INDUSTRIAL APPLICATIONS OF FRAMATOME'S STATE-OF-THE-ART CFD METHODS TO NUCLEAR REACTOR ANALYSIS

Julien DUMOND, Veit MARX, Markus REHM
Framatome GmbH
Paul-Gossen-Straße 100, 91052 Erlangen – Germany

Anca HATMAN
Framatome Inc.
3315 Old Forest Road, Lynchburg, VA 24501 - USA

Maxime BEZARD, Benjamin FARGES, Elodie MERY DE MONTIGNY, Julien PACULL
Framatome
10 rue Juliette Récamier, 69456 Lyon - France

Lise CHARLOT
Framatome Inc.
2101 Horn Rapids Road, Richland, WA 99354 - USA

ABSTRACT

Due to its ability to capture the effects of geometric details on local flow conditions, Computational Fluid Dynamics (CFD) has become a valuable tool for Framatome to design tailored products, meet customer needs, and find solutions to complex engineering problems.

Framatome developed CFD methods for accurate predictions of fuel performance. These techniques were rigorously validated against measurements obtained at test facilities such as Framatome's Technical Centers in Le Creusot, Karlstein, and Erlangen or CEA's HERMES. CFD assessments are routinely used to define test specifications and to support the development and optimization of nuclear fuel designs with high thermal hydraulic performance.

The paper focuses on a selection of typical industrial applications where CFD methods were applied to develop high performance nuclear fuels. The first part is dedicated to pre-test CFD-based evaluations and optimization of experimental setups where these CFD methods significantly reduced time to market and helped increase the safety margin. The second part highlights applications supporting the development and optimization of advanced fuel assembly designs for pressurized and boiling water reactors (e.g. GAIA and ATRIUM). Finally, the paper concludes with CFD application for predicting the fuel assembly distortion.

1. Introduction

The nuclear industry faces a continuous increase of demand for safer and more performant fuel designs and higher flexibility in addressing safety and reliability issues associated with nuclear reactor operation. Framatome developed a comprehensive portfolio of high fidelity Computational Fluid Dynamics (CFD) methods that can be used directly for performance evaluations or help overcome the limitations of the classical representation of the fluid domain and enhance the traditional subchannel codes. While Framatome uses all major commercial CFD codes the fuel analysis methodologies were built around STAR CCM+. The modelling techniques currently employed have built-in modelling flexibility as they were designed to be easily adapted to a wide range of engineering problems. Extensive validation was performed against experiments specifically designed to produce CFD-grade, high-resolution data for real-scale hardware, using a variety of traditional and advanced measuring techniques. While the core validation data was produced in Framatome's Le Creusot, Karlstein, and Erlangen Technical Center test facilities [18], data from other sources including the CEA's HERMES facility, collaborative projects such as the EPRI Round Robin benchmark [1], and open literature data was also used. Verification and Validation of CFD methods [3] relied on the ASME V&V20 [2] procedures and techniques, which coupled with standard experimental uncertainty assessments, provided a reliable measure of the methodologies predictive capabilities and ranges of applicability. Specifics of various methodologies were described in previous publications ([4], [5], [7], [9]) and are not the focus of this paper; current emphasis is on highlighting some of Framatome's CFD-based engineering applications including experimental test design, new product development, and enhancement of traditional codes and methods.

2. CFD-based design and optimization of experimental tests

One valuable application of CFD is the design of experimental tests, from optimization of the test stand and instrumentation placement to structuring the test campaigns such that the most relevant data is captured. Improving the experimental setup and the testing methodology relies on understanding the data before it is measured. Beforehand knowledge of the flow field allows early identification of critical areas and regions that exhibit unusual, sometimes counterintuitive flow behaviour. For example, undesirable test section "end-effects" can be quantified and corrected. Once the analytical results are benchmarked against the experimental data, the validated CFD setup provides a cost-effective platform for consistent "virtual testing" of various design variants, and "testing" at intermediate flow conditions or at conditions extrapolated outside the experimental capability limits. As opposed to physical testing that gives only limited information at discrete probe locations, the CFD results offer a continuum of multi-variable information that complements the measurements and help fully describe the problem of interest.
An example of CFD being used to redesign a test stand flow-path is given herein. A test campaign for the Flow Induced Vibration (FIV) characterization of a new fuel product was designed around an existing test loop. Repurposing the existent test setup required an iterative CFD-based optimization analysis aimed at minimizing the impact of end effects on measurements. As shown in Figure 1, the test section enclosing a complete 17x17 fuel assembly (FA) was equipped with additional features to force lateral flow in the lower part of the bundle. Two crossflow windows were placed between the lower and second spacer grids. The lateral flow entered and exited the bundle through perforated plates integral to the windows. The redesign focused on preventing interferences at the top and bottom of the crossflow windows with the adjacent spacer grids (A, B, C, and D), expanding the volume of the crossflow circuit plena (E), and optimizing the perforated plates' hole pattern (F) to improve the cross flow homogeneity over the span. The major local flow changes due to geometry adjustments are captured by Figure 2. The central plot shows the overall normalized bundle flow velocity, while the plots at A, B, C, and D locations compare the relative change in the normalized cross-flow velocity.

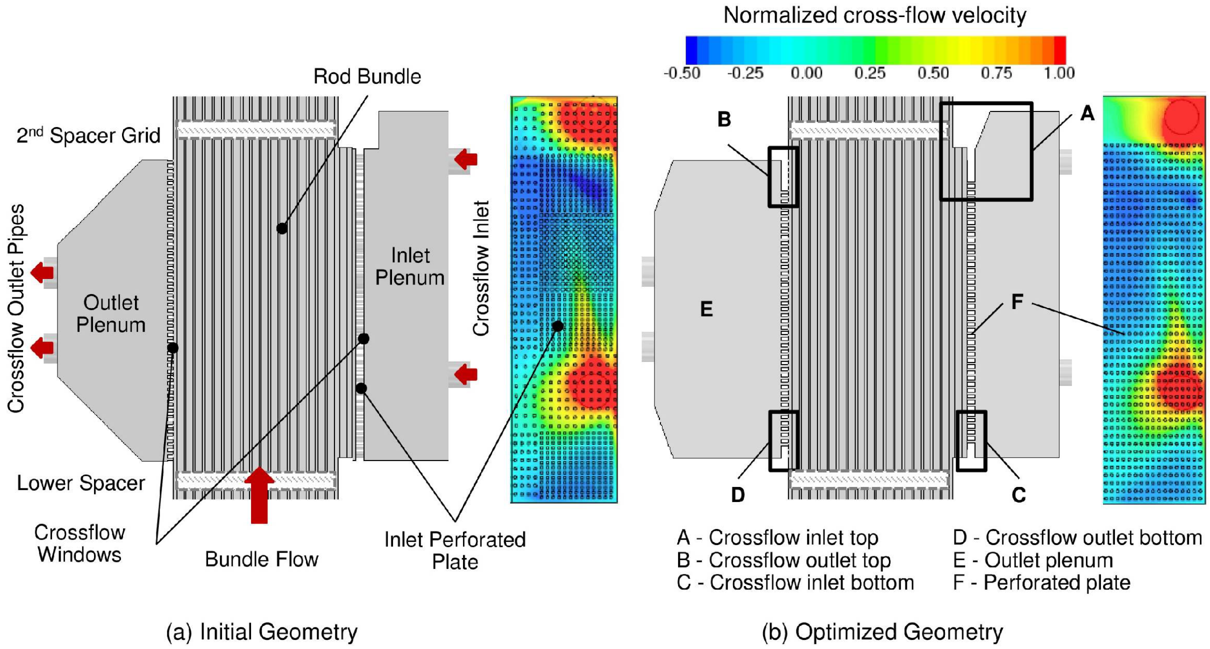

Figure 1: FIV test loop with lateral cross-flow

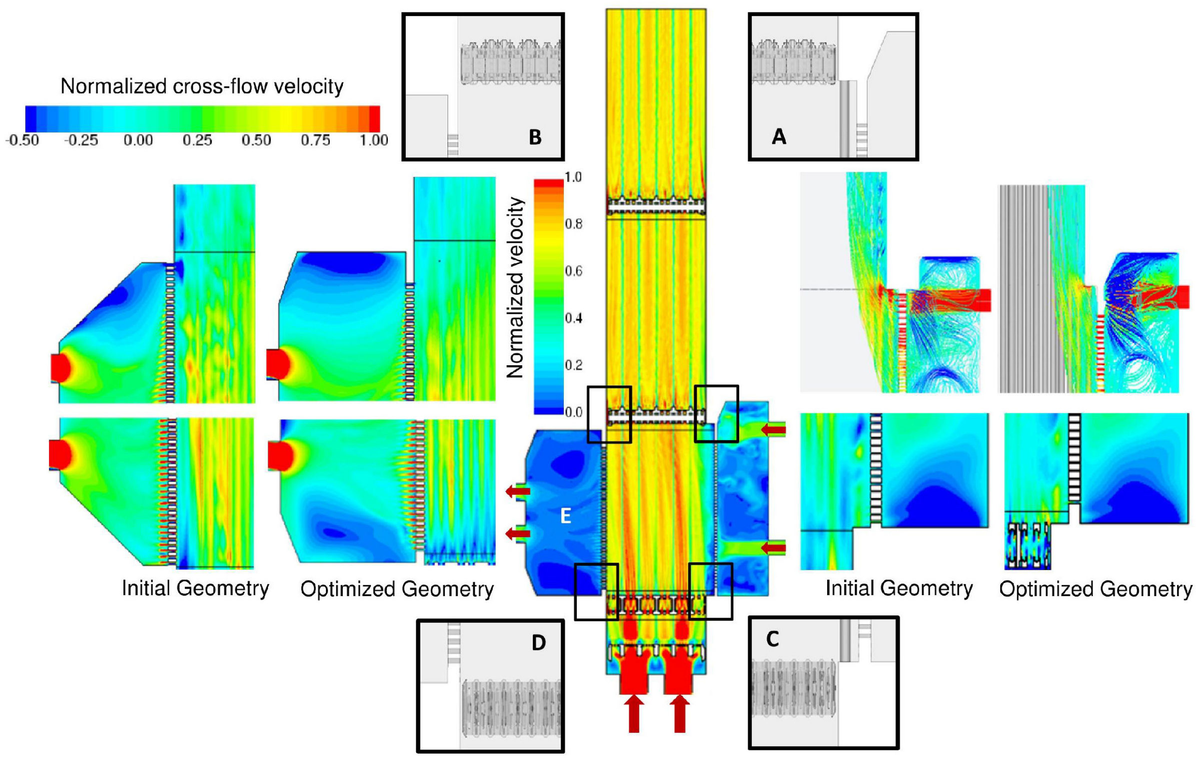

Figure 2: Local flow changes due to test section geometry adjustments

The most challenging aspect was preventing the inlet pipes impinging flow from penetrating into the bundle at the top of the inlet window (A) and inducing undesirable uncontrolled flow unsteadiness. Due to lack of space, the distance between the inlet pipes discharge plane and the inlet window could not be increased to reduce the impingement effect. The solution relied on blocking the flow at the top of the perforated plate, deflecting the impingement flow on an inclined surface, and slightly extending the top of the inlet plenum. The need to eliminate peripheral rows of holes at the top and bottom of crossflow windows led to the redesign of the perforated plates (F). Since the original targeted-pattern approach could not ensure uniform flow at the entrance to the bundle, a uniform hole-pattern was chosen instead. The initial design of the outlet plenum (E) was conducive to significant distortions of the bundle flow: strong outlet

suction on the lower part of the crossflow span and reverse flow at the top. The inflow into the bundle at the top of the outlet window was strong enough to significantly affect several downstream spans. Most of the inflow at the top of the crossflow span (B) was corrected by eliminating several rows of plate holes. However, reaching a more centered, uniform bundle flow required increasing the volume of the outlet plenum by extending the outer plenum surfaces until all available space was occupied. The modification (E) also eliminated any remaining back flow into the bundle.
The experimental program was successfully completed. The optimized test section geometry and the experimental data served as a basis for benchmarking the CFD results for the inlet region as well as for downstream spans.

3. CFD-based product design

3.1. ATRIUM 11 design study

CFD is an effective way to assess the impact of design changes [7]. Small modifications were proposed to enhance the structural robustness of the ATRIUM 11 design [6] for off normal conditions. These changes may impact the overall inlet pressure drop and the flow through the water channel as it is affecting the flow distribution close to the inlet holes of the water channel.

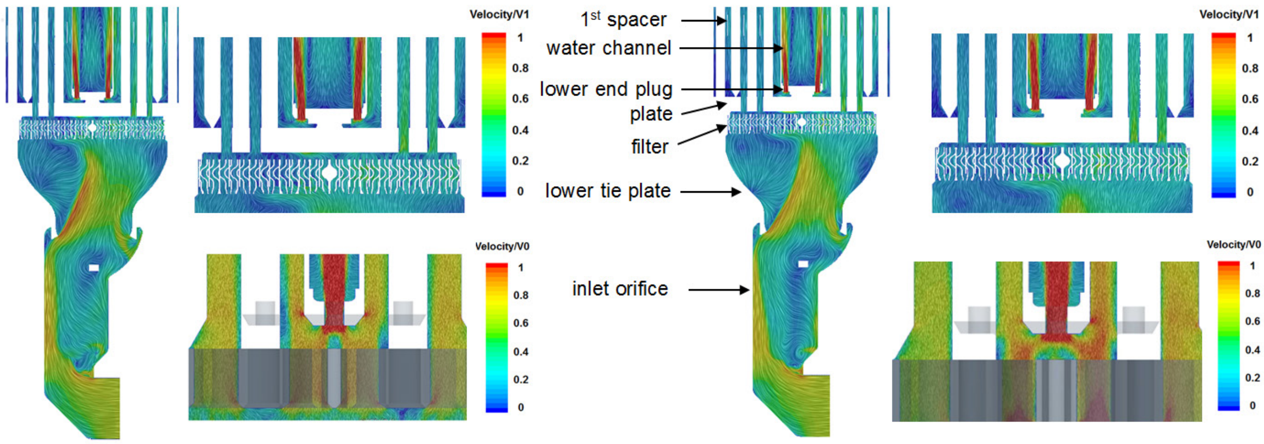

Figure 3: Impact of design change on the flow distribution around the water channel inlet. Left: current design, Right: proposed design.

The proposed design was compared to the existing design using CFD. The model was developed to properly simulate the flow in the region of interest under reactor conditions and predict the flow split between the water channel and the active bundle. The fuel elements were explicitly modelled around and upstream of the water channel inlet up to the inlet orifice (see Figure 3). The hydraulic resistance of the bundle from downstream the first spacer to the top is simulated with porous media, which is set to match the prototypic two phase pressure drop of a full bundle using measurement from the KATHY loop [17]. Figure 3 shows the flow distribution close to the inlet of the water channel for both designs. It shows that the flow is more constricted when entering the water channel with the new design. However, the impact on the pressure drop is negligible, and the flow through the water channel is reduced by only 0.1% of the total inlet flow, which is negligible as well. Due to insignificant change in performance parameters, no physical testing is required. CFD analysis was instrumental in quantifying the impact of different changes and confirming that the proposed design would preserve the thermal hydraulic performance of the ATRIUM 11 while improving its structural robustness.

3.2. Adaptation of the GAIA spacer design

Framatome has developed the GAIA fuel assembly design [12], [13], with the objective to provide an advanced FA regarding both robustness as well as performance. The GAIA product development relied on a step process where CFD-based improvements of the spacer grid geometry definition were followed by comprehensive experimental testing of the most promising variants. The iterative process yielded an optimized, high performing design.

CFD can be further used to support possible adaptations of the GAIA spacer and Mid Span Mixing Grid (MSMG) designs. As an example, the effect of a fuel rod diameter reduction, addition of vanes and outer straps modifications on the pressure loss coefficient (PLC) of the GAIA spacer and MSMG designs was determined. The modified geometries were simulated using CFD and the results were compared against the baseline geometries for which experimental data is available. The standard CFD methodology for PLC calculation was applied. As previously documented in [7], [8] the methodology relies on an accurate capturing of the true geometry of spacer details such as dimples, weld nuggets, strap cut-outs, strap edge rounding, and chamfers. RANS modelling with standard quadratic k-ε turbulence model [20], and second order spatial discretization is applied. The PLC is determined for one span of representative axial length with cyclic axial boundary conditions that simulate an infinite fuel bundle and half of a spacer geometry with periodic boundaries that take advantage of the 180° symmetry. While employing a relatively economic mesh, the current methodology produced results with a validation comparison error within the measurement uncertainty [7]. The deviation in PLC relative to the tested configuration is plotted in Figure 4 for the entire range of Reynolds numbers for the GAIA spacer and MSMG designs. With the help of CFD it can be shown that the relative change in performance is small.

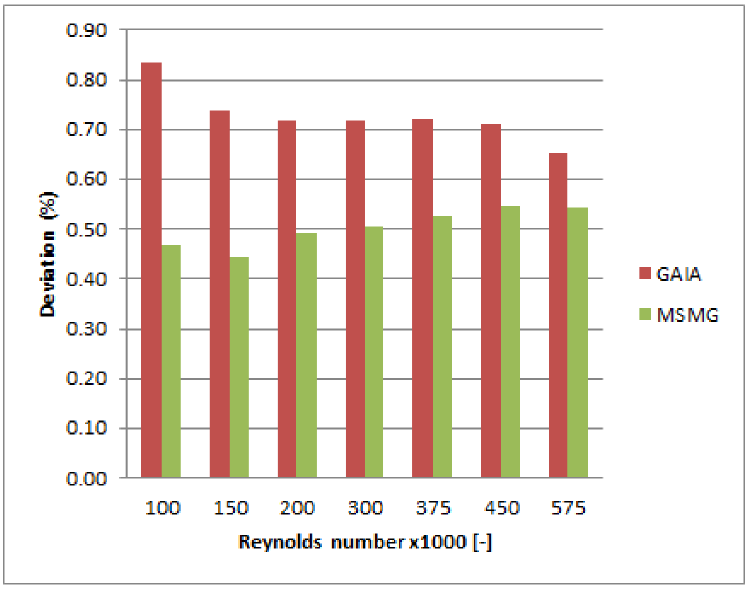

Figure 4: Normalised PLC values against Reynolds number for GAIA spacer, and MSMG.

These calculations take advantage of a significant effort put into streamlining and optimizing the PLC calculation workflow in terms of turnaround time for standard cases. It relies on a high degree of automation through JAVA macros, robust CAD generation guidelines that enable automated setup and meshing process, and a high performance cluster environment. With the improved workflow, the automated simulation set-up, including meshing, takes about 2 hours per geometry and the calculation time for the full range of Reynolds numbers containing 7 state points is about 10 hours.

3.3. Local PLC calculation for subchannel codes

Under elevated temperature and high heat flux, rod bowing can occur. One potential outcome is unwanted Departure from Nucleate Boiling (DNB). Thermal hydraulic predictive capability is required to identify this potential risk and to extract the required key data to identify the impact as accurately as possible. Thermal-hydraulic subchannel codes need local PLC values to predict DNB correctly. In order to provide a local PLC map, accounting for a realistic rod bowing, the standard CFD methodology for PLC calculation [7] was applied and extended: from a converged CFD solution the map of the local PLCs was extracted via post processing. As stated in the previous section, the PLC methodology is applied for one span and half spacer geometry, with cyclic axial and periodic lateral boundary conditions. To mimic the subchannel discretization used in thermal hydraulic subchannel codes, the CFD geometry is separated into several derived parts. This is automated by user-built Java macros. The discretization is schematically shown in Figure 4. Each colour represents a subchannel type: green depicts water gap peripheral subchannels, white depicts typical subchannels (only surrounded by rods other than guide tubes), orange depicts the guide tube subchannels (subchannels with at least one adjacent guide tube), and blue depicts the corner subchannels. Getting the PLC values for each type of subchannel cells relied on using various STAR CCM+ physical monitors such as pressure differential, and velocity. This forms a PLC map that serves as input for further use in subchannel codes.

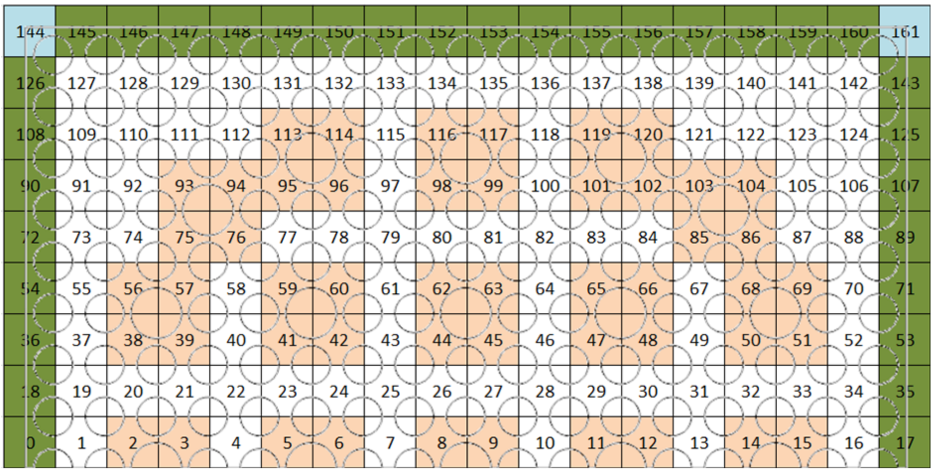

Figure 5: Subchannel discretization and numbering for a 17x17 spacer design.

3.4. Flow induced vibration (FIV) assessments

CFD simulations are also used by Framatome to assess the flow-induced vibration behaviour of fuel structures. The turbulence in the coolant flow generated by the spacer grids induces small-amplitude vibrations on the rods which may in turn lead to fretting wear. In addition, fluid-structure instabilities can be observed for fuel structures under specific hydraulic conditions which may be generated by either reactor core design features (e.g. jetting flow at baffle slots or LOCA holes) or by spacer grid design (e.g. self-induced excitation). For both of these flow-induced vibration mechanisms, CFD tools can be used to investigate design performance and support product optimization. Framatome has developed an expertise in the simulation and analysis of turbulence-induced vibration of fuel structures. Unsteady CFD simulations have been used to determine the turbulent forces acting on the fuel rods both for single-rod and industrial-scale test configurations, including full-size fuel assembly cases. A sample fluid domain considered for such type of simulation is presented in Figure 5, left and includes the lower core plate (LCP) flow passage, the bottom nozzle and a portion of the fuel bundle. Typical results consist of turbulent excitation force spectra (see Figure 5, right) which describe the excitation force amplitude and frequency content along the length of the fuel rod. This kind of spectrum is used advantageously in place of empirically-adjusted spectra that were historically used to simulate rod vibrations. The approach provides greater sensitivity to design features as the CFD simulations provide excitation spectra which are representative of the actual flow topology in the region of interest. It was demonstrated that the method developed by Framatome [14] has the potential not only to predict realistic vibration amplitudes for the fuel rods, but also to identify penalizing rod locations with regard to turbulent excitation, in agreement with reactor experience feedback.

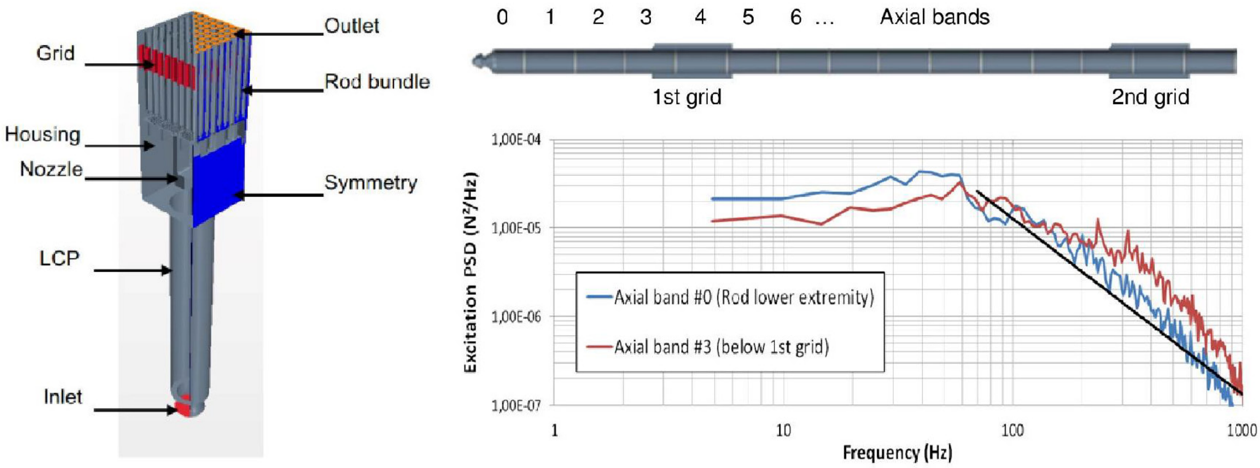

Figure 6: FIV applications sketch of a typical modelled geometry (left), and typical turbulent force excitation spectra (right).

3.5. CHF performance enhancement

A major application of CFD to fuel-related assessments is the critical heat flux (CHF) calculation. Over the last few years, a methodology for simulating the onset of Departure from Nucleate Boiling (DNB) has been developed at Framatome [9], [10], [11]. This methodology relies on the CFD code STAR CCM+ and in-house two-phase flow models implemented via user defined functions.

It was validated against CHF tests performed at the KATHY loop [16], [17]. The goal of the calculations is to estimate the critical power (i.e. the power in the rod bundle) for which DNB onset is numerically detected. This critical power value is then compared to its experimental counterpart. The KATHY loop located in Karlstein uses electrically heated rods with radial peaking as a nuclear rod surrogate. DNB is experimentally detected by sudden temperature excursion. 5x5 arrays of rods were tested in various thermal hydraulic state points, grid designs (with and without mixing vanes, with and without a central guide tube), and axial power distribution shapes (uniform or cosine).

The methodology is based on two Eulerian multi-phase calculations: one slightly above experimental critical power, the other slightly below. Results are then interpolated between the two values. Numerical DNB detection is based on local physical values provided by CFD post-processing. When those values are above a given threshold, then DNB is numerically detected. The threshold values were empirically established by calibration with a representative set of experimental data, including tube tests [15], and KATHY data for various grid designs. CFD mesh generation implements the Framatome's best practices for thermal modeling [1]; it relies on the STAR CCM+ trimmer module that generates a hex-dominant mesh. Specific focus is on small-scale details at the fuel rod and spacer. Calculations are steady state and the turbulence is modeled with the standard k-epsilon closure laws.

Over 60 state points were run and compared to experimental results for several Framatome grid designs. The results show overall a good agreement with experimental data. Indeed, simulated results are mainly located within +/- 5% of experimental critical power (see Figure 6). The main focus for the application of the methodology is on supporting next generation fuel product development and CHF correlation improvement.

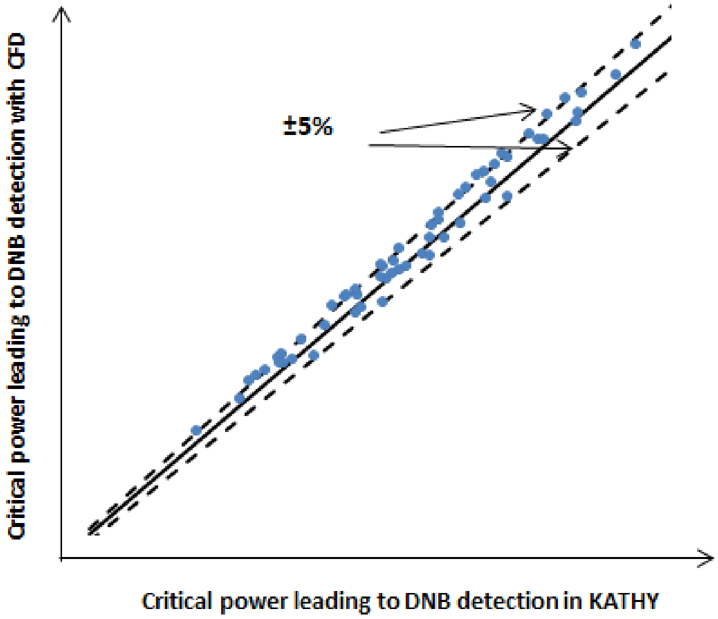

Figure 7: Comparison of experimental vs. simulation values for critical power leading to DNB detection.

4. Fuel assembly bow prediction

In pressurized water reactors (PWRs) the fuel assemblies are subject to distortions during their whole life cycle. FA lateral distortions or bow results from numerous complex and coupled phenomena: creep under irradiation, hydraulic forces, axial stresses, reactor design, plant type, operational history, etc.

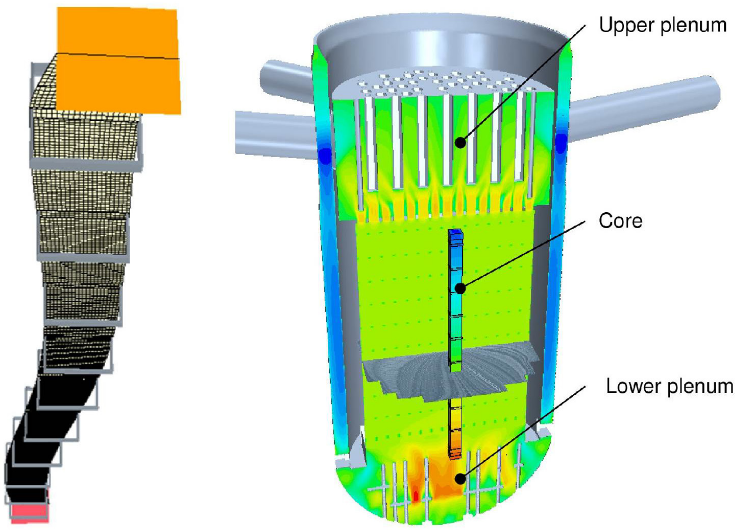

Figure 8: Schematic representation of the FA bow (left), simulation result of the 3D coupled tool (right).

The FA bow cannot be prevented but can be minimized by certain measures such as fuel design and core loading pattern. Framatome has developed a patented powerful tool describing the multi-physics to predict FA distortions in the reactor core under operating conditions [19]. A 3D coupled Fluid Structure Interaction (FSI) approach is employed: it couples a nonlinear mechanical model of the FA structure with CFD to model the flow in the full reactor vessel, including explicit models of the lower and upper plena and porous media for the core (see Figure 7). The FSI methodology was validated against full scale FSI tests. For a given fuel cycle, the simulation can predict the distortion of each FA of a complete core at every spacer grid position. Very good agreement with measured fuel bow was seen for real cycles.

The tool is used to perform generic simulations to obtain responses of the expected bow for new designs in an early stage of development. The tool was instrumental in the GAIA design process. Different core loading patterns could be analysed before or even during the outage. This allows selecting the best in-core fuel management regarding the expected bow behaviour and minimizing the impact on the reactor operation, core loading and unloading. Thus, the tool efficiently contributes to avoiding FA bow-related issues and finally leads to an improved reactor core performance.

5. Conclusion

CFD has become a valuable tool for Framatome. Validated CFD methodologies play an important role in the design phase, providing cost-effective means for optimizing all aspects of Framatome's fuel hardware functional features. CFD calculations are routinely used to perform relative performance assessments, to provide insights into challenging flow fields and phenomena outside the scope and capability of traditional subchannel codes, and to deliver better products and services to the customer.

This paper highlighted some of Framatome's CFD-based engineering applications. The first part was dedicated to pre-test CFD-based evaluations and optimization of experimental setups. The second part highlighted applications supporting the development of advanced fuel assembly designs for pressurized and boiling water reactors. Finally, the paper concludes with CFD application that is used to predict the FA distortion.

Nomenclature

CFD	Computational Fluid Dynamics	LES	Large Eddy Simulation
CHF	Critical Heat Flux	LOCA	Loss Of Coolant Accident
DNB	Departure from Nucleate Boiling	MSMG	Mid Span Mixing Grid
FA	Fuel Assembly	PIV	Particle Image Velocimetry
FIV	Fluid Induced Vibration	PLC	Pressure Loss Coefficient
LCP	Lower Core Plate	PWR	Pressurized Water Reactor

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", NURETH-16, Chicago, USA , 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", NURETH-15, Pisa, Italy, 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", NURETH-15, Pisa, Italy, 2013.

[5] 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", NURETH-15, Pisa, Italy, 2013.

[6] S.E. Cole, N. Garner, S. Mazurkiewicz, V. Schoss, and P. Mollard, "ATRIUM 11 Operating Experience", TOPFUEL 2016, Boise, USA, 2016.

[7] M. Martin, A. Hatman, A. Chatelain, K. Goodheart, "AREVA NP's industrial CFD single-phase methodology and applications for nuclear fuel", TOPFUEL 2016, Boise, USA, 2016.

[8] A. Hatman, A. Chatelain, K. Goodheart, M. Martin, T. Keheley, "A Review of Areva's Experimental validation State of the Art Single Phase CFD Methods with Application to PWR Fuel Analysis and Design", NURETH-16, Chicago, USA, 2015.

[9] A. Chatelain, K. Goodheart, N. Alleborn, C. Baudry, J. Lavieville & T. Keheley, "Two-Phase CFD Simulation of DNB and CHF Prediction Behind Spacer Grids with NEPTUNE_CFD and STAR CCM+", TOPFUEL 2012, Manchester, UK, 2012.

[10] K. Goodheart, N. Alleborn, A. Chatelain & T. Keheley, "Analysis of the Interfacial Area Transport Model for Industrial 2-Phase Boiling Flow Applications", NURETH-15, Pisa, Italy, 2013.

[11] K. Goodheart, A. Ylönen, V. De Cacqueray, H-M Prasser, "CFD Validation of Void Distribution in a Rod Bundle with Spacer", ICONE 22, Prague, Czech Republic, 2014.

[12] G. A. Thomas, J.S. D'Orio, G. Gentet, P.-H Louf, M. Mindt, "GAIA: Areva's Advanced PWR Fuel Design", TOPFUEL 2013, Charlotte, NC, USA, 2013.

[13] O. Wieckhorst, H. Gabriel, M. Bär, M. Anghelescu, R. Harne, O. Martinie, „ORFEO - A CHF Correlation for GAIA, Areva's advanced PWR Fuel Assembly Design", NURETH-16, Chicago, USA, 2015.

[14] J. Pacull, Y. Akariouh, A. Hatman, H. Hadzic, G. Williams, K. Karazis, "Nonlinear analysis of fuel rods subjected to turbulence excitation", IMECE, Tampa, Florida, 2017.

[15] D.C. Groeneveld, "The 2006 CHF look-up table", Nuc. Eng. and Design 237, 2007.

[16] O. Wieckhorst, J. Kronenberg, H. Gabriel, S. Opel, D. Kreuter, T. Berger, R. Harne, K. Greene, F. Filhol, „Areva's test facility KATHY: Robust Critical Heat Flux measurements, a prerequisite for reliable CHF prediction", ICONE 22, Prague, Czech Republic, 2014.

[17] D. Kreuter, O. Wieckhorst, T. Berger: „KATHY: AREVA's Test Facility for Critical Heat Flux measurements", TOPFUEL 2015, Zürich, Switzerland, 2015.

[18] 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", NURETH-15, Pisa, Italy, 2013.

[19] C. Lascar, J. Champigny, A. Chatelain, B. Chazot, N. Goreaud, E. Méry de Montigny, J. Pacull, H. Salaün: "Advanced predictive tool for fuel assembly bow based on a 3D Coupled FSI approach", TOPFUEL 2015, Zürich, Switzerland, 2015.

[20] E. Baglietto, H. Ninokata, T. Misawa, "CFD and DNS Methodologies Development for Fuel Bundle Simulations", Nuclear Engineering and Design, Vol. 236, pp. 1503-1510, 2006.