

SCALING OF A PWR STEAM GENERATOR SIMULATOR WITH THE HELP OF CFD

A. Dehbi¹, H. Badreddine¹

¹ Paul Scherrer Institut, Villigen, Switzerland
abdel.dehbi@psi.ch, hassan.badreddine@psi.ch

Abstract

A project is planned at PSI to experimentally simulate natural circulation mixing in a PWR steam generator during a severe accident whereby superheated steam is generated and secondary water cooling is lost. To assist in the scaling of the mock-up, CFD with the Reynolds Stress Model is used. The methodology is first validated against the Westinghouse 1/7th scale experiments. Thereafter, the flow field is simulated in the PSI mock-up which consists of 270 U-tubes with 0.005 m diameter and 4.9 m length. We show that the scaling assumptions (recirculation ratio, fraction of tubes seeing the hot plume, etc) are globally confirmed by the CFD simulations.

1. Introduction

In a PWR, an accident scenario of particular safety relevance is that of a station blackout (SBO) leading to a potential temperature-induced steam generator (SG) tube rupture. In such scenario, the hot leg becomes voided via venting of coolant through the pressurizer, and the cold leg loop seal is plugged with water. In addition, leakage in the secondary leads to loss of secondary side coolant and depressurization. Hence, the prevailing conditions are those of high primary pressure and dry, low pressure secondary (so-called “high, dry, low”). As the core uncovers, highly superheated steam is produced, and a counter-current natural convection flow pattern is established in the hot leg, whereby the hot steam flows above the cold return stream from the steam generator plenum. Failure of the SG tube bundle to remove the heat load can lead to tube failure.

The highly 3D flow field in the hot leg and steam generator plena determines the temperature distribution seen by the tubes. For high enough circulation and sufficient secondary side cooling, the maximum temperature seen by the tubes could be low enough not to threaten the tube integrity, and in this case the hot leg or pressurizer surge line will fail first, leading to depressurization into the containment. On the other hand, poor bundle recirculation and poor heat transfer to the secondary will lead to high temperature plumes into the bundle, with a high potential for tube breach followed by containment bypass.

Since system codes are one dimensional by design, they are not able to determine the 3D flow pattern in the SG plena and tubes, and hence rely on empirical hard-wired parameters to assign required inputs such as the mixing factor and fraction of the tubes seeing the hot/cold plumes. These factors come from a series of test conducted at Westinghouse on a mock-up SG which is 1/7th of the full scale SG [1]. To have better confidence in the mixing parameters, there is a need for CFD simulations which are validated with detailed experimental data. The main shortcomings of the Westinghouse 1/7th tests are: a) the rather scarce and integral type instrumentation (mainly thermocouples) which makes it hard for 3D codes to get properly validated against the data, and

b) the secondary side water cooling which leads to very efficient heat transfer by the SG bundle whereas in the real plant, heat transfer is severely hampered due a dry secondary. As a consequence, the total heat transfer in the Westinghouse 1/7th scale is almost a-priori known because the return flow is very close to the coolant water temperature. This means CFD simulations will invariably predict the global heat transfer quite well, and deficiencies in the computations cannot be readily detected.

To remedy these shortcomings, the Paul Scherrer Institut (PSI) is planning an experimental program that builds on the Westinghouse 1/7th scale experience. Better scaling approach is undertaken to ensure poor secondary side heat transfer, while high resolution 3D, CFD-grade data is planned to validate the CFD codes.

The present investigation consists of three steps: in a first step, we use ANSYS CFD-Fluent code to simulate flow and heat transfer in the Westinghouse 1/7th scale SG experiment [1]. This exercise serves as a basis for the validation of the CFD methodology. We compare results with experimental data on mixing phenomena. In the second step, we present a detailed 1-D scaling analysis of the steam generator simulator proposed by PSI. We focus on improving the Westinghouse 1/7th scale SG by matching more closely the dimensionless numbers that are important, in particular the Richardson numbers which measure the ability to transform buoyant energy into kinetic energy. In a third step, we present a CFD simulation of mixing in the new SG mock-up, and show that the design assumptions (mixing factor, hot tube fraction, etc) are globally confirmed by the CFD simulations.

2. Station Blackout Conditions using MELCOR

The system code MELCOR 1.8.5 was used to determine the conditions in a reference two-loop PWR following a SBO. It is assumed that the loop seal is plugged, and the secondary is dry and depressurized to atmospheric pressure. The SG consists of 3200 U tubes with inner diameter 0.0167 m, and mean total length of 18 m. The plena are hemispheres with a radius of 1.25 m. Since MELCOR is a 1-D code, assumptions were made to determine the extent of mixing in the SG. In particular, it was assumed that half the tubes see the hot flow, while the other half see the cold return flow. Additionally, it is assumed that the flow rate in the tube bundle is twice that prevailing in the hot leg (i.e. recirculation ratio of 2). Both of these assumptions are derived from the outcomes of tests conducted in the Westinghouse 1/7th SG facility [1]. The calculated conditions at the beginning and end of the transient that we simulate are shown in Table 1.

In the following sections, the average conditions from Table 1 are taken in order to compute the dimensionless numbers.

Time (s)	Hot leg flow rate, kg/s	T _{hot Leg} K	T _{tube in} K	T _{tube out} K
13000	4.47	833	703	623
18000	2.81	1294	987	877
Average	3.60	1063	845	750

Table 1: Range of conditions for the SBO scenario

3. Scaling of the hot leg

The counter-current natural circulation pattern which develops in the hot leg results from density differences between the inlet hot flow and the return cold flow. Based on the Westinghouse 1/7th scale tests, we assume that the hot and cold stream occupy respectively half the cross sectional area of the hot leg. In the absence of significant condensation, the resulting mass flow $\dot{m}_{HL}$ is obtained from the Leach and Thompson correlation [2] which reads:

$$\dot{m}_{HL} = 0.1153 \cdot \bar{\rho} (g \beta D_{HL}^5 \Delta T)^{1/2} \quad (1)$$

where ΔT is the temperature difference between the hot and cold stream reservoirs, $\bar{\rho}$ the mean density, β the thermal expansion coefficient, and D_{HL} the hot leg diameter. If one assumes that convection between the streams is the dominant heat transfer mechanism, it can be shown [3] that the mean temperatures in the hot leg satisfy the following equation:

$$\frac{\overline{T_{HL,hot}} - \overline{T_{HL,cold}}}{\overline{T_{HL,hot,in}} - \overline{T_{HL,cold,in}}} = \frac{4 \cdot (2 \cosh \alpha + 3 \sinh \alpha - 2)}{\alpha \cdot (12 \cosh \alpha + 13 \sinh \alpha)} \quad (2)$$

where:

$$\alpha = \frac{3}{4} \frac{h_{HL} L_{HL} \pi D_{HL}}{\dot{m}_{HL} c_p} \quad (3)$$

In the above, the overbar represents averages over the length of the hot leg, L_{HL} the hot leg length, c_p the fluid heat capacity, and h_{HL} the heat transfer coefficient between the two streams.

The hot leg equivalent diameter is defined in terms of the hot area and its corresponding wetted perimeter:

$$D_{eq} = \frac{4 \cdot A_{hot}}{P_w} = \frac{\pi D_{HL}}{(2 + \pi)} \quad (4)$$

Hence the Reynolds number in the hot leg based on the equivalent diameter is:

$$Re_{HL} = \frac{\rho_h U_{HL} D_{eq}}{\mu_h} \quad (5)$$

where ρ_h and μ_h are the hot fluid density and viscosity, respectively, and U_{HL} the mean velocity in the hot stream. The heat transfer coefficient between the streams can be estimated from e.g. the Dittus-Boelter correlation:

$$Nu_{HL} = \frac{h_{HL} D_{eq}}{k_{HL}} = 0.023 \cdot Re_{HL}^{0.8} \cdot Pr_{HL}^{0.3} \quad (6)$$

where Pr_{HL} is the fluid Prandtl number. To produce the high density necessary to approach the dimensionless numbers of the plant, the heavy gas SF₆ (molecular weight 146) is used at an absolute pressure of 20 bars. This yields densities on the order of 100 kg/m³.

The hot leg dimensions and test conditions are taken to be similar to the Westinghouse 1/7th scale. Given the chosen boundary conditions displayed in Table 2 and the formulations above, the mixing factors (equation 2) in the hot leg are similar and close to 1 for both the plant and planned mock-up, meaning very little mixing takes places in the hot leg and most of it will happen in the SG plenum and tubes.

	Reference NPP	Mock-up
Diameter of hot leg, m	0.76	0.0837
Length of hot leg, m	6	0.66
Inlet temperature, K	1063	448
Outlet temperature, K	845	365
Hot leg flow rate, kg/s	3.6	0.038
Mixing factor (equ. (2))	0.94	0.91

Table 2: Conditions for the hot leg in the NPP and planned mock-up

4. Scaling of the SG tube bundle

Scaling the SG plenum and tube bundle necessitates the use of momentum and mass balances on a typical flow path in a typical tube. We consider a “mean” U-tube with total length L and diameter D . Neglecting acceleration losses due to density changes as these are small, the pressure drop across the tube is given by:

$$\Delta P = \int_0^L \rho a \cdot dx - \frac{1}{\bar{\rho}} \left(\frac{\dot{m}}{A} \right)^2 \left(\frac{f}{2} \frac{L}{D} + \frac{K}{2} \right) \quad (7)$$

where ρ is the fluid density, a the gravity acceleration, $\bar{\rho}$ the mean density, $\dot{m}$ the flow rate, A the flow area, f the friction factor, and K the form loss coefficient.

We consider now the pressure loss around the whole loop consisting of a “hot” tube and the return leg, or “cold” tube. Since one circulates in a closed loop, the integration of the pressure drop around this loop is zero. In the inlet plenum side, we assume that the hot flow goes up through a fraction of the available tube bundle flow area ϕ_{hot} and that the other fraction ϕ_{cold} of the tubes sees cold, downward flow. Hence, from these definitions, the integration of the momentum equation around a closed loop consisting of a “hot” and “cold” tube reads:

$$\oint dP = 0 = \int_{hot\ tube} \rho a \cdot dx + \int_{cold\ tube} \rho a \cdot dx - \frac{1}{\rho_{hot}} \left(\frac{\dot{m}_t}{A_t} \right)^2 \left[\frac{1}{\phi_{hot}^2} \left(\frac{f}{2} \frac{L}{D} + \frac{K}{2} \right)_{hot} \right] + \frac{1}{\rho_{cold}} \left(\frac{\dot{m}_t}{A_t} \right)^2 \left[\frac{1}{\phi_{cold}^2} \left(\frac{f}{2} \frac{L}{D} + \frac{K}{2} \right)_{cold} \right] \quad (8)$$

In the above, $\dot{m}_t$ is the total mass flow through the hot tubes (respectively cold tubes), and A_t is the total flow area of the tubes. In order to solve the above equation, the dependency of the density on temperature along the length of the tube is necessary. This dependency is obtained via an energy balance. In steady state, the loss of energy of the fluid is balanced by the heat transfer from the outside tube wall to the secondary side reservoir assumed to have a constant temperature. Hence:

$$\dot{m}_s c_p \frac{dT_f}{dx} = -h \pi d_i (T_f - T_c) \quad (9)$$

In the above, T_f is the local fluid temperature in the tube, T_c the secondary side temperature, $\dot{m}_s$ the mass flow rate in a single tube, h is the total heat transfer coefficient between the fluid in the tube and the cold reservoir and d_i the inner tube diameter. h is given by:

$$h = \frac{1}{\frac{1}{h_{in}} + \frac{(d_i/2) \cdot \ln(d_o/d_i)}{k_w} + \frac{1}{(d_i/d_o) \cdot h_{out}}} \quad (10)$$

where the inner side and outer side heat transfer coefficients are denoted h_{in} and h_{out} , respectively, while d_o is the outer tube diameter and k_w the tube wall thermal conductivity.

To first order, it is assumed that the heat transfer coefficients are independent of location along the tube. Hence one can integrate (9) to yield:

$$T_f - T_c = (T_{in} - T_c) e^{-\lambda x} \quad (11)$$

where T_{in} is the inlet fluid temperature and:

$$\lambda = \frac{h\pi d_i}{\dot{m}_s c_p} \quad (12)$$

We assume moreover that the density changes are small such that the Boussinesq approximation holds. Therefore:

$$\rho \equiv \rho(T_f) = \rho_c (1 - \beta(T_f - T_c)) \quad (13)$$

With some algebra, it can be shown that equation (8) can be cast to yield the Richardson number (based on tube length L) which characterizes the efficiency to transform buoyant energy into kinetic energy:

$$Ri_{tube_L} \equiv \frac{Gr_{tube_L}}{Re_{tube_L}^2} = \frac{\rho_c^2 g \beta (T_{hot,0} - T_c) L}{\left(\frac{\dot{m}_t}{A_t}\right)^2} \approx \frac{\frac{1}{\phi_{hot}^2} \left(\frac{f}{2} \frac{L}{D} + \frac{K}{2}\right)_{hot} + \frac{1}{\phi_{cold}^2} \left(\frac{f}{2} \frac{L}{D} + \frac{K}{2}\right)_{cold}}{\frac{(1 - e^{-\lambda_h L/2})^2}{\lambda_h L} + \frac{(1 - e^{-\lambda_c L/2})^2}{\lambda_c L}} \cdot e^{-\lambda_h L} \quad (14)$$

Following the results of the Westinghouse tests, we assume that the flow rate into the tube bundle is about 2 times larger than the flow rate in the hot leg, and that roughly half the tubes see the hot gas. CFD simulations by Boyd et al. [4,5] show similar results. Hence, expression (14) simplifies to:

$$Ri_{tube_L} \equiv \frac{Gr_{tube_L}}{Re_{tube_L}^2} = \frac{\rho_c^2 g \beta (T_{hot,0} - T_c) L}{\left(\frac{\dot{m}_t}{A_t}\right)^2} \approx \frac{4\lambda L \left(f \frac{L}{D} + K\right)}{(1 - e^{-\lambda L/2})^2 \cdot (1 + e^{-\lambda L})} \quad (15)$$

Another Richardson number can be defined in terms of the Grashof number Gr_{SG} based on the SG bundle radius and the hot leg Reynolds number Re_{HL} . This would be represented by:

$$Ri_{HL-SGB} \equiv \frac{Gr_{SGB}}{Re_{HL}^2} = \frac{\rho_c^2 g \beta (T_{hot,0} - T_c) R_{SG}^3}{\mu_c^2 Re_{HL}^2} \quad (16)$$

Hence, four dimensionless numbers are defined as outlined in Table 3. A good similarity between plant and mock-up requires that these numbers be of the same magnitudes. In particular, the Richardson numbers need to be close. After some optimization, we settled on a design which consists of a pressure vessel (not modeled here), a hot leg with length 0.66 m and diameter 0.0837 m, and 270 U tubes with 0.005 m inner diameter, and total mean length 4.9 m. The tubes are arranged in a rectangular configuration with 0.0225 m pitch. Each plenum is a hemisphere with a radius of 0.21 m. The plena are separated by 0.1 m to allow for optical instrumentation access.

Table 3 shows a comparison between the dimensionless numbers for the plant, the PSI mock-up, and the Westinghouse 1/7th scale mock-up. Re_{tube_D} indicates the tube Reynolds number based on the tube diameter. While the exact replication of all dimensionless numbers is not possible, one should strive to ensure a) that the flow regime is matched in all sections of the system, and b) the most important dimensionless numbers are as close as possible. In the plant case, the flow is highly turbulent in the hot leg, and moderately so in the tubes. Both of these features are met in the mock-ups. In addition, the Richardson numbers are the most important factors determining mixing. It is seen that the Richardson number for the PSI mock-up tubes is much closer to the plant number than the Westinghouse 1/7th scale one. This was made possible by the increase in the tube resistance compared to the Westinghouse 1/7th scale (i.e. reduced diameter and extended length). Because of secondary water cooling, the Westinghouse mock-up was criticized [3] for being too efficient -compared to the plant- in converting buoyant energy to kinetic energy. This resulted in atypical thermal conditions as the flow in the tubes came very quickly into thermal equilibrium with the secondary reservoir. With the geometry selection and air secondary cooling, the PSI mock-up transforms buoyant energy into kinetic energy as inefficiently as the NPP does, which allows improved similarity in replicating the mixing physics (recirculation ratios, fraction of hot tubes, etc).

Parameter	PWR SG	PSI SG	W. 1/7 th SG
Re_{HL}	210000	35300	46500
Re_{tube_D}	9800	7200	9200
Ri_{tube_L}	670	630	220
Ri_{HL-SGB}	190	360	350

Table 3: Dimensionless numbers for plant and mock-ups

5. CFD simulations

CFD simulations are required to validate the assumptions of the 1-D scaling analysis made earlier. Following the methodology of Boyd [4,5], the CFD domain is decomposed into two parts. The

first part consists of the hot leg and inlet and outlet SG plena, and in this region the real geometry is meshed. The second part consists of the tube bundle, and is simplified such that a porous media treatment is employed there to considerably reduce the size of the mesh. Each tube flow channel is replaced by a square section with a porous interior. Separate computations are done featuring a real tube geometry and the porous tube geometry counterpart. The porous media parameters (pressure loss coefficients, porous media thermal conductivity) are thereafter optimized such that the pressure and temperature distribution inside the porous tube closely match those of the real tube over all the range of expected velocities.

5.1 CFD simulations of the Westinghouse 1/7th scale test

Simulations are done first on the Westinghouse 1/7th SG, since it is the only geometry where mixing data is available. The Westinghouse 1/7th facility consists of a pressure vessel (not modeled here), a hot leg with length 1 m and diameter 0.1016 m, and 216 U tubes with 0.00775 m inner diameter, and total mean length of 2.5 m. The tubes are arranged in a triangular configuration with 0.0206 m pitch. Each plenum is a hemisphere with radius of 0.24 m. In the test simulated here, SF₆ is the working fluid. A mesh consisting of about 700,000 hexahedral cells is built. Owing to symmetry, only one half of the facility is modeled. The inlet temperature is 448 K and cooling in the secondary side is accomplished with water at 338 K. The main instrumentation consists of 51 fluid thermocouples placed inside the tubes close to the tube-sheet which show whether a tube sees the hot rising plume or the cold return plume.

The Ansys Fluent CFD simulation uses the Reynolds stress model (RMS) with wall functions. The grid is built such that the cell closest to the wall has a y^+ of 20 or more, while the boundary layer consist of 5 to 10 cells. Second order accuracy is used for the momentum and energy equations. The heat transfer coefficient to the water side is taken to be 500 W/m²K such that the main resistance is on the inner side of the tubes. The inlet face of the hot leg is divided into 60% flow area for the hot flow and 40% for the cold return flow, consistent with the experimental observations. On the hot face, a mean velocity giving 0.06 kg/s flow rate is imposed, whereas a zero gauge pressure is set for the cold fluid outlet face. A transient calculation was necessary in order to drive the solution to convergence.

We show on Figure 1 the temperature distribution on the vertical symmetry plane. In Figure 2, we show the temperature in the tubes just upstream of the tubesheet. The thick line represents the experimentally observed boundary between the hot and cold fluid plumes as deduced from thermocouple measurements. Because of the high heat transfer rates to the secondary, the cold return flow temperature is essentially that of the secondary coolant. The hot plume location is reasonably well predicted by the CFD simulation, and so is the fraction of hot tubes: CFD predicts 39% versus 35% for the data. The flow recirculation factor in the bundle is predicted to be 2.0, matching exactly the data. This very good agreement needs however be tempered by the fact that the very efficient secondary cooling leads to an exit fluid temperature which is close to the water reservoir. Hence the total heat removal rate is almost a-priori known, and thus large CFD deviations will have little incidence on this global prediction. Therefore, a good CFD prediction for the heat removal rate in this particular problem is not a guarantee for the accuracy of the simulation. As a result, more challenging test conditions (poor secondary heat transfer) and high

resolution 3D data are needed to fully determine the quality of the CFD simulations. This is what PSI intends to do by proposing a test campaign in the new mock-up.

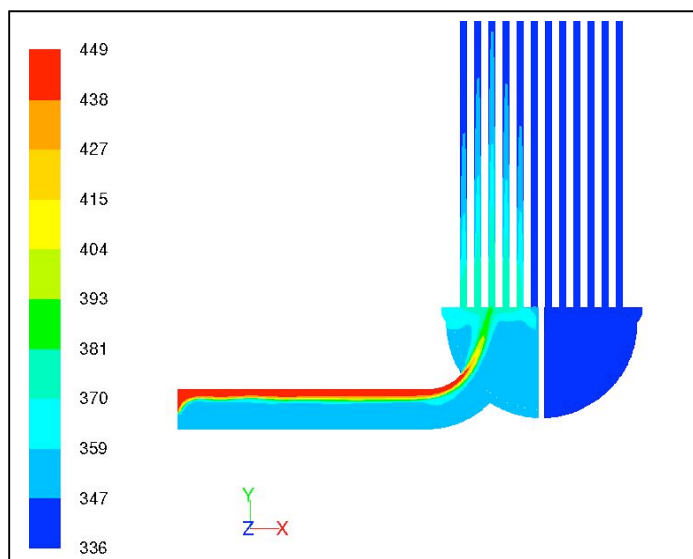


Figure 1: Contour of the temperature distribution in symmetry plane

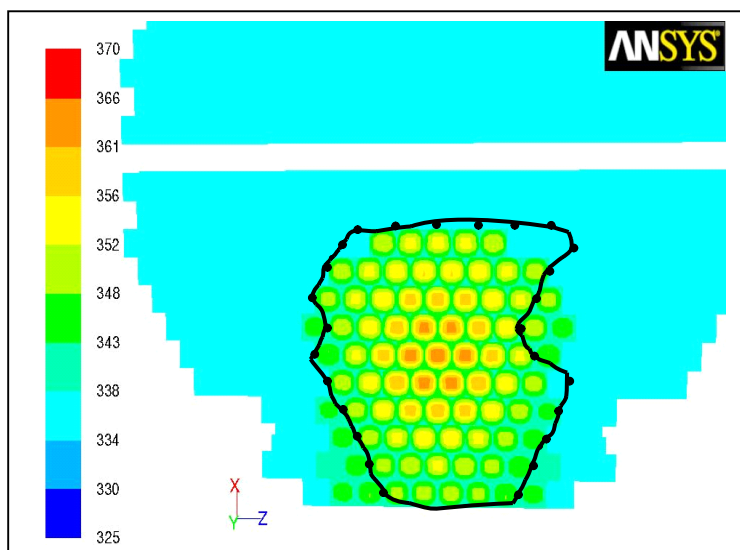


Figure 2: Contour of the temperature distribution in tubes. (Thick line on right represents experimental data)

5.2 CFD simulations of the PSI steam generator mock-up

For the PSI SG mock-up, a mesh consisting of about 960,000 hexahedral meshes is built (Figure 3). As in the Westinghouse model, the hot leg intersects the plenum symmetrically, so that only half the geometry needs be modeled. The inlet temperature is 448 K and cooling in the secondary side is done with air at 338 K. The CFD simulation uses the same models as outlined previously in section 5.1. The heat transfer coefficient to the air side is taken to be 10 W/m²K, a reasonable number for gas cooling on the secondary. On the inlet hot face of the hot leg, a mean velocity is prescribed to yield a flow rate of 0.038 kg/s, whereas a zero gauge pressure is set for the cold fluid outlet face. We show on Figures 4 and 5 the temperature distribution on the vertical symmetry plane and in the tubes just upstream of the tubesheet. In Figure 6, we show the porous upward velocity the tubes, just above the tubesheet.

Finally, Figure 7 shows the complex flow pattern via a velocity vector plot in the inlet plenum. We note that the velocity and temperature distribution in the tubes are more uniform than in the Westinghouse 1/7th scale simulation. This is due to the greater flow resistance induced by longer tubes with smaller diameter.

It is found that the hot flow is significantly cooled in the inlet plenum as the maximum temperature experienced by the tubes is about 392 K, which is just about the average between the hot stream and the coolant. 37% of the tubes see the hot fluid plume, while the recirculation rate is 1.7. Both of these numbers are comparable to the parameters assumed for the 1-D scaling (50% and 2.0, respectively). Thus, the scaling analysis is globally validated, and one can be reasonably confident that the proposed PSI mock-up will replicate the main phenomena expected in the full plant SG under the assumed scenario. Full validation of the simulations will only be possible, however, after extensive comparisons with the actual 3D data that will emerge following the project first tests.

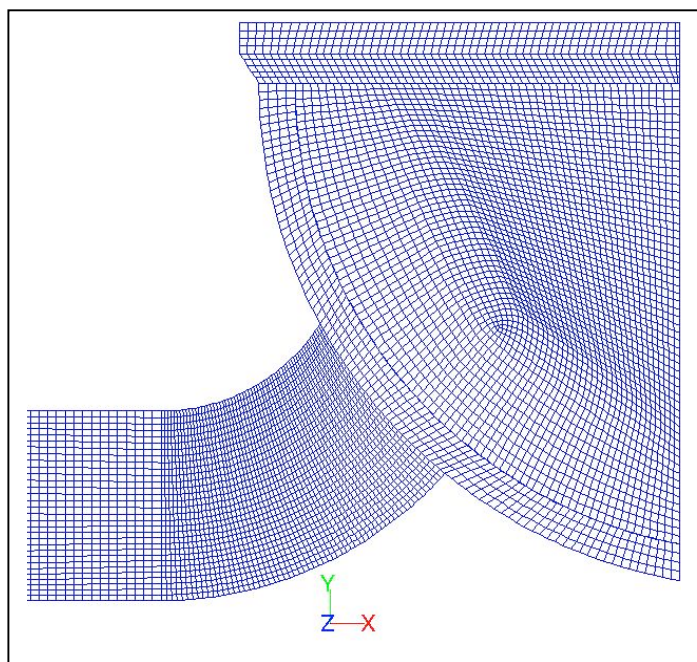


Figure 3: Mesh in the symmetry plane

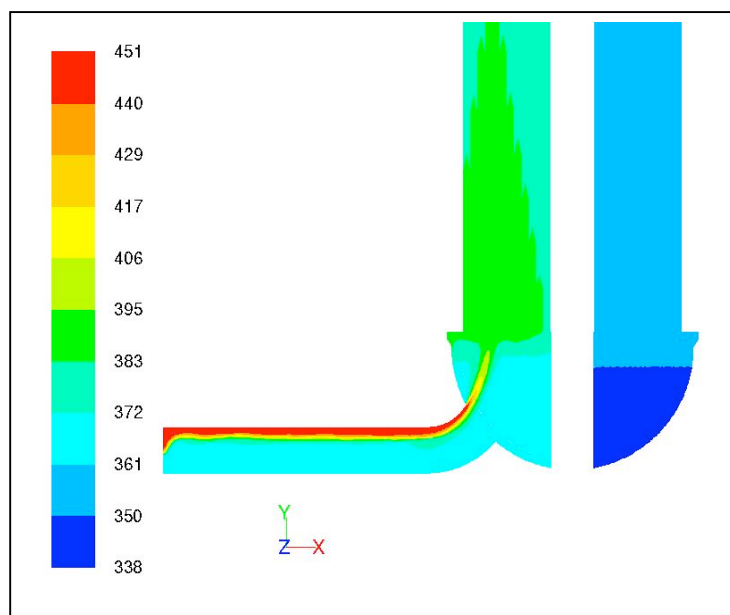


Figure 4: Contour of the temperature distribution in symmetry plane

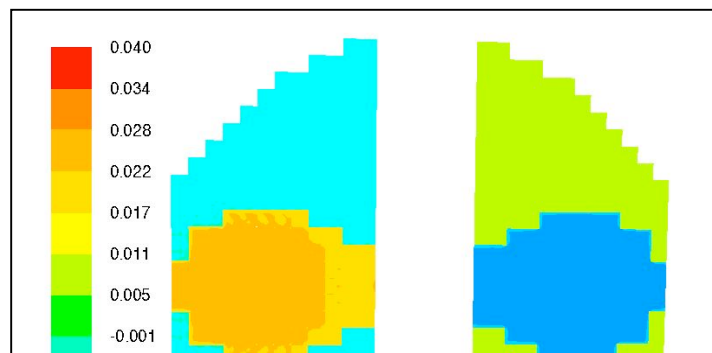
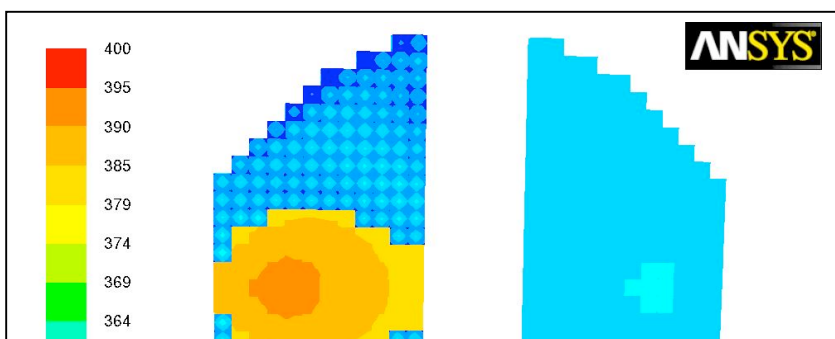


Figure 5: Contour of the temperature distribution in tubes

Figure 6: Contour of the upward porous velocity

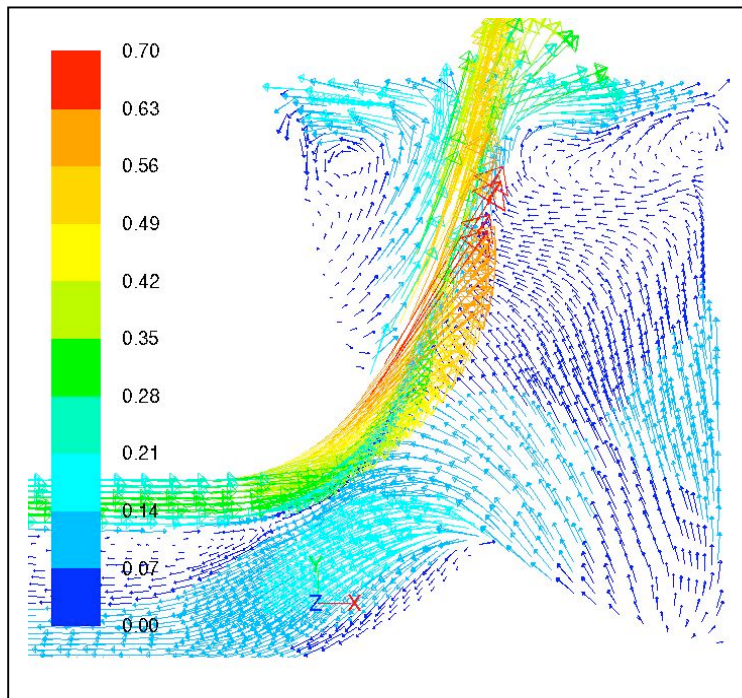


Figure 7: Velocity vector plot in plenum symmetry plane.
Colored by velocity magnitude

6. Conclusion

This investigation consists of three steps: in a first step, we use ANSYS CFD-Fluent code to simulate flow and heat transfer in the Westinghouse 1/7th scale SG experiment. This exercise serves as a basis for the validation of the CFD methodology. We compare results with limited

experimental data on mixing phenomena and find the simulations to be in good agreement with the data.

In the second step, we present a detailed 1-D scaling analysis of a proposed steam generator simulator to be built at PSI with the aim of producing high resolution, CFD-grade data. We focus on improving the Westinghouse 1/7th scale SG by matching more closely the dimensionless numbers that are important, in particular the Richardson numbers in dry secondary side conditions which are more typical of the plant. In a final step, we present a CFD simulation of mixing in the new SG mock-up, and show that the design assumptions (bundle recirculation factor, hot tube fraction, etc) are globally confirmed by the CFD predictions. Full validation of the CFD simulations will only be possible after extensive comparisons with the actual 3D data that will emerge following the project first tests.

7. References

- [1] Westinghouse Electric Corporation, “Natural Circulation Experiments for PWR High-Pressure Accidents,” Research Project 2177-05, Final Report, August 1993.
- [2] S.J. Leach, H. Thompson, “An investigation of some aspects of flow into gas cooled nuclear reactors following an accidental depressurization,” J. Br. Nucl. Energy Soc., 1975, 14, July, No. 3, 243-250.
- [3] P.D. Bayless, D.A. Brownson, C.A. Dobbe; K.R. Jones, J.E. O’Brien, D.J. Pafford, L.D. Schlenker, V.X.Tung, “ Severe accident natural circulation studies at the INEL, Appendix B, Scaling Review of the Westinghouse Test Apparatus Natural Circulation Experiment”, NUREG/CR-6285, February 1995.
- [4] C. Boyd, K. Hardesty, “CFD prediction of severe accident steam generator flows in a 1/7th scale pressurized water reactor,” , ICONE10, Arlington, USA, 2002, April 14-18.
- [5] C. Boyd, “Prediction of severe accident counter current natural circulation flows in the hot leg of a pressurized water reactor”, ICONE14, Miami, USA, 2006, July 17-20.

Acknowledgements: the authors wish to thank Dr. C. Boyd of the USNRC for his generous help during this work.