

# Core Exit Thermocouple Response to Inadequate Core Cooling Using CFD Dynamic Mesh

**Liping Cao, Josh Hartz, Hong Xu and Justin Figley**

Westinghouse Electric Company LLC

1000 Westinghouse Drive, Suite 228

Cranberry Township, PA 16066

caol@westinghouse.com; jhartz@westinghouse.com; xu1h@westinghouse.com;

figleyjt@westinghouse.com

## ABSTRACT

The CFD study presented in this paper simulates the thermal response of a Westinghouse Incore Instrument Thimble Assembly (IITA) core exit thermocouple (CET) to the core uncover and heat up for a nuclear power plant loss of coolant accident (LOCA) scenario. The IITA thermocouples are used to monitor core cooling in pressurized water reactors. They represent new generation of these instruments and are inserted in the middle guide/thimble tube of a fuel assembly. During some LOCAs, the CETs can indicate high core exit temperature in response to rapid heatup from reactor core uncover. Because they are not in direct contact with the reactor coolant, there are potential time delays associated with the local CET temperature indications.

The analysis in this study predicts the time delay associated with the CET measurement using a 3 by 3 fuel rod array model with a guide/instrument tube containing an IITA located in the center. The model simulates the uncovered portion of the rod array cooled only by single phase vapor evaporated from the two-phase mixture present below the uncovered portion of the fuel rods. As the two-phase mixture level decreases during the core boil off transient, the model domain expands accordingly using the ANSYS® FLUENT® dynamic mesh. The transient expansion of the mesh, as well as the fuel rod power decay and the steam flow rate are defined by the known data resulting from a LOCA transient simulation using a thermal hydraulic system code. The results show that the CET measured temperature lags the coolant vapor temperature of about 1000 °F with as much time delay of >200s. The transient period of interest studied is from just prior to core uncover to when the mixture level reaches the approximate mid-plane of the core. The core heat up predicted by the CFD simulation matches that obtained from the thermal hydraulic system code simulation.

## KEYWORDS

CFD, Core Exit Thermocouple, Dynamic Mesh, Inadequate Core Cooling

©2015 Westinghouse Electric Company LLC.

All Rights Reserved.

*ANSYS, ANSYS Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.*

## 1. INTRODUCTION

The Core Exit thermocouples (CET) are used worldwide in nuclear industry in the accident management (AM) of pressurized water reactor (PWR) power plants [1]. They are the indications most directly related to the reactor core cooling condition and help detect the fuel rod temperature excursion during the progression of an inadequate core cooling accident, such as in a loss of coolant accident resulting from the break of the reactor coolant pressure boundary. Typically, the plant AM package contains procedures and guidance associated with the CET indications (set points) of the reactor coolant system (RCS) thermal hydraulic conditions, e.g. the subcooling margin, the loss of subcooling or onset of saturation and the degree of superheating. Thus, the definition of the CET set points plays important role in initiating AM actions to effectively restore the reactor core cooling and mitigate the consequence of an accident. The ultimate goal of these AM measures is to ensure the safe operation of nuclear power plants and protect the environment and public health.

It is obvious that the accuracy of the CETs as indications of the reactor core cooling conditions has direct impact to the effectiveness of the pertinent preventive and protective AM actions. Particularly it is important to be aware that CETs respond to the temperature variations of the reactor coolant and fuel rods with significant delays due to poor heat transfer of the low velocity vapor in inadequate core cooling accidental transients. Significant delays were observed in multiple experiments [2-7] investigating the behavior of the CETs in some accident scenarios, which were attributed to factors such as the three dimensional profiles of the flow and temperature above and at the upper part of the core, thermal non-equilibrium between the vapor and liquid and specific mounting configuration of the CETs [1]. As a consequence, it is crucial to take into consideration pertinent physical process/phenomena causing the CET delays when defining the associated set points for the AM measures. The definition of the CET set points should account for not only the instrument uncertainties/bias, but also the process measurement accuracies based on sound and validated technical models.

In this paper, the CET response is studied for the specific Westinghouse CET design in which a CET is embedded in an Incore Instrument Thimble Assembly (IITA). Since the CET is enclosed in sealed IITA sheathe, it is not in direct contact with the reactor coolant, thus temperature difference exists between the CETs and reactor coolant whenever coolant temperature varies in a transient. For the current study the limiting (worst case) transient in terms of the CET time delay or temperature difference, is identified as the boil-off period during a small break loss of coolant accident (SBLOCA) of a PWR power plant based on the fact that significant delays are expected for the CET to respond to the heat up of the uncovered core due to the poor heat transfer of the low speed vapor flow in this transient.

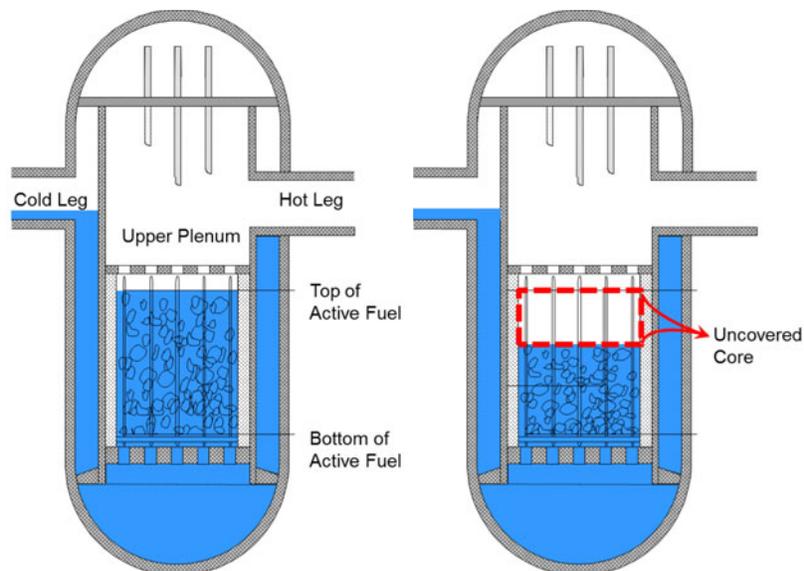
The simulations performed to fulfill the current study include the prediction of the global RCS transient behavior using the system thermal hydraulic code as well as the CFD modeling of the local region where one of the CET is located. The typical system code performs accidental simulation for the entire RCS system with coarse computational nodes much larger than the geometry scales needed to conduct accurate prediction of an individual CET and local regions impacting its behavior. On the other hand, CFD has the advantage of resolving the detailed 3D geometries using fine mesh however limitations exist on resolving the system global behavior under accident conditions. Thus, an approach is taken in this study to make the best use of the

capabilities of simulation tools at different levels of resolution for the improved robustness of the modeling results. This paper presents only the CFD portion of the simulation which couples with the system code through the use of system code results to define the CFD model boundary/initial conditions.

For the purpose of predicting the CET response, it is apparent that the CFD transient and geometry model only need to be subsets of the entire accident transient when and where the core uncover occurs. Therefore, for the CFD study, the transient period of interest is from just prior to core uncover to when the mixture level reaches the approximate mid-plane of the core and CET temperature reaches the typical Westinghouse plant set point value of about 650 °C for an inadequate core cooling scenario.

## 2. SIMULTIAON TECHNIQUES AND ASSUMPTIONS

The physical process of importance for this study is the core boil off period in a plant SBLOCA transient. Just prior to the boil off when core uncover happens, the core is filled with liquid-vapor two phase mixture due to heat addition from the fuel rod decay power and continuous depressurization of the RCS system by losing mass to the containment. The subcooled/saturated nucleate boiling prevails in the core region where superior heat transfer exists between the powered fuel rods, coolant and unpowered guide thimble tube. Afterwards, the two-phase mixture level decreases because the liquid supply (high pressure safety injection) to the core is less than the amount evaporated by the fuel rod decay heat. The uncovered top portion of the core is thus heated up due to the poor heat transfer between the single phase vapor and the fuel rods. The core heats up more with decreasing mixture level until enough ECCS (Emergency Core Cooling System) liquid is injected to recover the core cooling. Figure 1 schematically shows the reactor fluid distribution at the beginning (left) and during the period of (right) boil off.

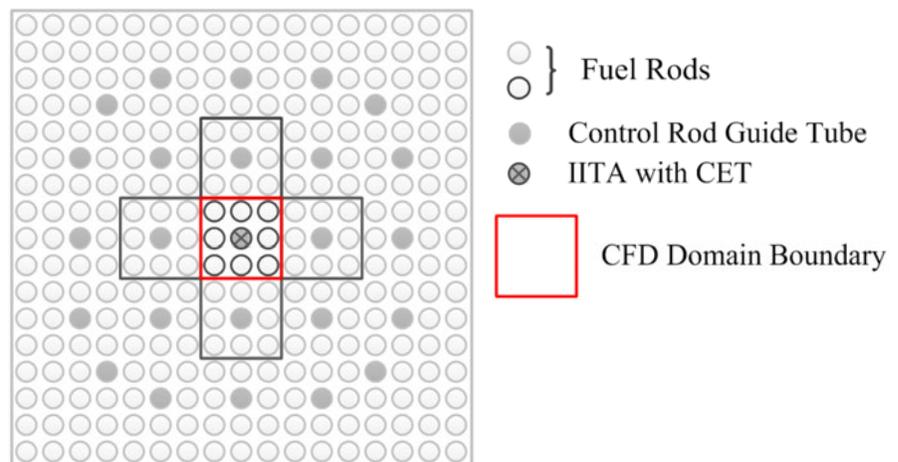


**Figure 1. Schematics of the Reactor Coolant Distribution during Boil-off of a SBLOCA**

To investigate the CET response to the core heat up using CFD, the proper model and time domains relevant to CET behavior of the interest need to be determined first. As the predicted RCS behavior is readily available from the system code results, the CFD simulation can limit its model and time domain within boundaries of known values in computationally realistic and economic ranges to investigate the CET transient behavior resolving the fine details of local 3D geometry.

First, the time domain of the CFD study starts at when reactor core is about to uncover, and continues until high fuel rod/CET temperature is reached with almost half of core uncovered. This period is part of the cold leg SBLOCA accident sequences simulated using the thermal hydraulic system code with input models constructed for all relevant components in the entire RCS system. The SBLOCA transient covers the time period from when break occurs to when the core is safely recovered by the ECCS actuation. With the initial time selected for the CFD model, the core region coolant fluid is at the system pressure (known from system code results) and its saturation temperature; the temperature of the both powered and unpowered solid components in the core is also close to the saturation temperature due to high heat transfer coefficient in nucleate boiling regime.

For the model geometry, the domain can be limited to a single CET and surrounding fuel rods and coolant region to calculate the most delayed CET response in the limiting conditions. In a fuel assembly (FA) with IITA guide thimble in the middle, the center region of 3 by 3 rod array can be chosen with approximately symmetric boundaries around it, taking into consideration of where the IITA is located in a fuel assembly (FA) relative to other unpowered guide tubes and powered fuel rods. The top view of the selected model domain (bounded by red square in the center) is illustrated in Figure 2, in which the gray solid circles represent the guide tubes and empty grey and black circles the fuel rods.



**Figure 2. Top View of the CFD Model Domain in a Fuel Assembly**

Axially along the length of the rod, the top of the model (outlet) is where CET is located and is 5.0 inches (0.127 m) above the active fuel rod elevation. The bottom of the model (inlet) is defined at the two-phase mixture level, which is initially at the top of the active fuel rods and moving downward as the core uncovers more during the simulated transient. By defining the moving inlet boundary this way, the benefit is, on one hand, only single phase vapor heat transfer in the uncovered portion of the core needs to be included and two-phase flow and heat transfer simulation with much uncertainties are avoided in the CFD simulation; on the other hand, the inlet vapor flow and temperature are known from the system code calculation.

The simulations using the CFD and thermal hydraulic system code supplement each other to cover broader range of resolutions of the physical phenomena from the system global behavior to the local 3D CET thermal response to the coolant and fuel rod conditions. Moreover, the models of the system code were validated and licensed for predicting the plant SBLOCA accident transient, thus their results provide validation to the CFD results which potentially involve large uncertainties attributed to the CFD turbulence models and mesh density of the input geometry.

The following approximations and simplifications are made to set up the input model and simulation:

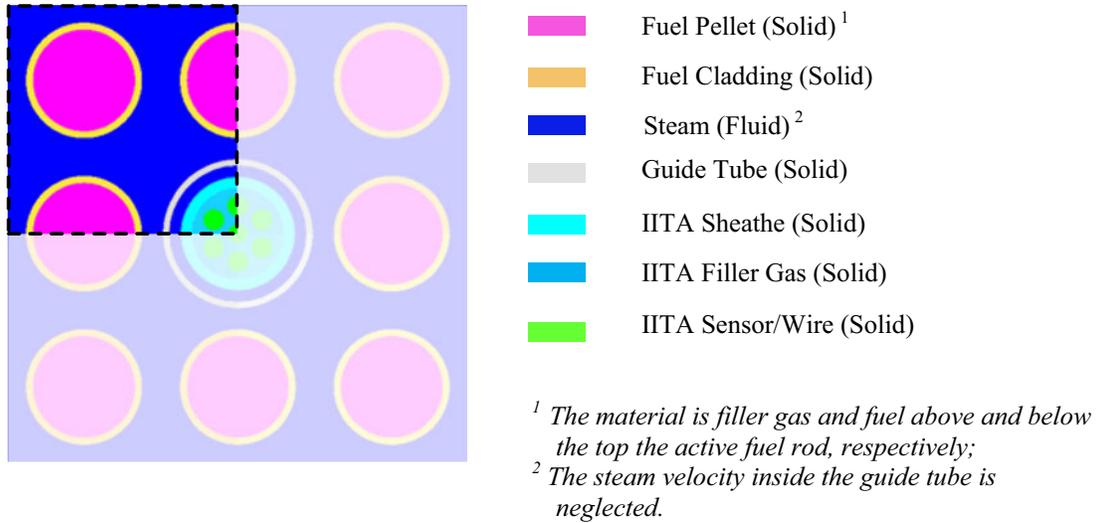
- Although the ideal symmetrical boundaries is one eighth of the FA square along the diving lines through the center of the square, approximately symmetrical boundaries are assumed around the center 3 by 3 rod/thimble array to compromise the computer time needed to complete the desired transient period;
- The fuel rod cladding and fuel pellet deformation are neglected during the heat up period simulated;
- The FA grid spacers are not included in the model to simplify the model geometry. Neglecting the grid spacers and their enhancement to both the heat transfer and mixing is considered conservative and acceptable in the study;
- Incompressible flow due to low vapor velocity;
- Radiation heat transfer is conservatively neglected.

### **3. INPUT MODEL AND BOUNDARY CONDITIONS**

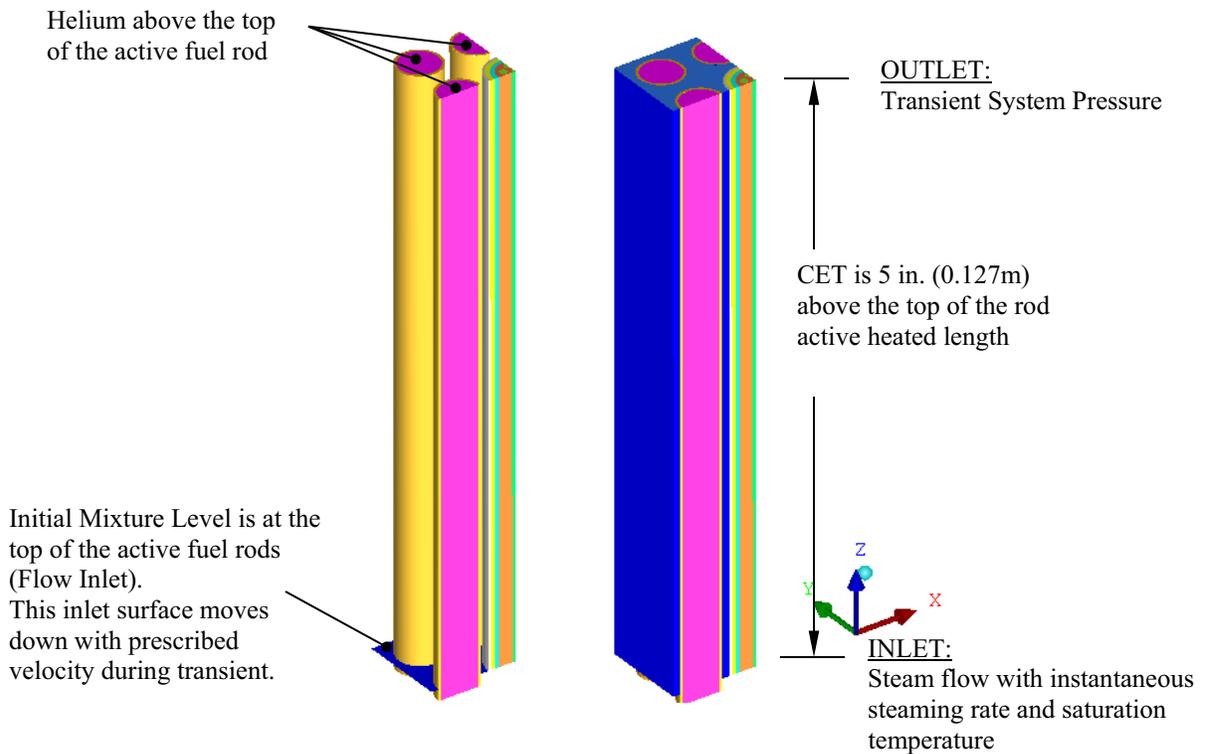
#### **3.1. Input model and Mesh**

Due to symmetry, the center 3 by 3 rod/thimble array is modeled with one fourth of the region as shown in the top view of the model regions in Figure 3. The regions only heat conduction is modeled include the fuel pellet (with heat source defined based on the transient decay power), fuel cladding, guide tube, the steam inside the guide tube (steam velocity is neglected for conservative heat transfer), IITA sheathe, IITA filler gas and IITA CET and other sensors. The only fluid region is the coolant vapor where convective heat transfer with the fuel cladding and thimble tube surfaces is modeled.

The input model is further displayed in Figure 4 to show different regions in 3D view and its axial length at initial time ( $t = 0.0$  s).



**Figure 3. Top View of the Input Model (Unshaded Upper Left Corner)**



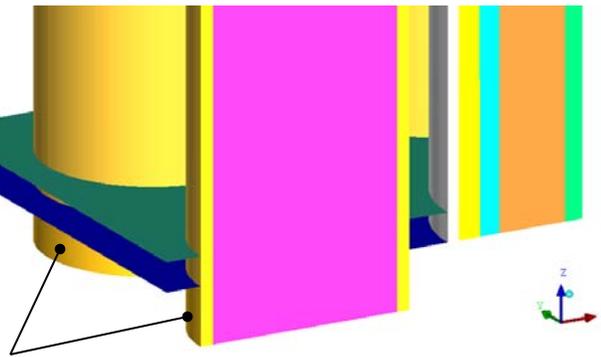
**Figure 4. Input Model Regions and Boundaries at Time = 0.0 Second**

Figure 5 illustrates detailed geometry setup at the inlet region and explains how dynamic mesh is defined to simulate the expanding upper core region with moving inlet surface.

Interior faces are defined at this elevation to separate the upper cells that are stagnant in dynamic motion from the bottom cells that move with the mixture level at a prescribed downward speed

The two-phase mixture level where flow inlet boundary conditions are defined

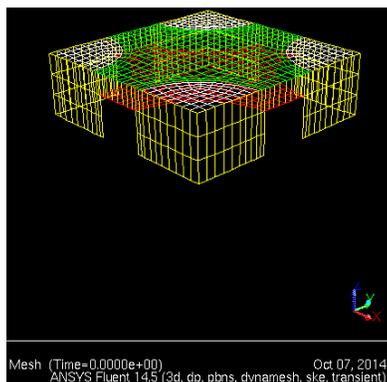
The extended rod regions to define the cooled material below the two-phase mixture level



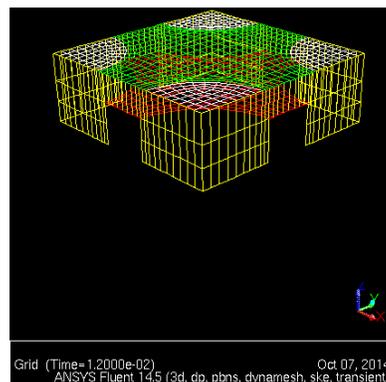
**Figure 5. Input Model Setup at the Inlet Regions**

The input model is meshed with hexahedral cells and the capability to expand the meshed volume is implemented using ANSYS® FLUENT® solver dynamic mesh feature used to model flows where the shape of the domain changes with time due to movement of the domain boundaries [8]. Specifically, the modeling method of ‘dynamic layering’ is adopted to add layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface. The layer of the cells adjacent to the moving boundary is split once the height of the adjacent layer grows to a ratio specified by the user input. Details of the feature can be found in the FLUENT® solver theory and user manuals [8-9].

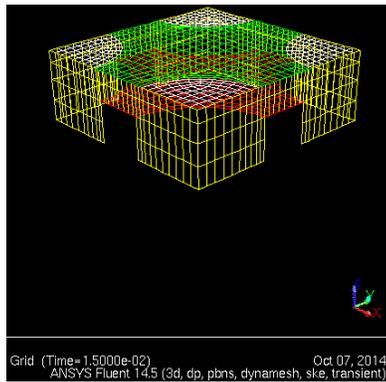
Figure 6 displays in sequence how the dynamic mesh changes with time and the growth and split of the cell layer once the ratio limit is hit, using an example model similar to the input model but only includes one subchannel and adjacent rods/tube.



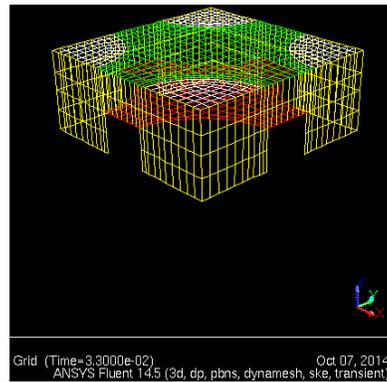
Time = 0.0 (s); Original mesh with stationary interior (white and green), side symmetry (yellow) and flow inlet (red) faces



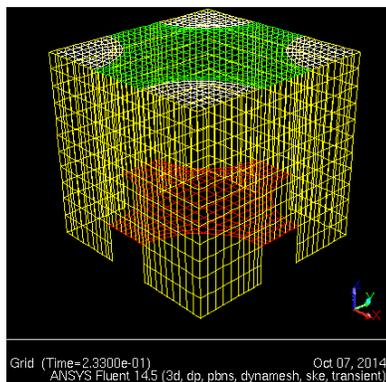
Time = 0.012 (s); The thickness of the first layer below the stationary interior faces grows, flow inlet face moves down accordingly



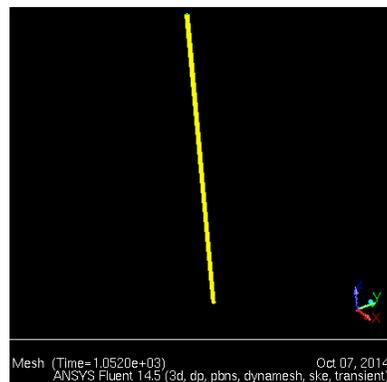
Time = 0.015 (s); The first layer below the stationary interior faces splits at 1.4 times of the thickness of the original, and a new layer inserted



Time = 0.032 (s); The added layer grows and splits, the process repeats itself to add more layers so the flow inlet moves down



Time = 0.233 (s); A total of 11 layers below the stationary interior faces (added 8 layers to the original 3 layers)



Time = 1052 (s); At the end of the transient, the flow inlet face is 60.2 inches (1.529 m) below the original position.

**Figure 6. Illustration of the FLUENT® Dynamic Mesh**

### 3.2. Boundary Conditions

The boundary conditions are mostly defined by the known transient results from the system simulation. During the simulation transient, the key system parameters varies following the accident sequences, including the system pressure, fuel rod decay power, steaming rate at the two-phase mixture level and the decreasing of the core mixture level. Besides, the fuel rod axial power profile is simulated in consistency with the system simulation.

Transient profiles of the CFD boundary conditions are done in general by curve-fitting of the system code results, with smoothing of oscillations in steaming rate which are typical in system

simulations due to the unstable nature of the two-phase flow and heat transfer. The high frequency oscillations of the core steaming rate are smoothed out as shown in Figure 7 so that only the general transient trend on the time scale of the CFD transient is followed. In addition, the surge of steaming rate around time = 400.0 seconds corresponds to the mixture level surge (Figure 8) and it is decided not to capture this behavior to simplify the dynamic mesh motion while predict the results with conservatively higher heat up of the core and worse CET response.

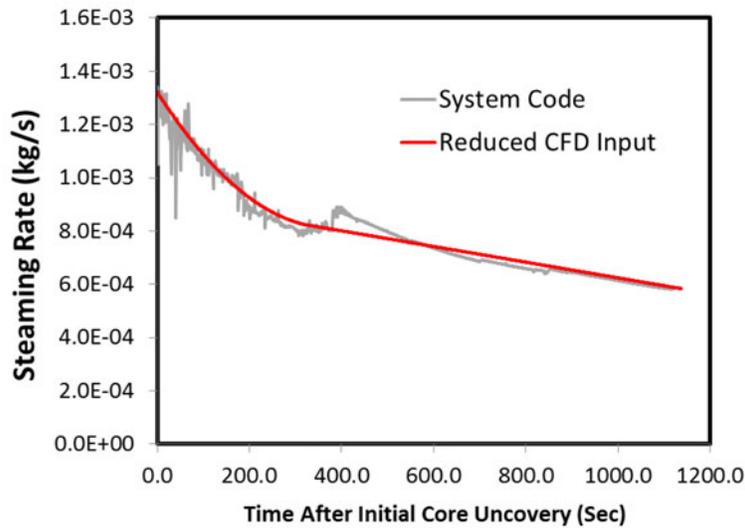


Figure 7. Reducing the CFD Inlet Flow Rate from the System Code Results

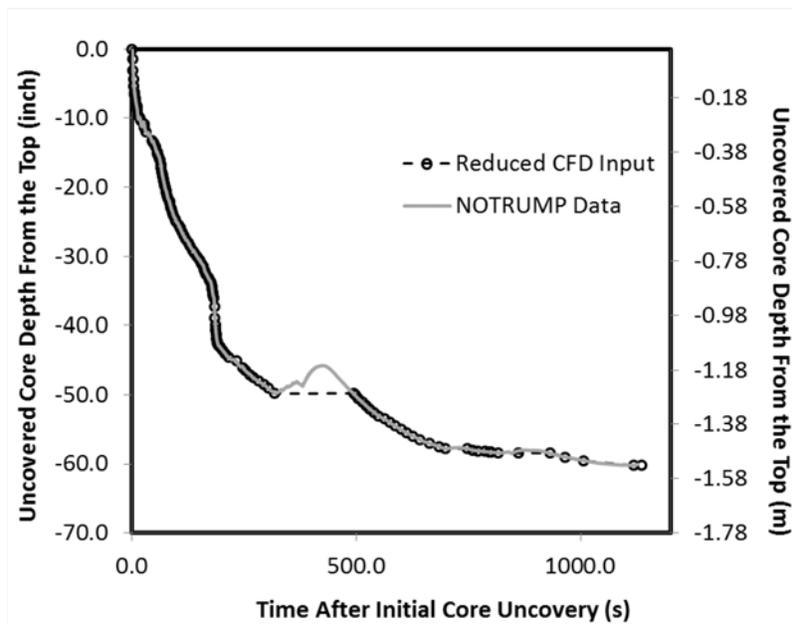
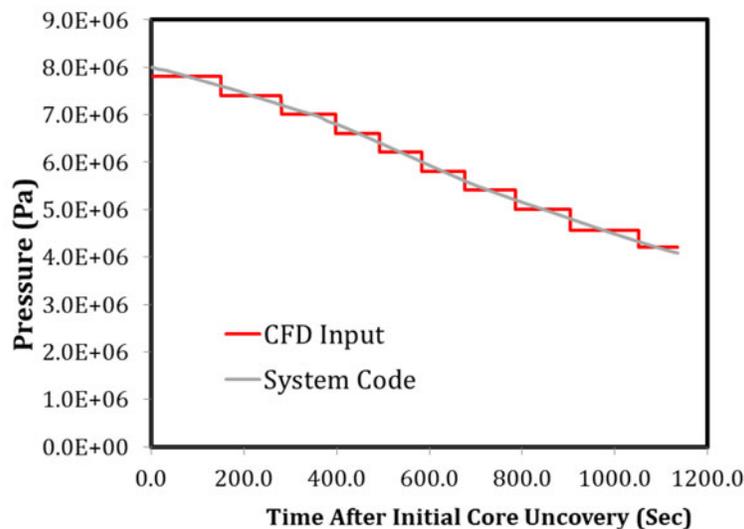


Figure 8. Reducing the CFD Inlet Surface Position from the System Code Results

For the low flow rate in this study, it makes sense to simulate the vapor flow assuming it is incompressible as it saves significant wall-clock time of the computer execution of runs. The computer execution time required for the desired physical time of about 700 seconds for this study is approximately two weeks on Westinghouse's computer system, thus it becomes important to consider saving of run time. It was found in the preliminary runs that the time saving is significant with incompressible flow assumption. Nonetheless, it is also important throughout the transient to capture the varied vapor density with the decreased system pressure (as function of time) which is not trivial.

As a compromise between the computer running time and capturing important property changes of the flow, the full transient is divided into multiple transient periods within which system pressure is assumed to be a constant value representative of the averaged pressure of the period. Each subsequent period restarts from the previous one with redefined constant pressure and associated vapor density as function of only temperature. By doing so, the incompressible flow is assumed within each period while the variation of the vapor density with system pressure is captured, although the system pressure transient is approximated in this approach by a step-wise function following the global trend (Figure 9).



**Figure 9. CFD Transient Pressure Reduced from the System Code Results**

The transient power of the fuel rods is a function of time and fuel rod axial position. The change is implemented by defining FLUENT<sup>®</sup> user defined function (UDF) called every time step by the solver.

#### 4. RESULTS AND ANALYSIS

The flow and temperature fields in the defined domain in this study are solved selecting the pressure-based algorithm implemented with the FLUENT<sup>®</sup> segregated solver. The standard  $k-\epsilon$

turbulence model with the option of scalable wall function is selected, in an attempt to relax the dependence of the turbulence wall function accuracy on the coarse mesh adopted in this study. Nonetheless, the CFD predicted temperatures were found insensitive to the choices of the wall functions in this simulation.

The predicted temperatures are shown in Figure 10 for CET, vapor, guide tube surfaces and IITA surfaces. There are three indications of the predicted vapor temperatures as labeled in the legends of the plot, representing vapor temperatures at the top elevation averaged over different selection of the subregions at the outlet. The “T\_vapor\_avg.” in the legend is the mass flow weighted average vapor temperature over the total outlet area; the “T\_vapor\_hot” is the mass flow weighted average temperature over the hot subchannel region of the outlet; the “T\_vapor\_cold” is similar but over the cold subchannel region of the outlet. The cold and hot subchannels are indicated in Figure 11 on the outlet surface, as result of the arrangement of the powered fuel rods versus unpowered IITA tube. Other legends in Figure 10 include the “T\_GT\_o” and “T\_GT\_i” representing the outer and inner surface temperatures of the guide tube, respectively; the “T\_IITA\_o” and “T\_IITA\_i” being respectively the outer and inner surface temperatures of the IITA tube. The centerline temperature of the thermo couple sensor is shown as “T\_CET” in Figure 10.

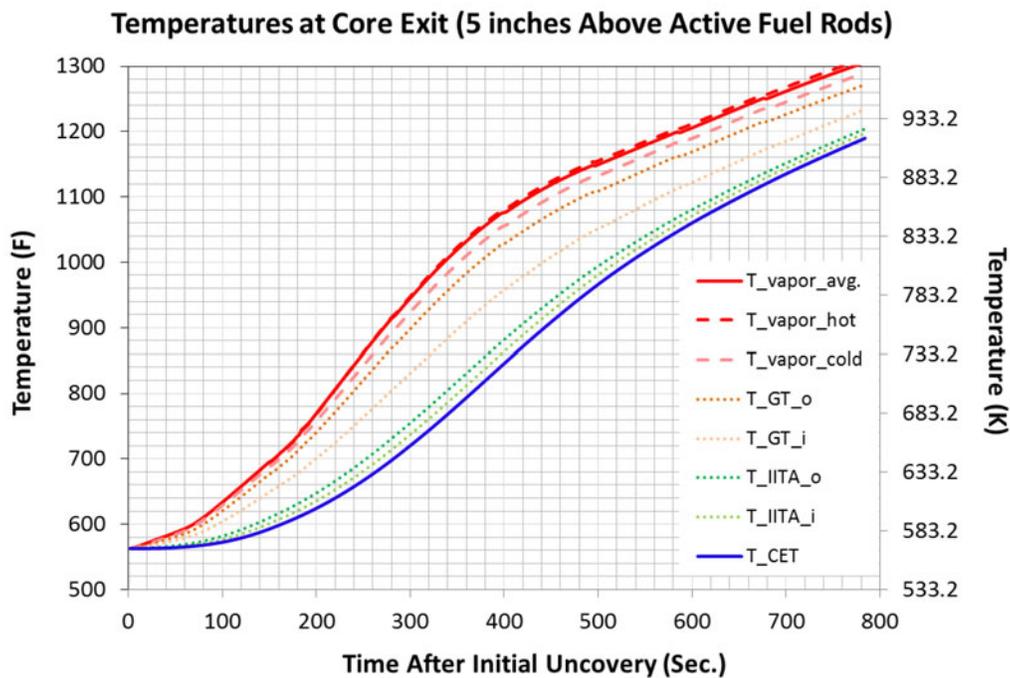
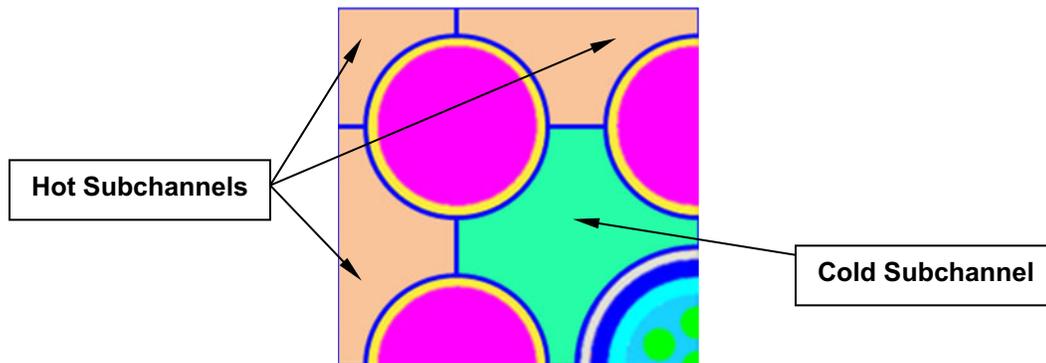


Figure 10. CFD Predicted Temperatures of CET, Vapor and Structures

The predicted temperature curves in Figure 10 indicate significant delay of the CET temperature following the initial core uncover and quick heat up of the reactor core, or equivalently speaking the CET temperature reading is significantly lower than the increased vapor temperature. The delay starts immediately when core begins to uncover and increases first to reach a maximum of about 234 seconds when the hot channel vapor temperature reaches 1130 °F

(610 °C), then the delay becomes slightly less. In terms of temperature difference between the hot subchannel vapor temperature and CET, similar trend follows with a maximum temperature difference of 242 °F (135 °C) at around 360 seconds starting from the beginning of core uncover; afterwards the difference decreases.



**Figure 11. Hot and Cold Subchannels at the Model Outlet Surface**

When assessing the CET process measurement accuracy (PMA) in defining CET set points of EOPs and/or SAMGs, the CET delays at prescribed set point values in terms of temperature difference between the CET and the measured coolant temperature are typically desired. For Westinghouse PWR plants, two of the generic CET set points relevant to current study are ~370 °C and ~ 650 °C , indicating degraded core cooling (DCC) and Inadequate Core Cooling (ICC), respectively.

For the DCC set point, positive measurement uncertainty (positive means CET indicates higher than the measured coolant temperature) needs to be accounted for in the set point definition to initiate relevant EOP action sequences when RCS reaches saturation at the highest possible pressure (3052 psia, or 210 bar) to avoid unnecessary AM actions due to CET uncertainty. For ICC set point in transition from EOP to SAMG, negative measurement error (meaning CET reads lower than the measured coolant vapor temperature) has to be considered so that the initiated AM actions can effectively restore core cooling or mitigate accident consequences.

Based on the results of the current simulation on SBLOCA core uncover transient, the CET PMA for DCC and ICC set points of 370 °C (698 °F) and 650 °C (1202 °F) are -58 °C (-104 °F) and -89 °C (-160 °F) respectively.

The CFD simulation can resolve the local 3D geometry of detailed configuration and structures of the rod and thimble tube array as well as the coolant subchannels, however concerns could arise on the result uncertainties caused by the basic models embedded in the CFD solver and the density of the mesh cells. Therefore, the two concerns specific to the current study are, first, how adequate is the selected turbulence model in predicting the flow and heat transfer in the expected range of flow conditions of the simulation transient; second, the mesh density generated for the

input model in this study is at minimum and no mesh sensitivity is done due to the issues encountered with the FLUENT® solver dynamic mesh when finer mesh is attempted.

To assess the adequacy of the CFD models, the system code results are utilized. This is justified because the system code models were validated against extensive experimental database for the important phenomena/processes during the licensing process. The validation is possible since the average vapor temperature at the FA outlet, which is the ideal parameter to evaluate the accuracy of the heat transfer and turbulence models is predicted directly in the system simulation and can also be derived from the field results in CFD simulations. The FA vapor outlet temperature results from the system simulation is compared with the CFD results in Figure 12 below.

As explained in Section 2, since the boundary and initial conditions are consistent between the system and CFD simulations, the reasonable agreement of the vapor outlet temperature indicates explicitly the adequacy of the CFD models and meshes.

Since the heat transfer of vapor to the fuel rod cladding surface is predicted with reasonable accuracy as supported by comparison shown in Figure 12, it is reasonable to extend the same conclusion on the validity of the predicted heat transfer from the vapor to the unpowered guide tube outer surface.

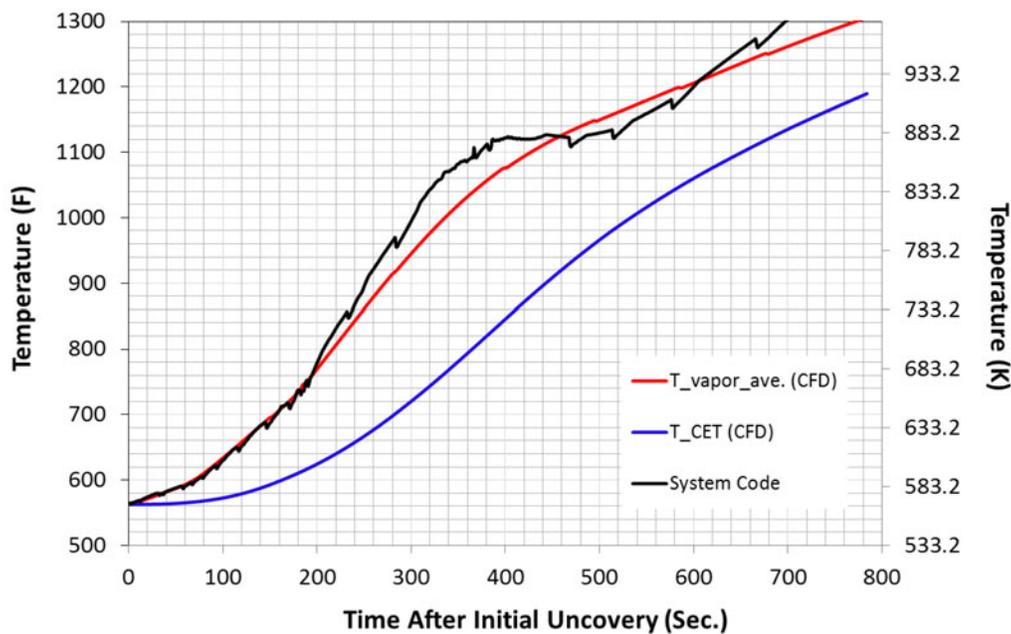


Figure 12. Comparison of the Outlet Vapor Temperature between CFD and System Code

## 5. CONCLUSIONS

The CET response during the boil off period of a PWR SBLOCA accident transient is predicted successfully using advanced dynamic mesh capability of the ANSYS® FLUENT® solver. The results will be included in plant AM packages to account for the CET process measurement uncertainties to improve the reliability and robustness of the AM measures whose ultimate goal is the safe operation of the plant and protection of environment and public health.

The results of the current study show that CETs respond with as much as >200 seconds of delay to the core heat up during the boil off of a SBLOCA accident.

The CFD models of the current simulation are validated using the licensing system code results and proved to be adequate in predicting the important heat transfer process during the studied transient.

## REFERENCES

1. Organisation de Cooperation et de Developpement Economiques, "Core Exit Temperature (CET) Effectiveness in Accident Management of Nuclear Power Reactor," *NEA/CSNI/R(2010)9*.
2. J. P. Adams and G. E. McCreery, "Detection of Inadequate Core Cooling with Core Exit Thermocouples: LOFT PWR Experience," NUREG/CR-3386 (1983)
3. T. Mull, M. Perst and K. Umminger, "Effectiveness of Accident Management Procedures under Small Break LOCA Conditions," "The 5<sup>th</sup> International Conference on Nuclear Engineering (ICONE-5), Nice, France, May 26-30, Paper 2192 pp. 1-8 (1997)
4. K. Umminger, B. Schoen and T. Mull, "PKL Experiments on Loss of Residual Heat Removal under Shutdown Conditions in PWRs," 2006 International Congress on Advances in Nuclear Power Plants (ICAPP'06), Reno, USA, June 04-08 (2006)
5. M. Suzuki, et al., "Performance of Core Exit Thermocouple for PWR Accident Management Action in Vessel Top Break LOCA Simulation Experiment at OECD/NEA ROSA Project," The 16<sup>th</sup> International Conference on Nuclear Engineering (ICONE-16), Orlando, Florida, USA, May 11-15 (2008)
6. M. Suzuki, et al., "Characteristic Response of Core Exit Thermocouples during Inadequate Core Cooling in Small Break LOCA Experiments Conducted at LSTF of ROSA-IV Program," The 2<sup>nd</sup> International Conference on Nuclear Engineering (ICONE-2), San Francisco, California, USA, March 21-24 (ICONE-2), Vol. 1, pp. 63-68 (1993)
7. M. Suzuki, et al., "Reliability of Core Exit Thermocouple (CET) for Accident Management Action during SBLCOA and Abnormal Transient Tests at ROSA/LSTF," Proceedings of the 13<sup>th</sup> International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-13), Kanazawa, Ishikawa, Japan, September 27 - October 2 (2009)
8. "ANSYS Fluent Theory Guide," Release 15.0, ANSYS, Inc. (2013)
9. "ANSYS Fluent User's Guide," Release 15.0, ANSYS, Inc. (2013)
10. "LOCTA-IV Program: Loss-of-Coolant Transient Analysis," WCAP-8301, Westinghouse proprietary report (1974)
11. "Westinghouse In-Core Information Surveillance & Engineering," [http://www.wecng.com/Products\\_&\\_Services/docs/flysheets/NS-ES-0014.pdf](http://www.wecng.com/Products_&_Services/docs/flysheets/NS-ES-0014.pdf) (2008)