CFD METHODOLOGY AND VALIDATION FOR SINGLE-PHASE FLOW IN PWR FUEL ASSEMBLIES

Michael E. Conner [1,*], Emilio Baglietto [2], Abdelaziz M. Elmahdi [1]

[1] *Westinghouse Nuclear Fuel, 5801 Bluff Road, Columbia, South Carolina, USA 29250*
[2] *CD-adapco, New York Office, 60 Broadhollow Road, Melville, New York, USA 11747*
[*] *Corresponding Author; Tel: +1-803-647-3685, connerme@westinghouse.com*

Abstract – This paper presents the CFD modeling methodology and validation for steady-state, normal operation in a PWR fuel assembly. This work is part of a program that is developing a CFD methodology for modeling and predicting single-phase and two-phase flow conditions downstream of structural grids that have mixing devices. The purpose of the mixing devices (mixing vanes in this case) is to increase turbulence and improve heat transfer characteristics of the fuel assembly. The detailed CFD modeling methodology for single-phase flow conditions in PWR fuel assemblies was developed using the STAR-CD CFD code. This methodology includes the details of the computational mesh, the turbulence model used, and the boundary conditions applied to the model. The methodology was developed by benchmarking CFD results versus small-scale experiments. The experiments use PIV to measure the lateral flow field downstream of the grid, and thermal testing to determine the heat transfer characteristics of the rods downstream of the grid. The CFD results and experimental data presented in the paper provide validation of the single-phase flow modeling methodology. Two-phase flow CFD models are being developed to investigate two-phase conditions in PWR fuel assemblies, and these can be presented at a future CFD Workshop.

1. INTRODUCTION

Traditionally, the key thermal performance parameter for Pressurized Water Reactors (PWR) fuel has been the critical heat flux (CHF). This parameter has been defined and used to ensure that the nuclear reactor core is operated in a manner that maintains safe operation even during assumed failure events. In recent PWR operation in the United States, several leaking fuel rod events have occurred resulting from excessive local crud build up on the fuel rod. The excessive crud build up occurred due to local subcooled boiling during steady-state, normal operation. With a trend in the US PWR plants toward increased power upratings, longer cycles, and higher fuel burnup, understanding the local conditions in a PWR fuel assembly has become necessary to minimize the risk of future crud leakers in PWR fuel.

Therefore, both single-phase normal operation and two-phase accident conditions are of interest for investigation for PWR cores. Local subchannel and fuel rod thermal-hydraulic conditions need to be obtained and understood. Thermal-hydraulic testing is important in understanding critical parameters, such as CHF. However, testing can only provide limited information at specific locations where the instrumentation is placed. Computational Fluid Dynamics (CFD) can be used to calculate key parameters in the whole modeling domain. Of course, the complex flow field in a PWR with mixing vanes provides a very challenging problem for CFD. Therefore, a rigorous benchmarking effort is needed to ensure that the CFD results obtained are reasonable.

Two-phase flow is much more complex to measure, model, and understand than single-phase flow. This is especially true in nuclear fuel assemblies which implement mixing vanes on the structural grids. Therefore, it is logical to start with CFD models of single-phase conditions. Then, when the methodology is developed with confidence on single-phase applications, application of 2-phase conditions can be developed. Not only does this strategy make sense from the complexity of the physics, it also makes sense from a benchmark testing perspective (single-phase testing is much easier and less costly) and CFD modeling perspective (two-phase flow CFD is more sensitive to grid quality and requires larger memory and longer computational time).

This paper presents the Westinghouse CFD methodology to model single-phase, steady-state conditions in PWR fuel assemblies. In support of this, a description of the benchmark testing used to qualify the methodology has been provided. As a validation case, a CFD model of the full lateral domain of a 5x5 rod bundle test geometry is discussed, and results are compared against test data. Now that this knowledge and experience has been gained, two-phase flow CFD models are being developed to investigate two-phase conditions in PWR fuel assemblies. The methodology and results from that effort can be presented at a future CFD Workshop.

2. PWR CFD METHODOLOGY DEVELOPMENT

Westinghouse has been using Computational Fluid Dynamics (CFD) to model the flow downstream of grids with mixing vanes in PWR fuel assemblies for over 10 years. Westinghouse has worked closely with CD-adapco (maker of STAR-CD CFD code) to develop a methodology that creates CFD models that effectively predict the flow field and heat transfer within a PWR fuel rod bundle.

An initial CFD application was to investigate the crud related fuel failures that occurred in high-powered 17x17 Westinghouse fuel assemblies at Seabrook Station Nuclear Power Plant. A two-subchannel CFD model was created and run. CFD results show good qualitative results compared to Post Irradiation Examinations (PIE); specifically high temperature streaks (in the axial direction) in the CFD model were very similar to crud deposit streaks on the actual fuel rods.

To quantitatively benchmark the CFD results and provide confidence in the detailed results, small-scale experiments (i.e., 5x5 rod bundle versus full size 17x17 rod bundle) have been developed. To investigate the flow field downstream of mixing vanes, a testing methodology was developed to measure the lateral flow field at various elevations downstream of the grid. Particle Image Velocimetry is the technique used in this testing (McClusky, 2002). To investigate the average heat transfer coefficient, a testing methodology was developed to measure the heat transfer coefficient as a function of axial distance downstream of the grid (Holloway, 2004). To investigate the axial streaks of high temperature and crud deposition, a testing methodology was developed to investigate the azimuthal heat transfer coefficient variation around the fuel rod (Holloway, 2005). All of this benchmark test data was obtained at low temperature and Reynolds number.

A CFD model was developed to predict some of this benchmark test data. The mesh, turbulence model, computational domain, and boundary conditions were used to improve the CFD prediction versus the test data. The final CFD model showed reasonable agreement versus the test data (Smith, 2002). With confidence established between the CFD model methodology and test data, application of CFD modeling was extended to PWR conditions, which are at higher temperature and Reynolds number versus the benchmark data.

To improve the heat transfer test data, a test methodology was developed that utilized fully-heated rods over the entire axial grid span (Conner, 2005). All of the rods in the 5x5 test bundle were heated over several grid spans to provide the appropriate thermal boundary condition for heat transfer measurements. Air was used as the fluid to reduce the power required to heat the rods. A specially designed thermocouple holder that could be moved axially down the rod bundle and azimuthally within a test rod was developed and used in this testing. This testing methodology was used to provide detailed heat transfer data for different mixing vane designs.

To do comparisons between different mixing vane designs, a larger lateral domain is needed in the CFD model. Test data shows that just downstream of the mixing vanes, the lateral flow field is dominated by the vanes and the CFD lateral domain is not important. However, further downstream of the grid and approaching the next grid, the overall vane pattern and lateral boundary conditions have an important effect on the flow within a subchannel. This is seen in the test data (McClusky, 2003). Fortunately, the computing speed of computers and CFD solvers has improved dramatically over the last 10 years. Therefore, transitioning from a two subchannel model to the 36 subchannels (5x5 rod bundle) in a test has become achievable. Therefore, a direct comparison can be made between the CFD model results and the experimental results of the same lateral geometry.

As can we seen above, the CFD methodology has evolved over time with an iterative process. An example of a step in the CFD methodology development is as follows:

- A CFD model is created based on the best known modeling practices and/or results of comparison with benchmark data.
- Results of the CFD model show a new hydraulic or thermal characteristic of the rod bundle that needs validation.
- A new experiment is developed to investigate the new characteristic.
- The new test data quantifies the hydraulic/thermal characteristic.
- If the CFD prediction does not predict the new test data well, the CFD modeling methodology is improved so that it captures the characteristic adequately.

The paper shows the test data and CFD results for a 5x5 rod bundle test. Comparison of the data and predictions provides validation of the single-phase flow CFD modeling methodology that was developed using the benchmark data described above.

3. MIXING VANE GRID VALIDATION CASE

To demonstrate the CFD methodology, a validation case of a 5x5 rod bundle is presented in this paper. The 5x5 rod bundle uses a concept grid (neither the mixing vane design nor the grid strap design is used in actual PWR fuel assemblies). This concept grid provides similar flow field and heat transfer characteristics as actual PWR grid designs. Therefore, this concept grid provides a good validation case.

The 5x5 rod bundle evaluated in the validation case is representative of a 17x17 fuel assembly design. The rod bundle pitch (12.6 mm), rod diameter (9.5 mm), and grid features are identical to a 17x17 fuel assembly. The only reduction in scale is in the overall 5x5 array versus a 17x17 array. The 5x5 rod bundle does not contain any thimble tubes, so it has 25 simulated fuel rods. The concept grid used in this validation case is shown in Figure 1.

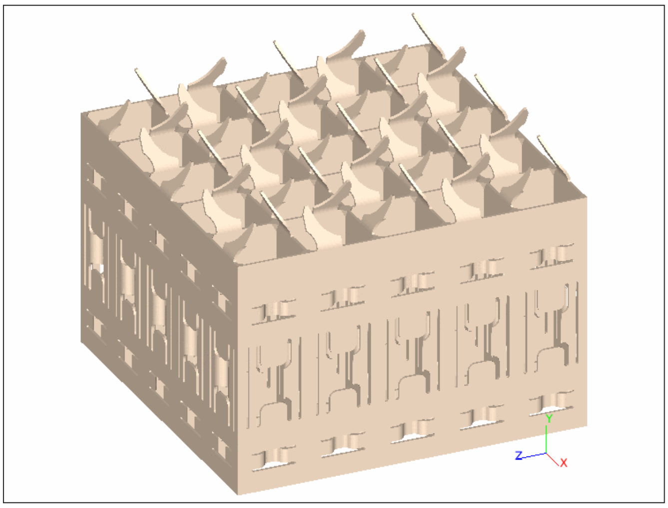


Fig. 1: 5x5 Concept Grid used as CFD Validation Case.

The 5x5 concept grid and associated rod bundle were modelled in CFD. Modelling of the entire lateral domain has the advantage of allowing direct comparison between the CFD model results and experimental results of the same geometry. The CFD model is described in Section 4. Experimental data was obtained from a 5x5 test bundle which used this concept grid design. Data from this testing is described in Section 5. Comparison of the CFD results and the test data is provided in Section 6.

Benchmark data from the experiments were obtained at low temperature and Reynolds number. In contrast, the CFD evaluation was run at in-core conditions so that heat flux values could be used that are typical of PWRs. This results in a large difference in the Reynolds number from the testing versus the CFD results. Impacts of this Reynolds number difference are discussed as part of the data comparison.

4. CFD ANALYSIS

The CD-adapco code STAR-CD has been the primary CFD platform for investigation of single-phase flow and heat transfer in PWR fuel rod bundles at Westinghouse. Westinghouse has used the three leading general purpose commercial CFD codes (STAR-CD, Fluent, and CFX) in PWR rod bundle modeling (Liu, 2005). Based on extensive benchmarking of CFD results, experience with various sized rod bundle arrays (i.e, 2-subchannel, 5x5 rod array, etc.), and a successful partnership with CD-adapco, Westinghouse has developed a CFD methodology for fuel rod bundle modeling that uses the STAR-CD code. The STAR-CD code generates high quality grids and contains accurate numeral schemes and turbulence models which lead to good results versus benchmark test data.

The CFD methodology used in single-phase flow and heat transfer modeling in PWRs is discussed in this section. The specific 5x5 rod bundle CFD model discussed in this section is used in the validation case described in Section 3. The conditions for this CFD evaluation are in-core conditions at an average axial velocity of 5.7 m/s.

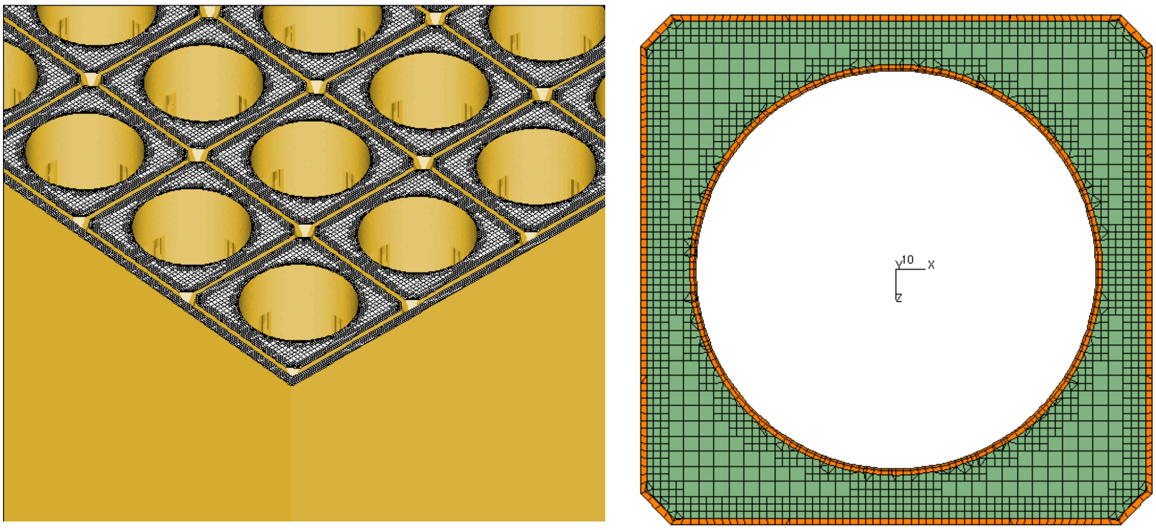


Fig. 2: Trimmed hexahedral mesh detail near the upper section of the spacer.

4.1 Mesh Generation

The model of one span length of a 5x5 nuclear fuel rod bundle, including the very complex grid spacer and mixing vane area, has been created using the pre-processing capabilities of the code STAR-CD. The software allows generating a high quality, predominantly hexahedral, unstructured grid; Figure 2 shows a representative detail of the mesh in region of the spacer surrounding one fuel rod. The core region is composed of uniform cubic hex cells, the grid is consistently refined in proximity to all walls, and a multi layer of body fitted hex cells is adopted in the near wall region and connected to the core grid with the use of trimmed transition elements.

The adopted meshing approach allows placing a sufficient number of computational points efficiently, even in complex regions such as the springs of the fuel spacer, as shown in Figure 3, and to maintain the turbulence model wall treatment requirements as a result of the precise location of near wall cells. In order to further reduce the number of necessary grid points, while cubic hex cells are adopted in the spacer region, elongated hex elements are used further away, as represented in Figure 4 where both the stretching, preceding and following the spacer, are presented.

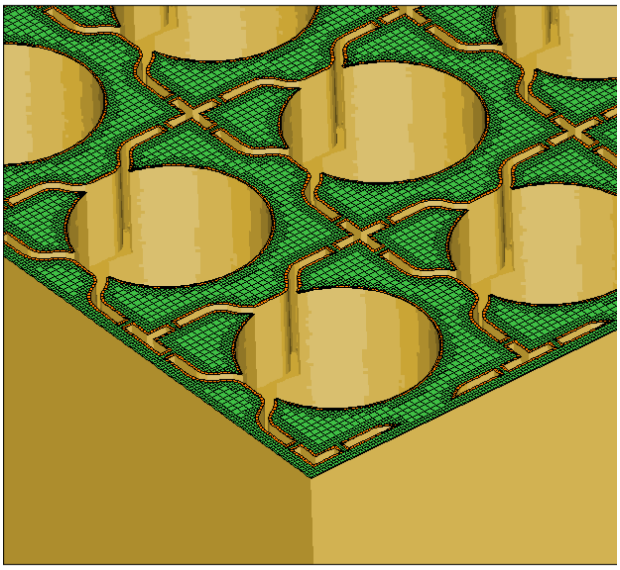


Fig. 3: Mesh detail in the spacer spring region.

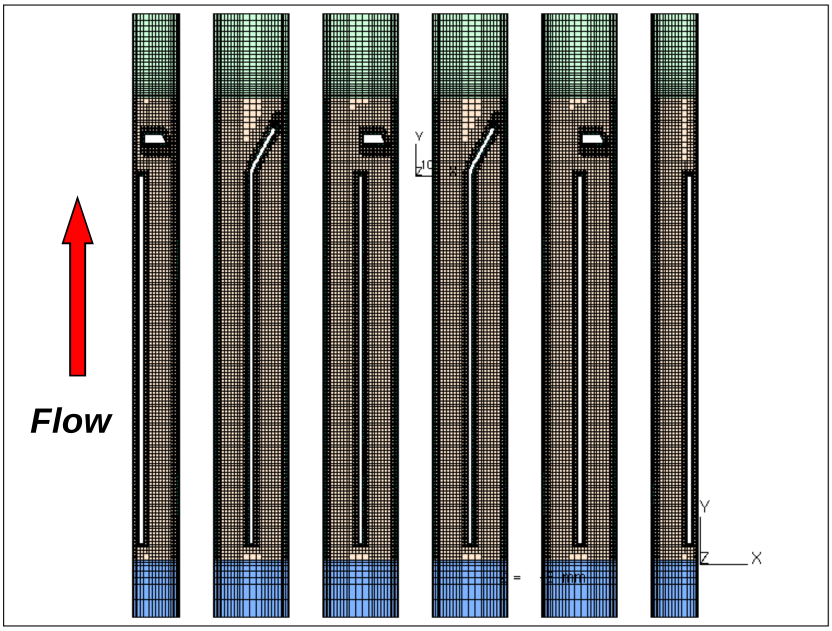


Fig. 4: Mesh section showing the axial stretching away from the spacer region.

The use of flow aligned cubic hex cells has the clear advantage of guaranteeing the grid quality, and eliminating aspect ratio and skeweness issues typical of tetrahedral meshes. Furthermore this approach guarantees the consistency of the grid refinement study. In the present work the absolute sizing of the computational cells has been derived from previous mesh convergence studies on a reduced 2x1 rods configuration, where they have demonstrated the ability of accurately capturing the average and local flow and thermal fields. The total size of the mesh is about 20 Million cells.

4.2 Discretization Practices and Turbulence Modelling

The numerical method of the STAR-CD code is based on a finite volume approach on unstructured grids. STAR-CD provides a large variety of differencing schemes for both temporal and spatial discretization of the momentum and energy equations, which can be used to solve 3D steady and transient problems; in this work steady state calculations have been performed by entirely dispensing with the temporal derivatives and iterating to the steady state with the help of under-relaxation.

As is well known, the discretization practices, together with the discussed grid quality, require special attention in order to guarantee the independency and accuracy of the results. The adopted second order discretization for momentum and turbulence quantities with a first order discretization for the temperature field, in combination with the high quality trimmed mesh, has demonstrated its validity in a large series of benchmarks, and in the present work has been further validated in separate analyses, confirming the ability to accurately capture the average and local peaks in heat transfer (Liu, 2005).

The applicability of turbulence models to the simulation of fuel rod bundles has been tested and validated by Westinghouse in their extensive research (Smith, 2002), and the renormalization group (RNG) k-ε model (Yakhot, 1992) had been found to produce the closest agreement with available experimental data. The improvement in the prediction obtained with the RNG model is assumed to be related to the somewhat inherited sensitivity to curvature and rotation contained in its turbulence dissipation rate equation. The model is applied in conjunction with a law-of-the-wall where the discussed boundary fitted grid construction near the walls, with local control of mesh thickness, results in Y+ values for the first computational points between 40 and 100.

4.3 Computation Domain and Boundary Conditions

The optimal mesh construction and efficient parallel solution performances of the CFD code have allowed modelling a complete one span region of a 5x5 bundle configuration; the capability of extending the simulation domain to a 5x5 bundle carries the significant advantage of being able to directly link the results with the hydraulic and heat transfer test performance. The steady state simulations of the discussed 20 million cells mesh have required 18 hours of computation to reach full convergence, using 14 Opteron 2.0 GHz CPUs in a LINUX cluster.

Since only one span of the whole assembly is represented in the CFD model, some assumptions need to be made in regard to the boundary conditions. In order to correctly reproduce the experimental setup, on the reduced computational model, consecutive calculation would need to be performed in which the outlet results of a calculation is mapped as inlet for the following run, (partial cyclic boundary conditions could also be used to model an infinite series of spacers). Previous experience has shown that the highly turbulent conditions and the strong influence of the spacer geometry allow adopting a simpler Inlet/Outlet configuration, producing almost identical prediction as a partial cyclic boundary condition in the region of interest, i.e. past the spacer grid. A schematic of the model and location of the boundaries and reference frames is shown in Figure 5.

Uniform inlet velocity and temperature profiles have therefore been applied at the inlet plane, which is located 10 hydraulic diameters upstream of the mixing grid, a flow split boundary condition is applied at the outlet and no-slip conditions are used for the fuel rods, spacer geometry, and walls of the flow housing. Regarding the thermal boundary conditions, a rod-by-rod heat flux distribution is adopted. Azimuthally uniform heat flux is imposed in the calculations so that variations in the HTC will only be related to the flow, and furthermore the heat flux is imposed directly to the fluid so that conduction through the fuel rod is neglected, emphasizing the local flow effects on the rod surface temperature distribution (as previously shown by Liu, 2005).

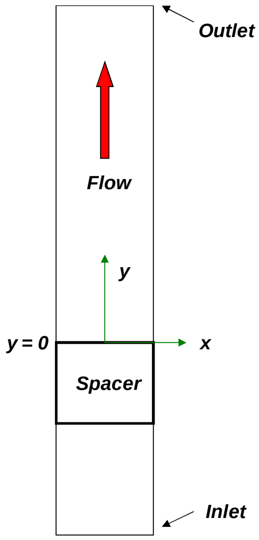


Fig. 5: Modelled 1 span flow domain.

5. TEST DATA

The 5x5 rod bundle with the concept grid design was tested using the experimental techniques introduced in Section 2. Each of the experimental techniques used a 5x5 rod bundle array like the sketch shown in Figure 6.

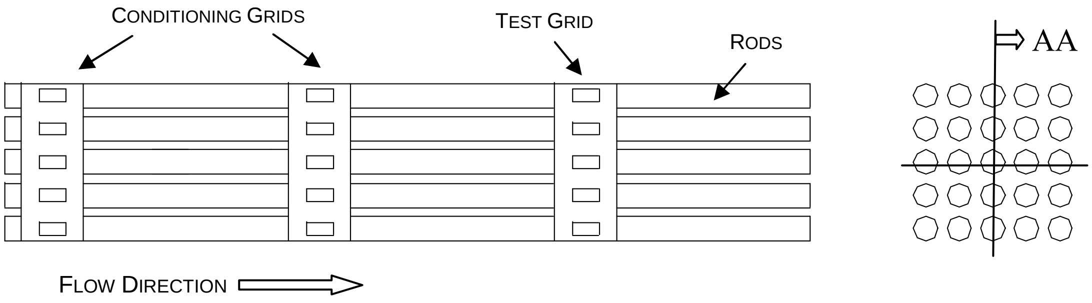


Fig. 6: Sketch of 5x5 Test Bundle
(AA = Azimuthal Angle)

All of the experimental test data discussed in this paper was obtained at low temperature and Reynolds number. For the data described in Sections 5.1, 5.2, and 5.3, water was used as the coolant. For Section 5.4, air was used.

5.1 Experiment to Measure Lateral Flow Field Velocity

The experimental methodology to measure the lateral flow field velocities is described in this section. Particle Image Velocimetry (PIV) is the technique used. This technique is described in detail by McClusky (2002). PIV measurements require illumination of particles in the flow field (water at ambient conditions) using a laser, and image capture using an optical borescope attached to a camera that transmits the data to a computer for post-processing. Figure 7 provides a sketch of PIV experimental setup. To allow optical access of the laser sheet into the subchannel of interest, a transparent Lexan housing and hybrid zircaloy/quartz rods were used. Two pulses of laser light, separated by a known amount of time, illuminate the particles. To capture an image of the particle positions for each laser pulse, a borescope extends into the flow field from downstream of the measurement plane so as to not interfere with the flow field being measured. The image pair is analyzed in small interrogation regions to determine an average velocity vector for that region.

The PIV data is typically taken at an axial velocity of 2.45 m/sec. The laser sheet and the borescope are positioned at different axial locations downstream of the grid so that the lateral velocity development and decay downstream of a grid with mixing vanes can be measured. Each image pair represents an instantaneous velocity field. A conservative estimate of the uncertainty in the magnitude of each velocity vector is estimated as ±6%. Multiple velocity fields are averaged to obtain the time-averaged velocity vector fields. This time-averaged velocity vector field is compared against the velocity results from steady-state CFD simulations.

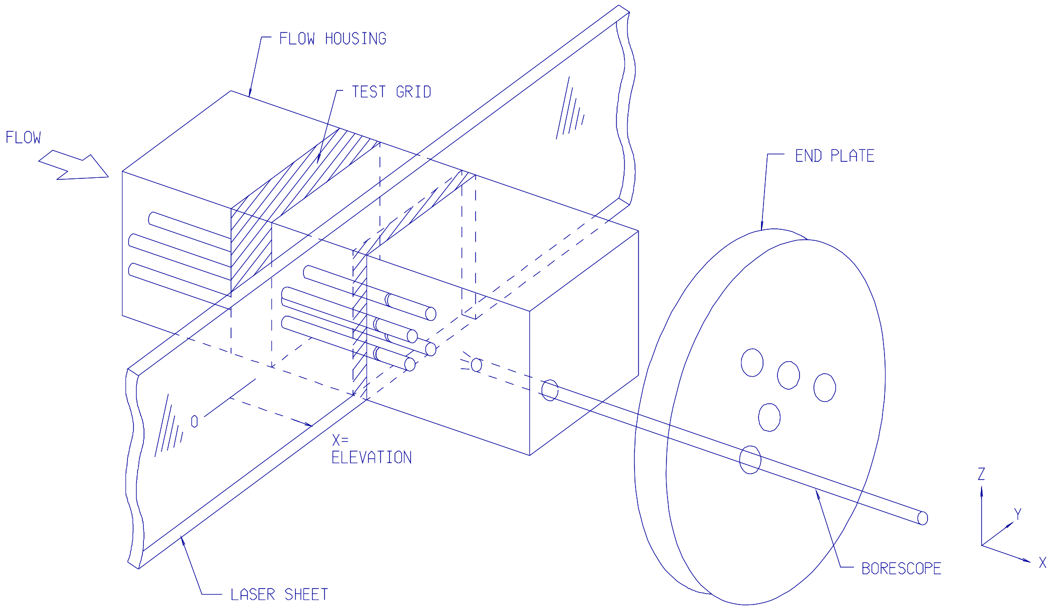


Fig 7: Sketch of PIV Measurement in a Rod Bundle

5.2 Experiment to Measure Average Heat Transfer

The experimental methodology to measure the heat transfer coefficient as a function of axial distance downstream of the grid (in ambient water flow) is described in detail by Holloway (2004). This methodology involves replacing a short axial distance of the rod (28.6 mm long) with a sensor. The sensor contains four embedded thermocouples and is heated with a concentrically inserted cartridge heater. The entire rod can be moved axially, repositioning the sensor at different axial elevations downstream of a grid. This experiment provides a challenging benchmark for CFD in that this sensor creates a localized region of thermally developing flow. The hydraulic boundary, on the other hand, changes at each axial measurement location, with many elevations being in essentially hydraulically fully developed flow. This benchmark has been performed using this CFD modeling methodology (Liu, 2004) and will not be repeated in this paper, especially since the CFD model has fully heated rods.

5.3 Experiment to Measure the Azimuthal Heat Transfer Variation

The experimental methodology to measure the azimuthal heat transfer coefficient variation around the fuel rod (in ambient water flow) is described in detail by Holloway (2005). An azimuthal heat transfer probe was developed for this experiment. The probe is a thin film sensor, 28.6 mm long, which is flush mounted on a polycarbonate holder that has the same outer diameter as the test rods (9.5 mm). The holder replaces a short axial distance of the rod, similar to the average heat transfer sensor discussed in Section 5.2. The film, which takes up 20 degrees of the circumference of the holder, is operated as a resistance heater, and a thermocouple is mounted in the holder beneath the film. The holder can be rotated to position the film sensor in different azimuthal locations, and the whole rod can be moved axially to position the probe at different axial elevations. The uncertainty of the azimuthal variation in Nusselt number from this methodology is ±4%.

5.4 Experiment to Measure Heat Transfer in Fully-Heated Rod Bundle

The experimental methodology to improve the heat transfer test data by utilizing fully-heated rods over the entire axial grid span is described in this section. This technique is discussed in detail by Holloway (2008). This experiment uses air at ambient temperature as the inlet boundary condition for the testing. As noted in Conner (2007), the different Reynolds number between air at ambient temperature and water at in-core conditions results is different axial decay characteristics of the average heat transfer. However, the relative difference in average heat transfer coefficient between different grid designs can be measured using this test. Additionally, the azimuthal heat transfer variation at a specific elevation is consistent with prior testing described in Section 5.3 (Conner, 2005). The advantage of this testing methodology versus the average heat transfer testing in Section 5.2 is that the thermal boundary layer is driven by the flow field and not impacted by a short heated region attached to an unheated rod.

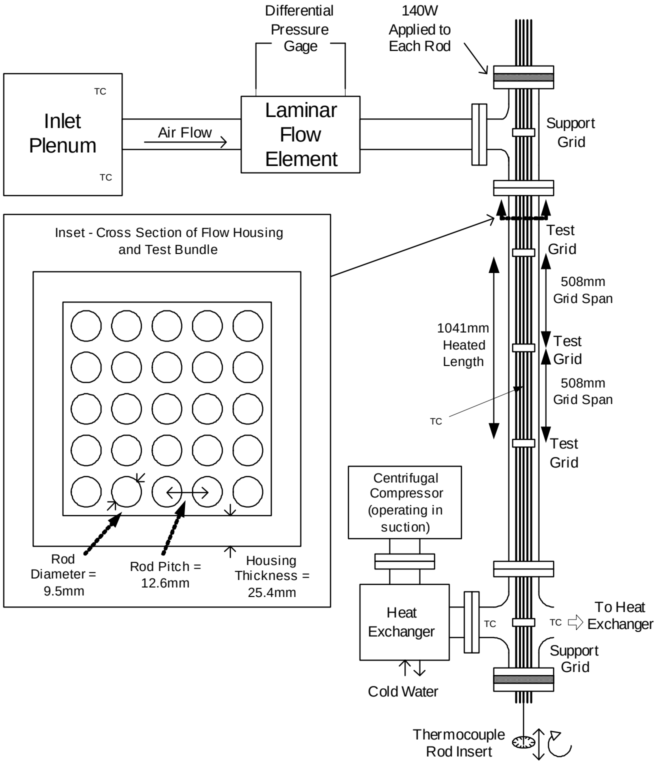


Fig. 8: Schematic of Air Test Facility

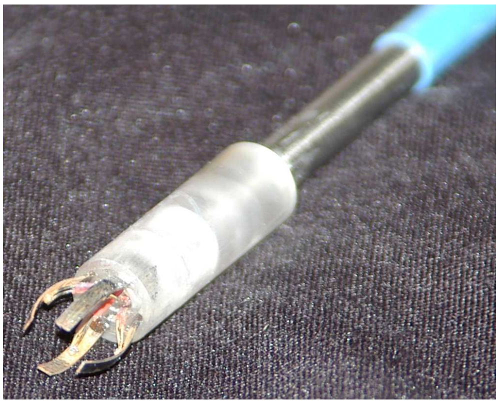


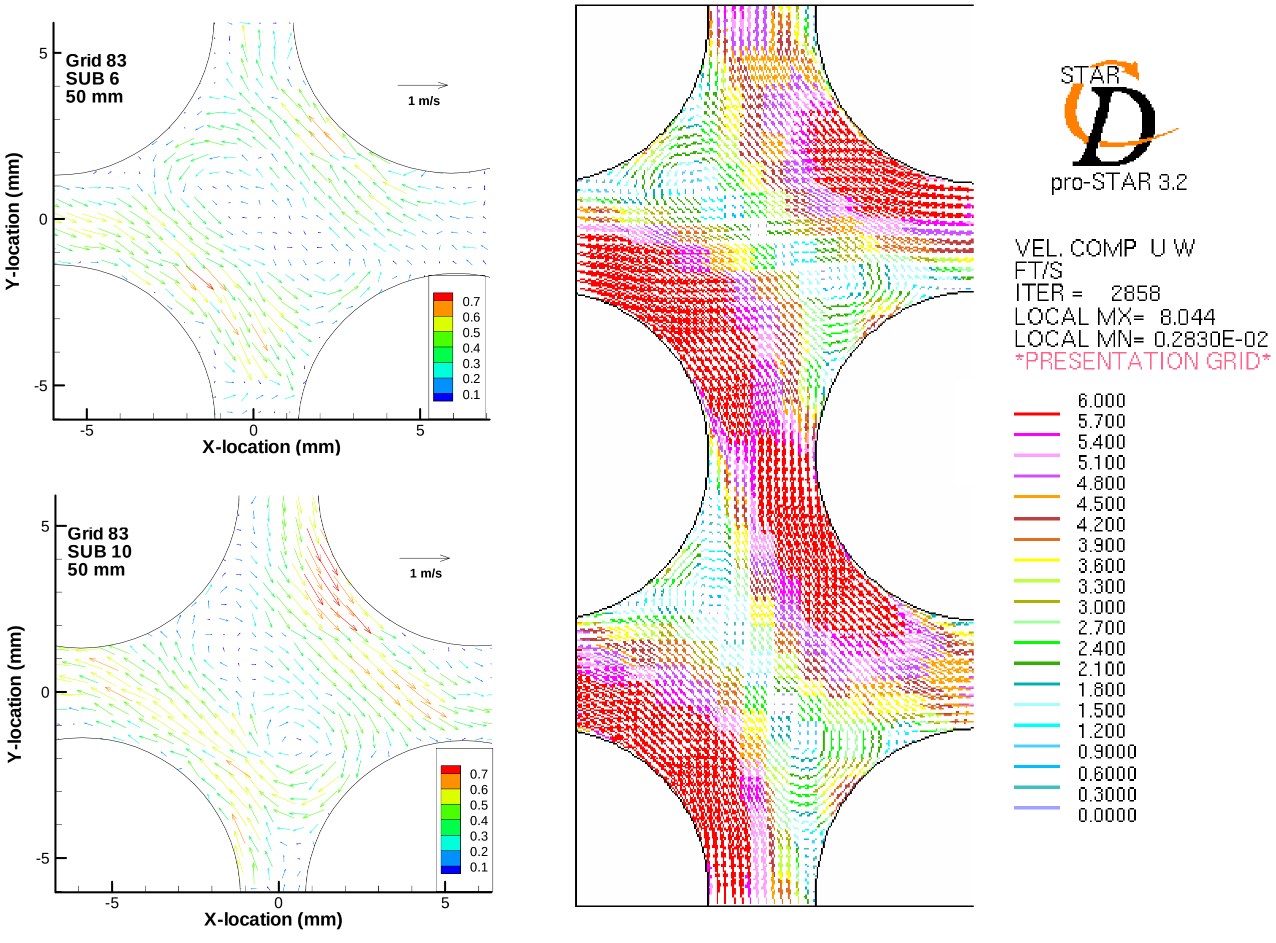
Fig. 9: Photograph of Thermocouple Holder in Air Test Facility

A schematic of the air heat transfer test facility is provided in Figure 8. The same test grid can be used in this testing as in the water testing described in Sections 5.1, 5.2, and 5.3. The test rods are different since the rod bundle is electrically heated over 1041 mm. Inconel 625 is used as the cladding in the heated length due to its high electrical resistance. The hollow cavity of the rods allows room for instrumentation to be inserted inside the tube for temperature measurements. A thermocouple holder design has been created for this experiment (a picture is provided in Figure 9).

Four thermocouples, separated by 90 degrees, are attached to springs that press against the ID of the rod wall. The springs are designed in such a way that the entire thermocouple holder can be moved axially and rotated azimuthally to measure the entire rod ID surface between two grids. The uncertainty in Nusselt number from this methodology is ±8.4%.

6. COMPARISON OF CFD RESULTS AND TEST DATA

To validate the single-phase CFD methodology described in Section 4, detailed comparisons are made between the CFD model results and the experimental data. As noted in Smith (2002), average parameters such as pressure drop across a 500 mm axial rod bundle span (including a grid) and average turbulent intensity have been compared in prior models showing good agreement between CFD and test data. For this paper, local comparisons of lateral flow field and heat transfer will be provided.



PIV Experiment CFD Results

Fig. 10: Lateral Velocity Vector Comparison at 50 mm Downstream of the Grid

6.1 Comparison of Lateral Flow Field Velocity

The lateral flow field at specific elevations was compared between the CFD simulation, at in-core conditions, and the PIV test data, which is at room temperature. Prior studies have shown the flow structures are independent of temperature and Reynolds number (Conner, 2005). In this validation case, the CFD code predicts very similar flow field structures as the test data. Of course the lateral velocity magnitude is directly related to the axial velocity so the magnitude of lateral velocity does not match. Figure 10 shows a comparison between the PIV data and CFD results at an elevation of 50 mm above the top of the grid strap. The subchannels shown are to the NW and SW of the center rod in the 5x5 array. Note that the vector density is much higher in the CFD case than the PIV data giving an obvious visual difference in Figure 10. Figure 10 does show that the CFD code predicts the same lateral flow structure as the test data: two vortices in the subchannel center, same vortex rotation, same rod swirling flow, same inter-assembly flow direction in gap. At other elevations, the axial development and decay of the vortices is also predicted well by the CFD simulation.

6.2 Comparison of Heat Transfer Data

Heat transfer coefficients were compared for the CFD simulation, at in-core conditions, and the air test heat transfer data, which is at room temperature. As noted in Section 5.4, the axial decay rate between the air heat transfer test and the CFD simulation of PWR conditions are not the same. This is due to the much lower Reynolds number of the air heat transfer test. Despite this difference in the physics of the axial heat transfer decay, there are relative heat transfer experimental results that can be used to validate the CFD simulation.

The azimuthal variation of heat transfer around the center rod in the 5x5 rod bundle was measured in the air heat transfer experiment. As stated in Section 5.4, the azimuthal heat transfer variation obtained from the air heat transfer test is consistent with the azimuthal variation obtained in the water heat transfer test using the methodology of Section 5.3. For the concept grid, the azimuthal heat transfer variation is compared against results from the CFD simulation in Figure 11. The data plotted in Figure 11 is variation of the axial average heat transfer coefficient as a function of azimuthal location around the center rod in the 5x5. This figure shows that the CFD simulation predicts the same kind of heat transfer variation as is measured in the test; the areas of increased heat transfer occur in the same azimuthal location for the experiment and the CFD simulation. This is also true of the areas of reduced heat transfer. The relative magnitudes between the experiment and the CFD simulation are not the same, but that is to be expected based on the different axial decay sensitivity of the air heat transfer test relative to water. This close prediction of the complex azimuthal heat transfer characteristic gives further validation that the CFD model is predicting the actual flow and heat transfer downstream of the grid.

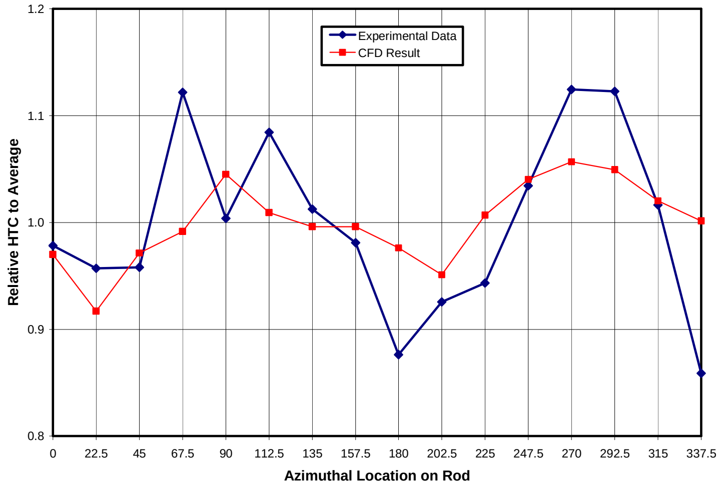


Fig. 11: Comparison of Azimuthal Heat Transfer Variation Downstream of the Grid

7. CONCLUSIONS

This paper has presented the CFD methodology that Westinghouse has developed to model single-phase, normal operation in a PWR core using the STAR-CD code. This paper has also described the benchmark experiments used to validate the CFD methodology. Finally, this paper presented a specific 5x5 rod bundle CFD model and showed how the results of the CFD simulation match the test data, thus validating the CFD methodology.

This methodology has evolved over time with an iterative process. Typically, each iteration in the methodology has resulted in a larger CFD model (domain and/or mesh quantity). Fortunately, the computational capability (computer speed, CFD code solving speed) has continued to improve over time allowing for larger models.

The experimental data used to benchmark the single-phase methodology is based on low temperature and low Reynolds number conditions. Understanding of how this data can be applied to CFD predictions of in-core conditions (high temperature, high Reynolds number) is important in determining if the CFD model is successfully validated by the test data. This understanding can be gained by modeling the identical test conditions, including temperature and flow, in the CFD model. The lower Reynolds number in the test requires a different surface mesh to provide the correct range of Y+ values in the CFD model. Westinghouse has performed rod bundle benchmarking of low Reynolds number test data in this manner (Liu, 2004).

Westinghouse has begun development of a two-phase CFD modeling methodology. One of the initial tasks that has been performed is to model benchmark cases (non rod bundle geometry) of two-phase flow to understand how the commercial CFD codes (i.e., like STAR-CD) predict these simple geometries and flow fields. This mirrors what was done in the development of the single-phase methodology (Liu, 2004). Additionally, rod bundle cases have been modelled and run. After the two-phase flow CFD methodology is established, Westinghouse can present this methodology at a future CFD workshop.

REFERENCES

M.E. Conner and M.V. Holloway, "Heat Transfer Coefficient Variations in Nuclear Fuel Rod Bundles," *Proceedings of the 2007 International LWR Fuel Performance Meeting*, San Francisco, California, USA, September 30 – October 3, 2007.

M.E. Conner, L.D. Smith, III, M.V Holloway, and D.E. Beasley, "Heat Transfer Coefficient Testing in Nuclear Fuel Rod Bundles with Mixing Vane Grids," *Proceedings of the 2005 Water Reactor Fuel Performance Meeting*, Kyoto, Japan, October 2-6, 2005.

M.V. Holloway, D.E. Beasley, and M.E. Conner, "Single-phase Convective Heat Transfer in Rod Bundles," *Nuclear Engineering and Design*, Vol. 238, 848-858 (2008).

M.V. Holloway, T.A. Conover, H.L. McClusky, D.E. Beasley, and M.E. Conner, "The Effect of Support Grid Design on Azimuthal Variation in Heat Transfer Coefficient for Rod Bundles," *ASME Journal of Heat Transfer*, Vol. 127, 598-605 (2005).

M.V. Holloway, H.L. McClusky, D.E. Beasley, and M.E. Conner, "The Effect of Support Grid Features on Local, Single-Phase Heat Transfer Measurements in Rod Bundles," *ASME Journal of Heat Transfer*, Vol. 126, 43-53 (2004).

B. Liu, L. D. Smith, III, M. E. Conner, M. B. Dzodzo, D. V. Paramonov, Z. E. Karoutas, R. P. Knott, M.Y. Young, "CFD Approach for Investigating Flow and Heat Transfer in PWR Fuel Assembly", *Proc. 13th International Conference on Nuclear Engineering,* Beijing, China, May 16-20, 2005.

H.L. McClusky, M.V. Holloway, T.A Conover, D.E. Beasley, M.E. Conner, and L.D. Smith, III, "Mapping of the Lateral Flow Field in Typical Subchannels of a Support Grid with Vanes," *Proc. 11th International Conference on Nuclear Engineering*, Tokyo, Japan, April 20-23, 2003.

H.L. McClusky, M.V. Holloway, D.E. Beasley, and M.E. Conner, "Development of Swirling Flow in a Rud Bundle Subchannel," *ASME Journal of Fluids Engineering*, Vol. 124, 747-755 (2002).

L.D. Smith, III, M.E. Conner, B. Liu, M.B. Dzodzo, D.V. Paramonov, D.E. Beasley, H.M. Langford, and M.V. Holloway, "Benchmarking Computational Fluid Dynamics for Application to PWR Fuel", *Proc. 10th International Conference on Nuclear Engineering*, Arlington, Virginia, USA, April 14-18, 2002.

STAR-CD Manuals, Version 3.2, CD-adapco Group (2005).

V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, and C.G. Speziale. "Development of turbulence models for shear flows by a double expansion technique", *Phys. Fluids*, A4(7), pp. 1510–1520 (1992).