A REVIEW OF AREVA'S EXPERIMENTAL VALIDATION OF STATE-OF-THE-ART SINGLE-PHASE CFD METHODS WITH APPLICATION TO PWR FUEL ANALYSIS AND DESIGN

A. Hatman
AREVA Inc.
3315 Old Forest Road, Lynchburg VA 24501, USA
Anca.Hatman@areva.com

A. Chatelain
AREVA NP
10 rue Juliette Récamier, 69456 Lyon, France
Alexandre.Chatelain@areva.com

K. Goodheart
AREVA GmbH
Paul-Gossen-Str. 100, 91052 Erlangen Germany
Kevin.Goodheart@areva.com

M. Martin, T. Keheley
AREVA Inc.
2101 Horn Rapids Road, Richland WA 99352, USA
Mathieu.Martin@areva.com; Thomas.Keheley@areva.com

ABSTRACT

During the course of the last decade, AREVA has developed Computational Fluid Dynamics (CFD) methods capable of accurately predicting flow fields and thermal mixing within fuel bundles, as well as pressure losses and hydrodynamic forces on fuel assembly components such as spacer grids, lower and upper nozzles. The present work summarizes the evolution of AREVA's single-phase CFD methodology and the extensive validation efforts, with focus on improving confidence level in CFD results. Currently, it is possible to produce models for large sections of a fuel assembly and produce accurate results with a CFD simulation uncertainty proven to be similar to the corresponding measurement uncertainty. Validated CFD methods have become a reliable and necessary tool used to optimize new designs and enhance performance of all the elements of a fuel assembly, being an integral part of any design effort.

KEYWORDS
CFD Validation; Pressure Loss, Hydraulic Force, Thermal Mixing

1. INTRODUCTION

Computational Fluid Dynamics (CFD) analyses provide numerical solutions to the conservation of mass, momentum, and energy equations that govern the fluid flow through reactor cores. AREVA has a long standing practice of using CFD analysis to support fuel development and design. Detailed pressure, velocity and temperature fields are currently used to evaluate fuel components thermal hydraulic performance, optimize new designs, supplement experiments, and enhance understanding of physics behind deleterious phenomena such as grid-to-rod fretting (GTRF) or particulate deposition leading to Crud-Induced Power Shift and Crud-Induced Localized Corrosion (CIPS/CILC), and eliminate excessive conservatism due to lack of knowledge. Methods development and validation is a continuous process; over the years, AREVA's methods and modeling best practices for specific applications evolved in terms of physics, turbulence models, geometry treatment, and mesh optimization. The methodology changes were driven by multiple factors, including advancements in commercial CFD codes, significant growth in the accessibility of affordable high power computing, and most importantly the availability of experimental data for validation. While all major commercial computer programs are used within AREVA, the fuel analysis methodologies were built around the CD-adapco™ software products[1]. The original single-phase CFD methods were developed in the '90s using the STAR-CD® code. The current STAR-CCM+® methods leveraged the legacy code modeling best practices and evolved to take advantage of the advanced meshing capabilities, improved physics models, and superior solver.

Methodology validation against experiments relevant to a particular application is a prerequisite for building confidence in the CFD calculations' results. Multiple benchmarking efforts were based on data from test facilities such as Le Creusot Technical Center or the CEA's HERMES facility as well as on data from tests specifically designed to produce CFD-grade high-resolution data, under a wide range of Reynolds numbers with real-scale hardware, using a variety of traditional and advanced measuring techniques. The validation work also relied on available data from open literature and participation in collaborative benchmarking projects such as the recent EPRI Round Robin CFD exercise [1]. The ASME V&V20 [2] procedure for Verification and Validation of CFD methods, which encompasses techniques for code verification and CFD simulation uncertainty assessment, coupled with standard experimental uncertainty assessments, provided a reliable means to evaluate the prediction errors and set confidence intervals.

While details of specific methodologies were presented in previous publications, the purpose of this paper is to highlight AREVA's current CFD development and validation status, with emphasis on single-phase CFD for flow fields, pressure loss, lift force, and thermal mixing predictions. The paper also provides an overview of direct applications to engineering problems, support of new product development and licensing of PWR fuel designs. The development of PWR and BWR methodologies was synergetic, and most modeling details covered by the current work translate to BWR designs; however there are many subtle differences in the specific approaches to warrant separate methodology description.

2. SINGLE-PHASE CFD METHODOLOGY DEVELOPMENT FOR FUEL ASSEMBLY COMPONENTS ANALYSIS

During the course of the last decade, single-phase CFD calculations became an integral part of the PWR fuel design and analysis. As the analysis methodologies evolved, several rigorous validation efforts were undertaken to ensure an optimal trade-off between accuracy and computational cost. Different CFD methodologies were developed to specifically address two specific needs: accurately predict pressure loss coefficients (PLC) under isothermal flow conditions, and velocity flow fields and thermal mixing. The accuracy requirements for the two types of analyses led to some differences, especially in the turbulence model setup and near wall mesh refinement. It has been proven that, in conjunction with specific spatial discretization requirements and wall treatment, the quadratic k-epsilon turbulence model performs best for pressure loss coefficient predictions, while the realizable k-epsilon turbulence model is more appropriate for velocity and temperature field predictions.

---

[1] CD-adapco, STAR CD and STAR CCM+ are registered trademarks or trademarks of CD-adapco in the USA or other countries.

2.1 Validation of CFD Methodology for Predicting Spacer Grids Pressure Losses

The CFD methodology for calculating pressure losses in rod bundles had been originally established using the STAR-CD® code. Best modeling practices were developed and benchmarked against a multitude of PWR spacer designs over a wide range of Reynolds numbers; the methodology and the validation results were documented in [6], [7]. As shown in Figure 2, the rod bundle pressure loss predictions for three PWR spacers of fundamentally different types: a vanned spacer (Type A), a spacer design without vanes (Type B), and a helical spacer (Type C), were in agreement with the experiments, being within -2% to +3% of the test results across the entire Reynolds number range. The results are presented in terms of pressure loss coefficient (PLC), calculated as:

$$PLC_P = \Delta P \,/\, 0.5 \rho V_{ref}^2 \tag{1}$$

where: ΔP is the span pressure loss, ρ is the inlet fluid density, and V_{ref} is the average inlet velocity, normalized by the measured PLC value at the highest tested Reynolds number. The measured values are plotted with error bars and are fitted with a standard power law. Next to the normalized PLC plots, Figure 2 includes bar plots of the corresponding validation comparison errors, defined as the percent difference between the measured values and the predicted values. **Error! Reference source not found.** provides a summary of the definitions for the terminology used in this paper to describe errors and uncertainties.

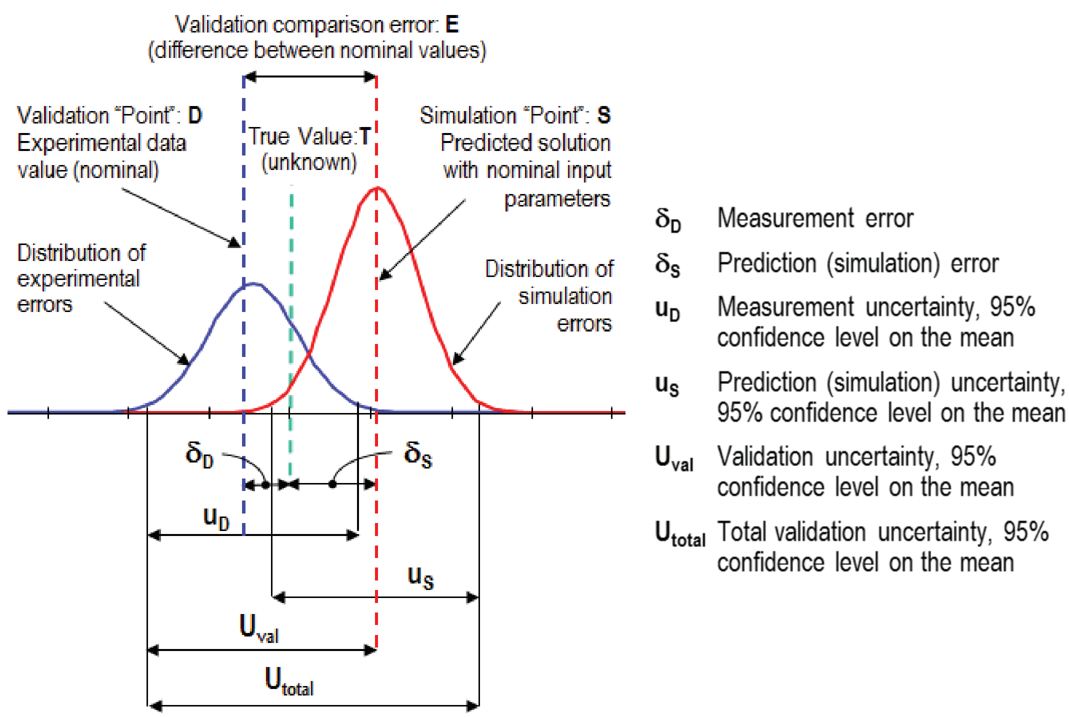
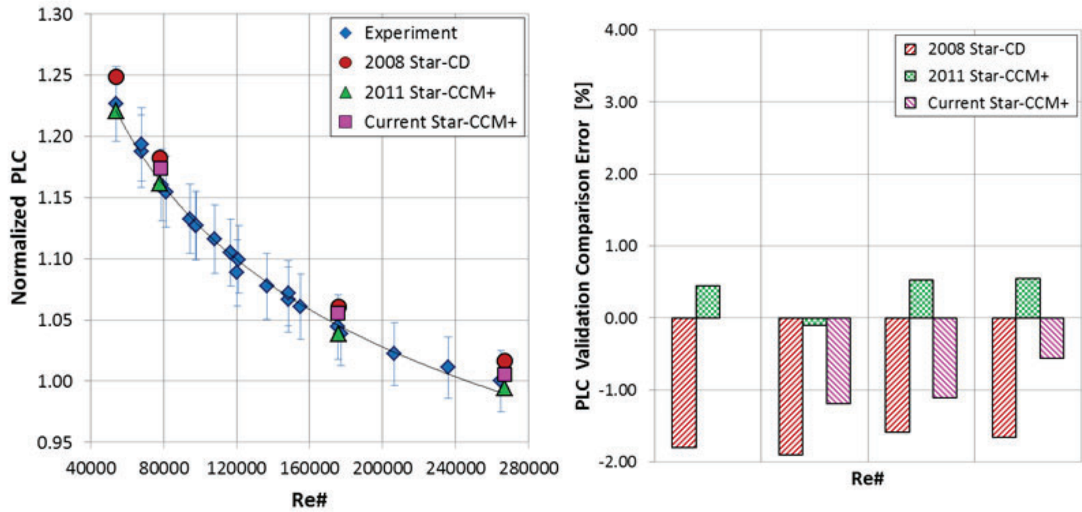
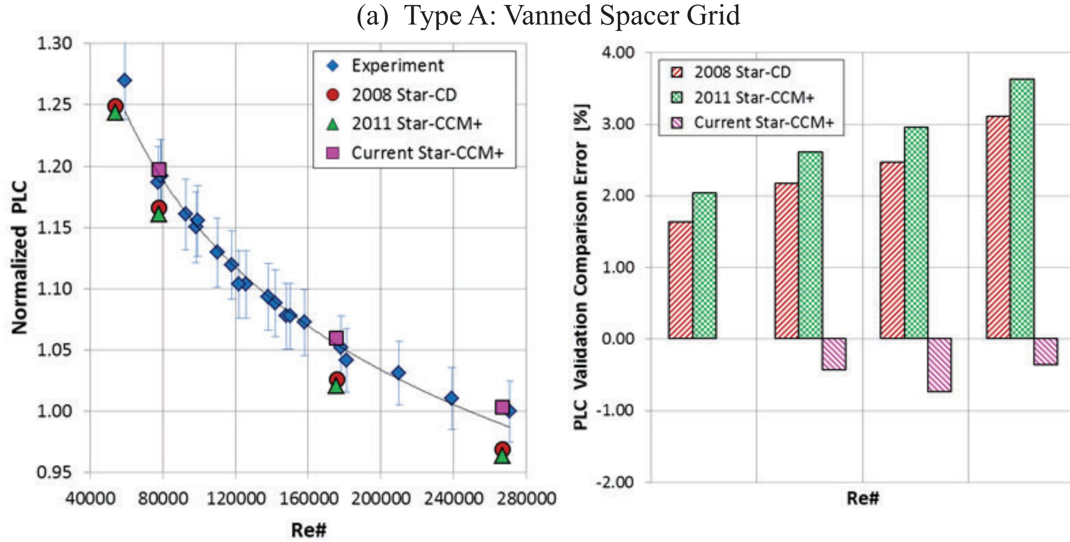


Figure 1. Nomenclature and definition of errors and uncertainties.

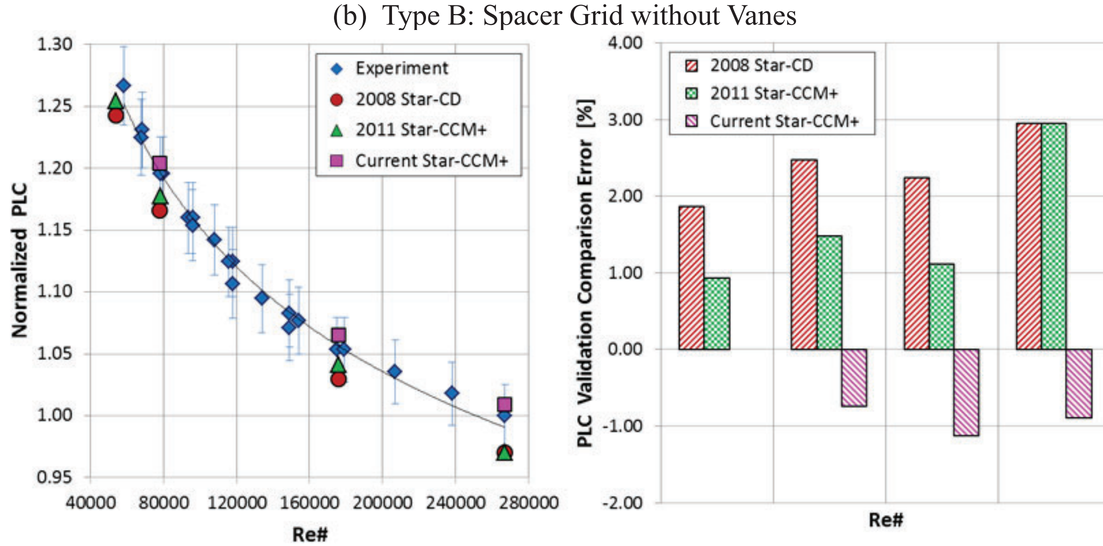
By 2010, the transition to the STAR-CCM+® code, led to new methods and improved PLC calculation accuracy. A new comprehensive validation effort was performed for the same set of PWR spacer designs and the results compared to the original STAR-CD® predictions. As shown in Figure 2, the STAR-CCM+® calculations were in excellent agreement with the experiments, being within -0.5% to +3.5% of the test results. By 2011, CFD analysis was established as a fast, cheap, and reliable tool for evaluating spacer grid pressure losses; however further improvement was achieved by giving special attention to the representation of geometry details.



(a) Type A: Vanned Spacer Grid



(b) Type B: Spacer Grid without Vanes



(c) Type C: Helical Spacer Grid

Figure 2. Validation of Pressure Loss Coefficient CFD prediction method for various spacer grid types.

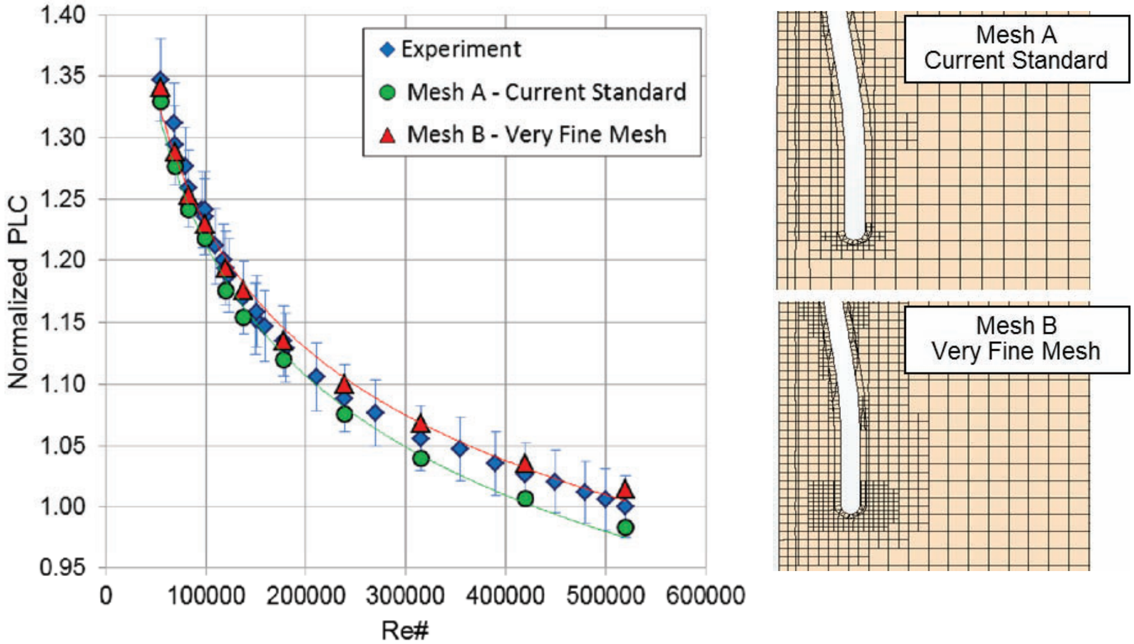


Figure 3. Mesh Refinement impact on prediction accuracy on a 17x17 mixing vane spacer model for PLC predictions in support of product licensing.

The updated methodology had been optimized through parametric studies and adjustment of mesh parameters to allow a much closer representation of the refined geometry definition. As a result, the current modeling capability can accurately capture the true geometry of dimples, welds nuggets, strap cutouts, strap edge rounding and chamfers. While still employing a relatively economic mesh, the current methodology produced results with a validation comparison error within +/-1.25%.

Figure 3 presents more recent results for a different Type A spacer grid; it shows that further mesh refinement could reduce the validation comparison error to about 0.5%; however, any increase in accuracy beyond the current standard method does not justify the added computational cost as long as the validation comparison error is much lower than the measurement uncertainty.

The ASME V&V20 method [2] had been successfully applied to assess the accuracy of CFD simulations based on AREVA's CFD pressure loss calculations for rod bundles with spacer grids [3]. The total validation uncertainty, which gives an upper limit for the error due to modeling assumptions and approximations, was calculated by estimating the standard deviation of the combination of errors due to the code numerical solution, the errors in the simulation input parameters, and the measurement errors. The simulation uncertainty due to input parameters was calculated based on 9 physical parameters and 17 geometrical parameters. Five of the geometrical parameters had the highest contribution to the overall input uncertainty, namely the vane angle, the spacer thickness, the weld nuggets, the spring width, and the central thickness; the uncertainty due to manufacturing tolerances was determined to be about 2.5% of the span pressure loss. The uncertainty propagation method estimated the bounding value of total validation uncertainty to be about 6% of the pressure loss, which is much higher than the validation comparison error. It was concluded that the simulation error was within the "noise level" relative to the total validation uncertainty. The exercise also proved that the technology is mature, and could replace pressure testing of spacer grids; it is currently used to quantify impact of geometry changes on components performance and it is instrumental in pre-screening and optimizing designs prior to physical testing. CFD PLC predictions are also used to supplement tests for minor changes to final designs. Consequently, this type of analysis has been successfully used in support of licensing of new products.

2.2 Validation of CFD Methodology for Predicting Flow within Rod Bundles with Spacer Grids

The applicability of the CFD methodology to other thermal-hydraulic applications such as thermal mixing, fuel performance and ultimately DNB prediction is conditioned by the ability to accurately predict the complex flow downstream of spacer grids. Adequate turbulence models and fine resolution computational grids are required for the characterization of enhanced thermal mixing designs. A new methodology, capable of capturing the details of the sub-channel swirl effects and the inter-channel cross-flows downstream of spacers and mixing vanes, has been developed and validated specifically for the accurate prediction of flow fields. To validate the methodology, a new experimental setup [4] [5], capable of producing high-resolution, "CFD-grade data" was built at AREVA's Technical Center in Erlangen, and two non-intrusive measurement techniques PIV (Particle Image Velocimetry) and LDA (Laser Doppler Anemometry) were implemented to measure the flow pattern downstream of several 5x5 spacer grids for a wide range of Reynolds numbers. Several spacer grids, producing complex swirling flow within sub-channels, cross-flow between sub-channels or a combination of the two strong currents were used for testing. The low pressure test facility, the LDA and PIV measurements and comparison between prediction and test results are fully described in [4], [5]. A reference case that modeled a representative spacer grid was built and run at multiple Reynolds numbers using AREVA's 2008 best-practices [6] using the quadratic standard k-ε turbulence model with customized parameters as defined in [8]. The results for the reference case were compared to the corresponding PIV and LDA measurements. While most of the averaged bundle velocity results were within 6% of measurements, the predictions overestimated the peak cross velocity values and did not capture small vortical structures clearly present in both LDA and PIV measurements, especially at high Reynolds numbers, which prompted the need for a methodology update.

Since the accuracy of the flow field calculations using the modeling best practices for PLC predictions was inadequate, a few significant changes to the CFD methodology were needed to address the near-wall treatment, the turbulence model setup and the mesh resolution requirements. Parametric studies were run to optimize the mesh and the turbulence model. The methodology improvement focused first on refining the mesh by reducing the global target size and optimizing the size and number of prism layers. Since the methodology was intended for use in routine analyses of a variety of spacer geometries for industrial applications, a trade-off between accuracy and computational cost led to a relatively high density mesh with a single prism layer sized for optimal y+ distribution. However, adjusting the mesh parameters alone did not achieve the desired level of accuracy. Since the LES (Large Eddy Simulation) and DES (Detached Eddy Simulation) models could not be considered due to prohibitive mesh size and computational time requirements, several Reynolds-averaged Navier-Stokes (RANS) turbulence models, besides the quadratic k-ε model, were tested on the adjusted mesh. The standard cubic k-ε model and the Reynolds Stress Model (RSM) were unsatisfactory. Amongst all tested turbulence models, the Realizable k-epsilon model with default parameters performed best, predicting the flow fields with acceptable accuracy. Once the modeling parameters were optimized on the reference case, the methodology was further evaluated on the remaining spacer grid types, over the entire Reynolds number rage, at multiple axial elevations.

Figure 4 uses Figure 5 of [4] to qualitatively illustrate the good visual match between the measured flow pattern and the predicted flow field in an axial cross-section eight hydraulic diameters (Dh) downstream of the upper edge of the spacer grid. Similarly, Figure 5, which replicates Figure 9 of [4], quantifies the differences between the predicted cross-flow velocity profiles in an axial cross-section two hydraulic diameters downstream of the spacer grid upper edge and the measured tangential velocities using both PIV and LDA measurement techniques. It must be noted that, while the agreement between predictions and data was excellent throughout the analyzed space, the closer the cross planes were to the spacer grid, the better the agreement (e.g. 2 Dh, Figure 5).

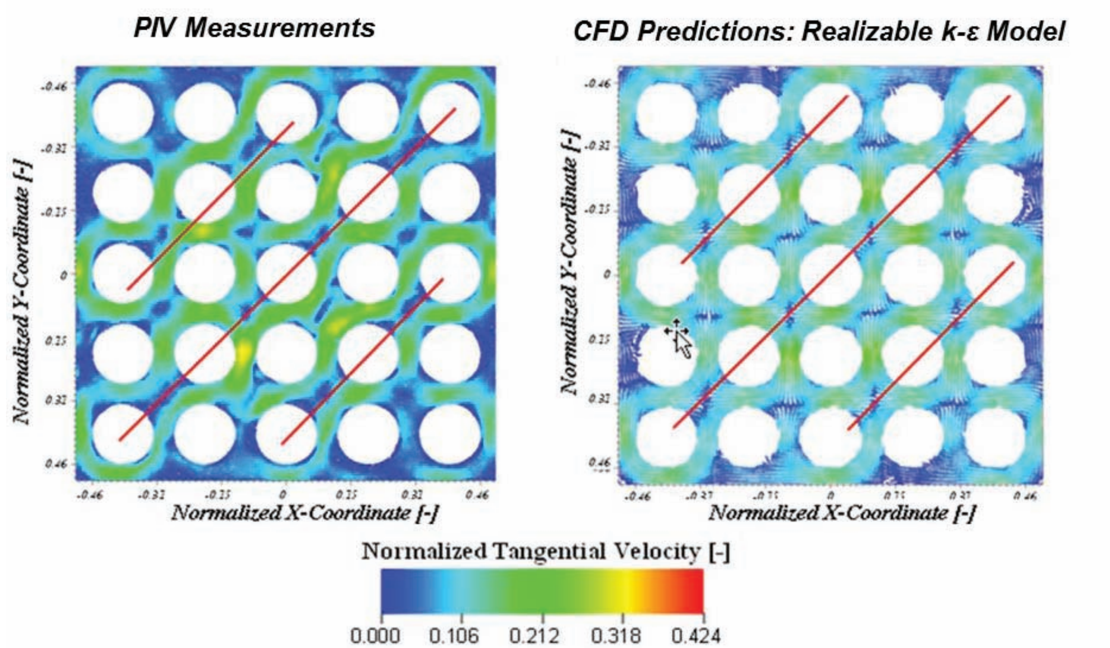


Figure 4. Comparison between the measured and predicted velocity profiles 8 Dh downstream of the spacer grid.

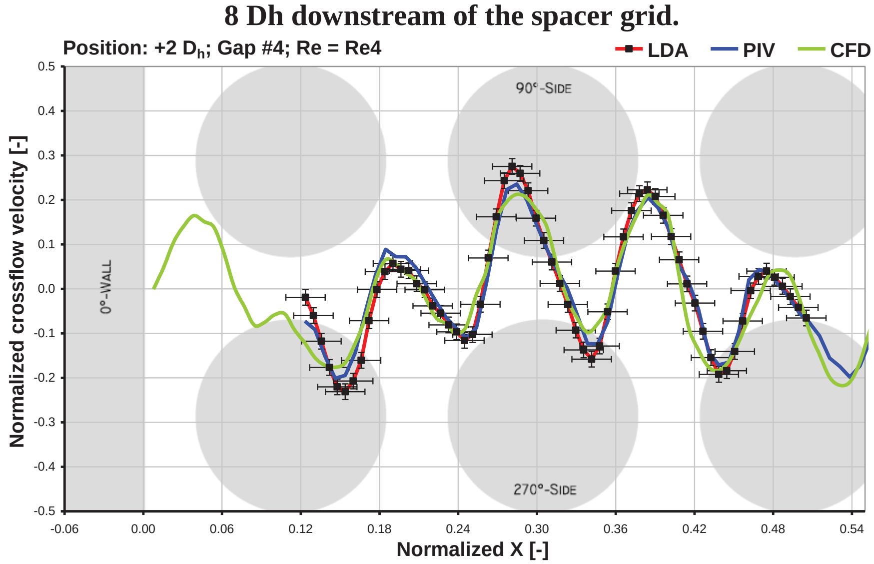


Figure 5. Comparison between the measured and predicted cross-flow velocity profiles 2 Dh downstream of the spacer grid.

An uncertainty analysis had been performed to determine the velocity measurement standard uncertainty for the LDA and for the PIV techniques; the calculation accounted for variability due to experimental devices and technique, due to the mounting of the test facility, as well as the statistical uncertainty related to repeatability. For LDA measurements, the velocity measurement uncertainty represented 1.8% of the largest tested axial velocity, while the position uncertainty was about 1.5% of the size of the test section. Similarly, the uncertainty of the PIV measurements represents 1.5% of the largest tested axial velocity. The simulation uncertainty was also calculated; it did not exceed 1.8% of the largest averaged bundle velocity for any of the analyzed geometries, being of the same order of magnitude as the measurement uncertainty. The averaged point-to-point deviation between the CFD prediction and measurements varied between 1.5 and 4.2% of the average bundle velocity. It was concluded that the accuracy of the single-phase CFD methodology for predicting flow fields within PWR rod bundles was adequate for quantifying the impact of geometry changes on spacers' performance. Separate thermal mixing benchmarking efforts (not detailed in the current paper), which were intended to improve the calculation of thermal diffusion coefficients for sub-channel codes, relied on low resolution, cold water injection thermal mixing experiments on spacer grids with and without vanes. Two data sets from separate experimental loops, were used to validate STAR-CD® and later STAR-CCM+® methodologies for thermal field predictions; it was determined that, to increase accuracy of thermal results, a second prism layer and further global mesh refinement is necessary.

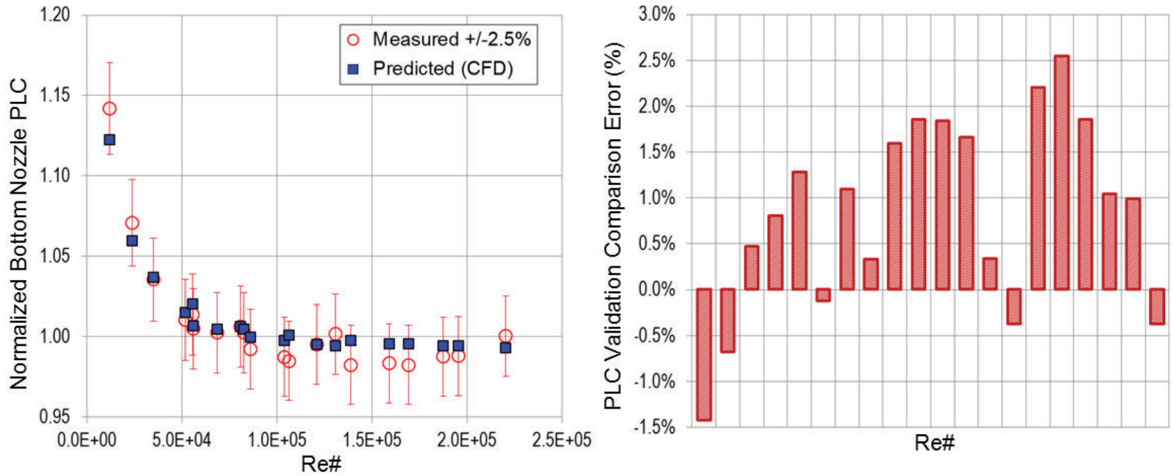
Recent work [1] used high fidelity data from the CEA's MANIVEL loop to further improve and validate the methodology for predicting single-phase temperature fields in fuel bundles. The exercise proved that AREVA's methodology, enhanced with explicit modeling of the cladding, can provide conjugate heat transfer results in excellent agreement with the data and confirmed the current high mesh resolution methodology can be used in support of product development and mixing design optimization; however, the validation efforts will continue as CFD-grade temperature data becomes available.

2.3 Validation of CFD Methodology for Bundle Inlet and Outlet Pressure Loss Predictions

Within AREVA, a methodology for predicting pressure losses in bottom and top nozzles of PWR fuel assemblies under isothermal conditions was established and improved over many years. Due to fundamental differences in the flow characteristics, the PLC methodology for nozzles differs slightly from the PLC methodology for spacer grids in terms of both mesh resolution requirements and turbulence treatment. A recent validation effort took advantage of experiments carried out at the Le Creusot Technical Center of AREVA in 2012 on several different nozzle geometries. The experiments were primarily designed to measure the hydraulic forces, but were also instrumented with multiple pressure taps to record the pressure losses throughout the test section. This provided an opportunity to validate the PLC methodology for the nozzles such that it would also provide accurate hydraulic force predictions. For brevity, this paper will present results for only one design of a bottom nozzle. The experimental inlet test section setup included the lower core plate, the bottom nozzle and the mock-up of the lower part of the 17x17 fuel bundle. The test section was equipped with 17 pressure taps that allowed flexibility for static pressure measurements at various elevations. Each measurement elevation was instrumented with two redundant pressure taps placed on neighboring walls. The tests were performed at three inlet water temperature values and Reynolds numbers ranging between 12,000 and 220,000.

The updated PLC prediction methodology was developed as an extension of the single-phase methodology for predicting flow fields by using the same STAR-CCM+® trimmer meshing technology and the RANS Realizable k-epsilon turbulence model. The results were strongly dependent on both the global mesh size and the surface target size; optimal meshing parameters were determined through mesh convergence studies for the target surface mesh and the number of prism layers also had a significant impact on results; 4 prism layers provided a good trade-off between accuracy and mesh size.

A parametric study was conducted to identify a set of robust settings that would perform well for both pressure and hydraulic force predictions. Figure 6 (a) plots the test data with error bars against the predicted pressure loss coefficients for a the entire inlet section, including the core support plate passages, the lower tie plate, the filter plate and the bottom spacer grid, while the bars plotted in Figure 6 (b) show that the validation comparison error is less than 2.5% across the entire Re# range.



(a) (b)

Figure 6. Validation of Bottom Nozzle PLC prediction method.

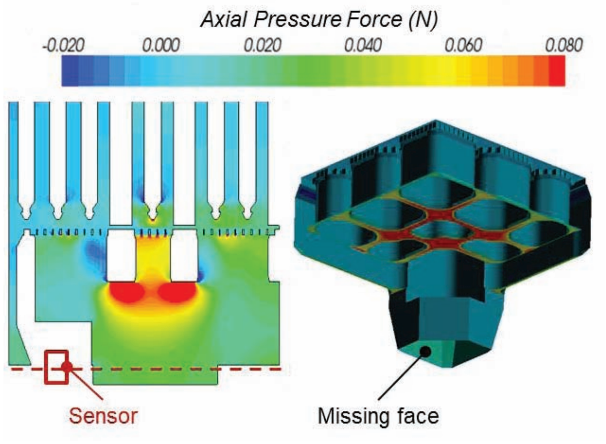
2.4 Validation of CFD Methodology for Hydraulic Force Calculations

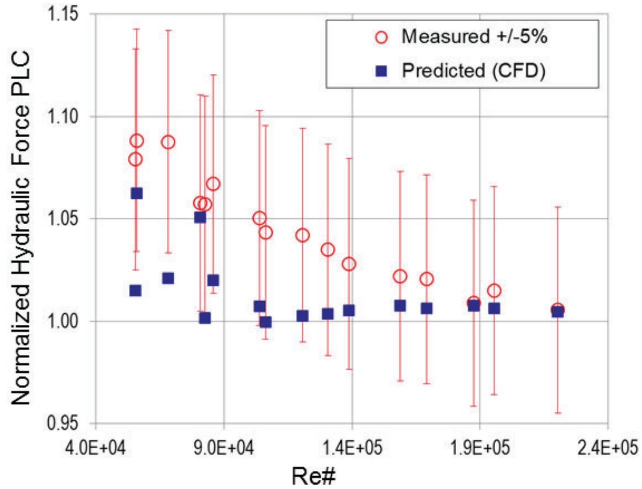
The hydraulic load on the bottom nozzle was directly measured using four sensors placed at the bottom of each lower tie plate legs, as shown in Figure 7 (a). To isolate the lower nozzle from the adjacent hardware the bundle was lifted above the top face of the nozzle, under the assumption the flow distribution was not affected by the flow passage between the filter and the bottom of the rod bundle. Using the CFD results, the hydraulic forces were calculated using both, traditional methods as well as direct STAR-CCM+® integration method. The control volume-based methods gave excellent results, however the procedure is cumbersome and does not take advantage of the built-in STAR-CCM+® force calculation capability. However, the direct use of the STAR-CCM+® force report function that integrates the pressure and shear on component surfaces is not straightforward if faces are missing due to intersection with the model envelope boundaries, and could lead to errors as high as 20%.

Figure 7 (a) points to the “missing face” at the bottom of the lower tie plate leg, where the component rests on the face of the core support plate. The STAR-CCM+® pressure force report calculation requires a reference plane for the Force Report reference pressure (P_{ref}) calculation. For closed bodies, the force calculation is correct irrespective of the chosen reference pressure; the reference pressure cancels out as the sum of face normals on a closed surface is zero. Also, “the sign” of the relative pressure value gives the direction of the resulting force, since an option for separate input of the face normal direction does not exist, [10]. For the shear force integration, the reference pressure can be set arbitrarily as the cell shear forces are absolute values. Figure 7 (b) plots the hydraulic force PLC calculated as:

$$PLC_F = F \,/\, 0.5\rho\, A_{ref} V_{ref}^2 \tag{2}$$

where: F is the hydraulic force, ρ is the inlet fluid density, and V_{ref} is the average inlet velocity and S_{ref} is the fuel bundle area, and normalized by the measured PLC_F value at the highest tested Reynolds number. While the CFD simulation uncertainty does not exceed that of the pressure and shear stress calculations, the error associated with the user handling of the “missing face” could be excessive if not handled properly. Significant deviation and scatter is visible for the low Reynolds numbers, where the viscous effects are dominant. However, the method performed very well for Reynolds numbers exceeding 100,000, producing results within -5% to +0.5% of the measured values. Since the validation comparison error for the operating conditions of interest is much smaller than the measurement uncertainty, it was concluded that the methodology can be used for practical engineering applications.





(a) (b)

Figure 7. Validation of CFD methodology for Hydraulic Force prediction.

3. METHODOLOGY FOR LARGE-SCALE MODELING

All the methodologies described in the previous sections were developed on models representing single spans or components in isolation, which required simplifying assumptions for the model bounding faces, which inevitably lead to under or over-constrained boundaries, or in the case of models replicating experimental setups used for validation, to inability to capture test sections' end-effects. Also, for many years, the analyses in support of product design were limited to 5x5 rod models with symmetry and/or cyclic boundary conditions on the cut planes. In recent years, an increase in computing resources allowed the use of multi-span models and full 17x17 rod bundle models, with limited need for simplifying assumptions on boundaries. The flow field methodology is currently used for product development of full scale 17x17 spacers, for developing correlations for large scale core models, and for providing analysis support to flow induced vibration (FIV) assessments of fuel assemblies. The current modeling and computing capabilities allow for the development of large models replicating full test sections, multiple spans, or large sections of the reactor core that offer the possibility of performing large scale, high fidelity CFD calculations for core thermal hydraulics analysis that can enhance the understanding of the system interactions, help improve system and sub-channel codes and reduce unnecessary conservatism.

3.1 Classical Porous Media Approach

While the inclusion of detailed representation of the spacer grids and mixing vanes for a full core calculation is not currently possible due to mesh size limitations, high fidelity models can be developed using a porous media representation of the fluid domain that simplifies the geometry and reduces the mesh size to a manageable size. A porosity-based, 250 million cells, high fidelity model of 1/4 of a reactor core was presented in [12] to demonstrate the capability. The spacer grids and the lower tie plate inserts were modelled as porous media, for which the flow resistance parameters were established through calibration against AREVA-proprietary pressure test data; for each flow condition, the porosity parameters were adjusted until the predicted pressure loss matched the test. Regression analysis was used to establish Reynolds-dependent correlations for the flow resistances of individual spacer grids and lower tie plate which were subsequently implemented in the reactor model. The simulation was run at in-core normal operating condition with 3D heat flux distribution mapped on the heated length of the fuel rods and standard k–ε turbulence model with non-linear quadratic constitutive equations and high y+ wall treatment.

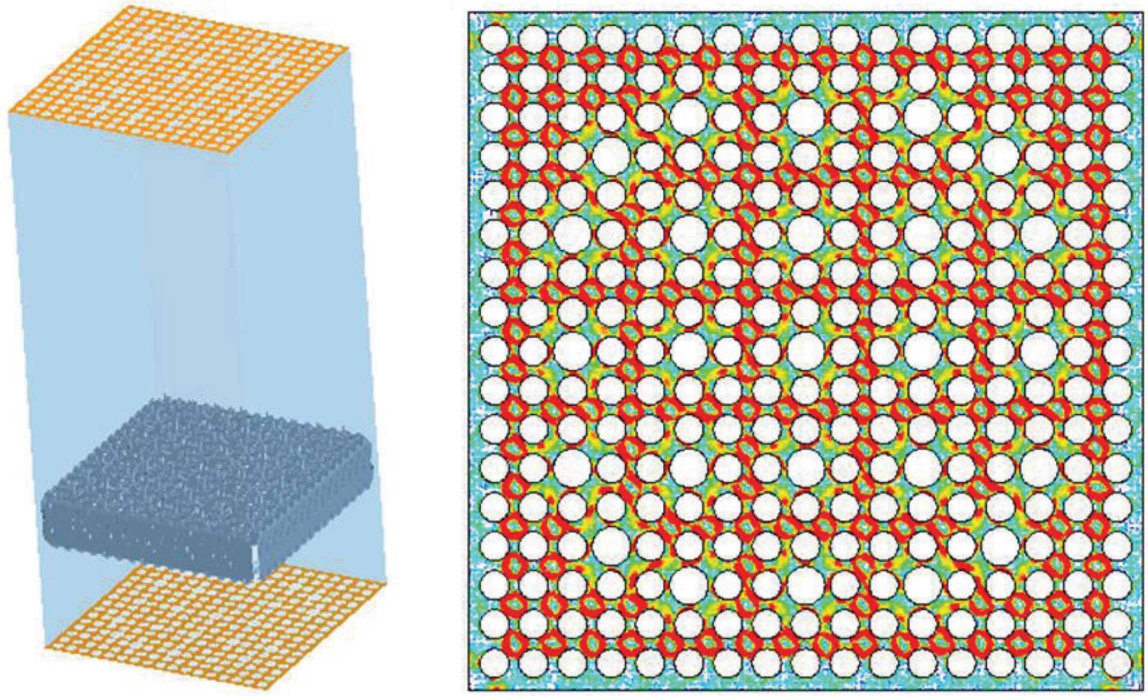
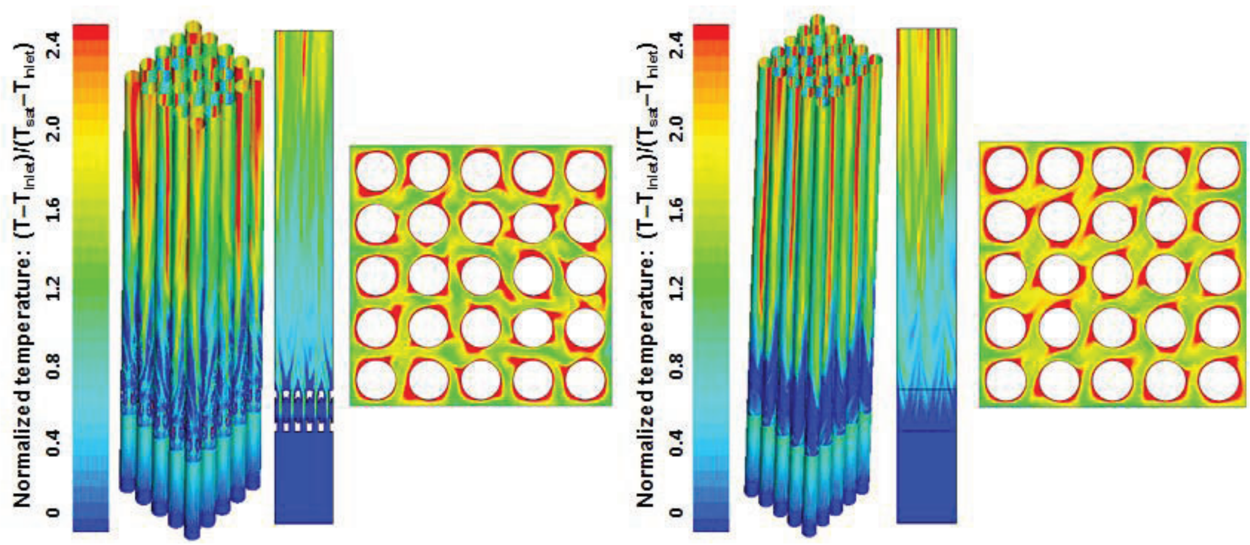


Figure 8. Model geometry and flow field for a typical single-span 17x17 spacer grid simulation.

Since turbulence quantities are not calculated within the porous media or transported from the upstream to the downstream side of a porous region the turbulence scales within the porous regions were directly specified such that the predicted turbulent quantities matched the average values and the decay rates downstream of the spacer grids predicted by explicit CFD models.

3.2 Advanced Simplified Spacer Grid Modeling

Recently, an improved modeling approach was developed to replace the classic porous media modeling for spacer grids and mixing vanes with higher fidelity source-based approach [11]. Several studies were conducted to test the ability of different non-explicit spacer grid modeling options to match the results obtained with explicit spacer grid models. A modeling option relying on porous baffles for modeling the spacer inlet and outlet flow resistances and momentum and turbulent sources for modeling the spacer straps and the mixing vanes was selected for being implemented in the next reactor core model. A 5x5 single span CFD model of a reference spacer grid, built based on the established methodology for predicting flow fields within rod bundles with spacer grids, was used to calibrate the appropriate flow resistances and source parameters of the corresponding sources-based model, which was built using the mesh settings of the open rods sections of the explicit model to ensure minimal distortion of results because of dissimilar meshes. Figure 9 compares the flow field and the temperature predictions obtained with the explicit and source-based models. Despite the fact the source model employs about 80% smaller mesh than the explicit model, visually there is very good agreement between the fine-mesh model and the explicit model in terms of flow pattern, swirl production and temperature predictions.



(a) Explicit Spacer Grid Geometry (b) Source Terms

Figure 9. Calibration of source-based modeling of spacer grids against explicit geometry.

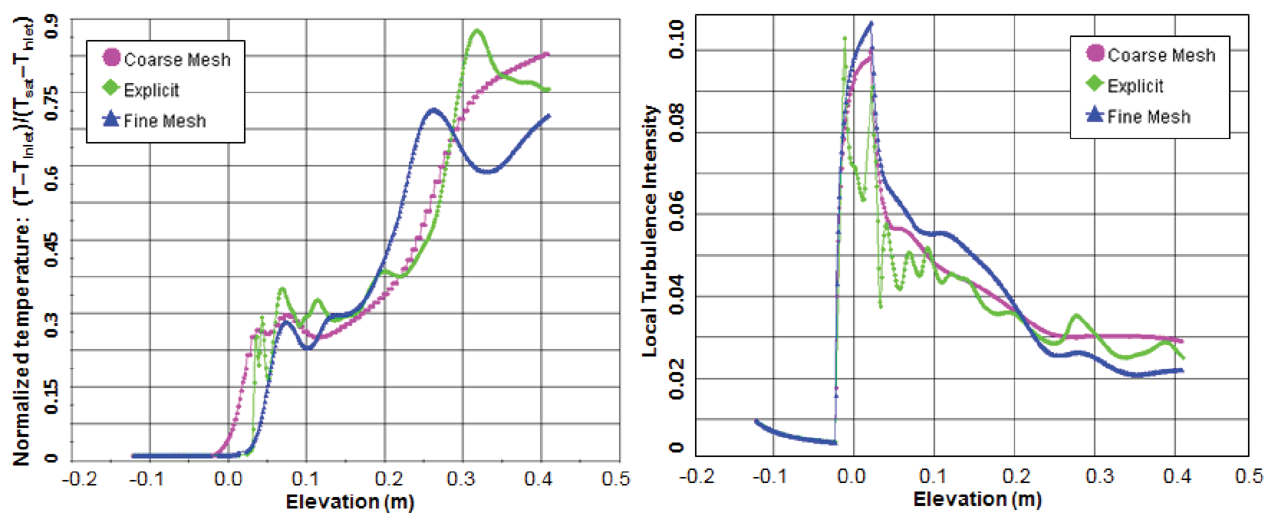


Figure 10. Source-based local predictions dependence on mesh refinement.

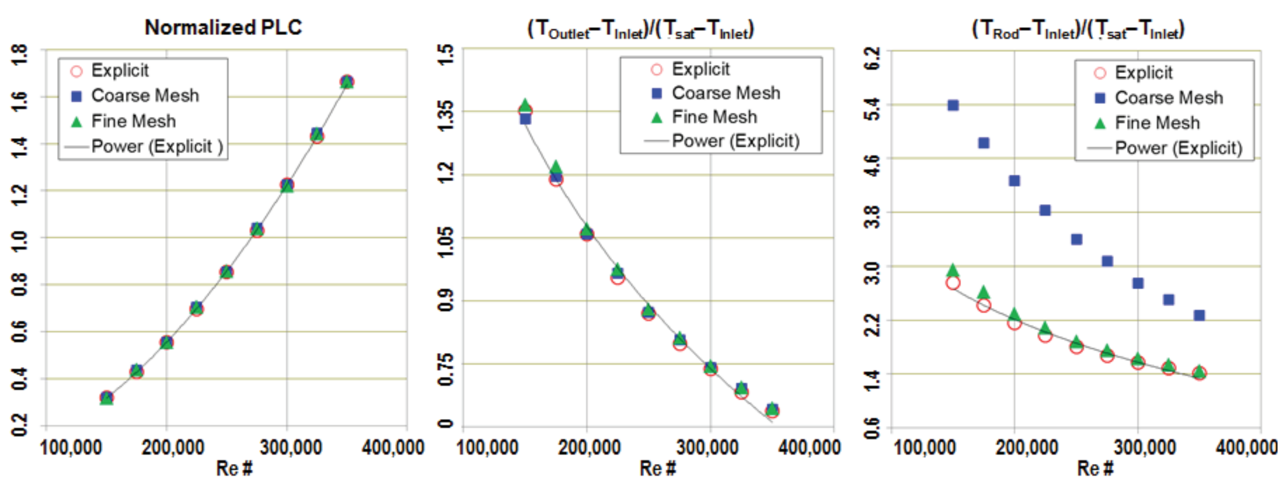


Figure 11. Source-based sub-channel average predictions dependence on mesh refinement.

The apparent mismatch in the cross section flow pattern in Figure 9, that plots the thermal fields at exactly the same elevation immediately downstream of the spacer, can be explained by the axial variation of local parameters in Figure 10; right downstream of the spacer, the complexity of the flow field generated by the explicit model cannot be fully replicated by sources terms. Better agreement could be achieved at select locations further downstream; however, the intent of this type of modeling is to replicate the overall mixing effects, which as shown in Figure 11, has been achieved. Reynolds number-dependent correlations were developed for the source terms and flow resistances. To test the robustness, the setup was run on a model with a much coarser mesh than the calibration model. As shown in Figure 11, the coarse source-based model performed well across the entire range of Reynolds numbers in terms of sub-channel average pressure and temperature. Moreover, the agreement between the local variations of flow parameters along the sub-channels axis predicted by the three models was excellent. However, for the coarse mesh, the accuracy of flow parameters in the proximity of the wall was compromised, producing erroneous wall quantities. With available computing resources, the method is still prohibitive for modeling full cores because of the mesh density requirements; however it can be used to extend explicit models by modeling regions away from the area of interest, or help replicate test sections in their entirety without affecting overall accuracy due to simplifying assumptions for end effects.

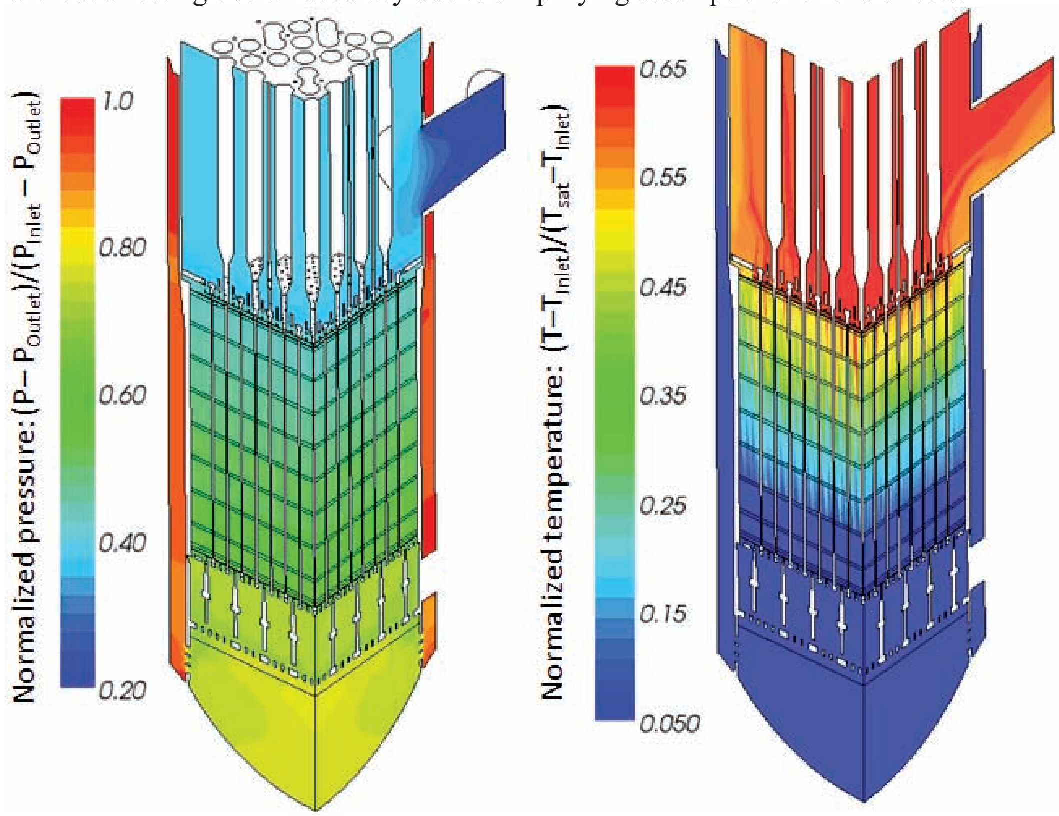


Figure 12. Source-based quarter reactor core thermal analysis.

To demonstrate the capabilities of the source-based modeling approach in solving complex thermal-hydraulic problems, a coarse mesh quarter reactor model previously built for an isothermal analysis [12] was used as the starting point for a thermal model with refined mesh and source-based representation of spacer grids. The model was run at nominal plant operating condition and determined flow and temperature fields throughout the reactor core that were in agreement with results obtained by traditional 0D and sub-channel analysis means. A graphical representation of the thermal solution is shown in Figure 12. While the results are considered qualitative in nature, the exercise demonstrated the technology is viable and with adequate computing resources could provide an advanced analysis platform and be used for routine core thermal hydraulics analyses with emphasis on inlet, outlet and peripheral effects.

4. CONCLUSIONS

The objective of this work was to summarize AREVA's state-of-the-art single-phase CFD methods and validation status. The methods for predicting pressure losses of PWR fuel components and flow fields through fuel bundles are mature and produce accurate results with the simulation uncertainty comparable to the experimental uncertainty. The validated CFD methodologies have direct applicability to a wide range of geometries and an integral part of any design effort; they are used to optimize new designs, quantify impact of design changes on performance, and to support licensing of new products. The validated methods also represent the building blocks for emerging technologies and for future CFD-based thermal-hydraulic analysis environment.

5. REFERENCES

1. M. Martin, T. Keheley, K. Vogel, K. Goodheart, A. Hatman, A. Chatelain, "Validation of AREVA's Best Practices in the EPRI Round Robin Benchmark," *Proceeding of the 16th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-16* (2015).
2. ASME, "Standard for verification and validation in Computational Fluid Dynamics and Heat Transfer", ASME V&V 20-2009, The American Society of Mechanical Engineers (2009).
3. C. Lascar, E. Jan, K. Goodheart, T. Keheley, M. Martin, A. Hatman, A. Chatelain, and E. Baglietto, "Example of Application of the ASME V&V20 to Predict Uncertainties in CFD Calculations," *Proceeding of the 15th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-15-070*, Pisa, Italy, May 12-17 (2013).
4. C. Lascar, M. Pierre, K. Goodheart, M. Martin, A. Hatman, and J-P. Simoneau, "Validation of a CFD Methodology to Predict Flow Fields within Rod Bundles with Spacer Grids," *Proceeding of the 15th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-15- 069*, Pisa, Italy, May 12-17 (2013).
5. D. Hille, B. Schrödel, I. Ganzmann, H. Schmidt, "PIV (Particle Image Velocimetry) Measurements for Detection of the Cross Flow Distribution Downstream of 5x5 Test Spacer Grids," *Proceeding of the 15th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-15- 142*, Pisa, Italy, May 12-17 (2013).
6. M. Leberig, N. Alleborn, M. Glück, J. Jones, Ch. Kappes, C. Lascar, G. Sieber, "AREVA NP's Advanced Thermal Hydraulic Methods for Reactor Core and Fuel Assembly Design", *Top Fuel 2009*, Paris, France (2009).
7. C. Lascar, N. Alleborn, M. Leberig, J. Jones, M. Martin, "Recent developments in CFD and their impact on fuel assembly optimization", *KTG Jahrestagung Kerntechnik*, Berlin Germany, (2010).
8. E. Baglietto, H. Ninokata, "Turbulence Models Evaluation for Heat Transfer Simulation in Tight Lattice Fuel Bundles," *Proceeding of the* 10th *International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-10,* Seoul, Korea, October 5-9 (2003).
9. Baglietto, E., Ninokata, H., Misawa, T., "CFD and DNS Methodologies Development for Fuel Bundle Simulations," *Nuclear Engineering and Design*, **Vol. 236**, pp. 1503-1510 (2006).
10. CD-adapco™, Inc., "User Guide – STAR-CCM+® Version 8.06," (2013).
11. L. Capone, "CFD Analysis of Nuclear Fuel Bundles and Spacer Grids for PWR Reactors," Ph.D. Dissertation, *Texas A&M University*, (2012).
12. A. Hatman, G. Williams, M. Martin, T. Keheley, C. Lascar, K. Goodheart, and A. Chatelain, "CFD Analysis of Reactor Core Flow Field in Support of FIV Diagnosis," *Proceeding of the 15th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, NURETH-15- 064*, Pisa, Italy, May 12-17 (2013).