TransAT Report Series
– Applications –

TransAT for Nuclear Science & Technology

Advanced CMFD for Gen. III Nuclear Reactor Issues

Ascomp Switzerland
Edited by: Dr Djamel Lakehal
Release date: Sep, 2014.
References: TRS-A/ 02-2014
Table of Contents
1. Dealing with Intricate Issues in 3rd Generation Nuclear Reactors ........................................... 2
2. Passive Containment Cooling Systems ........................................................................................ 3
3. Pressurized Thermal Shock: PTS .................................................................................................... 5
4. Emergency Core Cooling Systems ................................................................................................. 7
5. Boron Dilution .................................................................................................................................... 10
6. Mixing in Steam Generators ........................................................................................................... 13
   6.1 The Beznau (KKB) SG .................................................................................................................. 14
   6.2 The NRC 1/7 scaled SG ............................................................................................................... 16
Conclusion ........................................................................................................................................... 17
Abstract:
This note shows capabilities of the CMFD code TransAT for various flow scenarios in third generation nuclear reactors. The document highlights the unique features of the code compared to others, including meshing complex geometries using the coupled Immersed Surfaces Technology combined with Block Mesh Refinement (IST/BMR), and phase-change physics including steam condensation.

1. Dealing with Intricate Issues in 3rd Generation Nuclear Reactors
ASCOMP has been active in the simulation of various flow scenarios in third generation nuclear reactors, including APR1400, APWR, ESBWR and AP1000 (passive cooling system). Very recent simulation studies using TransAT deal with advanced design and enhanced safety features of the KOPEK APR1400 (active cooling systems using pumps). The AP1000 is a two-loop 1000MWe PWR where passive safety systems are used to provide significant and measurable improvements in plant simplification, safety, reliability, investment protection and plant costs. The AP1000 and its predecessor AP600 are the only nuclear reactor designs using passive safety technology. The AP1000’s greatly simplified design complies with NRC regulatory and safety requirements and the EPRI advanced light water reactor (ALWR) utility requirements document.

Korean APR1400 reactor design was selected as the first-ever nuclear power plant in UAE, 2009 and became a main brand representing the Korean nuclear power technology. Examples of issues that have been dealt with are shown in Fig. 1 below, including (1) the flow and convective heat transfer in Steam Generator (SG), (2) the flow in the reactor vessel (RV) including boron dilution, (3) the pressurized thermal shock in case of LOCA, (4) and the internal condensation, and (5) cooling of the IRWST by spargers, including condensation.

Figure 1: AP1000 RCS and passive core cooling system (Schulz, 2006)
2. Passive Containment Cooling Systems

Figure 2: AP1000 passive safety system simplifications (Schulz, 2006).

PCS provides the safety-related ultimate heat sink for the AP1000 plant (Fig. 3). The PCS cools the containment following an accident so that design pressure is not exceeded and pressure is rapidly reduced, dropping to ~40% of the design pressure in less than 5h (Schulz, 2006). The steel containment vessel provides the heat transfer surface that removes heat from inside the containment and transfers it to the atmosphere. Heat is removed from the containment vessel by the continuous, natural circulation of air. During an accident, air cooling is supplemented by water evaporation. The water drains by gravity from a tank located on top of the containment shield building. The containment height has been increased to provide additional free volume. This additional free volume, with a change of material for the vessel shell, provides increased margin to vessel design pressure from accident pressures over AP600. Analysis shows that during severe accidents the AP1000 containment is likely to remain intact and not be bypassed. As a result, the plants have a significantly reduced frequency of release of large amounts of radioactivity following core damage in an accident.

The dimensions of the containment vessel considered here are 5.2 m height and 3.5 m width. The pressure is 4.07 bars, corresponding to (close) conditions of the Westinghouse AP1000. In this example, the steam is initiated in the vessel, without representing the hypothetical steam leak from the SG or any other tubular system. Temperature is fixed (400° K) as a boundary condition at the upper height of the steel containment vessel. At the lower boundary, it was set to 450° K; there an open pressure boundary conditions has been used to allow the flow to circulate freely. Here use is made of the level set to track interfaces, with the DCC model for phase change heat transfer (solving energy balance across steam-water interfaces directly). The mesh was generated using the IST technique, where the dome was created using a CAD generator then simply immersed in a Cartesian grid. Simulations were conducted in 2D only, under laminar flow conditions.
Images shown in Fig. 3 are taken at 0.5 and 1 s. The results show some steam condensation occurring, forming a liquid film that slides along the vessel internal walls by gravity. The evaporation of the film occurs only at a later stage when the film reaches 25% height of the vessel, where the wall temperature is high. The thermal stratification as shown in the lower two panels induces a strong natural circulation, exactly as postulated in the sketch shown in Fig. 2. The average flow velocity of the steam is \( \sim 1 \text{m/s} \) and is \( \sim 3 \text{m/s} \) for the water film. These early results show clearly the unique capability of Transat in dealing with such complex flow conditions, involving buoyancy driven natural circulation and mixing, steam condensation and water film evaporation. The extension to 3D is straightforward, even including a real pipe with a break. The same is true of turbulent conditions, where TransAT can run using URANS (including Explicit Algebraic Stress Models that are better suited for strong buoyancy driven flows), VLES or even LES if required. Note too that the film height was initiated and is rather thick compared to the reality. This could be adjusted using a finer mesh near the steel containment vessel to better resolve the film. This is now underway, where use of BMR capability is resorted to. There is however no experimental data to compare to.

Figure 3: Results of film formation and drainage by gravity on the containment vessel walls subsequent to steam condensation. The water film starts to evaporate at the lower height of the vessel. Natural circulation is created by buoyancy forces due to thermal stratification.
3. Pressurized Thermal Shock: PTS

Emergency Core Cooling (ECC) injection of cold water is one of the most severe scenarios of the global Pressurized Thermal Shock (PTS), referring to the occurrence of thermal loads on the Reactor Pressure Vessel (RPV), under pressurized conditions (Lucas et al., 2008). Here cold water is injected into the cold leg during a hypothetical Small Break Loss Of Coolant Accident (SB-LOCA). The injected water mixes with the hot fluid present in the cold leg and the mixture flows towards the downcomer where further mixing with the ambient fluid takes place. Very steep thermal gradients may damage the structural components while the primary circuit pressurisation is partially preserved. Therefore, the transient fluid temperature must be reliably assessed to predict the loads upon the RPV and the pressure wall toughness. The coolant can be single- or two-phase flow, depending on the leak size, its location and the NPP operating conditions. The PTS has been the objective of a number of cooperative programs in the past, e.g. the OECD-ICAS as given by Sievers et al. (2000).

The computation of this severe scenario is now within reach of the averaged two-fluid formulation. Yao et al. (2005) solved for the purpose the 3D steady-state Navier-Stokes equations within the two-fluid framework, using a modified turbulent K-e model that accounts for the production by interfacial friction. The CATHARE code was then used; though recent attempts by the same group using the NEPTUNE-CFD code developed within the NURESIM project have been reported in several conferences. The time-averaged velocity profiles are somewhat well predicted, but not the turbulent quantities, in particular
the turbulent kinetic energy and the shear-stress distributions. The question here is whether the latest transition in computational thermal-hydraulics mentioned before relying on interface tracking than two-fluid modelling and LES or V-LES instead of RANS is feasible in this complex context or not. This is the reason why the test case presented here was selected – as ‘feasibility study’- towards more detailed simulation using the LEIS concept. The flow features more physics to deal with than the pipe flow: two-phase flow, heat transfer, phase change, complex geometry with two inflows.

The flow conditions selected here for the simulation are taken from the D2.1.2 deliverable for the NRESIM EU project (Lucas and Bestion, 2007). More specifically, the pressure of the system is set to 20 bar, the water mass flow rate to 0.58 Kg/s, the steam flow rate to 0.2 Kg/s. The coolant fluid and the water in the leg are initially at temperature 296 K, the steam is fixed at saturation temperature: 485 K. Here we have again resorted to the MILES approach because the Reynolds number is again not very high, within the level set context. The level set equation was modified to account for direct phase change, i.e. the rate of mass transfer is rather directly inferred from the balance of energy fluxes across the interface.

The grid was generated using the IST method described in Section 3 above, whereby a CAD file of the piping systems was embedded within a Cartesian grid; the geometry included the wire. The diameter of the pipe is 118 mm, the length is 1400 mm, the diameter of the IS port is 21.9 mm, with an inclination of 30 deg. The BMR technique for multi-block refinement was used, as shown in Figure 6. The grid was overall rather coarse, consisting at the end of 520,000 cells. The cross section was covered by 20x20 grids, which is rather coarse, but that does not mean that the resolution does not prevent capturing most of the
flow details. This is obviously coarse, set for the sole purpose of feasibility study of the approach. An initial perturbation of the flow based on the shear Reynolds number was set in the leg. Qualitative flow features are depicted in Figure 5, showing the interface deformations subsequent to coolant-jet impingement on the surface of the hot water in the leg, together with the temperature field as it mix downstream. The deformations of the sheared surface are most intense in the region below injection; the subsequent waves that form travel in the flow direction up to the wire level.

Figure 6 depicts several cross-flow planes inside the leg, showing secondary velocity vectors, interface level and temperature contours. The images start from left to right with flow direction: the first four panels depict the flow prior to injection; the fifth one depicts the flow at injection level; the 8th panel depicts the flow at the wire. The first panel shows heated water in the leg subsequent to steam injection at the inflow; the steam penetrates there quite deep and thus rising the temperature of the water to almost the steam temperature. The temperature of the steam does not remain constant at saturation conditions because the steam releases heat to the pipe walls, the temperature of which was set constant (296 K), too (isothermal boundary conditions). The picture would have substantially changed if indeed adiabatic boundary conditions were set at the wall pipe surfaces. The heat is shown to diffuse from the steam to the water in the core leg rather gradually, in contrast to the case if saturation conditions of the steam were forced by neglecting the heat transfer flux, keeping only that in the liquid. The 7th panel shows that just before the wire the water temperature mixes with the steam and reaches 350 K.

4. Emergency Core Cooling Systems

In AP1000, the passive core cooling system (PXS), shown in Fig. 1, is designed to protect the plant against RCS leaks and ruptures of various sizes and locations. It provides core residual heat removal, safety injection, and depressurization. Following a rupture of the main reactor coolant pipe, the PXS cools the reactor with enough margins to the peak clad temperature limit. It uses three sources of water to maintain core cooling through safety injection sources, including core makeup tanks, the accumulators, and the in-containment refuelling water storage tank (IRWST). The PXS provides depressurization using the four stages of the automatic depressurization system to permit a relatively slow, controlled RCS pressure reduction, and provides the heat sink for the passive residual heat removal heat exchanger PRHR HX as well, where the water absorbs decay heat for more than one hour before the water begins to boil. Once boiling starts, steam passes to the containment. The steam condenses on the steel containment vessel and, after collection, drains by gravity back into the IRWST. Similar PXS systems are used in Korean APR1400, albeit under different setup and operating conditions (not passive). Here the condensation of a steam jet discharged into the subcooled water pool is used to condense a large quantity of high pressure, high temperature steam without introducing complex devices. It is equally used for pressure suppression pools in boiling water reactors (BWRs). The main unknown safety issues here are: to keep the pool temperature below 90°C, and to quantify the effect of pressure loads induced by bursting of large steam bubbles on the IRWST wall integrity.

The example shown below relates to the steam injection via specific (the design will not be shown) spargers in the APR1400 IRWST. The work has been undertaken in two steps: first, the flow and thermal mixing and general circulation in the containment annulus is simulated to infer boundary conditions to be further used in the simulation of the complete condensation process from one isolated sparger. Use was made of the IST technique to mesh the geometry shown in Fig. ?, represents a huge advantage in terms of reducing time for mesh generation compared to other techniques. The flow is shown in Fig., depicting a strong recirculation and thermal mixing, as was to be expected. Here the simulation is
unsteady, using URANS modelling for turbulence. Steam injection spargers are represented by momentum and heat point sources. The feedwater from below is also accounted for through the three stumps.

![Figure 7: IST/BMR grid for the IRWST of APR1400](image)

![Figure 8: CFD analysis samples showing general flow patterns](image)

The results have been used in a later stage to define the computational domain for a single condensing jet (this is not shown here for NDA reasons). The details will thus not be shown here, but two prototypical examples of condensing jets at similar flow conditions are shown instead. Here use was made of the level set technique to track interfaces, combined with the surface divergence model for condensing phase change heat transfer (see Lakehal and Labois, 2011). Turbulence is treated using advanced scale-resolving techniques, namely LES (only in 3D) and V-LES (see Labois and Lakehal, 2011); the latter is used in 2D axisymmetric conditions.
The single jet has been simulated in detail combining level set for interface tracking (ITM) and VLES for turbulence. Phase change is dealt with using direct condensation approach (unique feature of TransAT) that solves the thermal gradients across the interface, and two other models implemented in TransAT, namely the Surface Divergence (SD) model and the Adaptive SD variant (Lakehal and Labois, 2011). These are unique modelling features of TransAT as well. It is clear that resolving the thermal boundary layer at the interface to capture its gradients and thus the heat flux is elusive; the simulation provided a very low rate of interfacial phase change, of the order of 0.001kg/s. Moving to the SD models, the rate predicted is at least two orders of magnitude higher, as shown in Fig. 10 left.

Figure 9: Experimental visualization of a condensing jet in subcooled water

Early and very recent condensing jet simulations mimicking the examples show above are presented below. Both cases are 2D axisymmetric. Obviously, the flow features different topologies, and as such it is not within reach of one single approach; neither interface tracking alone, nor phase averages Eulerian variant. Still, we have compared the two approaches and only ITM’s results are presented below. What is interesting to note in the two figures below (two different jets with different flow conditions, including the inflow pipe; vertical and horizontal) is how well the interface is captured together with the rate of mass transfer (mdot). This unique feature of the code TransAT outperforms compared to any other code. It is obvious that 2D axisymmetric simulations cannot reproduce a perfect jet with a cone like in the experimental image in Fig. 9. It is perhaps important to note that these detailed simulations are used to develop a coarse-grained phase change model to be implemented for simulation like the entire annulus (previous images).

Figure 10: CMFD of a 2D axisymmetric condensing jet in subcooled water (vertical feeding)
Figure 11: CMFD of a 2D axisymmetric condensing jet in subcooled.

5. Boron Dilution

Figure 12: The ROCOM CAD files.
Various measurement campaigns were performed worldwide to understand the phenomena in boron dilution in pressurized water reactors (PWR). The Rossendorf Coolant Mixing Model (ROCOM) operated by the Forschungszentrum Rossendorf e.V. is one of these (Prasser et al., 2003; Grunwald et al., 2002; Höhne et al., 2003). The transparent 1:5 linear scaled ROCOM test facility shown in Fig 12 simulates German PWRs and includes all important details for coolant mixing. The facility has four loops each equipped with an individually controlled pump to enable the performance of tests in a wide range of PWR flow conditions: ranging from natural convection flow-up to forced convection flow at nominal flow rates, including flow ramps (e.g., due to pump start-up). ROCOM is operated with water at ambient temperatures as the RPV mock-up and its internals are made of Perspex. For the investigation of boron dilution transients, disturbances are created by computer-controlled injection of a tracer (salt water solution) into the cold leg of one or two of the loops near the inlet nozzle. The facility was equipped with fast-acting pneumatic gate valves (opening time 3 s) to cut-off the part of the loops where disturbances are generated. The initial test conditions were checked by a wire mesh sensor at the RPV mock-up inlet nozzles. Since the boron content influences the fluid density, this was adjusted by adding ethyl alcohol.

Steady state and transient flow simulations were performed by TransAT using the K-e model for turbulence. The CAD file shown in Fig 13 including the details of the drum (right) was immersed in Cartesian grid consisting of 2 and 4.4 million cells (Fig. 14). The IST method was applied. OpenMP parallel computation took 8h on 4 CPU Linux PC cluster for the fine grid simulations. The results of the simulation are discussed in the context of Figs. 15 and 16. The left panel of Fig. 13 displays the velocity iso-contours, showing the details of the flow captured by the method without use of the porosity approach. The right panel displays the boron dilution and distribution in the downcomer. Figure 16 compare experimental and CFD results of the scalar mixture at a cross flow section located just above the perforated drum. The simulation results agree pretty well with the data, which proves that the overall IST/BMR approach can be used for practical thermal-hydraulics problems.
Figure 14: Flow and scalar mixing obtained by TransAT.

Figure 15: Measured vs. simulated boron mixing in a cross section above the perforated drum.
6. Mixing in Steam Generators

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. 16). 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 $k$-$\varepsilon$ 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.
6.1 The Beznau (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), cf. Fig. 17. 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).

![Figure 17: Overview of the CAD file representing the KKB SG (courtesy from PSI)](image)

![Figure 18: Two planes depicting the IST grid used in TransAT for the KKB SG](image)
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. 18 in two planes. The grid consists of $144 \times 154 \times 241 = 5.3$ 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. 19 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 19: Overview of natural circulation flow pattern in the KKB SG](image)
6.2 The NRC 1/7 scaled SG

Figure 20: Overview of the NRC 1/7 scale SG CAD file used for IST grid generation

Figure 21: 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
0.02064 m triangular pitch array. The average tube length is 2.499 m and the overall bundle height is 1.124 m from the bottom face of the tube sheet. The average U-bend radius is 0.1014 m. A geometry describing the full experimental facility has been created; see Fig. 19, 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 $326 \times 157 \times 204 = 10.4$ million cells (Fig. 20). The $k-\varepsilon$ 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 335 K is fixed in the wall boundary conditions, while the incoming fluid (SF6) has a temperature of 447 K. 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 22 and 23 show selected results obtained so far with TransAT. The upward-downward motion of the flow is well depicted in Fig. 22 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 a tremendous impact on the final results. New simulations with a longer tube are underway. In the present TransAT simulations, 110 'hot tubes' were found (Fig. 23), 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%.

**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 22: Temperature and velocity fields in the steam generator

Figure 23: Temperature and vertical velocity in the tube bundle
References


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


