TransAT Report Series - Applications

TransAT for Nuclear Science & Technology

Fluid Flow and Heat Transfer in Steam Generators (TransAT vs. Fluent)

ASCOMP GmbH

Edited by:
Dr D. Lakehal

Release Date: November, 2009

Reference: TRS-A/02-2009

Abstract:

This note presents results of a practical thermal hydraulics problem, i.e. mixing in a PWR Steam Generator, obtained with TransAT. The document highlights in particular the robustness of the coupled Immersed Surfaces Technology and Block Mesh Refinement (IST/BMR) meshing approach for such complex problems. Results are compared to the data of Fluent obtained by NRC (USA). As a first example, TransAT was used to predict the mixing in the Beznau (KKB, Switzerland) steam generator.

1. MIXING IN STEAM GENERATORS

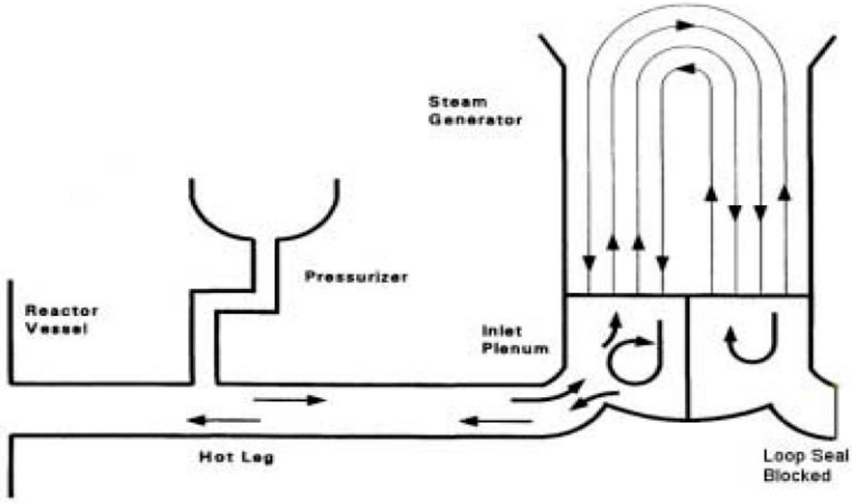

Figure 1: Overview of natural circulation flow pattern (Boyd & Hardesty, 2003)

Mixing in steam generator inlet plenum is a very important issue during postulated severe accidents in pressurized-water reactors (PWRs), where thermal stresses in steam generator tubes can be significant. Briefly, Steam Generator (GS) are resorted to for heat removing from the primary circuit to the secondary circuit in a PWR. The hot water comes from the reactor core, distributes between the U-shaped tubes of the SG, and cooled by the main feed water that is coming from the secondary circuit (Fig. 1). Once it is cooled, the water is pumped again to the nuclear core. Steam generators can measure up to 22 meters in height and weigh as much as 800 tons. An SG can contain anywhere from 3’000 to 16’000 U-shaped tubes. We should precise that detailed CFD simulations of the flow in steam generators are rare, due to the complexity of the configurations. In most cases, the number of U-shaped tubes is simply reduced.

U.S. Nuclear Regulatory Commission (NRC), among others, has implemented an action plan to assess the thermal-hydraulic conditions during a PWR severe accident (Boyd & Hardesty, 2003). One objective of this plan is to investigate the time-dependent thermal-hydraulic conditions in the hot leg and steam generator. This research is supposed to ultimately lead to a reduction in the uncertainties in modelling these severe conditions. One aspect of this research involves using state-of-the-art CFD techniques to predict inlet plenum mixing. The first part of this plan was to perform experimental study of such a case using a 1/7th scale facility. From this experiment, temperature in the hot tubes and recirculation inside the steam generator is evaluated. Selected CFD results of the NRC group for their 1/7th scale facility are detailed in various internal reports, e.g. (Boyd & Hardesty, 2003). The CFD exercise presented next (using TransAT) is a repetition of the simulations undertaken by Boyd & Hardesty (2003), who used the code FLUENT in steady-state conditions, using the k-ε model. The TransAT results will be compared to NRC simulation and experiment data. To validate TransAT calculations, resort was made of comparable physical models and computational techniques to what has been employed by Boyd & Hardesty (2003) using the CFD code Fluent.

The Beznau (KKB) SG

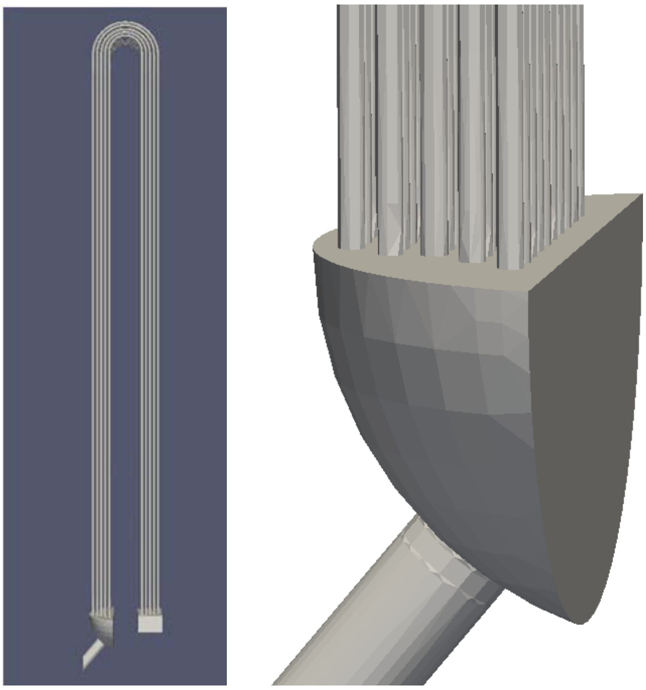

Figure 2: Overview of the CAD file representing the KKB SG (courtesy from PSI)

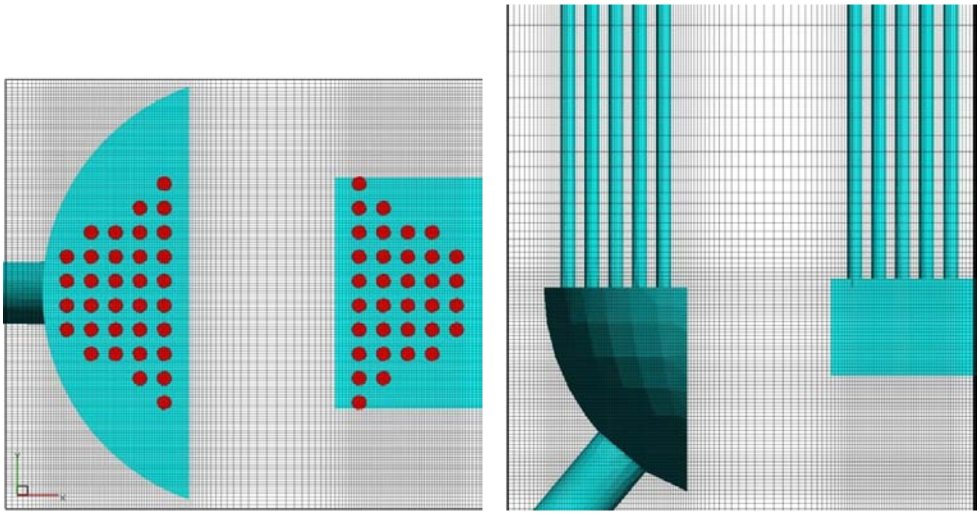

Figure 3: Two planes depicting the IST grid used in TransAT for the KKB SG

Prior to that, in the first part of this evaluation, a simplified test case was simulated, in particular to address the feasibility of the Immersed Surfaces Approach for this class of flow. This work has been performed in collaboration with the Paul Scherrer Institute (PSI) in Switzerland, which aims at developing a scaled steam generator and carry on experimental research by applying scenarios similar to the one studied by NRC. The steam generator in question is borrowed from the Beznau NPP (KKB), c.f. Fig. 2. It has 3233 U-shaped tubes; each has a height of 8 meter, and inner diameter of 16.87mm. The flow was simulated with TransAT (at ASCOMP) and Fluent (at PSI).

Obviously, one cannot simulate the 3233 U tubes; instead, the problem was reduced to 34 tubes, compensating by using the porous medium approach to rescale pressure losses along the bundle of tubes. The IST grid is presented in Fig. 3 in two planes. The grid consists of 144 x 154 x 241 = 5.3 million cells. It is shown that the tubes are covered by a Cartesian grid with sufficient grid resolution (about six cells per tube). This may be insufficient to properly resolve the boundary layers inside the tubes, but is enough for a qualitative representation of what might be expected in real conditions. The natural circulation flow patterns are shown in Fig. 4 at two planes; the upper panel clearly shows how the flow is driven by buoyancy upwards to the tubes, with obviously no backflow. The lower panel shows the flow from the pipe discharged into the plenum before pumped up by gravity forces. The panels are both coloured with pressure.

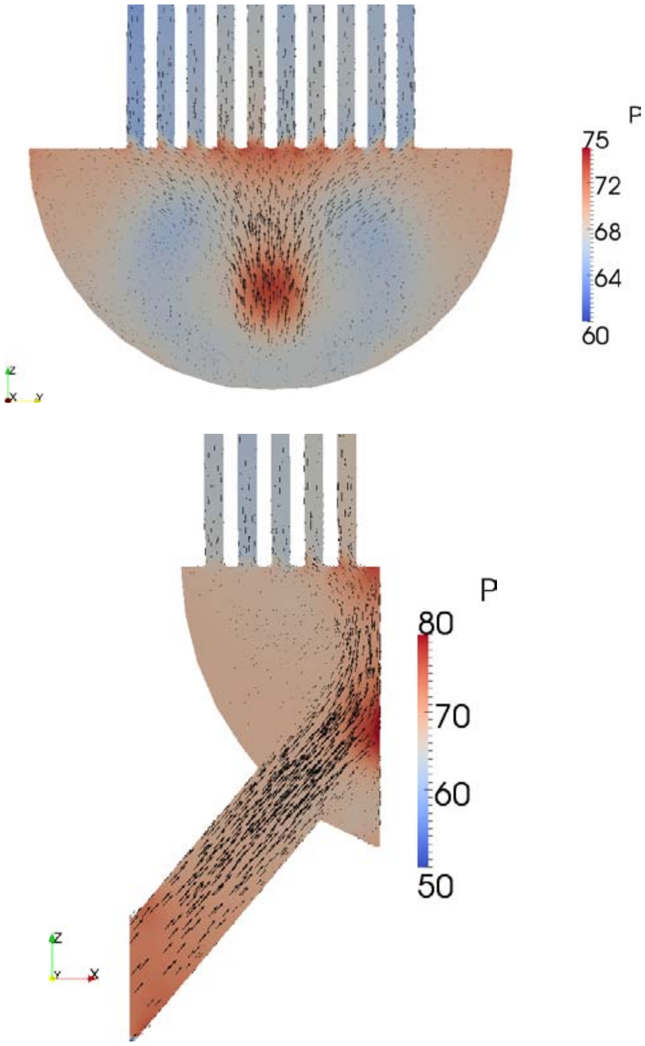

Figure 4: Overview of natural circulation flow pattern in the KKB SG

The NRC 1/7 scaled SG

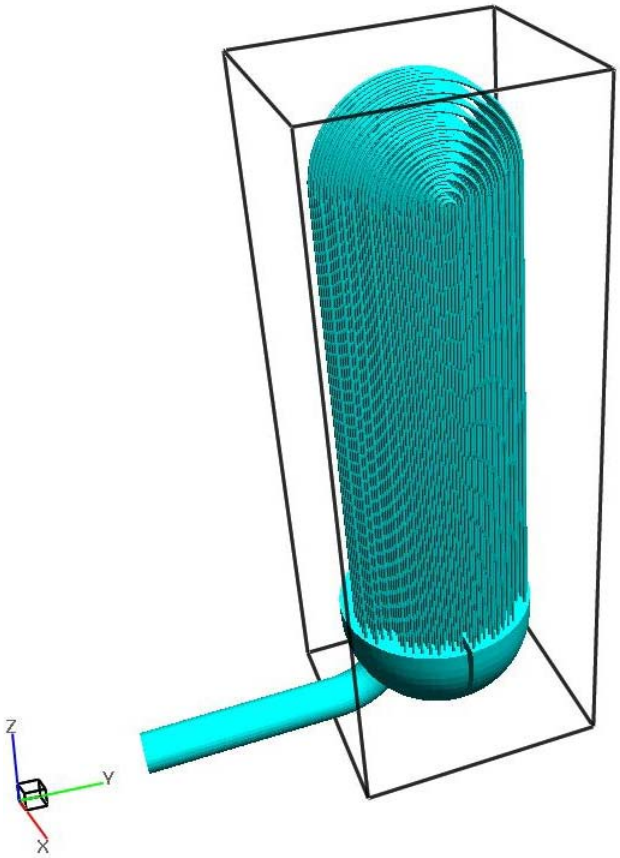

Figure 5: Overview of the NRC 1/7 scale SG CAD file used for IST grid generation

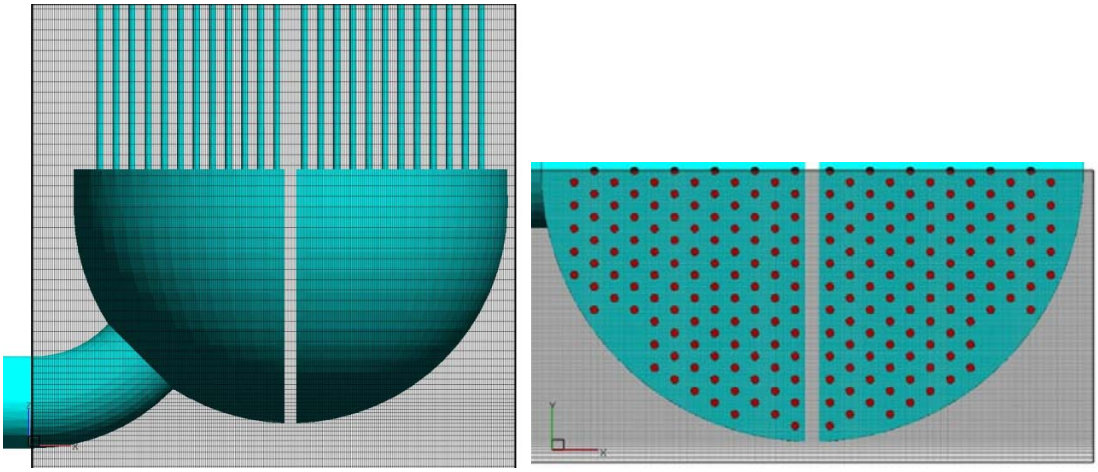

Figure 6: IST grid of the NRC SG at two planes

The NRC test facility tube bundle (ID: NUREG-1781) consists of 216 U-shaped tubes with a 0.007747m inner diameter and a 0.009525m outer diameter, anchored into a 0.1143m thick tube sheet that caps off the inlet and outlet plenums. The tubes are arranged in a

0.02064m triangular pitch array. The average tube length is 2.499m and the overall bundle height is 1.124m from the bottom face of the tube sheet. The average U-bend radius is 0.1014m. A geometry describing the full experimental facility has been created; see Fig. 5, in which the cold leg is assumed to be blocked, resulting in a recirculation inside the steam generator. Inlet tube plays thus the role of both inlet and outlet planes. Preliminary, steady-state simulations have been performed with the CMFD code TransAT. In order to reduce the number of cells, a symmetry axis has been used in the middle of the steam generator, as proposed by Boyd & Hardesty (2003).

Immersed surface technology has been used, which results in a Cartesian mesh of 326x157x204 = 10.4 million cells (Fig. 6). The k-ε model has been used for Turbulence with wall functions and conjugate heat transfer is solved in the solid so that heat exchange between primary and secondary loops is modelled in the tube bundle region. A constant temperature of 335K is fixed in the wall boundary conditions, while the incoming fluid (SF6) has a temperature of 447K. It is perhaps important to note that in the TransAT simulations, the Boussinesq approximation has been employed to cope with thermal stratification; in the work of Boyd & Hardesty (2003), however, use was made of the weak-compressibility approach based on the equation of state. The simulation lasts 5 days on an 8-core PC machine using OpenMP parallel protocol.

Figures 7 and 8 show selected results obtained so far with TransAT. The upward-downward motion of the flow is well depicted in Fig. 9 in particular, together with the thermal stratification occurring inside the feed tube. The thermal-induced upward motion of the flow is seen to occur almost on the left part of the plenum, which is may be due to the way inflow conditions were set here. Indeed, an important point to rise to this regard is that in contrast to the early Fluent simulations performed by the NRC group, in the present case the feed pipe has been considerably shortened to minimize gridding, which has been revealed later to have an impact on the final results. New simulations with a longer tube are underway. In the present TransAT simulations, 110 'hot tubes' were found (Fig. 8), reflecting the fact that the flow goes from the inlet plenum to the outlet plenum. Compared to the experimental result, which shows only 85 hot tubes, the present simulation is qualitatively acceptable, with an error of 20%. As to temperature, the error is of the order of 12%.

Turning now to the NRC (USA) results and simulation campaign using the code Fluent. The grid employed in their case is a nonstructured one, requiring a huge amount of time to adjust it around the complex configuration, in particular around the tubes (Fig. 9). Their results shown in Fig. 10 are similar to the one obtained by TransAT, but with less efforts in terms of grid generation using IST7BMR technique. Of course since in Fluent simulations they have considered the entire feed-pipe, their results of the mixing are more logic, since in TransAT an inlet boundary conditions was imposed to mimic the entire feed-pipe.

CONCLUSION

This note describes the way computational thermal-hydraulics is migrating to more sophisticated meshing techniques for problems involving complex geometries. The proposed technique, called IST/BMR, helps describe the wall-surface of any component simply using CAD-based information. The CAD file is immersed in a Cartesian grid. TransAT recognizes the wall-surface and applied the wall boundary conditions as appropriate. The method can be successfully combined to generate realistic transient simulations of turbulent flows in reasonable computing times (of the order of 24H on PC Linux clusters), since it reduce the grid size and thus the simulation time.

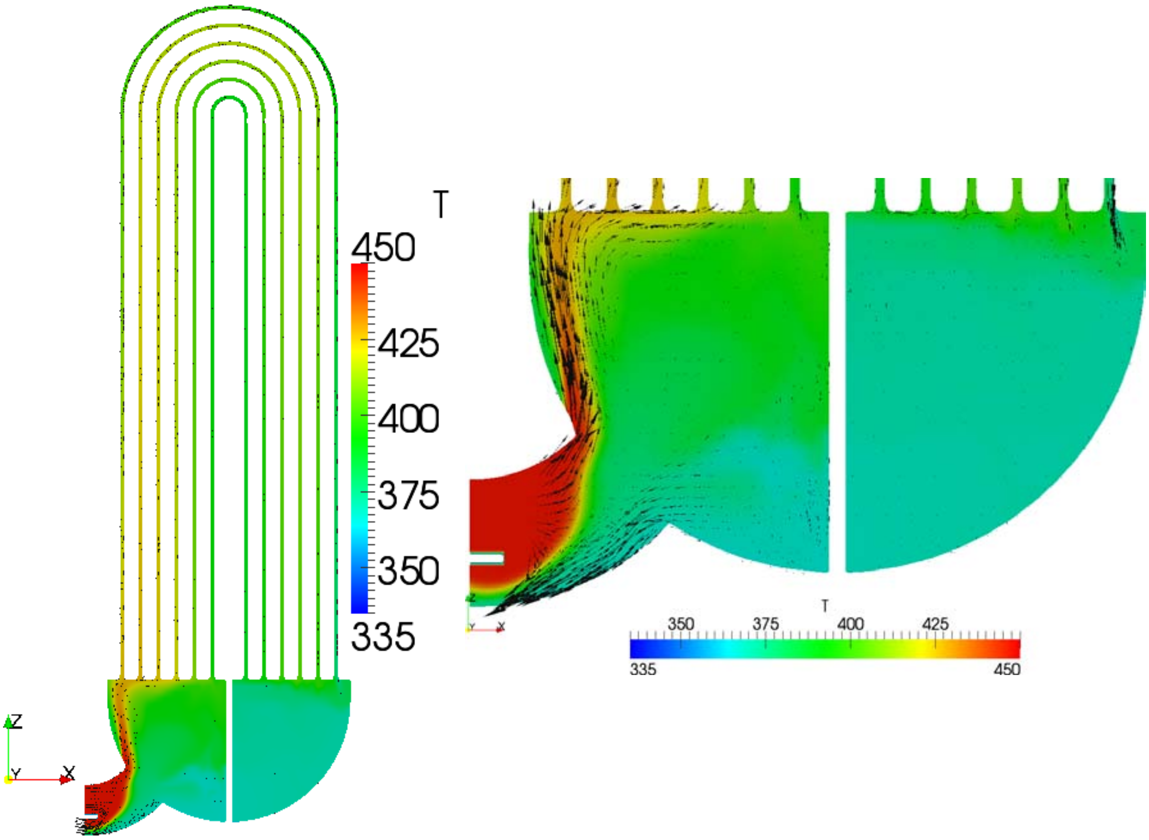

Figure 7: Temperature and velocity fields in the steam generator

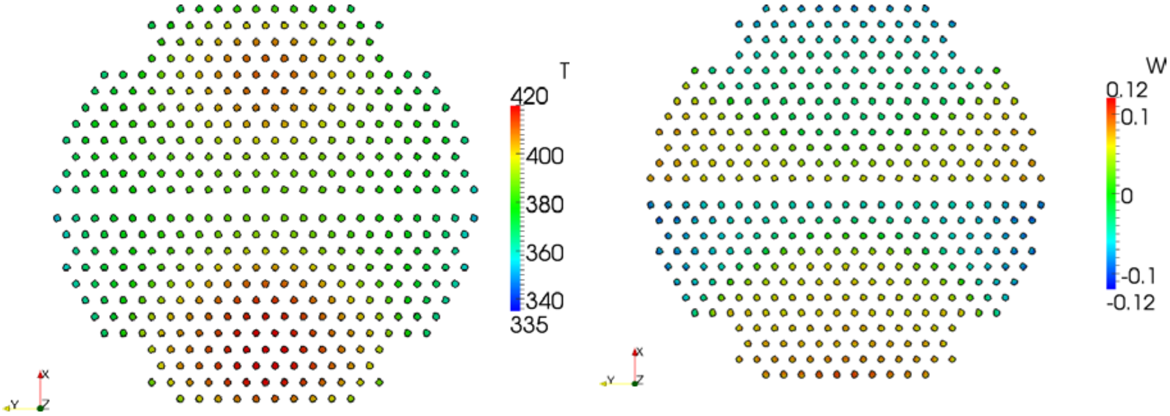

Figure 8: Temperature and vertical velocity in the tube bundle

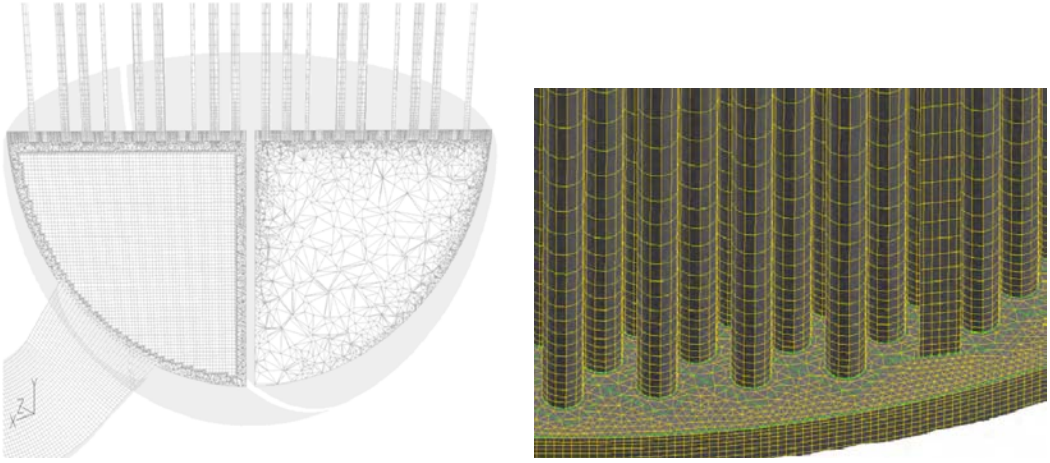

Figure 9: Unstructured grid details for FLUENT

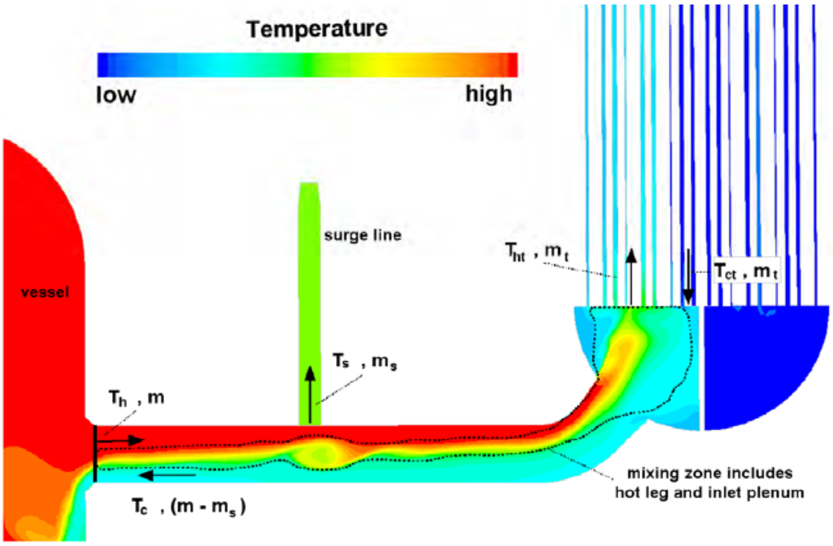

Figure 10: Temperature and velocity fields in the steam generator (FLUENT; NRC, Chris Boyd)

References

U. Bieder, E. Graffard, Qualification of the CFD code Trio U for full scale reactor applications, Nucl. Eng. Design 238 (2008) 671–679

C. Boyd, K. Hardesty, CFD analysis of 1/7th scale steam generator inlet plenum mixing during a PWR severe accident. USNRC report NUREG-1781, 2003.

Höhne, Th, Kliem, S., Bieder, U., 2006. Numerical modelling of a buoyancydriven flow experiment at the ROCOM test facility using the CFD codes CFX and Trio U. Nucl. Eng. Des. 236 (12), 1309–1325.

G. Yadigaroglu, D. Lakehal, “New trends in computational thermal hydraulics”, Nuclear Technology, 152 (2), 239-251, 2005.