338X-F7E6A3J6H9 **NURETH14-338**

ADVANCED THERMAL HYDRAULIC METHOD USING 3X3 PIN MODELING

Z. Karoutas[1], Jin Yan[1], M. Conner[1] and A. Mandour[1]
[1]Westinghouse Electric Company LLC, Columbia, SC
KaroutZE@westinghouse.com

Abstract

Advanced thermal hydraulic methods are being developed as part of the US DOE sponsored Nuclear Hub program called CASL (Consortium for Advanced Simulation of LWRs). One of the key objectives of the Hub program is to develop a multi-physics tool which evaluates neutronic, thermal hydraulic, structural mechanics and nuclear fuel rod performance in rod bundles to support power uprates, increased burnup/cycle length and life extension for US nuclear plants. Current design analysis tools are separate and applied in series using simplistic models and conservatisms in the analysis. In order to achieve key Nuclear Hub objectives a higher fidelity, multi-physics tool is needed to address the challenge problems that limit current reactor performance. This paper summarizes the preliminary development of a multi-physics tool by performing 3x3 pin modeling and making comparisons to available data.

Introduction

A multi-physics tool is needed to address the challenge problems that limit current reactor performance. Some of these key challenge problems include crud/corrosion, grid to rod fretting, pellet clad interaction, fuel assembly bow, departure from nucleate boiling transients, loss of coolant accident, reactivity insertion accident, advanced fuel material implementation and the extension of reactor vessel and internals lifetime. The first step or strategy in developing a single multi-physics tool is to focus the analysis to a small control volume of the rod bundle. This control volume would be a 3x3 pin array. If the neutronic, thermal hydraulics, structural mechanics and nuclear fuel rod performance equations can be properly solved for this control volume then it can easily be expanded to all bundles in a fraction of the core including vessel and internals.

A 3x3 pin model was developed with a mixing vane type spacer grid. In this paper the 3x3 pin model is applied at typical PWR conditions to demonstrate the flow patterns in single phase flow and the rod heat transfer downstream of the spacer grids. To help validate the CFD model the model was compared to PIV (Particle Image Velocimetry), rod bundle test data for single phase flow from the test facility at Texas A&M. The neutronics and thermal hydraulics were linked together by coupling the CFD code, STAR-CCM+ [1] together with the neutronics code, DeCART [2]. A fuel rod heat transfer model was implemented to investigate the impact of rod heat transfer downstream of the grid in single phase flow conditions. The model predictions indicate that there is a significant azimuthal variation in rod heat transfer downstream of mixing vane spacer grids which can lead to excessive crud deposition and localized corrosion as observed in the reactor. Turbulence modeling and rod vibration was also investigated by performing 3x3 pin modeling. This paper shows some preliminary results and provides an introduction or overview of the multi-physics 3x3 pin modeling. Further detail on the 3x3 modeling and test data is provided in referenced papers [3] [4] [5]. These papers are also presented in this conference. Future work includes further development of two-phase flow models, implementing crud and corrosion models, fuel rod performance modeling and benchmarking these models to available test data.

1. Description of CFD Model

The 3x3 pin CAD model is shown in Figure 1 for radial and axial geometries. Figure 2 provides further detail on the axial geometry of the model in one grid span. The fuel pellet, cladding, dimples, springs, mixing vanes, grid strap thickness, and the weld nugget were all included in the CFD model.

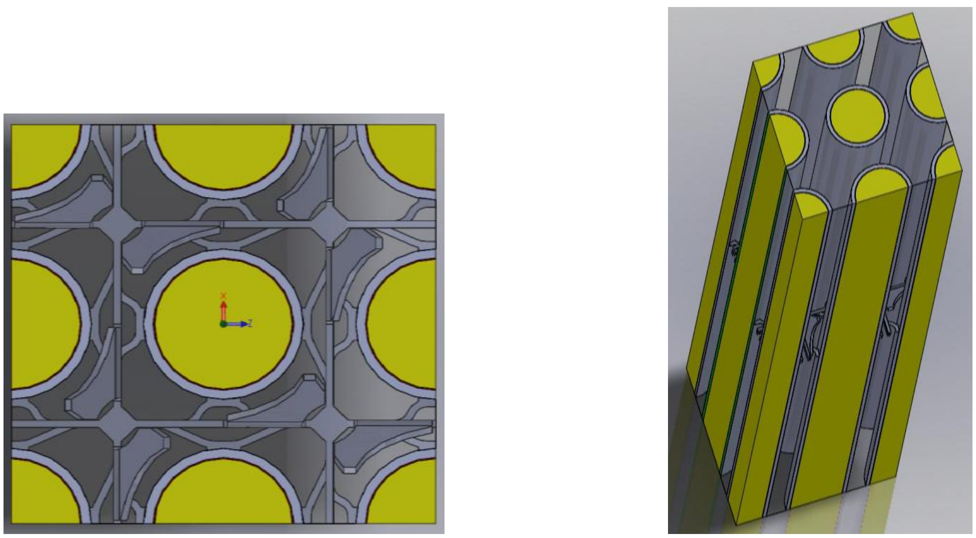


Figure 1 Radial and Axial Geometries of 3x3 Pin Model

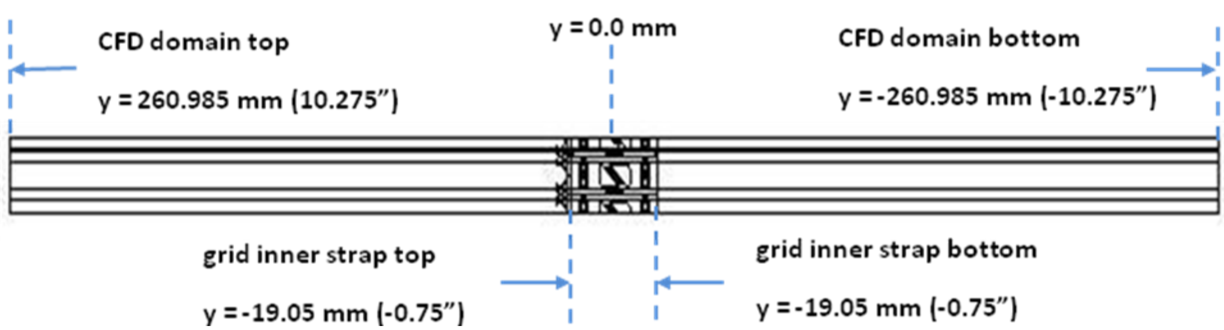


Figure 2 Axial Geometry Details of 3x3 Pin Model in Grid Span

The model contains a 3x3 section of a typical mixing grid used in reactors. From the 3x3 pin CAD model the CFD model is generated using the STAR-CCM+ software. A computational mesh as shown in Figure 3 with 17M hexahedral cells was generated. Sufficient mesh density was used to capture the details of the spacer grid. Periodic boundaries were used on the both sides of the computational domain as shown in Figure 4. The operating conditions were set to be the same as in the test. The meshing strategy and parameters used in the model are from the validations and benchmarks performed in Westinghouse including a mesh sensitivity study. The Realizable k-epsilon turbulence model with high y+ wall function was used. The segregated solver with second order upwind schemes was used and the simulation was isothermal and steady.

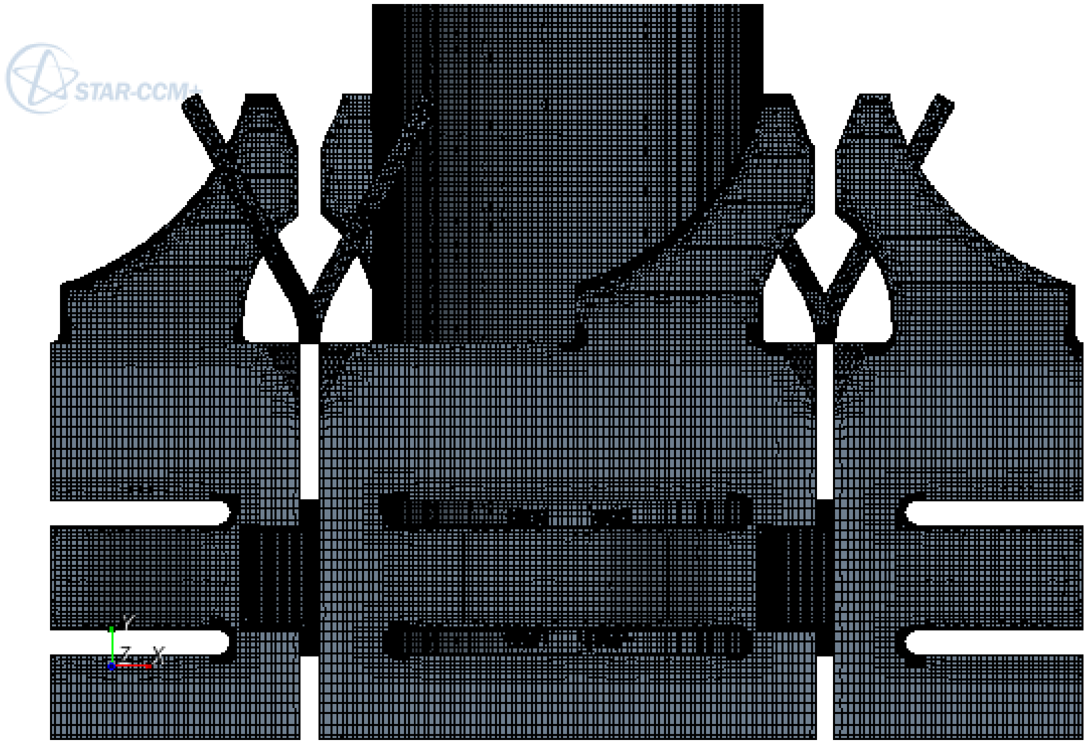


Figure 3 CFD Model Mesh

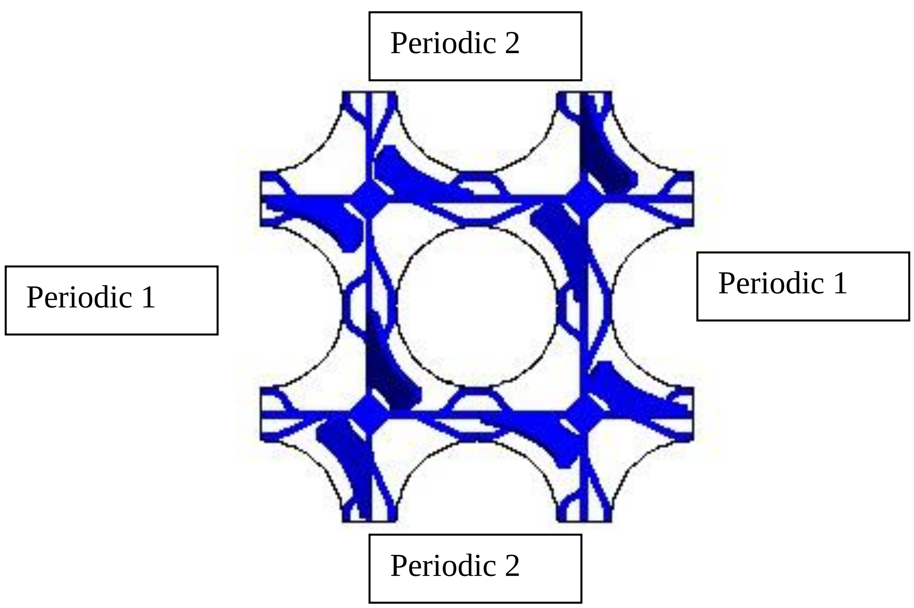


Figure 4 CFD Model Periodic Boundary Conditions

2. Validation of CFD Model to Available Test Data

To provide some validation of the CFD model, comparisons were made to available 5x5 PIV test data for the selected 3x3 spacer grid section. The experiments were performed at the Optical Multi-phase Flow Research Laboratory in Nuclear Engineering Department of Texas A&M University. The experimental facility is shown in Figure 5. One of the unique characteristic of this set-up is the use of the Matched Index of Refraction technique which consists of immersing plastic rods with a similar index of refraction as the one for water to achieve optical transparency zones in the measurement region near the spacer grids. This unique feature allows flow visualization and measurement within the bundle without rod obstruction. This approach also allows the use of high temporal and spatial non-intrusive dynamic measurement techniques to investigate flow evolution below and immediately above the spacer. Further details of this test facility and data are described in reference [3].

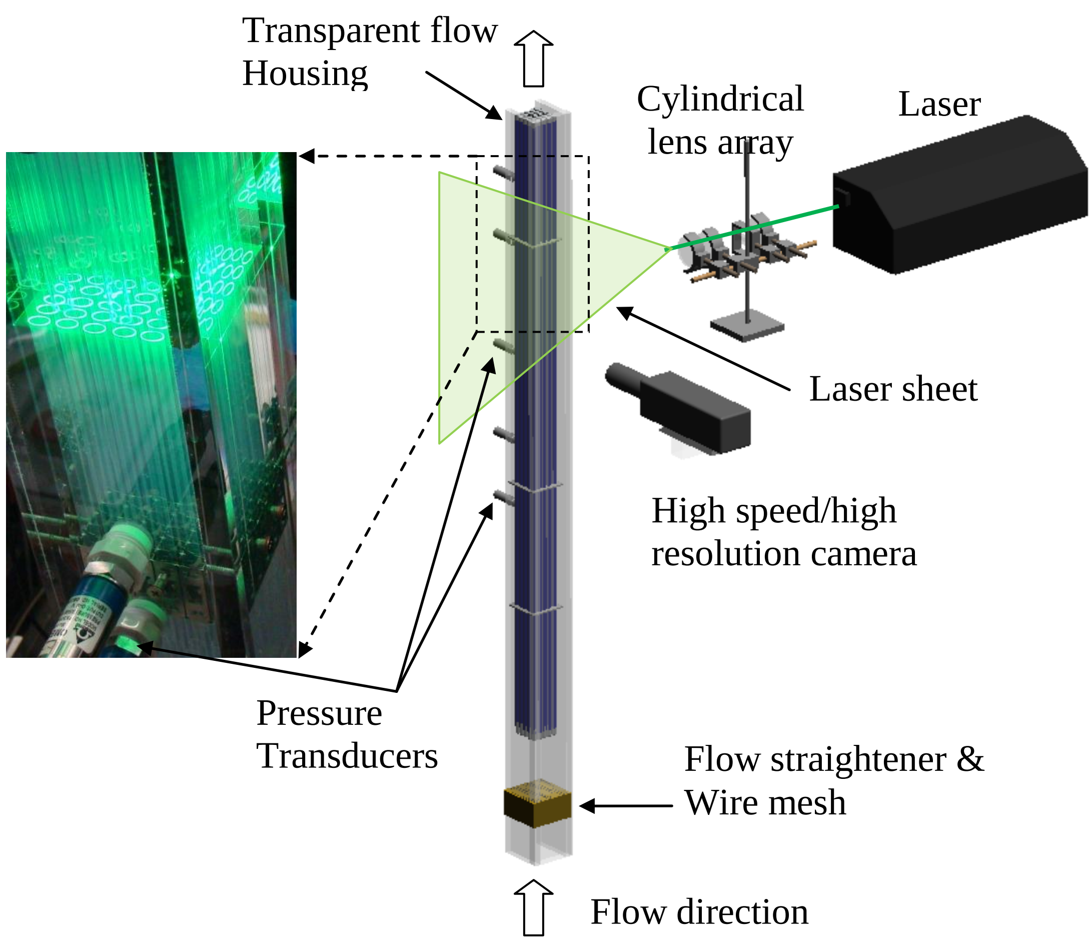


Figure 5 Texas A&M 5x5 PIV Test Facility

The accuracy of the PIV measurements has been estimated based on the testing and data analysis techniques used. Average values from the PIV measurements are calculated from 8000 instantaneous velocity fields yielding total number of about 1,000,000 vectors in a viewing area of

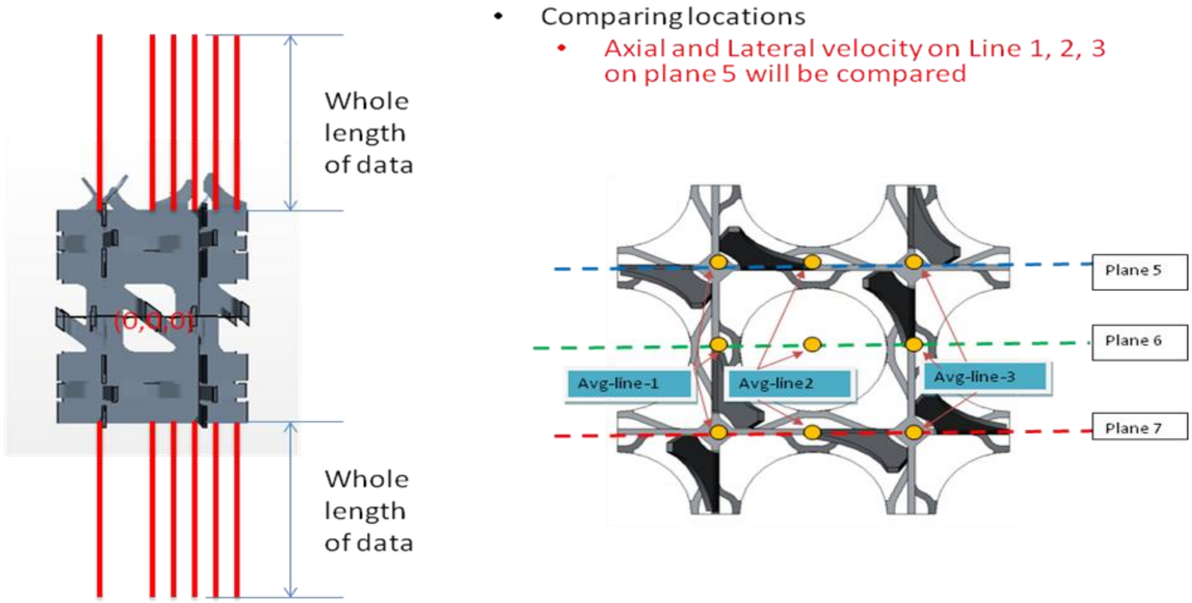
71 mm x 51 mm. The temporal averaging corresponds to about 1.2 seconds of elapsed continuous time with a time resolution of 0.000125 second. The instantaneous velocity fields are obtained using a Particle Tracking Algorithm without interpolation. Therefore, averaging and statistics are calculated from real vectors ordered in a mesh but without using any interpolation scheme.

The resolution of the measurements are in the range of 0.4 to 0.7 mm of spatial resolution, while the time resolution range is 0.125 ms to 0.250 ms. The measurement error is estimated to be 8% and includes uncertainties in particle location, time between laser pulses, and optical distortion in the testing technique.

The CFD model was run based on conditions defined in Table 1 using the axial and radial geometries defined in Section 1 of this paper. Axial and lateral velocity test data was extracted from lines and planes described in Figure 6 for comparing to CFD predictions. The test data was obtained on those planes through laser sheet. Then the data along the lines were extracted from both the test and the CFD for comparisons. The lateral positions of the red lines in the side view are described by the yellow dots in the top view.

Table 1 CFD Model Input Conditions

Turbulence Model	Realizable k-ε turbulence model with high y+ wall
Solid surfaces (rest)	No-slip wall
Fluid inlet	Velocity inlet at 2.44 m/s, and temperature 27 Deg-C
Fluid outlet	Pressure outlet with a constant gauge pressure of 0
Fluid side surfaces	Periodic
Reference pressure	1 ATM



Side View Top View

Figure 6 Locations of Axial and Lateral Velocity Lines/Planes for Comparing CFD Predictions to Test Data

Lateral and axial velocity comparisons versus distance downstream of grid are made between CFD predictions and PIV test data (experimental error is about 8%-15%. The error will approach 15% near the vanes) for Lines 1 to 3 in Figures 7 to 9. Reasonable agreement is observed.

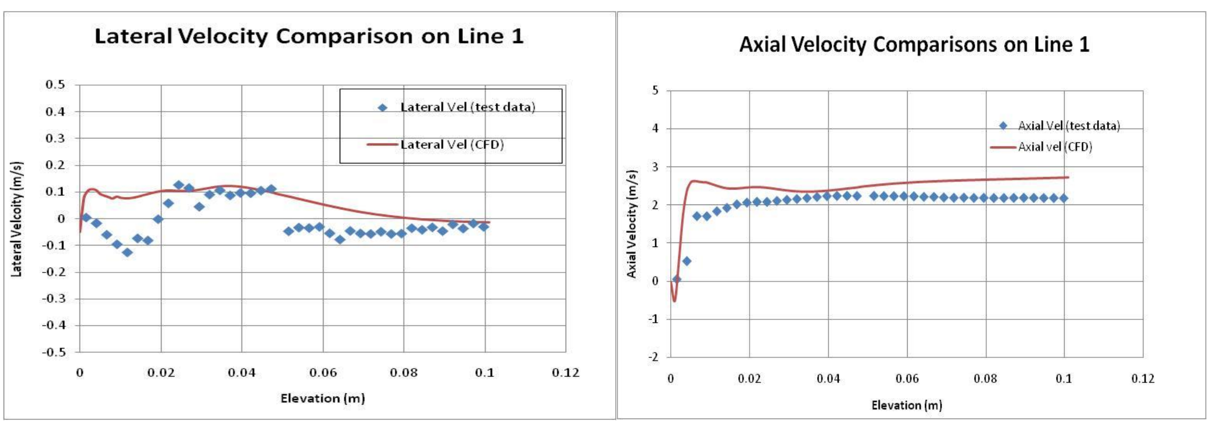


Figure 7 Lateral and Axial Velocity Comparisons for Line 1

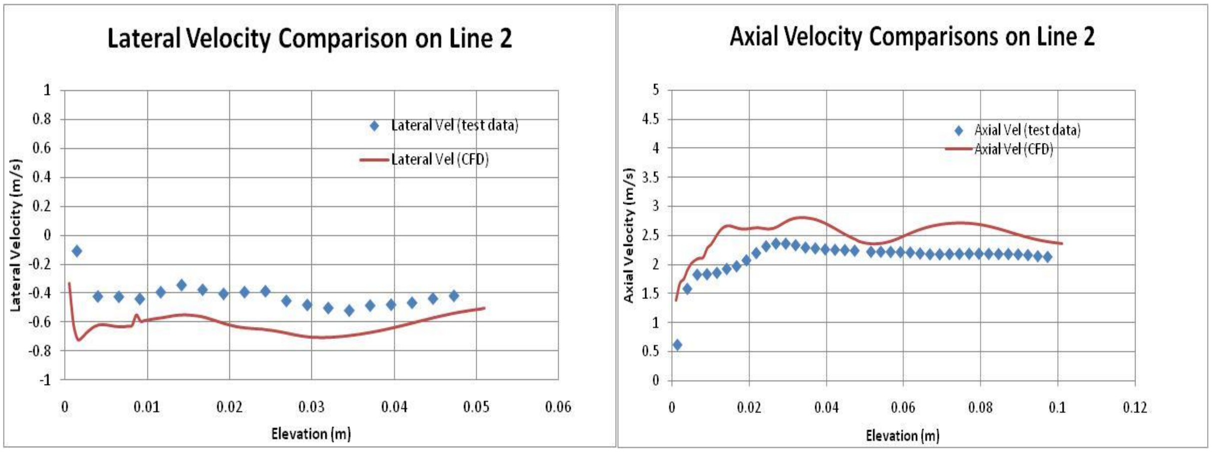


Figure 8 Lateral and Axial Velocity Comparisons for Line 2

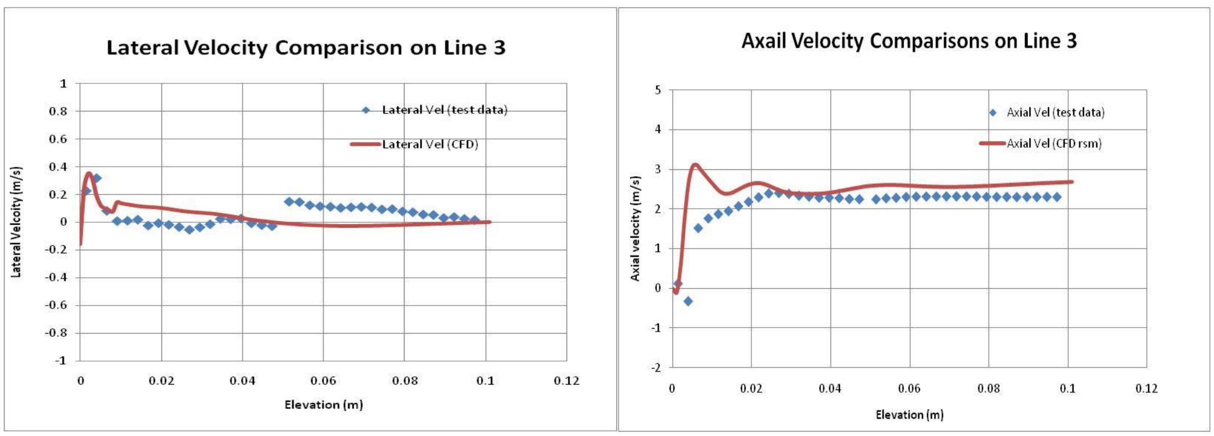


Figure 9 Lateral and Axial Velocity Comparisons for Line 3

3. Neutronics Coupling

To achieve multi-physics coupling the neutronics and thermal hydraulics must be coupled together. This was performed by coupling the STAR-CCM+ CFD and DeCART neutronics codes and models. The 3x3 pin model was extended over the entire length of the fuel rod including all the grid spans. All the grids were simulated as a momentum source except for one grid the model was a detailed one as described in Section 1 to demonstrate the detailed flow patterns downstream of the mixing vanes. The CFD model includes the coolant, cladding, and fuel pellet. The conjugated heat transfer model and Realizable k-ε turbulence model were used. The NIST data base was used to calculate the coolant density. For each coupled iteration temperature, density and enthalpy are passed between the codes. The mesh of the model must also be aligned between the two models as shown in Figure 10 for a typical 3x3 pin arrangement. Figure 11 show typical thermal results from the coupling of the tools. . Further detail on the coupling and results are provided in reference [4].

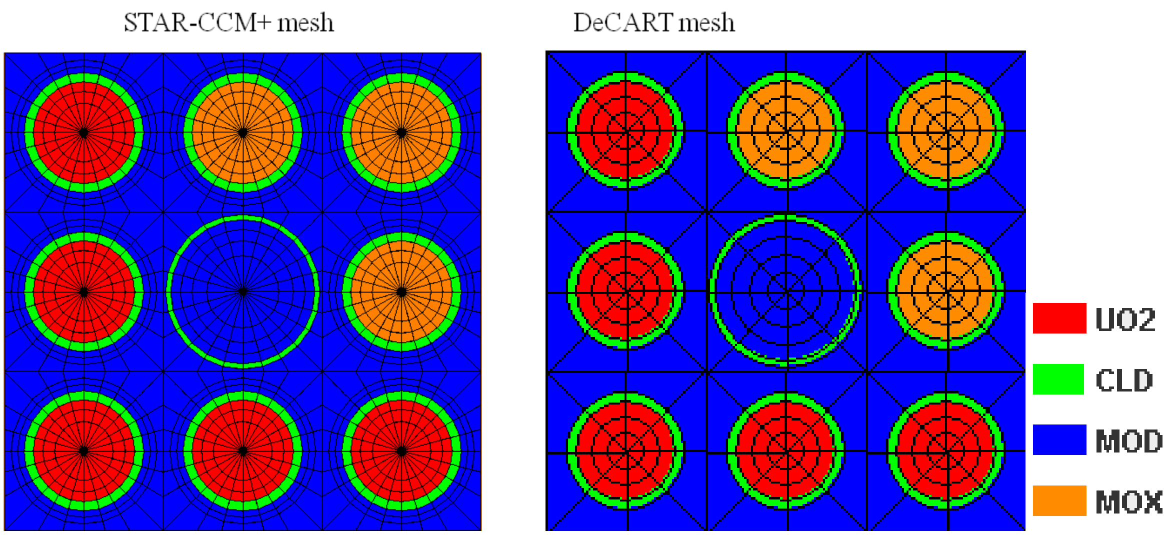


Figure 10 STAR-CCM+ and DeCART Meshes for 3x3 Pin Model

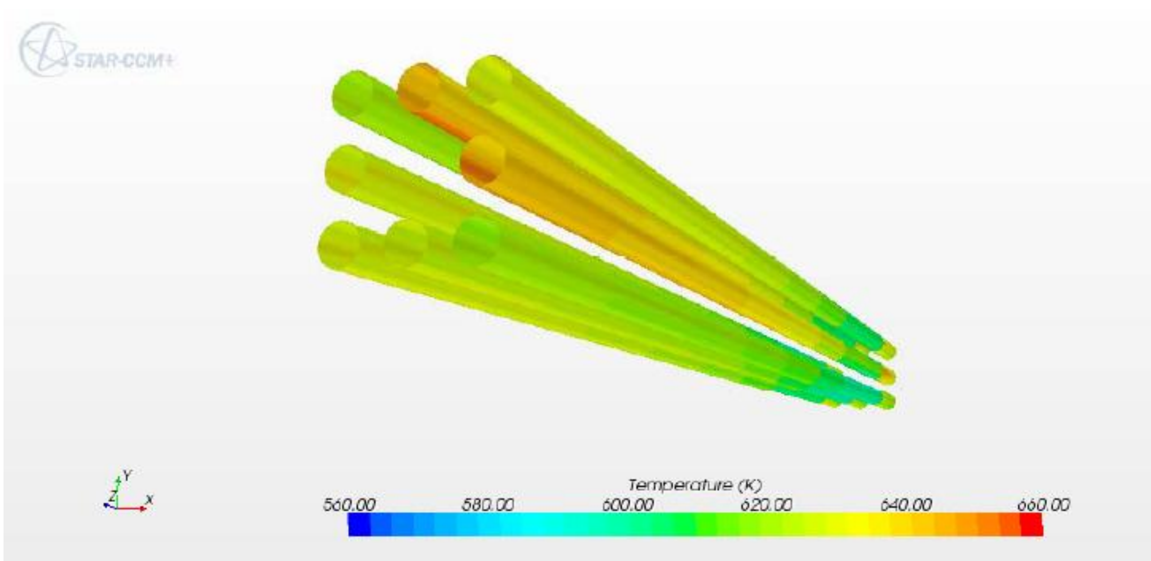


Figure 11 Clad Inner Surface Temperature from Coupled Tools

4. Fuel Rod Heat Transfer Modeling

A fuel rod heat transfer model is included in the STAR-CCM+/DeCART coupling described in Section 3. Several rings are used in the fuel rod to simulate the pellets and the clad. The model was applied at single phase reactor operating conditions in Table 2.

Table 2 CFD Model Input Conditions

Turbulence Model	Realizable k-ε turbulence model with high y+ wall
Solid surfaces (rest)	No-slip wall
Fluid inlet	velocity inlet at 4.95 m/s and temperature at 288.9 deg-C
Fluid outlet	Pressure outlet with a constant gauge pressure of 0
Fluid side surfaces	Periodic
Reference pressure	15.513 Mpa

The temperature distribution on the rod surface is shown in Figure 12. Hotspots tend to form near the end of the grid span due to a large azimuthal variation in heat transfer coefficient (about 80 to 120%). These hotspots can lead to excessive subcooled boiling, crud deposition and localized corrosion if a large crud source exists in the reactor. Figure 13 shows a localized crud deposit that occurred on a fuel rod at the end of the grid span in the reactor.

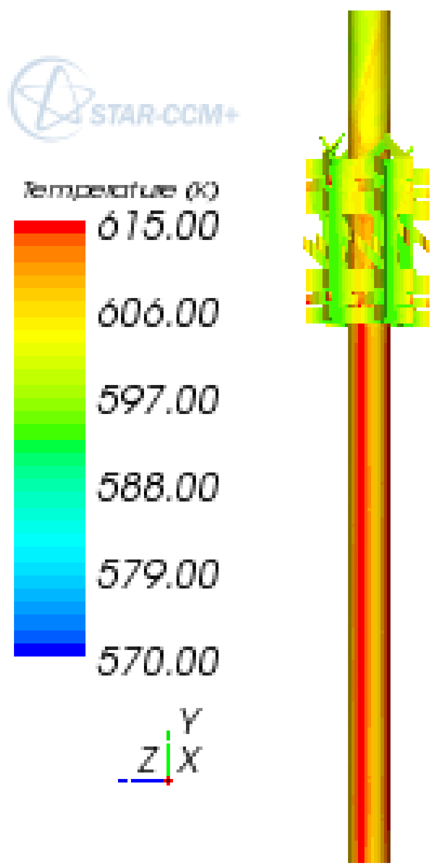


Figure 12 Temperature Distribution on Rod Surface

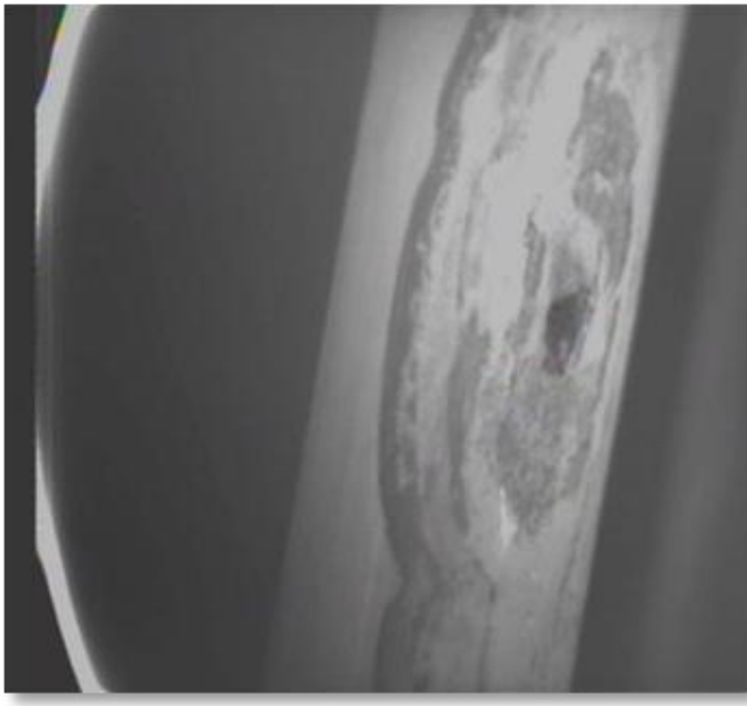


Figure 13 Localized Fuel Rod Crud Deposit in Reactor

5. Turbulence Modeling and Rod Vibration

The 3x3 pin model can be used to define turbulence excitation forces for fuel rod vibration. The turbulent excitation forces in combination with a grid to rod gap can lead to excessive fuel rod vibration and cause grid to rod fretting and fuel leakers in the reactor. The 3x3 pin CFD model was used to calculate the turbulent excitation force for the spacer grid with and without mixing vanes. LES model in STAR-CCM+ was used to calculate the turbulence downstream of the grids. The WALE subgrid model, in combination with bounded central differencing scheme for spatial discretization of the momentum equations and a blending factor of 0.1 was used. A second order implicit formulation is employed for temporal discretization and the physical time-step is chosen in order to produce an average Courant number of around 1.

For the purposes of the calculation of the transient forces on the fuel rod, the center rod in the CFD model is used. The center fuel rod is divided into segments of 25.4 mm (inch). Transient forces acting on the fuel rod surface from CFD model are integrated at each rod segment at each time step in two radial directions, for this analysis the two radial directions are X and Z directions. Figure 14 shows the turbulence force (standard deviation on force) downstream of the grid with and without mixing vanes in the Z-Direction. In vibration term, the standard deviation of a fluctuating data in time domain is the overall vibration amplitude, RMS. The resultant vibration forces are then used in another tool to predict rod vibration and wear. Further details on this work are provided in reference [5].

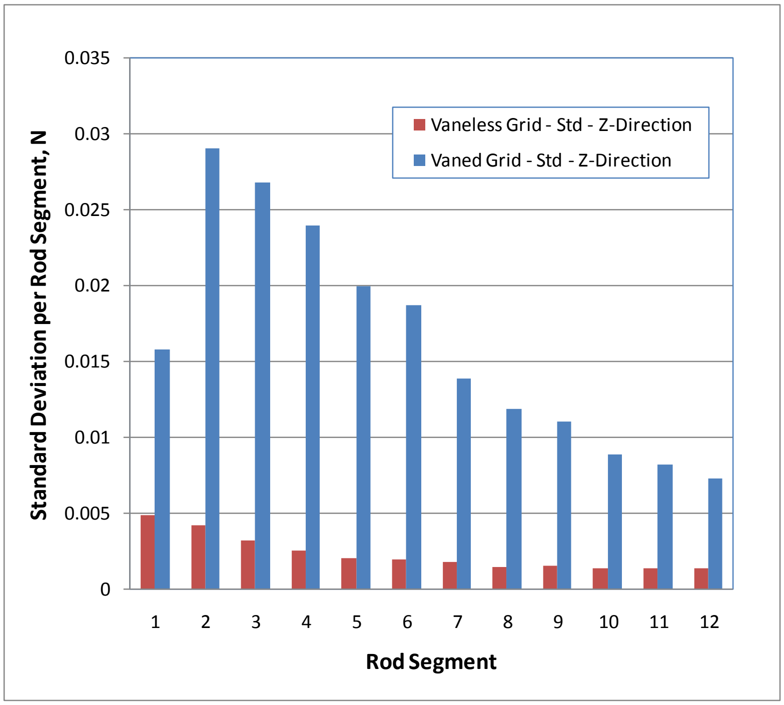


Figure 14 Standard Deviation on Force versus Distance Downstream of Grids with and without Mixing Vanes

6. Scaling from 3x3 Pin Modeling to Larger Geometries

After multi-physics tools are implemented in the 3x3 pin model and some benchmarking is performed to available test data, the next step will be to scale up the geometry to a larger portion of the reactor. Some work has been performed where a CFD model has been developed for ¼ of the vessel where all reactor internals and fuel rods are simulated [6]. Figure 15 provides a summary of the 3D CAD models for ¼ of the vessel including the fuel. The first vessel CFD model simulated the spacer grids with a porous body model (~122 million cells). Another vessel model has been recently simulated where spacer grids utilize a detailed mesh resulting in about a billion cells. This large vessel CFD model was run on ORNL supercomputers.

A steady RANS simulation was performed using STAR-CCM+, a fully-featured commercial finite-volume CFD code. The spatial differencing scheme used in the analysis was second-order upwind for all variables. The turbulence model used was the Realizable k-ε model with high-y+ wall treatment (wall functions). The STAR-CCM+ vessel CFD model was run on a Linux cluster using 80 processors (2 GB RAM per processor). The analysis was run for approximately 1100 iterations to establish the overall flow structures, with a corresponding reduction in solution residuals of 2-3 orders of magnitude. The accumulated CPU time over all processors was approximately 7700 hours and the elapsed time for the analysis was approximately 100 hours.

Figure 16 shows a typical view of the detailed axial velocities at the inlet of the fuel. The challenge is to apply the multi-physics tools for all rods in the core in a coarse mesh approach then zoom in with a more detailed mesh or higher fidelity where the limiting reactor phenomena are detected.

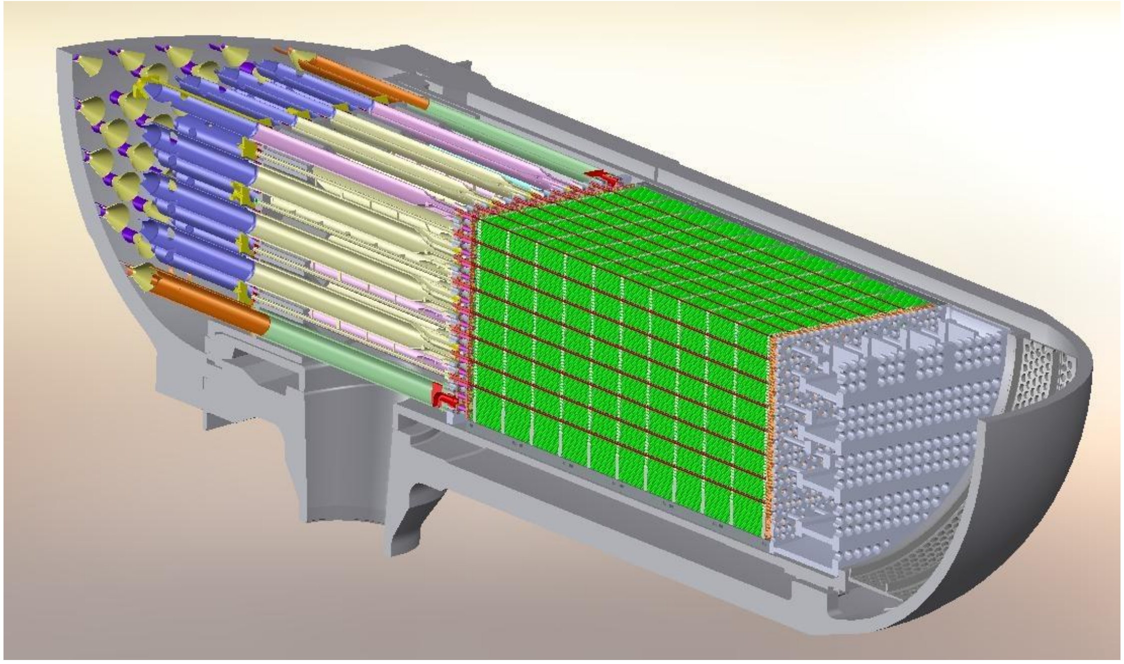


Figure 15 3D CAD Models for Vessel, Internals and Fuel

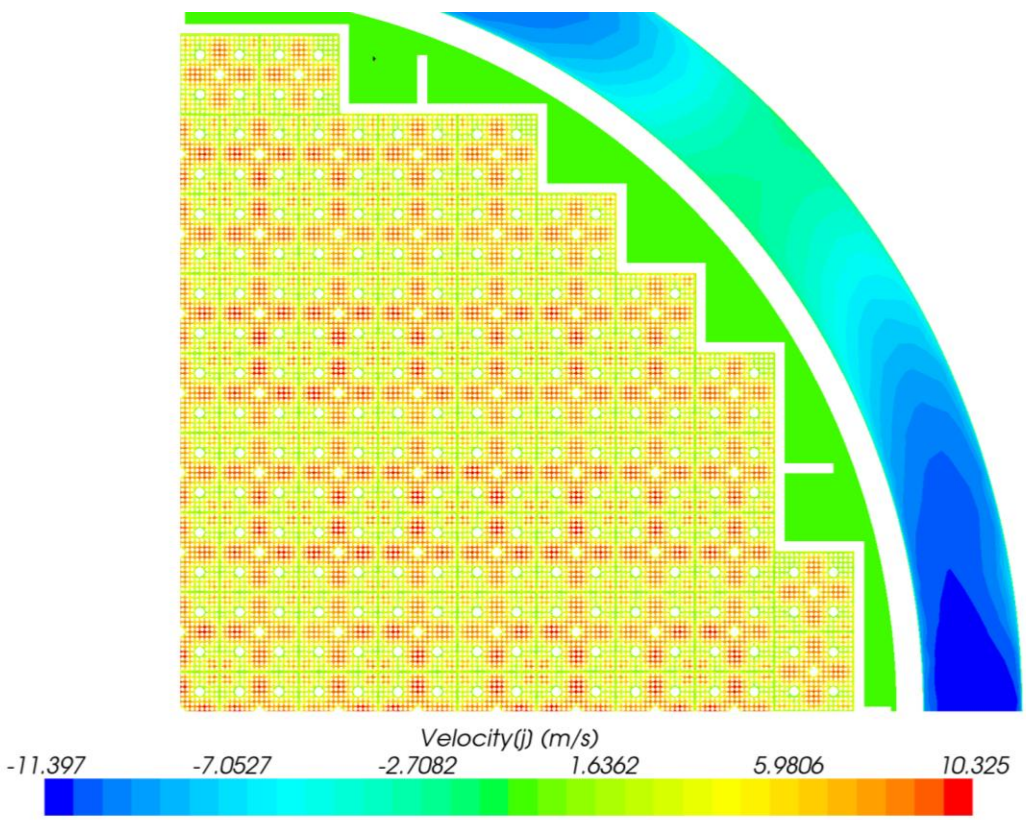


Figure 16 Detailed Pin CFD Modeling Results – Axial Velocities at the Inlet Region of Fuel

7. Conclusion

The preliminary analyses and initial data comparison indicate that the multi-physics; neutronics, thermal hydraulics, structural mechanics and nuclear fuel rod performance can be performed in a 3x3 pin volume. The goal is to complete the 3x3 multi-physics model development and perform validation to available test data. Figure 17 describes how the 3x3 multi-physics model development can be performed with data validation. The application of the multi-physics tools can then be scaled up to larger geometries such as the fuel assembly and the entire vessel. Future work includes further development of two-phase flow models, implementing crud and corrosion models, fuel rod performance modeling and benchmarking these models to available test data.

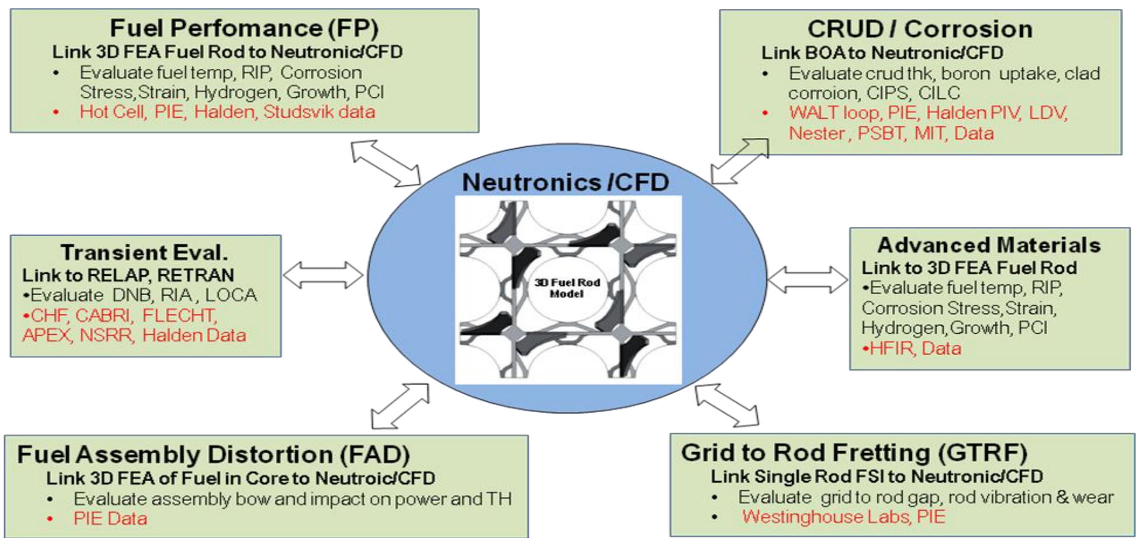


Figure 17 3x3 Pin Multi-Physics Model Development and Validation

8. References

[1] STAR-CCM+ by CD-adpaco, http://www.cd-adapco.com/products/STAR-CCM_plus/index.html

[2] B. KOCHUNAS, M. HURSIN and T. DOWNAR, "DeCART-v2.05 Theory Manual", University of Michigan, (2009).

[3] Conner, et al, “Hydraulic Benchmark Data for PWR Mixing Grid”, NURETH 14-337, Toronto, Canada, September, 2011

[4] Yan, et al, “Coupled Computational Fluid Dynamics and MOC Neutronic Simulations of Westinghouse PWR Fuel Assemblies with Grid Spacers”, NURETH 14-254, Toronto, Canada, September, 2011

[5] Elmahdi, et al, “Flow Induced Vibration Forces on a Fuel Rod by LES CFD Analysis”, NURETH 14-365, Toronto, Canada, September 2011

[6] Karoutas, et al, “Evaluating PWR Fuel Performance Using Vessel CFD Analysis”, Proceedings of 2010 LWR Fuel Perfomance/TopFuel/WRFPM Orlando, Florida, September 26-29, 2010, Paper 01