CHALLENGES FOR THE EXTENSION OF LIMITED EXPERIMENTAL DATA TO FULL-SCALE SEVERE ACCIDENT CONDITIONS USING CFD

C. Boyd, K. Armstrong

Office of Nuclear Regulatory Research - U.S. Nuclear Regulatory Commission

ABSTRACT

The U.S. Nuclear Regulatory Commission (NRC) has been studying thermal-hydraulic phenomena related to low-probability induced-failure scenarios in pressurized-water reactors (PWR) using the ANSYS/FLUENT computational fluid dynamics (CFD) code. A set of updated predictions is completed for use as a numerical experiment to determine mixing and entrainment in the hot leg, surge line, and steam generator inlet plenum during specific severe accident natural circulation flow conditions. These results are used to determine the distribution of temperatures for the flows entering the steam generator tube bundle. This determination is important for an accurate prediction of the potential for induced steam generator tube failures. The results are also used to adjust severe accident system code models to account for the three-dimensional nature of the natural circulation flows during certain periods of the severe accident scenarios. A challenge presented by the work is the extension of a limited set of available test data at 1/7th scale to the full-scale conditions. A benchmark exercise carried out at 1/7th scale does not include all of the flow phenomena and variations in geometry that are part of the full-scale behavior. The full-scale CFD predictions rely on best practice guidelines for CFD application along with engineering judgment to develop results that add to our understanding of these complex flow phenomena.

1. INTRODUCTION

The U.S. Nuclear Regulatory Commission (NRC) has recently completed activities related to a steam generator (SG) action plan that focused on reducing uncertainty and improving the regulatory basis for issues related to steam generator tube integrity. This action plan included evaluating the risk of temperature-induced tube ruptures during low probability severe accident scenarios. In order to determine if the tubes remain intact during these scenarios, a structural analysis must be completed on various parts of the reactor coolant system (RCS). This evaluation requires a prediction of the thermal and pressure loads on the various structures. System codes such as MELCOR and SCDAP/RELAP5 are used to predict the plant behavior under a variety of severe accident conditions. In specific scenarios, the system pressure remains high and the source of water to the SGs fails, resulting in a dry condition on the secondary side of the SGs. In addition, if significant leakage occurs on the secondary side, the mechanical (pressure) load on the tubes is increased. This low probability combination of events leads to a plant state referred to as high-dry-low. The high-dry-low condition refers to the high primary system pressure, the dry steam generators, and the low pressure on the secondary side. Background information on some of the issues related to the prediction of the thermal-hydraulic conditions related to severe accident-induced tube ruptures can be found in NUREG/CR-6995 (Fletcher, 2010).

Under high-dry-low conditions, the primary system temperature increases and the inventory boils down ultimately leading to substantial core uncovery and the generation of superheated steam and hydrogen in the primary system loops. In scenarios where the loop seals are filled with water and full loop circulations are blocked, a three-dimensional counter-current natural circulation flow pattern is established that transfers heat out into the RCS structures. Figure 1 shows a simplified sketch of this natural circulation flow pattern for a PWR with U-tube steam generators. This flow pattern has been experimentally observed at 1/7th scale (Westinghouse, 1993) and is the subject of interest to NRC for this particular study.

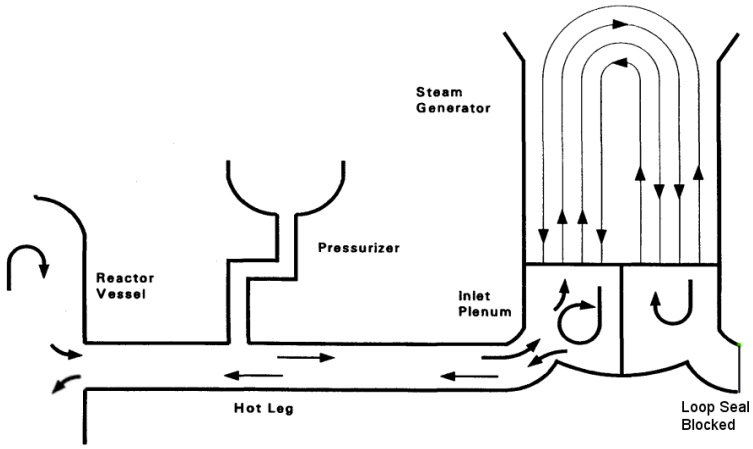


Fig. 1: Severe Accident Natural Circulation Flow Pattern of Interest

A tube rupture is likely to lead to a containment bypass, and because core damage is typically predicted during the scenarios of interest, the superheated steam in the system likely contains a significant amount of fission products. A containment bypass can be avoided if an RCS component fails inside of the containment prior to the time that an SG tube is predicted to fail. Significant failure locations include components such as the hot-leg nozzle region, the pressurizer surge line connection to the hot leg, and the reactor coolant pump seals. These types of RCS failures can depressurize the primary system and remove the load on the SG tubes.

MELCOR and SCDAP/RELAP5 are able to predict the behavior of the entire RCS over extended periods of time, but these types of models do not explicitly account for the details of the three-dimensional natural circulation flow phenomena illustrated in Figure. 1. Experimental methods or validated multidimensional codes are needed to support the system code modeling during the countercurrent natural circulation phase of the accident. The lack of data at reactor conditions is noted in the induced break section of the Nuclear Energy Agency (NEA) assessment guide for CFD (Smith et al., 2008). This assessment guide notes the need for CFD for induced failure analysis and provides recommendations that are considered in the preparation of this work.

2. INITIAL STUDIES

Prior to attempting the prediction of severe accident flows in full reactor geometry, a benchmark exercise was completed using a set of test data from a Westinghouse 1/7th scale facility (Westinghouse, 1993). The facility included one half of a simplified vessel, two hot legs, and the primary side of two steam generators (216 tubes each) such that the flow pattern in Figure 1 could be established. These data provide a valuable confirmation of the natural circulation flows and mixing under this specific severe accident scenario. The data, however, are limited to a single nonprototypical inlet plenum design, and there are concerns related to the scaling of the tube bundle heat transfer rates. The benchmark exercise completed by the NRC staff is documented in NUREG-1781 (Boyd, Hardesty, 2003). This study demonstrates that CFD predictions can adequately predict the inlet plenum mixing and tube bundle flows observed at 1/7th scale. Some background on this data and NRC's initial studies is provided below.

The relevant instrumentation in the 1/7th scale facility consists of a limited number of thermocouples placed around the loop. This includes 14 thermocouples on the vertical center-plane of the hot leg, several thermocouples in the inlet plenum, and a thermocouple located in the entrance region of about 20

percent of the tubes on the inlet side. The experimental results are used, along with a simplified mixing model, to determine an inlet plenum mixing fraction and a recirculation ratio.

Figure 2 illustrates the inlet plenum mixing model as outlined in several sources including Ebeling-Koning (1990).

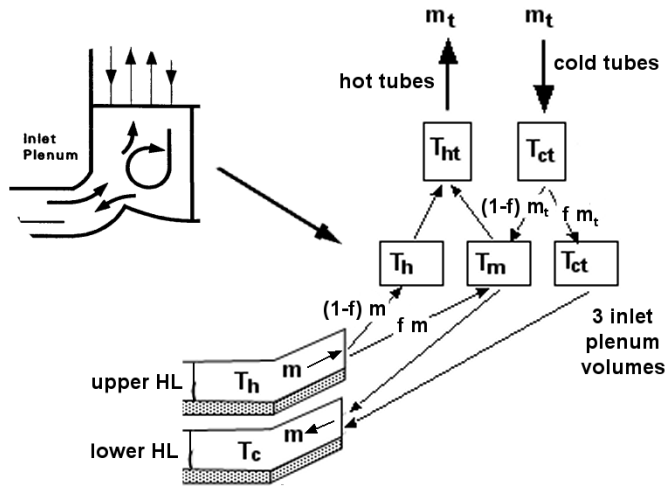


Fig. 2: SG Inlet Plenum Mixing Model Control Volumes

The mixing model formulation is defined using the flow paths and control volumes from Figure 2 and applying conservation of mass and the first law of thermodynamics in a steady-state-steady-flow formulation to the mixing volume. The following equations are derived from this setup.

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

$f = 1 - r\,(T_{ht} - T_m) / (T_h - T_m)$ (f = mixing fraction) (2)

$r = m_t / m$ (r = recirculation ratio) (3)

The experimental temperature measurements are used to estimate values for T_h, T_{ht}, T_{ct}, and r. From these values, the mixing fraction (f) is determined using the mixing model formulation. It is noted that the upper hot leg and hot tube mass flows (m and m_t) are obtained from an overall energy balance and not measured directly. The recirculation ratio and mixing fraction are key parameters used to adjust system code models used for the prediction of plant behavior during this type of scenario.

It is noted that a full uncertainty analysis for the data is not included with the experimental results, and this presents a challenge in the benchmark. Several factors add to the experimental uncertainty. These include the use of energy balances to determine mass flows from thermocouple readings and the limited number of measurement locations. For example, four thermocouples placed in a vertical line are available at the SG end of the hot leg. The lower two thermocouples are used to represent the cold return flow temperature. The third thermocouple from the bottom is in the mixing layer, and the experimentalists decided that this measurement could not be used due to its fluctuating values. The upper thermocouple (a single measured temperature in a region of large vertical temperature gradient) is assumed to represent the mass averaged temperature of the forward flow in the upper hot-leg region.

In NRC's CFD benchmark study (Boyd, Hardesty, 2003), a model of the hot leg, SG inlet plenum, and a simplified tube bundle was developed using the FLUENT 6.0 CFD code. Qualitatively, the predictions demonstrate all of the experimentally observed flow features in the hot leg and steam generator regions.

Predicted features such as the sloping hot-cold interface in the hot leg, the relative sizes of the hot and cold flow regions, and the path and size of the inlet plenum plume are consistent with the experimental observations. The global natural circulation flow pattern is also well established in the CFD predictions. These results added significant insight into the natural circulation flows and mixing and helped to fill in details of the flow phenomena that could not be discerned from the limited temperature measurements. The predictions are used like a numerical experiment to compute the mixing parameters of interest. Table 1 shows the results of the benchmark exercise for the key parameters of interest.

Table 1. Comparison of CFD Predictions to Experiment

Parameter	Westinghouse 1/7th Test Data	CFD Prediction	Difference
# Hot Tubes	75 / 216*	82 / 216	7 / 216 (3.2%)
T_h Hot Leg (average Hot T (^oC))**	159.3	155	-4.3 ^oC
m Hot Leg (mass flux (kg/s))**	.06	.0586	-2.3 %
T_{ht} Tubes (average Hot T (^oC))	100.8	100	-0.8 ^oC
T_{ct} Tubes (average Cold T (^oC))	64.7	64.7	0
m_t Tubes (mass flux (kg/s))	0.12	0.121	0.8%
m_t/m (recirculation ratio)	2.01	2.06	2.5 %
f (mixing fraction[2])	0.85	0.81	-4.7 %

* 216 tubes in bundle
** controlled by boundary condition

The mass flows reported for the CFD predictions come from integrated results across the flow surfaces and are reported at locations consistent with the experiments. The results in Table 1 are very encouraging. Most importantly, for a given hot leg mass flow and temperature measured at the SG end of the hot leg, the flows entering the SG tube bundle have essentially the same mass flow and temperature. This demonstrates a very good prediction of the SG inlet plenum mixing and entrainment behavior that are the key parameters of interest in this study. The results are within this author's estimate of the experimental uncertainty. Further details are provided in NUREG-1781 (Boyd, Hardesty, 2003).

Following the successful benchmark exercise at 1/7th scale, an initial set of full-scale analyses was completed using an identical modeling approach. The results are documented in NUREG-1788 (Boyd, et. al., 2004). This effort considered a few full-scale steam generators under severe accident conditions in an attempt to extend the experimental results to full-scale conditions. The predictions indicated that similar results could be obtained under full-scale conditions if the secondary side heat transfer rates and inlet plenum geometry were similar. Prototypical SG inlet plenum designs can be significantly different from the facility, however, and this work considers the design impact. The work also highlights the need to accurately model the heat transfer from the tube bundle. The benefits of using CFD modeling to extend the limited experimental results are demonstrated by this study. The report of Boyd, et al. (2004) lists a variety of assumptions and limitations for the work. In addition, the NEA/CSNI assessment guide (Smith et al., 2008) provides a list of state-of-the-art recommendations for modeling an induced break scenario. NRC has undertaken further studies of these scenarios using an updated CFD model that considers the

state-of-the-art recommendations and limitations of the prior studies. This updated modeling and its application is the subject of this report.

3. CFD MODELING

The updated full-scale predictions focused on a Westinghouse four-loop reactor with a Model 51 SG. The model addresses several assumptions and limitations of the previous work through a better representation of the geometry and additional modeling options. Where possible, the approach is consistent with the assumptions and limitations of the prior analysis and the initial benchmark at 1/7th scale. This consistency with the prior efforts maintains a linkage between these full-scale predictions and the positive benchmark at 1/7th scale. Some details of the modeling are provided below.

3.1 Computational Domain, Solver Settings, and Mesh

The domain of the updated CFD model builds upon the geometry of the previous efforts with the inclusion of a simplified vessel region and a significantly improved tube bundle. The vessel region represents a ¼ vessel inside of the core barrel and above the bottom of active fuel. The vessel is broken down into regions to represent the vessel in a highly simplified model. The inlet plenum and hot leg are identical to the model used in the initial NRC work for a Westinghouse plant described above. Two optional pressurizer surge lines are added to the hot-leg model—one on the side and one top mounted. These surge lines can be turned on or off in the model to simulate various accident scenarios and SG designs. The tube bundle consists of 371 full-height tubes with a total flow area to match the Westinghouse Model 51 design. Each SG tube in the model represents the flow area of nine (a three x three grouping) prototypical SG tubes. Figure 3 shows an overview of the updated model domain. Figure 4 illustrates the tube bundle layout on a horizontal plane just above the inlet plenum. The hot-leg nozzle is in the upper left-hand corner.

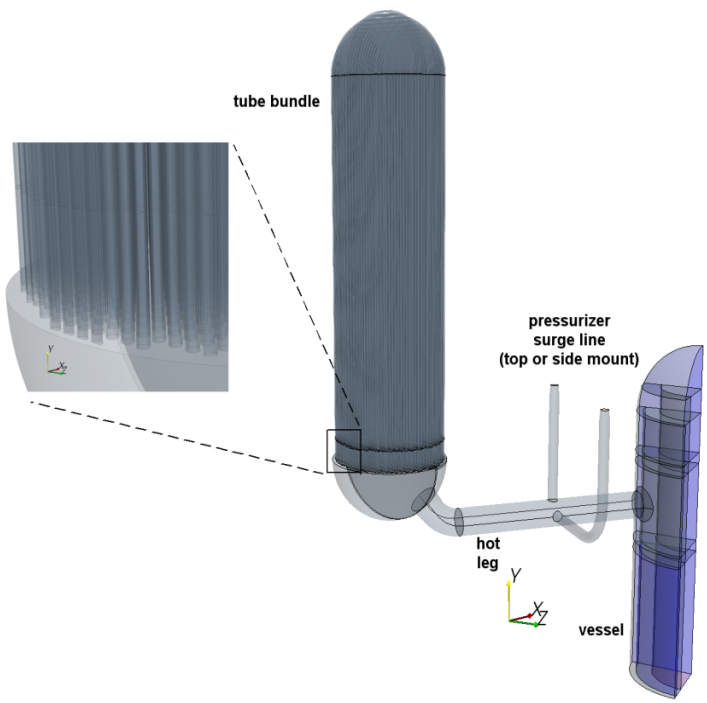


Fig. 3: Overview of CFD Domain Used for Study

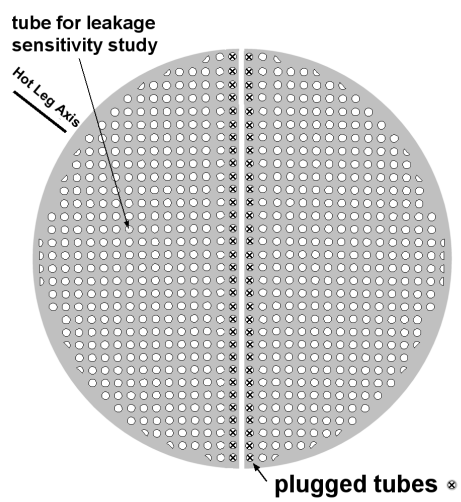


Fig. 4: Tube Bundle Layout

A basic overview of the CFD modeling options is listed below. The only changes from the previous benchmarking exercise are the addition of a ¼ symmetry vessel model and the addition of species tracking for hydrogen.

- Time-dependent RANS, 0.05s time step, steady BCs.
- ¼ symmetry simplified vessel and upper plenum model.
- Reynolds stress turbulence, full buoyancy effects on turbulence.
- Nonequilibrium wall functions.
- Temperature-dependent thermal properties (steam and hydrogen) at constant pressure.
- Hydrogen species tracking.
- Gravity.
- Segregated solver with second-order differencing on momentum and energy.
- Porous media flow loss coefficients in vessel region and simplified SG tubes.

The finite volume mesh consists of 7.8 million computational volumes (cells). The domain is composed mainly of hexagonal elements with the exception of a small transition region between the inlet plenum walls and a hex-core region and a small tetrahedral region in the upper dome of the vessel. About 1 million of the cells are used for the inlet plenum region where a hex-core approach is used with the volumes aligned with the hot-leg axis. This is considered to be an improvement over the approach used in the previous studies. The tube bundle itself contains about 6 million hexagonal cells. Special attention was paid to cell quality as outlined in the "NEA/CSNI Best Practice Guidelines for CFD" (Mahaffy, J., et al., 2007). The non-conformal interfaces used in the previous work are removed and where flow direction is known, the hexagonal mesh is aligned with the flow direction. Grid angles and grid-density changes are also controlled. The choice of the second-order RSM turbulence model carries over from the initial studies where this option was selected due to the non-isotropic nature of the expected turbulence in the hot counter-current hot-leg flows and the rising hot plume. This selection is consistent with the recommendations for RANS modeling in the Best Practice Guidelines.

3.2 Tube Bundle Modeling

Modeling the tube bundle directly using CFD best practice guidelines is beyond the capacity of most computer systems at this time. The approach used is consistent with the state-of-the-art recommendation for this type of modeling in the NEA/CSNI assessment guide for CFD (Smith, et al. 2008) that suggests a simplified but accurate nodalization of the tube bundle that "defines some equivalent" to reduce the size of the mesh. The SG studied is a Westinghouse Model 51 that contains over 3300 tubes. To reduce the mesh requirements, groups of nine tubes (three x three array) are modeled as a single tube. Total flow area and the height and length of the tubes are preserved. The resulting model contains a total of 371 tubes. Figure 4 shows a cross section of the tube bundle as modeled just above the inlet plenum. The inner row of tubes is plugged to be consistent with tube plugging assumptions used in recent NRC system code analysis for this scenario.

The simplified tube bundle does not directly represent the prototypical SG because the tube bundle model has a reduced wetted surface area and therefore will have reduced pressure-drop and heat-transfer rates for a given flow. Tube bundle flows are the principal driving force in the severe accident natural circulation flows and these must be established properly. The SG flow is established by a balance between the buoyancy driving forces and the viscous flow losses in the tube (as well as pressure drops at the tube entrance). To accurately model this balance, the heat loss and pressure-drop characteristics of the CFD model must be augmented to be consistent with the prototypal behavior.

The appropriate tube bundle characteristics are determined by comparing the heat-transfer and pressure-drop characteristics of the simplified tubes to a best-estimate prediction using best practice guidelines and the prototypical tube dimensions. The best-estimate prediction uses boundary conditions that are consistent with the expected tube-bundle flows as determined from system code analyses of the severe accident behavior. Once best-estimate predictions are completed for the range of conditions, the simplified tube is modeled with the same boundary conditions and the heat-transfer and pressure-drop deficits are determined for the simplified model. The porous media formulation is turned on in the simplified tube model to provide a means to account for the deficit in heat transfer and pressure drop.

Figure 5 shows a comparison of the pressure and temperature along an SG tube for the reference (best estimate) predictions and the simplified tubes augmented with the porous media formulation. These plots demonstrate that the larger tubes augmented with the porous media coefficients can be used to predict the heat-transfer and pressure-drop characteristics of the prototypical SG tubes. The accurate prediction of the temperature and pressure drop along the tubes is essential for an accurate prediction of the tube-bundle response during the severe accident scenarios. The benefit of this tube-bundle modeling approach is the significant reduction in the mesh requirements. The best-estimate prediction used about 1 million cells for each full-scale SG tube. This type of mesh density would result in over 3 billion computational volumes for an SG tube bundle. The simplified tube bundle approach resulted in a complete SG tube-bundle model that utilized a more realistic 6 million computational volumes.

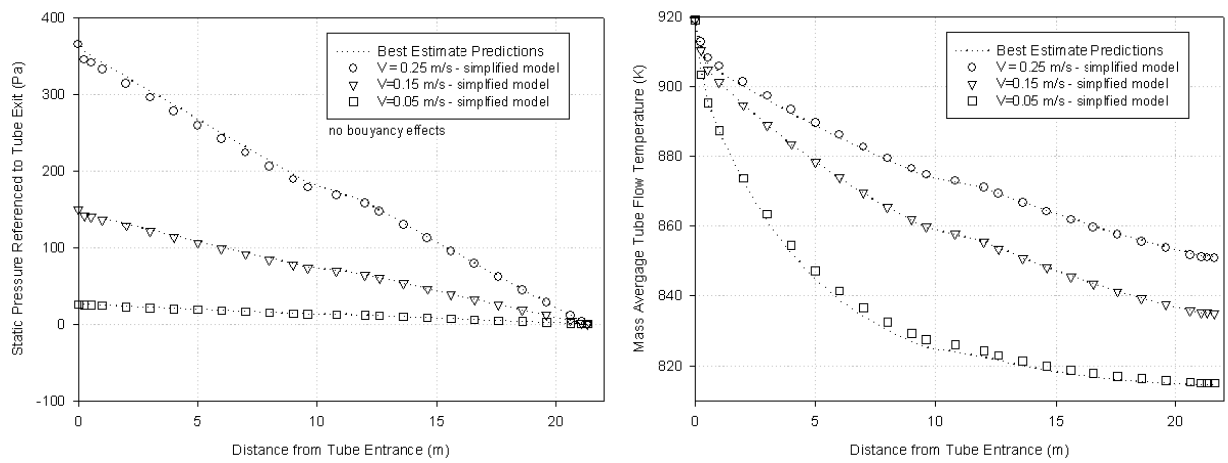


Fig. 5: Temperature and Pressure along Tubes for Best Estimate Predictions and Large Tubes

3.3 Boundary Conditions and Solution Procedure

Boundary conditions for the CFD model are adapted from SCDAP/RELAP5 system code predictions of the severe accident conditions in a Westinghouse plant with a Model 51 SG. Because the CFD predictions are quasi-steady (transient solution with fixed boundary conditions), most walls are assumed to be adiabatic. The boundary conditions transfer heat to the model in the vessel region and remove the heat from the tube bundle. No flows are forced into the hot leg as in NRC's initial CFD studies. The flows in the system are essentially natural circulation flows between the hot gasses in the upper plenum and the heat sink of the SG as shown in Figure 1.

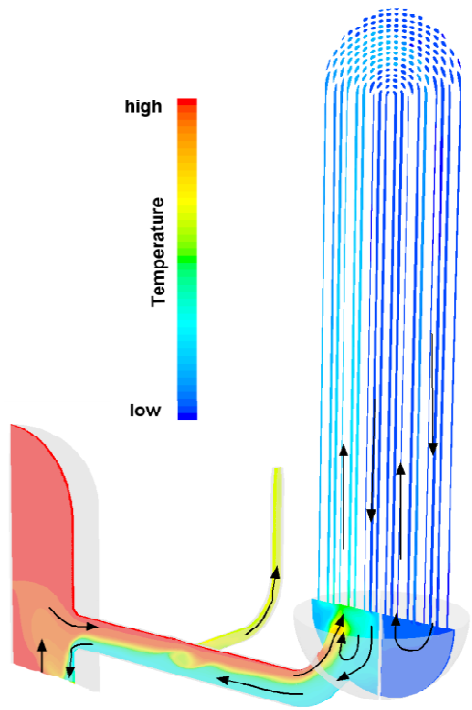
The boundary conditions in the vessel are established to create the fluid conditions in the upper plenum region of the vessel that are consistent with the severe accident conditions of interest. Initial modeling attempts at using a heat source resulted in unsteady flows, and it became very difficult to converge a solution with the upper plenum conditions of interest. A fixed temperature condition is applied in the fuel region and upper dome to efficiently control the conditions in the upper plenum region. The goal of

this approach is to create specific conditions in the reactor vessel upper plenum. The simplified vessel model ~~is basically a means~~provides and efficient means of adjusting the upper plenum conditions that feed into the hot leg. The boundary conditions on the SG tubes consist only of the secondary side temperature conditions and the associated heat transfer coefficient. The temperature is obtained directly from the SCDAP/RELAP5 predictions of the secondary side conditions, and the heat-transfer coefficient is determined to be consistent with those used in the tube bundle development studies outlined in Section 3.2 above.

The predictions of the natural circulation flows proved to be unsteady. Waves formed on the interface between the hot and cold flows in the hot-leg region, and the rising plume in the SG inlet plenum was predicted to oscillate in intensity and location. To obtain the average system behavior, the predictions were carried out for an extended period of time to confirm a long-term steady trend and then to record a series of 140 data sets at 2-second intervals that could be combined to obtain the average system behavior.

4.0 CFD RESULTS

The CFD model is used to predict the severe accident flows of interest using boundary conditions that are consistent with SCDAP/RELAP5 predictions (Fletcher, 2010) of the system behavior during a severe accident. The predictions qualitatively demonstrate the general natural circulation flows that have been experimentally observed. Figure 6 illustrates a temperature contour plot on a vertical plane through the hot-leg axis. Arrows are added to the figure to illustrate the general flow directions. The temperature contours are for one instance in time during the solution that predicted to be naturally unstable.



Vertical Plane

The simulations are used as a numerical experiment to determine mixing parameters and correlations used to augment system code models to account for the three-dimensional natural circulation flows and mixing. A hot-leg discharge coefficient, similar to that outlined by Leach and Thompson (1975), is determined from the predictions for direct application in system code models. This provides a physically based means of adjusting the hot-leg flow rates. The mixing fraction and recirculation ratio are predicted using the type of mixing model described above. The mixing volume is expanded to include mixing in the hot leg and the possibility of a side-mounted surge line as outlined by Boyd and Armstrong (2010). The result is a more realistic and repeatable analysis of the loop mixing prior to the flow entering the tube bundle. For the tube-bundle flows, each tube is sampled for temperature and mass flow to obtain the tube-bundle flow rates and the number and distribution of hot tubes. These predictions have been used in recent NRC system code evaluations of a Westinghouse PWR under severe accident conditions.

A lesson learned from the recent system code evaluations that utilized the CFD results is the importance of the hot tube temperature variations. The higher temperatures within the tube bundle pose the greatest thermal challenge to the tubes. The system code models typically consider only a single hot and cold flow tube, and the predicted temperatures and mass flows are bundle average values. The CFD predictions provide a means to look at the distribution of the temperatures across the tube bundle, and this aspect of the CFD predictions is important in the evaluation of tube-failure timing.

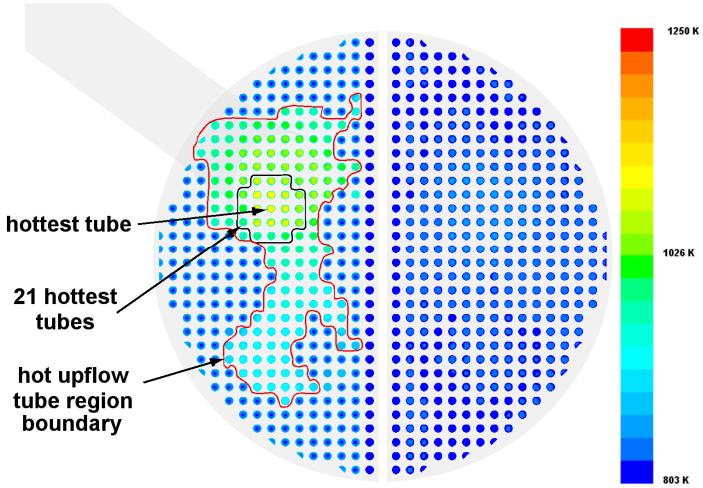


Fig. 7: Temperature Contours at Tube Entrance

Figure 7 shows the temperature contours in the tubes just above the tube-sheet entrance. The forward flowing (hot) tubes are clearly discernable by the higher temperatures. The return flow tubes are all very close to the secondary side temperature. A boundary is drawn between the hot and cold tubes to illustrate the hot-tube region. The extent and shape of this boundary changes along the edges with time during the calculation as some of the tubes change flow directions. The 21 hottest tubes in the region are illustrated by an inner boundary. This core region of the hot flow tubes does not change direction during the scenario, and the greatest thermal challenge exists in this region. Hot-leg orientation is indicated in the upper left corner of the figure.

A method of applying the CFD results to the transient system code simulation is developed by defining a dimensionless temperature scale to represent the tube entrance temperatures. It is found that the distribution of the dimensionless temperatures is fairly consistent across CFD simulations over a range of boundary conditions. The dimensionless (T_n) temperature scale is defined using the following relation.

$$T_n = (T_{ht} - T_{ct}) / (T_h - T_{ct}) \quad (4)$$

The values of T_h and T_{ct} represent the hot-leg inlet temperature and the cold-tube return flow temperature, respectively.

These are the hot and cold temperatures that flow into the mixing region. The dimensionless temperature, T_n, is bound between zero and one. A value of one indicates no mixing between the hot leg and the tube-sheet entrance. A value of zero represents flow into the tube bundle at the temperature of the flow returning to the inlet plenum from the cold tubes. The mass averaged temperature at each tube is obtained in this case from 140 snapshots in time at 2-second intervals, and an average value is computed for each tube. The results are grouped into 20 intervals over the dimensionless temperature range (0-1) and a histogram is created. Figure 8 shows the results are plotted as the fraction of tubes within each range. Individual tubes in the hottest region see temperatures on this scale as high as 0.6 to 0.65 for short periods of time. The averaging process clips off the upper tails of the distribution. The results from the CFD predictions are used to determine the range of tube entrance temperatures at any time in the scenario from the system code predictions of T_h and T_{ct}.

These results indicate that the hottest tube temperatures at the tube entrance have a long time average value that falls within the 0.4 to 0.45 range on the dimensionless scale. The single hottest tube averaged over the 140 data sets is found to have a dimensionless temperature of 0.43 +/- 0.1 (0.1 is the

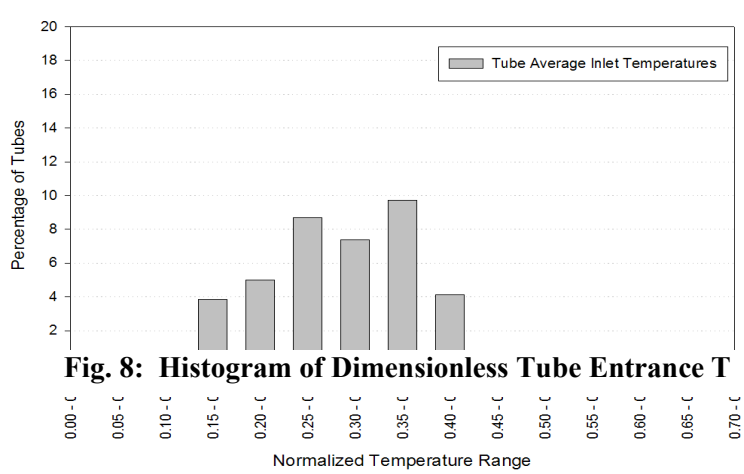


Fig. 8: Histogram of Dimensionless Tube Entrance T

standard deviation). To further judge the temporal variations, the hottest tube history is broken down into 7 periods containing 20 data sets each (i.e., seven 40- second intervals). Forty seconds is an estimate of the thermal time constant for the tubes. The average normalized temperature predictions for the hottest tube on each of these intervals are 0.51, 0.38, 0.44, 0.45, 0.44, 0.39, and 0.43. This indicates that the temperature fluctuations can be significant. For the purposes of NRC's screening calculations on induced failures, a peak average normalized temperature of 0.5 has been used.

Sensitivity studies are completed to consider some of the modeling and data reduction methods. One sensitivity relates to the choice of the relatively long sampling period (2 seconds) and limited number of data sets (140). The model is run again for a total of 240 seconds, and data are recorded at 0.1 second intervals on the 21 hottest tubes for a total of 2,400 data points. The resulting average normalized hottest tube temperature is again predicted to be 0.43. These new data are broken down into 40-second intervals, and the maximum normalized tube temperature for any of these intervals is found to be 0.5. It is concluded that the sampling frequency and number of data points used in the study are adequate for determining the average tube entrance temperatures.

In addition to the boundary condition sensitivities completed as part of the larger project and the averaging sensitivity noted above, a grid sensitivity study is completed. The previous work by Boyd and Hardesty (2003) demonstrated that the mesh was adequate for predicting the net inlet plenum mixing and entrainment. The flow-aligned cubic core mesh used for this study is an improvement over the prior mesh. As a sensitivity study, the 1 million cells in the inlet plenum are adapted (each cell is split into 8 cells) to create a total of 8 million inlet plenum cells. Figure 9 shows a cross-sectional cut of the inlet plenum for the base model and the refined mesh. The predictions from the refined grid model showed no significant change in the inlet plenum mixing or tube bundle flows. The hottest tube average normalized temperature rounded to two significant figures is 0.43 for the refined mesh. The extent of the temperature variations is also found to have no significant variations after the mesh refinement. It is concluded that the base-case mesh produces grid independent results for the key mixing parameters and the tube-temperature distributions.

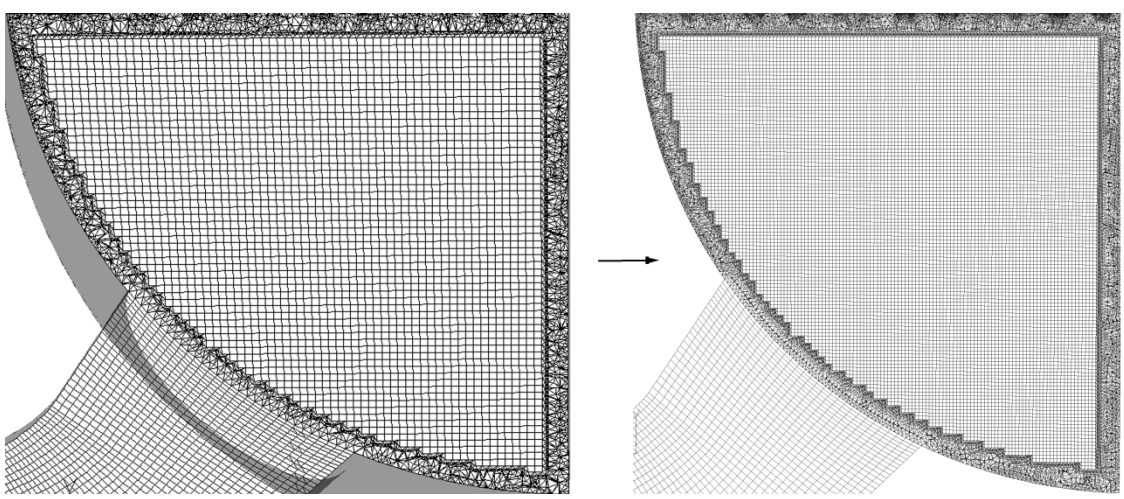


Fig. 9: Inlet Plenum Mesh before (left) and after (right) Grid Refinement

The CFD predictions provide valuable insights into the mixing and entrainment during the complex severe accident flows of interest to induced failure studies. The results of these predictions have been incorporated into the latest NRC system analysis evaluations of severe accident-induced failures. The

CFD predictions are used as a means to extend the limited available experimental results to a variety of full-scale conditions of interest to NRC.

5. LIMITATIONS AND CHALLENGES

The goal of the CFD evaluations is the development and refinement of parameters and mixing models used to support system code evaluations of severe accident natural circulation flows. A challenge for the analyst is to build confidence in the solutions. This is difficult due to the lack of experiments at full reactor conditions. The limited experimental results available do not cover the full range of conditions or geometries of interest to NRC. A limited benchmarking exercise undertaken at 1/7th scale indicates that CFD can be used to predict the average flows and mixing in the SG region but the follow-on studies indicate that the expected flow patterns can be significantly different than the experimental conditions. State-of-the-art recommendations in the NEA/CSNI CFD Assessment Guide (Smith, 2008) recognize the lack of data and layout some guidelines to consider. This limited study partially addresses these recommendations but further refinements are still possible. A few of the key challenges remaining are outlined below.

5.1 Validation

Experimental results under full reactor conditions are difficult to imagine given the extreme conditions associated with severe accident events. The limited available data are not sufficient for a proper CFD validation due to the limited nature of the data and the limited scale of the facility. Some specific flow attributes that are not accounted for include the unsteadiness of the hot-leg flows, the instability of the hot-cold interface, the unsteadiness of the hot plume in the inlet plenum, the mixed convection heat transfer in the hot leg, and the radiation exchange between the gas and the solid structures. Validation efforts on some of these individual phenomena ~~are needed to further refine the modeling~~using appropriate test data is an area recommended for future investigation. The nature of the severe accident conditions and the scale of the reactor loop make these types of data difficult to ~~find~~obtain. For instance, mixed convection heat transfer in a 1-meter diameter pipe with temperature differences approaching several hundred degrees is not readily available. Similarly, data for a rising buoyant plume in a constrained geometry impacting a perforated tube sheet are not common.

5.2 Boundary Conditions

The boundary conditions used for the CFD evaluation are adapted from system code predictions of the severe accident behavior. No suitable test data are available at these conditions, and the boundary conditions utilized for this CFD study are adapted from severe accident system code evaluations. The system code evaluations only provide simplified conditions and lack the detail necessary for a proper CFD evaluation. One sensitive boundary condition is the secondary side heat transfer rates. Spatial variations on the secondary side caused by natural circulation flows in the boiler region are not considered in this evaluation. Another limitation is the quasi-steady approach and the use of adiabatic boundary conditions for the hot leg and inlet plenum walls. Mixed convection in the hot leg during a transient scenario will lead to secondary flows in the hot leg that are not considered in this evaluation.

5.3 Radiation Modeling

The superheated steam in the system during these high-pressure events is optically thick and in the hot leg can be hundreds of degrees hotter than the hot-leg wall. Radiation exchange between the gas and the wall will remove heat from the system before the gas reaches the SG tubes. This effect is not modeled in the current evaluations. NRC addresses radiation exchange in the severe accident system code models

(Fletcher, 2010), but the impact of this exchange on the overall natural circulation flows is not considered in the present CFD evaluation.

6. SUMMARY

A set of CFD predictions is completed to study the flows that are relevant in the evaluation of low-probability severe-accident-induced failure scenarios. The predictions are used to adjust the flows and mixing in recent SCDAP/RELAP5 severe accident analyses for a Westinghouse four-loop plant (Fletcher, et al., 2010). CFD is important in this evaluation because of the three-dimensional nature of the flows and thermal loadings associated with the severe accident scenario and because of the lack of full-scale experimental results.

The CFD predictions are used like a numerical experiment to develop an updated mixing model, a hot-leg flow model, a surge-line-flow split model, and a distribution of the tube temperatures. The models build upon prior successful benchmarking at $1/7^{th}$ scale with improved modeling and nodalization. Challenges still remain in this area because of the lack of available data for CFD validation. The work proceeds with engineering judgment and CFD best-practice guidelines to produce results that further our understanding of these complex phenomena. Future experiments or CFD evaluations of induced failure conditions should consider the areas of concern outlined above to further enhance our understanding in this area.

7. REFERENCES

Boyd C., Armstrong K., "Modelling Improvements for System Code Evaluation of Inlet Plenum Mixing under Severe Accident Conditions using CFD Predictions," ICONE18-30262, May 2010, Xian, China.

Boyd C., Hardesty K., "CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1781, U.S. Nuclear Regulatory Commission, October 2003.

Boyd, C., Helton D., Hardesty K., "CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident," NUREG-1788, U.S. Nuclear Regulatory Commission, May 2004.

Ebeling-Konig D., "Steam Generator Inlet Plenum Mixing Model for Severe Accident Natural Circulation Conditions," 1990 ASME Winter Meeting, Dallas, TX.

Fletcher, C.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.

Leach, S., 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.

Mahaffy, J., et al., "Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications," NEA/CSNI/R(2007)5, April 2007.

Smith B, et al., "Assessment of CFD for Nuclear Reactor Safety Problems," NEA/CSNI/R(2007)13, January 2008.

Westinghouse Electric Corporation, "Natural Circulation Experiments for PWR High Pressure Accidents," Research Project 2177-05, Final Report, August 1993.