To: Jess Gehin – AMA FA Lead
Rose Montgomery – AMA Deputy Lead

cc: Z. Karoutas (W)

From: J. Yan (W)
Tel: (803) 647-2047
Fax: (803) 695-3985

Date: April 16, 2013
Our Ref: PFT-13-34

Subject: **CASL Technical Note: Fibrous Debris Pilot Simulation**

Keywords: Fibrous Debris Pilot Simulation, LOCA, CFD

Fibrous Debris Pilot Simulation

Pressurized water reactor (PWR) containment buildings are designed both to contain radioactive material releases and to facilitate core cooling in the event of a loss-of-coolant-accident (LOCA). Reactor core cooling following the LOCA may utilize water discharged from the LOCA pipe break and containment spray. This water is collected in a sump and is recirculated through the emergency core cooling system (ECCS) and the containment spray system (CSS). Although the sump contains several screens to remove any debris in the water, it is possible that fibrous debris from reactor insulation or other sources could be present in the

©2013 Westinghouse Electric Company LLC

All Rights Reserved

recirculated water. Concerns have been raised about the potential for the debris to affect long term core cooling during the LOCA event.

Nuclear fuel assemblies (FA) are equipped with bottom nozzle filters designed to trap debris prior to entry into the active fuel region. Industry tests have demonstrated that, when present, fibrous debris collects at the bottom nozzle or at the lowest spacer grid, causing higher resistance to the coolant flow. The redistribution of flow and effect of the blockage on the fuel temperatures due to the postulated scenario and presence of the fibrous debris is not fully understood.

In order to demonstrate some of CASL's simulation capabilities on a high-performance computing system, the Advanced Modeling and Applications (AMA) team piloted a full core simulation of a 4 loop Westinghouse NSSS with postulated fibrous debris following a LOCA event. The work was led by Westinghouse Engineering Fellow Dr. Jin Yan, and utilized the commercial computational fluid dynamics code STAR-CCM+ (version 7.04.006) provided by CD-Adapco. Dr. Yan noted, "The CFD results will give more details of flow and temperature fields of coolant and solids in the reactor core under complicated transient accidental conditions than have been available previously."

The simulation leveraged an existing mesh for the reactor and fuel. Due to the large model size and computational requirements, this effort provided valuable experience in identifying CFD model development issues (labor, time and code limitations) and challenging capacities of available computing resources. The CFD simulation includes a full 360-degree model of the reactor vessel, including the cold and hot legs, as shown in Figure 1. The upper head was eliminated to reduce computing cost and a solid wall was placed at the boundary. The geometries of the reactor internals were meshed in great detail, except for the spacer grids, which were represented by porous media. The mesh of the reactor core was built in Prostar and the remaining regions of the reactor vessel were meshed using the STAR-CCM+ polyhedral mesher; the total cell count was ~1.14 billion mesh cells. A typical structured mesh in the core region is depicted in Figure 2. The various meshes were assembled on the Westinghouse large memory workstation into a single domain connected through interfaces. The simulation was executed on Westinghouse computer clusters and the INL Fission computer. Post-processing was done on the Westinghouse large memory workstation.

The fibrous debris was modeled as a gradual time-depdendent pressure drop increase across the bottom nozzle region of each fuel assembly as the fibrous debris accumulates and was assumed to accumulate uniformly across the reactor core. The core decay heat (NS-RAT-DS-85-026) was applied to each fuel rod in the model using a power density table that was generated based on typical core conditions (calculated using Westinghouse's ANC neutronics code). Normalized axial power shapes were applied, with the fuel rod axial power shape determined based on its core nodal location and relative radial rod power.

The transient simulation was started 18 minutes after the postulated hot leg break LOCA. All of the coolant and all solid structures (including the fuel) were initialized at the RHR cooling temperature of 190ºF at the beginning of the simulated transient. A constant mass flow rate of 91.482 kg/s and temperature at 190°F was applied at each RHR inlet. One of the hot legs was assumed to be at 0 psi (relative to atmospheric pressure) as the outlet boundary condition; the rest of the hot legs were closed. Phase change was simulated with using a Volume of Fluid (VOF) model. The entire flow domain was assumed to be filled with liquid water initially.

The volume of liquid coolant present in the core and its depletion is a key parameter indicating whether fuel assemblies are uncovered during accidents. The simulation provided an indication of the collapsed liquid level in the core and the mass flow rate through the broken hot leg, indicating that the liquid mass flow increases substantially when boiling starts and quickly reduces in inventory. The water level in the core decreased sharply at the beginning of boiling and then slowly decreased as the liquid boiled off. The liquid inventory was depleted almost the same rate in all quadrants when boiling started. However, shortly after the onset of boiling, the quadrant closest to the broken hot leg depleted at a higher rate than the other quadrants, suggesting convection of entrained steam toward the broken loop. Steam continued to accumulate more rapidly in the broken loop quadrant until vapor began to discharge at the exit (break) after approximately 40 seconds. Figure 3 provides a cross-section of the reactor core with the volume fraction of vapor at 94 seconds into the LOCA event.

The maximum pellet and clad waterside surface temperatures increased rapidly in the first 15 s (due to decay heat application). Once the clad reached the saturation temperature and boiling started, the temperature increase rate slowed down, and the pellet centerline temperature responded about 20 seconds later. Figures 4 and 5 graph clad outer surface temperatures and fuel rod centerline temperatures, respectively, at various elevations as a function of LOCA event time.

An interesting finding uncovered by the simulation is the appearance of a strong circulating flow in the bypass channel providing enhanced core cooling for some time. The vector plot near the core inlet in Figure 6 shows the high speed jets through the holes on the lower former plates. This downward flow is attributed to greater hydrostatic pressure in the core bypass region relative to the fuel assemblies as boiling progresses in the core. A coincident increase in upward flow in the fuel assemblies is also suggested by the cladding temperature trends of Figure 4. The outer clad temperatures are shown to decrease at the lower elevations as subcooled liquid from the inlet plenum penetrates further upward in the assembly with the onset of boiling at higher elevations. These results demonstrate the important capability of the CFD analysis to resolve the influence of debris accumulation on the redistribution of flow and heat transfer in the core.

This computationally demanding model and analysis was run on 4800 cores to ensure memory use during execution did not exceed available capacity on any core. No scaling test was done as part of this initiative. The associated run statistics, provided in Table 1, illustrate the execution time. At the termination of the run, only half of the decay heat was transferred out of the vessel through boiling (vapor generation), clearly indicating longer simulations are necessary to reach peak fuel temperatures. Additionally, a more expeditious and realistic simulation could be achieved by applying a more representative initial temperature condition to the fuel rods. However, this simulation demonstrated, with careful consideration of plant accident conditions, i.e, asymmetric flow configuration, decay heat variations in time and in space, Conjugate Heat Transfer (CHT) between solids and the coolant and phase change due to possible boiling, that a CFD model can provide insights of fluid flow and temperature distribution under such complex situations.

Table 1 Run Time information for the Fibrous Debris CFD Simulation

Physical Time (s)	94
Total Solver CPU Time (s	8576077312.0
Total Solver Elapsed Time (s)	4766887.0
Average CPU Time per Iteration (s)	9.48

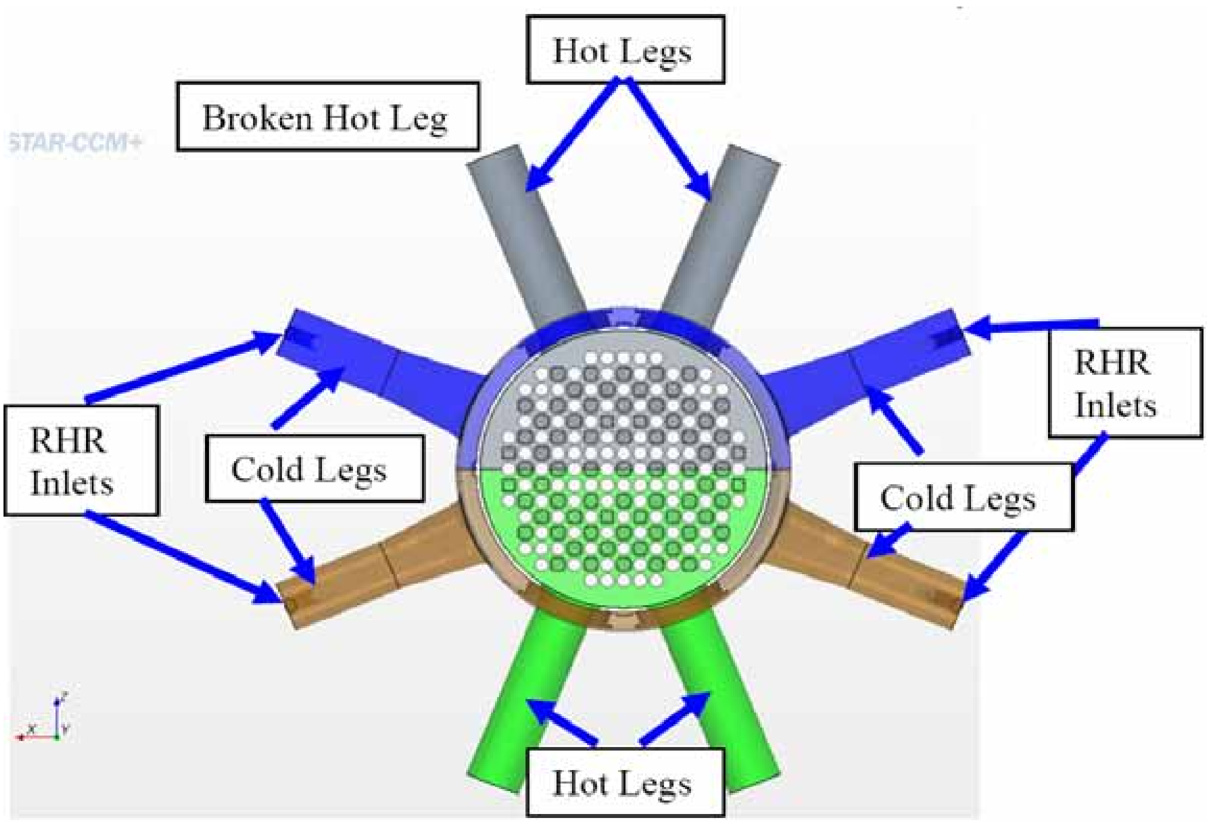

Figure 1 Hot and Cold Leg with RHR Inlet Configuration

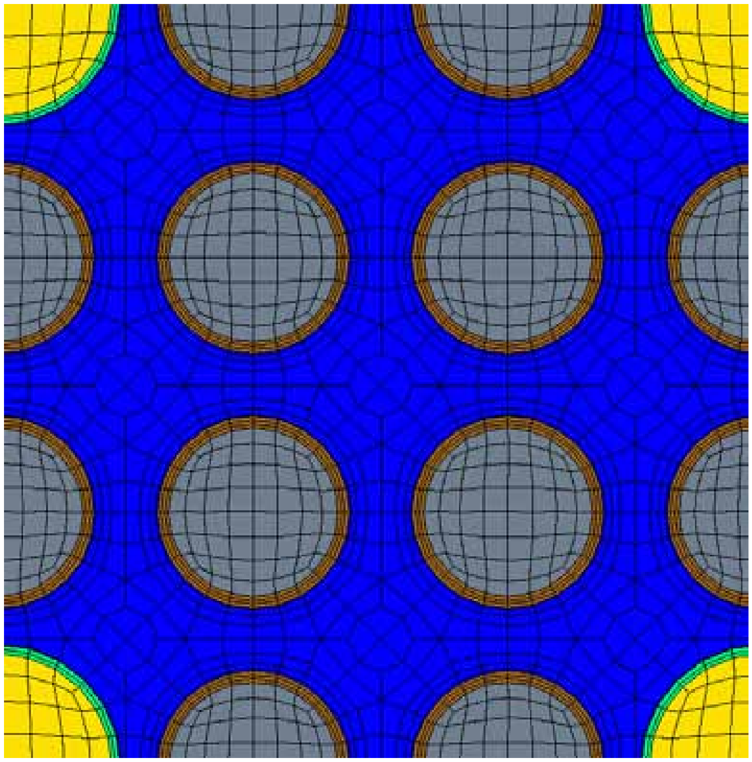

Figure 2 A typical mesh in the core region

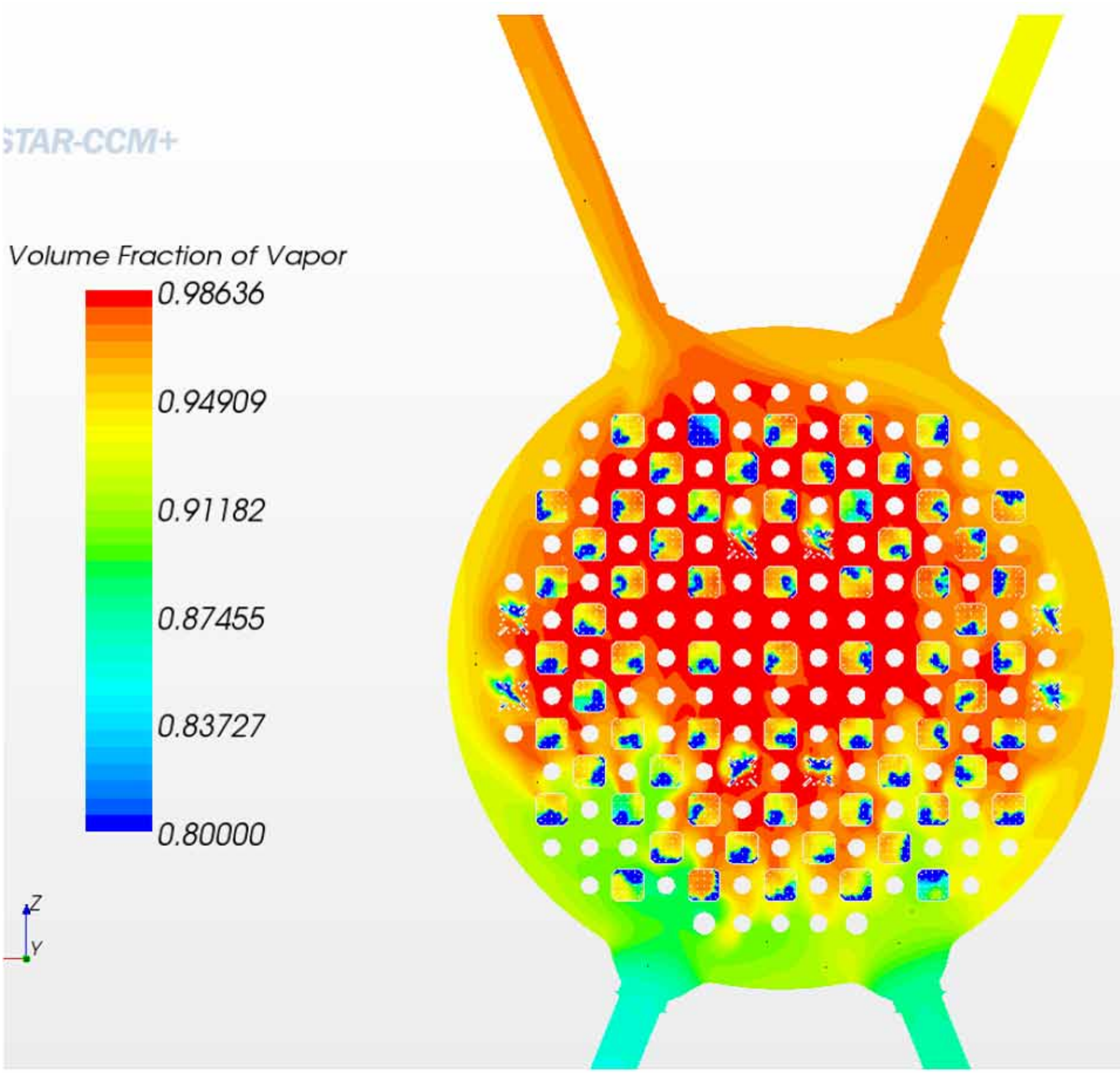

Figure 3 cross-section of the reactor core with the volume fraction of vapor at 94 seconds into the LOCA event (There is some liquid remaining in the guide tube assemblies at this moment).

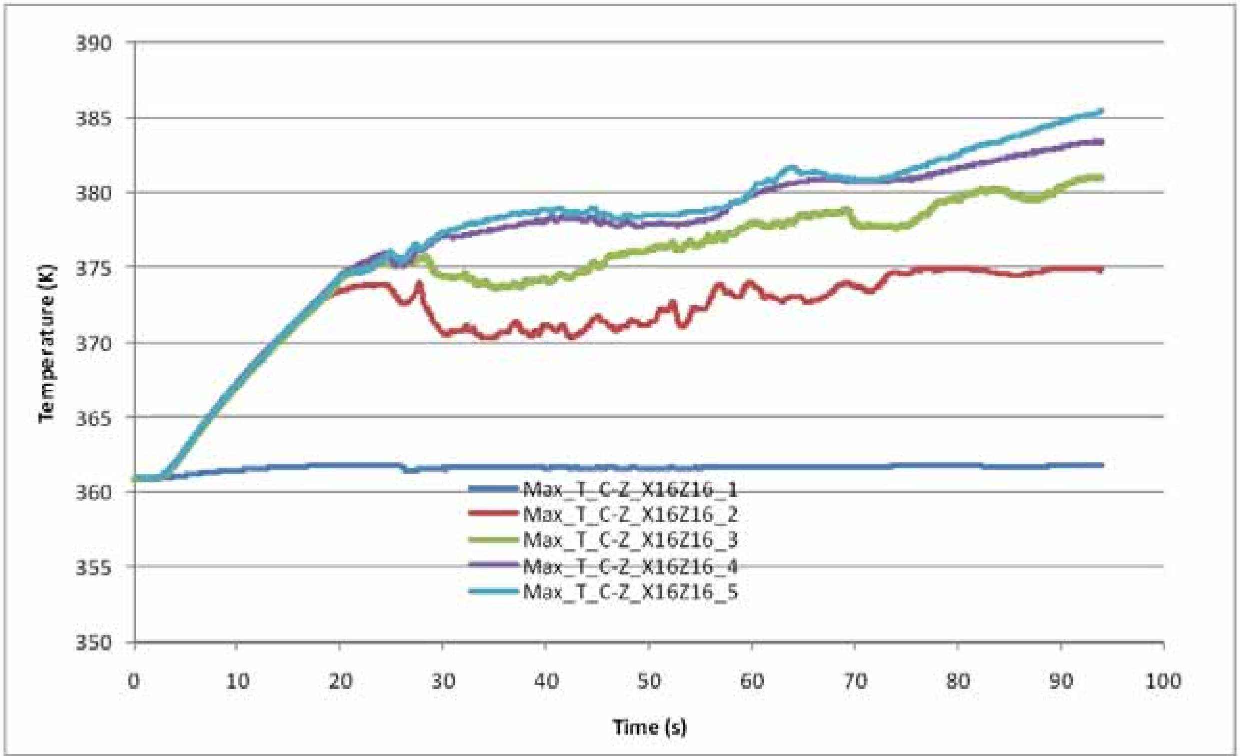

Figure 4 Cladding Outer Surface Temperature of Rod X16Z16 at various Elevations

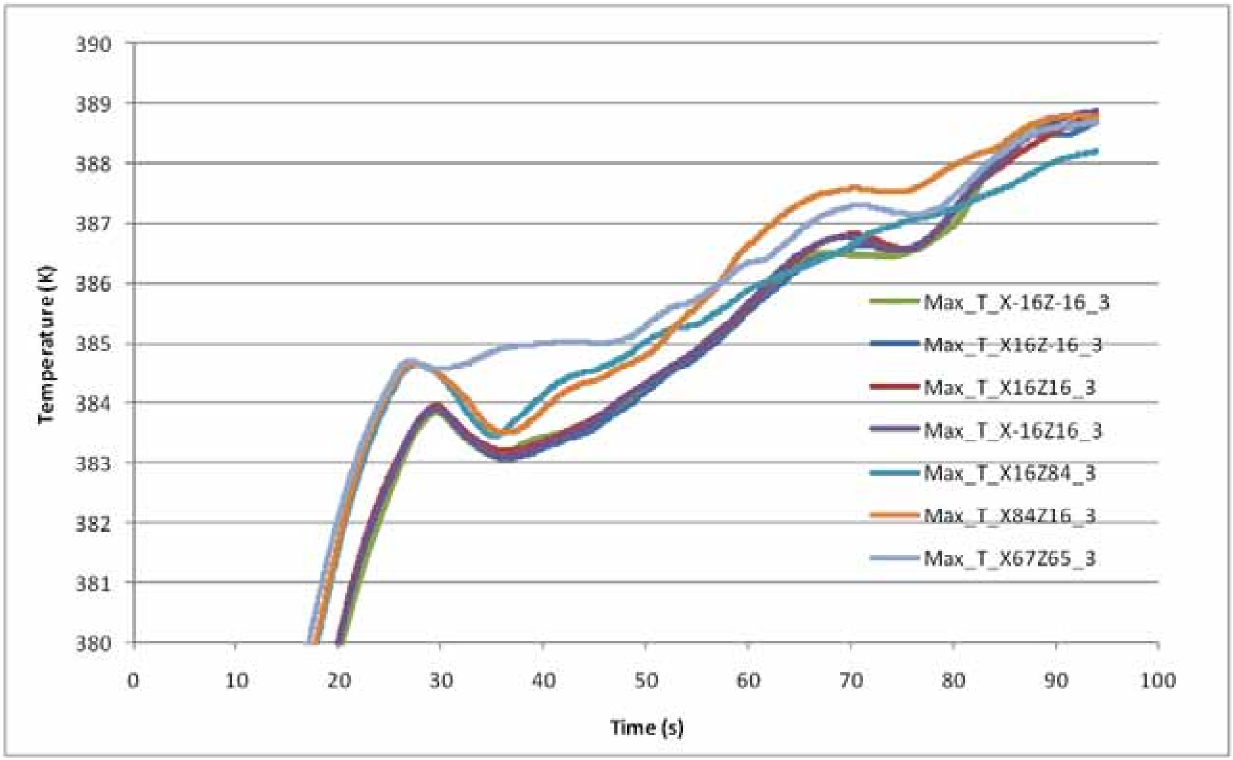

Figure 5 Fuel Centerline Temperature of Rod X16Z16 at various Elevations

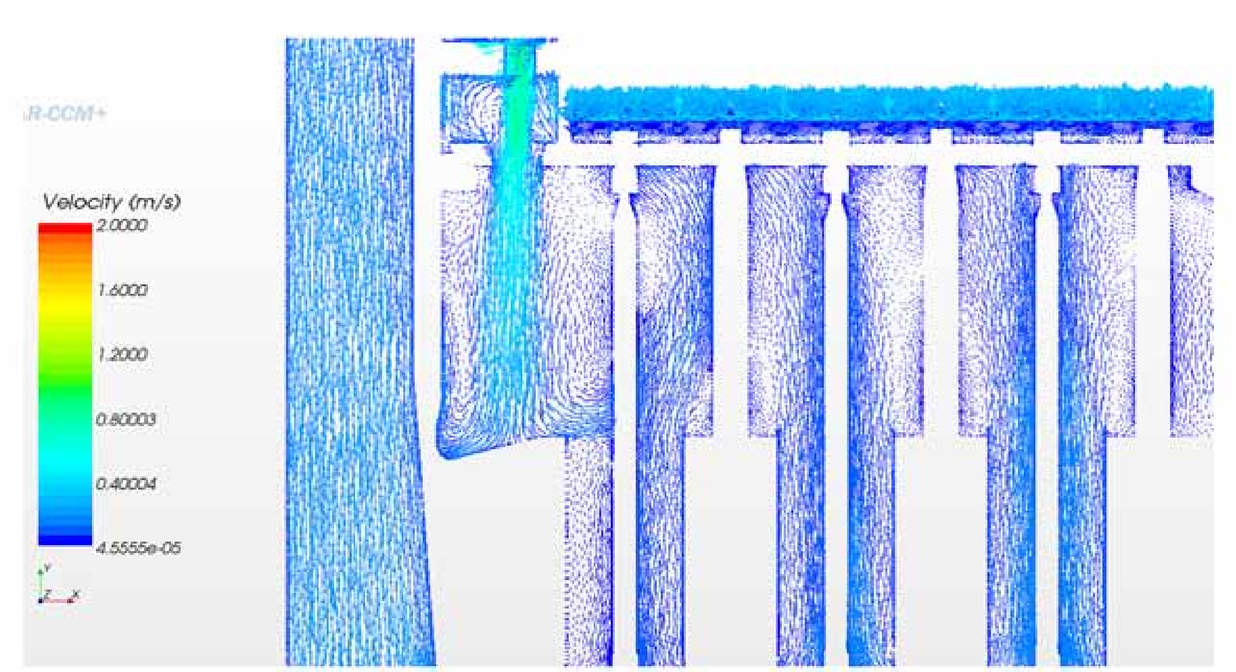

Figure 6 Recirculation jets through the Former Plates near the Core Inlet during Condensation and Recirculation (94 seconds)