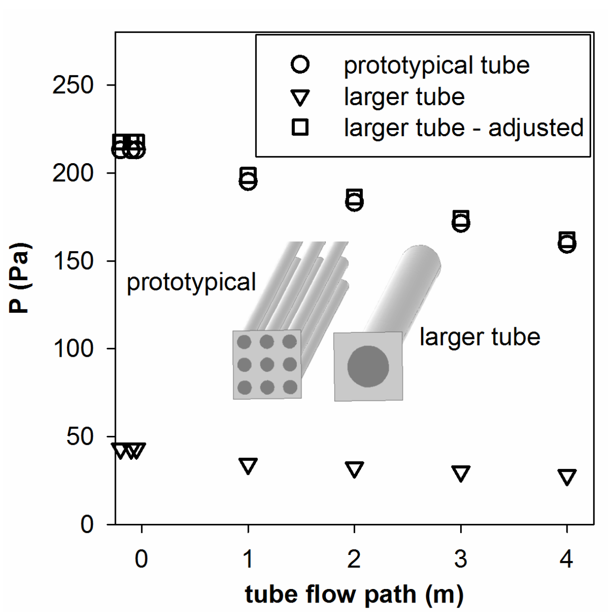

Figure 5. Tube Pressure Models

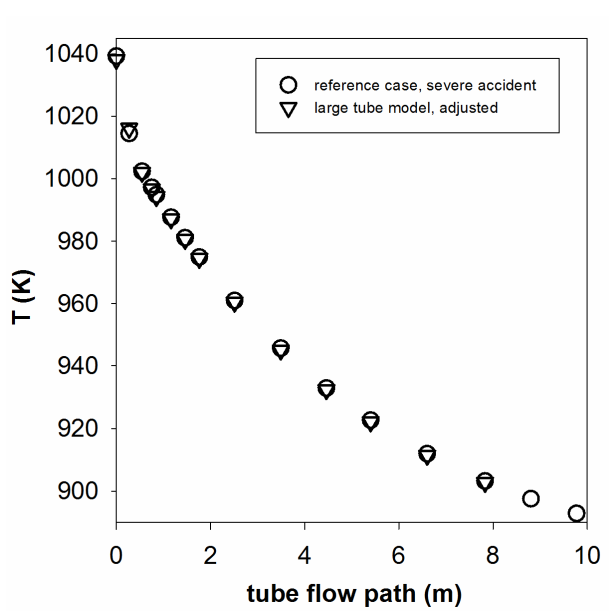

Figure 6. Tube Thermal Model

Tube heat transfer for the large tubes is adjusted using the options available in the porous media formulation of the FLUENT code. The tube sheets are modeled with a wall thickness that

accounts for the metal mass surrounding each tube. Fixed wall temperatures are used as an outer wall boundary condition. The thermal conductivity of the fluid is augmented to help drive more heat to the walls using the porous media option which assumes the flow is through some type of solid matrix. A solid material with a thermal conductivity of 100 W/m-K and a density and specific heat similar to the fluid is created for this purpose. A porosity value of 0.96 (solid material 0.04 by volume) is determined to yield the appropriate heat flow to the walls. In FLUENT, the porous media thermal conductivity is determined as a volume weighted average of the solid and fluid thermal conductivities (with the fluid value incorporating the turbulent component). The boundary conditions on the outer tube in the free span region are established using the external (secondary side) temperature from the reference CFD model along with an augmented heat transfer coefficient. The heat transfer coefficient is set to 300% of the reference case value. It is noted that the reference case CFD model has roughly 300% of the wetted surface area per unit flow compared to the large tube model. The results from this approach for one of the test cases are plotted in Fig. 6. Temperature of the flow along the flow path for the simplified (larger) tube model is consistent with the reference case. This approach ensures that the simplified tube bundle model can predict the temperatures in the tube bundle and ultimately the temperature dependent buoyancy driving forces.

During bundle model testing, it was observed that flow recirculation zones established in the entrance region of some of the tubes (especially when the flow entered at an oblique angle). These recirculation zones occupied the first few tube diameters. This effect, which is assumed to result from the use of oversized tubes, modified the heat balance and had some impact on the consistency of the results. Earlier studies of Westinghouse style steam generators, where the tubes were modeled with a smaller diameter and flow approached the tube sheet in a vertical direction, did not show this effect. Small flow straighteners added to the first 0.1 m (approximately 2 diameters) of the tubes on the inlet plenum side reduced (or eliminated) the recirculation zones. Fig. 7 illustrates the flow straighteners which break the entrance region of each tube down into 5 flow paths. Results with the flow straighteners appear to be more consistent.

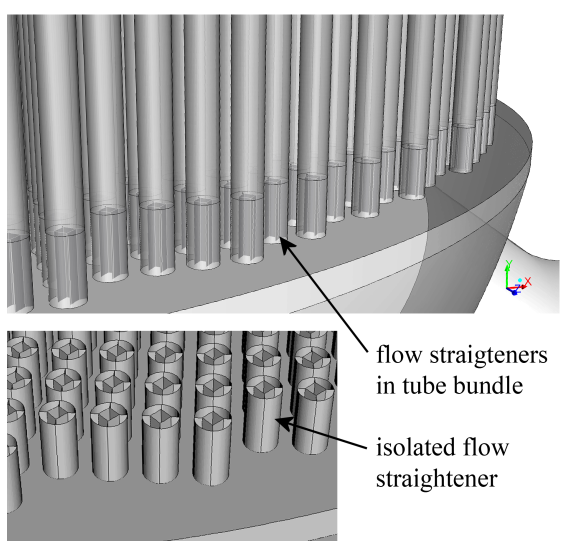

Figure 7. Flow Straighteners at Tube Entrance

Table II contains a summary of the boundary conditions used for the vessel, hot leg, surge line, and SG. These values represent a snapshot in time from severe accident conditions predicted by a system analysis code. The porous media conditions in the tube bundle are also included. Since the inlet tube sheet includes hot flow entering some tubes from below and cooled flow returning through the remaining tubes, the thermal boundary condition for the inlet tube sheet is established as the average of the hot and cold sections of the tube sheet from the severe accident system code predictions. The fluid is a steam and hydrogen (h2) mixture at 2250 psia. Pressure is assumed to remain constant and material properties vary with temperature.

Table II. Summary of Boundary Conditions

Location	Boundary Condition
vessel, fuel region	Fixed temperature 1250K, porous media
vessel, lower inlet boundary	Mass flow = 0.485 kg/s, T = 1250 K, h_2 = 0.00639 (mass fraction)
hot leg; plenum walls	adiabatic
surge line outlet	pressure outlet
tubes (porous media)	D (212,717/ m^2), C (1.2/ m), porosity (0.96), solid (k=100 W/m-K)
inlet side tube sheet wall	Twall (outer) = 911 K, wall thickness = 0.027 m, steel
outlet side tube sheet wall	Twall (outer) = 842 K, wall thickness = 0.008 m, steel
tube wall above tube sheet	convective, h = 105 W/m-K, T_∞ = 869K, wall thickness = 0

5. RESULTS

The solution is started with a steady state solver to create a representative initial condition for the transient solver. The transient solver is executed for approximately 80 seconds of flow time to let the predictions stabilize. Starting from this point, a final series of 5 data sets are recorded at 10 second intervals and these data are averaged to obtain final results.

The overall flow pattern is illustrated with arrows showing flow direction overlaid on temperature contours in Fig. 8. The countercurrent hot leg flow has a relatively thin mixing layer between hot and cold. The interface is sloped upwards toward the SG and the hot flow accelerates in this direction. A small flow exits the top of the hot leg through the pressurizer surge line. The distance from the hot leg exit to the tube sheet is relatively short and the hot flow momentum carries it towards the tube sheet at an oblique angle. Mixing is limited in this region. Hot gases rise into a portion of the tubes and buoyancy forces drive the flow towards the outlet plenum. The blocked loop seal forces the flow to return back to the inlet plenum through the remaining tubes. Temperature contours for the flow at the tube sheet entrance are illustrated in the upper right side of Fig. 8. These contours highlight the hot tube region and the location of the hottest tubes. Temperatures for the flow returning to the inlet plenum are close to the SG secondary side temperature. The cooled flow returns to the vessel through the lower portion of the hot leg. Vessel upper plenum temperatures are dominated by the hot upward flow from the core region along with a small amount of mixing with the colder returning flow from the hot leg.

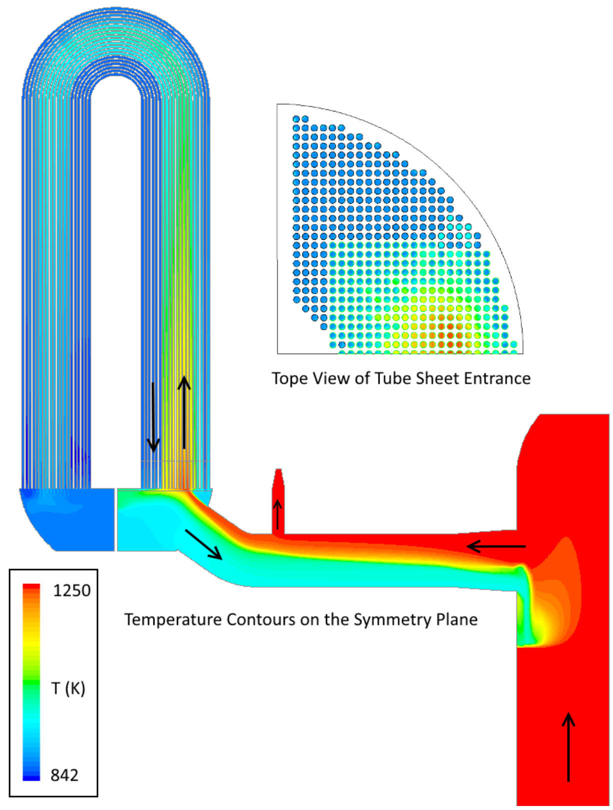

Figure 8. Temperature Contours on Symmetry Plane

The process of determining average mixing and flow parameters from the CFD predictions for use with system code models is outlined in NUREG-1922 [7]. The same procedures are used here. An important prediction is the hot leg flow rate which is related to the density differences at each end of the hot leg by a Froude based correlation that includes a discharge coefficient (Cd) to account for geometric details. This method is adapted from a paper by Leach and Thompson [8]. The discharge coefficient is found to be fairly constant over a range of conditions which makes this a good method for establishing the hot leg flows in this type of scenario. Tube bundle flows are defined based on a recirculation ratio (r) which is the ratio of tube bundle to hot leg mass flow. A mixing model is used to define a mixing fraction, f, which was originally developed to account for inlet plenum mixing [9]. This model has been adapted to include hot leg mixing as well as surge line flows [7] and this updated approach is utilized here. The CFD predictions also provide the fraction of tubes that carry hot flow which is important for setting up flow areas in system code models. The most significant result with respect to tube integrity is the distribution of temperatures in the hot tube region. Temperatures are normalized using Equation 2.

$$T_n = (T - T_{ct}) / (T_h - T_{ct}) \qquad (2)$$

The normalized temperature ranges from from 0 to 1 with 1 representing the hottest temperature in the loop, T_h, and 0 representing the coldest temperature, T_{ct}. T_h is defined as the mass-

averaged temperature of the flow entering the hot leg form the vessel. T_{ct} is determined as the temperature of the cooled tube bundle flow returning to the SG inlet plenum. The normalized tube inlet temperatures are found by considering the flow entering each tube at the tube sheet entrance. A summary of the mixing parameters as well as the hottest normalized tube inlet temperature is provided in Table III along with comparisons to previous Westinghouse SG predictions [7].

Table III. Mixing and Flow Parameter

Parameter	Westinghouse SG	CE SG
C_d – hot leg discharge coefficient	0.12	0.13-0.14
r– recirculation ratio	2.4	1.05
f – mixing fraction	0.96	0.65
hot tube fraction	41%	22%
T_n – (hottest tubes)	0.43	0.95 – 1.0

The hottest tube predictions represent the most significant difference between the CE and Westinghouse results. The high temperature predictions for the CE model indicate that a small portion of the gas entering the hot leg from the vessel flows up to the tube sheet entrance with very little mixing. In order to quantify the range of temperatures for the flow entering the tube bundle, a histogram of tube entrance temperatures is created in Fig. 9. The histogram contains the 5 final data sets (5 instances in time). For each data set, the mass-averaged temperature and flow direction at each tube entrance is collected. The downward cold flow tubes are eliminated and the remaining hot flow tubes are sorted by temperature. The results are binned in groups representing 5% of the normalized temperature range (0-1). This process is repeated for each of the 5 data sets. The lowest normalized temperatures fall in the range from 0.15 to 0.20. Results indicate that 1.25 to 2% of the tubes fall into the highest normalized temperature range of 0.95 to 1.0. Tubes in this highest range are exposed to the greatest thermal challenge.

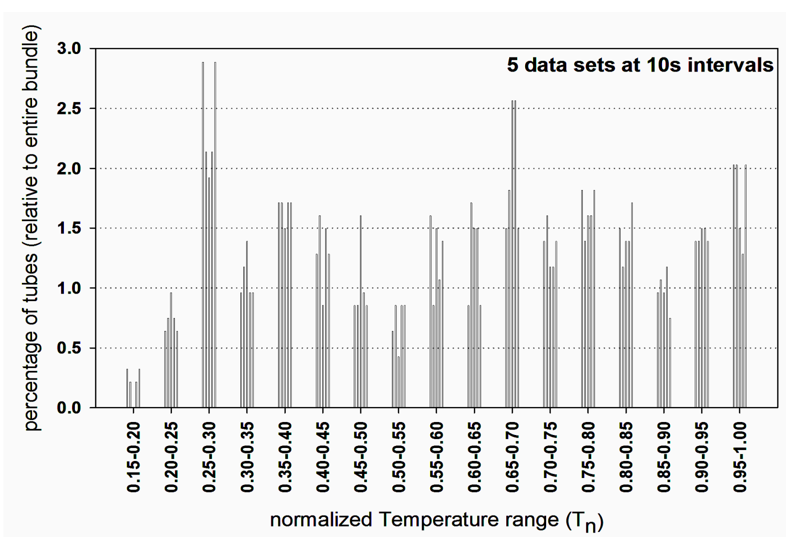

Figure 9. Histogram of Tube Entrance Temperatures

6. ASSUMPTIONS AND LIMITATIONS

The methods used in this study are built upon previous experience [7] with similar phenomena in a Westinghouse SG. Knowledge from these previous studies provide a level of confidence that the modeling options and mesh size are appropriate for this analysis. There are, however, assumptions and limitations that should be considered for future modeling efforts or as background information to help qualify the current predictions. For instance, the vessel region is highly simplified. Details of the geometry and thermal conditions within the vessel are not included and any flow predictions within the vessel are not considered to be prototypical. The impact of any vessel flow patterns on the overall flow and mixing in the SG loop has not been considered. Modeling the details of the reactor vessel internals to obtain more accurate vessel flow and heat transfer predictions will require a significantly larger computational effort.

Another assumption in these studies is the quasi-steady approach that assumes steady boundary conditions can be used to predict a snapshot of the full transient behavior. This assumption may impact the solution during the period of rapid core heat up when system temperatures climb rapidly. A full transient simulation with time dependent boundary conditions has not been attempted and would require significantly more computational resources.

The assumption of adiabatic walls in the hot leg and inlet plenum is made to simplify the boundary conditions used for the quasi-steady state solutions. The impact of connective wall heat transfer in the hot leg on the natural circulation flows is expected to be small based on the relatively short hot leg length and previous sensitivity studies that considered hot leg heat transfer. This affect has not been considered in the current CE model. Another heat transfer limitation is the lack of a radiation model. The application of the discrete ordinates radiation model in the FLUENT code was attempted but converged solutions were not obtained and the approach was dropped. The high pressure severe accident gases in the system are at conditions where thermal radiation is significant and these gases are expected to participate in the radiation balance with the wall. Heat transfer to the hot leg wall would reduce the temperature of the gas reaching the SG. The adiabatic hot leg wall assumption is considered a conservative assumption.

The tube bundle is simplified in order to facilitate practical solutions with the available computer resources. Limited benchmark studies at 1/7th scale [4] indicate that a simplified bundle can be used to predict bundle mass flows and temperature. This current study, which focuses on a larger steam generator with hot leg flows approaching the tube sheet at an oblique angle, showed some sensitivity to the use of larger diameter tubes. The addition of flow straighteners at the entrance effectively reduced the hydraulic diameter at the tube entrance region and reduced (or eliminated) the effect of the large diameter tubes. The tube bundle model also utilizes simplified thermal boundary conditions. The entire tube bundle above the solid tube sheet is modeled with a single convective heat transfer condition and the tube sheet regions have a single fixed temperature boundary condition for each side (inlet and outlet). Three-dimensional thermal boundary condition variations in these regions are expected but conditions are not available from the one-dimensional severe accident system codes used to define the boundary conditions.

7. CONCLUSIONS

CFD predictions are completed for a CE PWR under severe accident conditions to support the NRC's efforts to study severe accident induced SG tube failures. The results build upon a series of predictions that began with a benchmark study at 1/7th scale and a series of sensitivity studies and model developments focused on a Westinghouse PWR. The current model uses the approach refined in these previous studies to predict the countercurrent natural circulation flows in a CE PWR loop. The predictions are processed to determine mixing and flow parameters that can be used to adjust severe accident system code models to ensure these one-dimensional codes are consistent with the three-dimensional predictions. Parameters for a CE plant are determined and a comparison is made with the prior Westinghouse results. The results indicate that a small portion of the flow from the vessel enters the SG tube bundle with very little mixing. This reduced mixing, compared to the Westinghouse results, is attributed to the CE design which includes a large hot leg positioned relatively close to the tube bundle entrance. A distribution of tube inlet temperatures is provided to quantify the variation of temperature for the flow entering the tube bundle. These results support the NRC's current efforts to predict the potential for severe accident induced tube failures in a CE type plant using the MELCOR code.

REFERENCES

1. Nuclear Regulatory Commission (NRC), "Risk Assessment of Severe Accident-Induced Steam Generator Tube Rupture," US Nuclear Regulatory Commission, NUREG-1570, March 1998.
2. Fletcher, D. et. al., "SCDAP/RELAP5 Thermal-Hydraulic Evaluations of the Potential for Containment Bypass During Extended Station Blackout Severe Accident Sequences in a Westinghouse Four-Loop PWR, " NUREG/CR-6995, March 2010.
3. Westinghouse Electric Corporation, "Natural Circulation Experiments for PWR high-Pressure Accidents," Research Project 2177-05, Final Report, August 1993.
4. Boyd, C., Hardesty, K., "CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1781, October 2003.
5. Boyd, C., et. al., "CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1788, May 2004.
6. Dehbi, A., Badreddine, H., "Scaling of a PWR Steam Generator Simulator with the Help of CFD," NURETH-14, Toronto, Canada, September 2011.
7. Boyd, C., Armstrong, K., "Computational Fluid Dynamics Analysis of Natural Circulation Flows in a Pressurized-Water Reactor Loop under Severe Accident Conditions, " NUREG-1922, March 2010.
8. Leach, S. and Thompson, H., "An Investigation of Some Aspects of Flow Into Gas Cooled Nuclear Reactors Following an Accidental Depressurization," J. Br. Nuclear Energy Society, July 14, 1975, No. 3, 243-250.
9. Ebeling, D., et al., "Steam Generator Inlet Plenum Mixing Model for Severe Accident Natural Circulation Conditions," 1990 ASME Winter Meeting, Dallas, Texas.