CFD PREDICTION OF PRESSURE DROP FOR THE INLET REGION OF A PWR FUEL ASSEMBLY

Jin Yan, Kun Yuan, Emre Tatli, David Huegel, Zeses Karoutas

Westinghouse Electric Company; 5801 Bluff Road, Columbia, SC, 29209
yan3j@westinghouse.com

Abstract

Computational Fluid Dynamics (CFD) has been widely used in pressure drop/hydraulic loss calculations in many industries, including the aerospace and turbo-machinery industry as well as in the nuclear power industry. During the past 30 years, a significant amount of time and effort have been spent in benchmarking the CFD tools and models against test data in aerospace and turbo-machinery industry. The application of the CFD methods in the nuclear power industry is relatively new in comparison and has accelerated in recent years. This paper focuses on the benchmarking of the CFD modeling of Westinghouse fuel assemblies. The pressure drop across the fuel assembly is a very important characteristic to consider in the design of any new fuel assembly design. The ability to accurately predict the pressure drop across the fuel assembly is critical in nuclear fuel design. In this paper, the effect of the turbulence model in the CFD modeling is investigated. Different computational meshes are generated and evaluated. The results show that the turbulence model and mesh density both have a significant effect on the pressure drop predictions. A well calibrated modeling procedure is needed to meet the strict requirements of the nuclear power industry. This paper shows that a well calibrated CFD model shortens the design cycle and saves costs in the development of new fuel assembly components. Most importantly, it enables Westinghouse to more easily investigate new fuel assembly component designs, some of which were never thought possible before because of the involved development process.

1. INTRODUCTION

A typical Westinghouse Pressure Water Reactor (PWR) fuel assembly consists of a debris filter bottom nozzle, a Protective-Grid, support grids, spacer grids, fuel rods, skeleton, and a top nozzle. A fuel assembly sits on the lower core plate which has four flow holes per assembly. The lower core plate holes direct the flow from the reactor vessel lower plenum into the bottom nozzle of the fuel assembly. The overall fuel assembly pressure drop has to be determined for any change in design of the fuel assembly components. Traditionally the tests were required to evaluate the pressure drop at different elevations of the fuel assembly under low pressure conditions as compared to the real reactor operating conditions. Typically, only a single fuel assembly is tested within a rectangular wall surrounded enclosure. The test generally takes a long time to prepare due to the component design and fabrication time and assembly time. There are also uncertainties in the test results. Due to the complexity of the modern fuel assembly and the increasing demand from the customer on delivery time, it is important that the design cycle time be shortened. All of these factors limit the number of designs that can be evaluated in a given period of time. Therefore, the CFD approach was proposed as an evaluation tool for pressure drop and flow field evaluation for different component designs. Recent advances in CFD and computing resources have made it feasible to simulate the 3D flows through complex geometries in the reactor core and specifically in a fuel assembly. CFD has been used to investigate flow and heat transfer in fuel assemblies [1, 2]; estimate hot-spot conditions in rod bundles [3]; improve spacer grid mixing vane designs [4]; and to perform two-phase flow and CHF analysis [5]. However, CFD investigations of the fuel inlet region is rather limited, partly due to the complex geometry to be resolved with adequate mesh elements and limited experimental data to be compared with. A recent study [6] benchmarked the pressure loss through the perforated plates, which are similar to bottom nozzles used in the fuel assemblies. This study, however, considered only the perforated plates and did not include the P-grid, bottom grid and fuel rods; the maximum number of holes in this study was 64, while a typical bottom nozzle usually has more than 600 flow holes. Therefore, to improve the fuel assembly design process, this CFD study includes all necessary geometric features and compares the results with experimental measurements. In 2009, an effort to develop a CFD model for the pressure drop prediction for fuel assemblies took place in the Westinghouse Nuclear Fuel division. The purpose was to develop a CFD model based on the Westinghouse fuel thermal hydraulics test facility and to benchmark the CFD model against test data.

2. HYDRAULIC TEST FACILITY

The tests were performed in the Fuel Assembly Compatibility Test System (FACTS). This mobile, isothermal, closed-loop test system was designed to provide single-phase hydraulic data from which pressure drop characteristics of fuel assemblies could be determined. In the FACTS loop, loop flow, which is measured by a venturi flow meter, is controlled by a variable frequency drive for the main pump motor in conjunction with a pneumatically operated proportional control valve. Heat is injected into the system solely by the work done on the fluid by the main pump. Temperature is controlled by adjusting the flow of cooling water on the secondary side of the heat exchanger. Loop pressure is established by a pneumatically driven hydraulic pump acting against a backpressure regulator. A pressure relief valve provides protection against over-pressurization of the loop. A rupture disc provides additional protection. The design operating limits of the system are 250°F and 225 psig. All pressure boundary components were designed in accordance with Section VIII of the ASME Boiler and Pressure Vessel Code and ANSI/ASME Standard B31.1 for boiler external piping.

The instrumented flow housing is custom-designed for each fuel assembly type to strategically place pressure taps for optimum data acquisition. Likewise, the upper and lower core plates are customer designed in order to model the specific reactor geometry to assure correct hardware interface and to simulate correct inlet and outlet flow conditions. The core plates are positioned in the flow housing at the representative reactor cavity height for a Westinghouse 17x17 core.

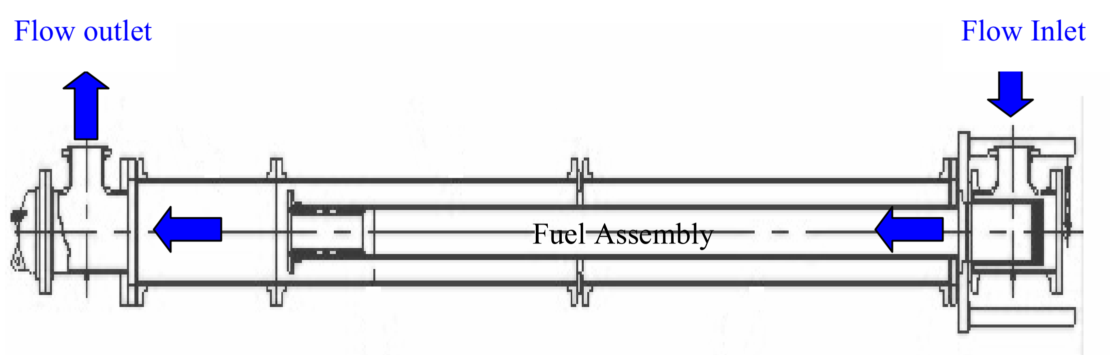


Fig 2-1: FACTS test facility

Figure 2-1 is a general illustration of the test vessel including the flow housing. The flow housing is mounted inside the pressure vessel. This figure is rotated 90 degrees from the actual test positions of the fuel assembly in a vertical configuration. Figure 2-2 shows a schematic of the elevations and connections of the pressure taps for the test.

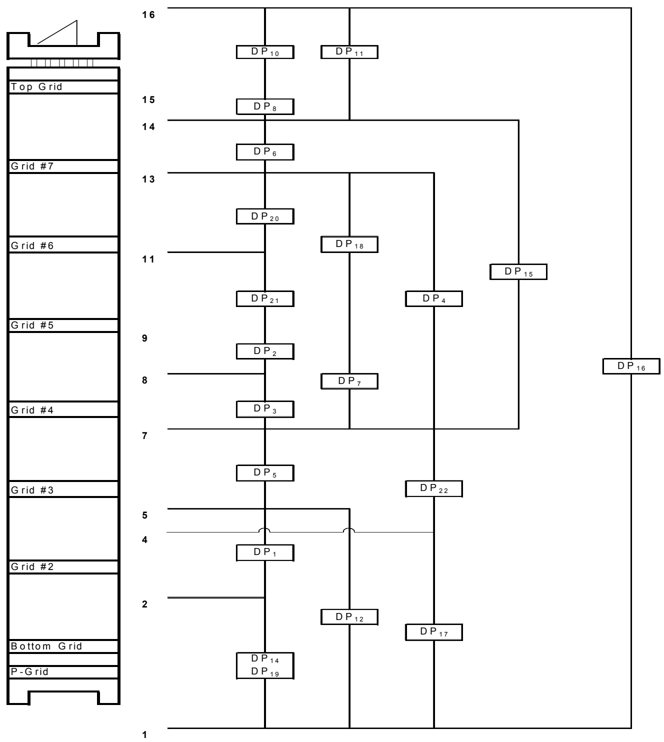


Fig 2-2: Pressure Tap and Grid Elevations

The data is then post-processed to reflect the Reynolds number effect on the fuel assembly pressure loss. The overall uncertainty of the test data is 5%. These data were then used as the basis for the purposes of performing detailed Computational Fluid Dynamics analyses (CFD), as discussed below.

3. CFD MODELLING

In this section, the CFD modeling approach of the FACTS test results is discussed.

3.1 Geometry

The whole fuel assembly has many spacers and each spacer has a complicated geometry. The focus of this study is the fuel inlet section. The computation domain includes the lower core plate, the standard bottom nozzle, P-Grid and Bottom Grid as shown in Fig 3-1. Full details of the geometry are included. Because of the symmetry of the geometry, a triangle portion of a quarter of the fuel assembly was used in the CFD model. As can be seen, an inlet duct was modeled below the lower core plate to establish the flow development in the entrance region. Figs 3-2 and 3-3 show the detail of the standard bottom nozzle, P-Grid and Bottom Grid used in the simulation. A computation fluid domain was extracted from the detailed solid model and all the geometry features were kept in the fluid domain. Fig 3-4 illustrates the bottom nozzle region and bottom grid region of the fluid domain.

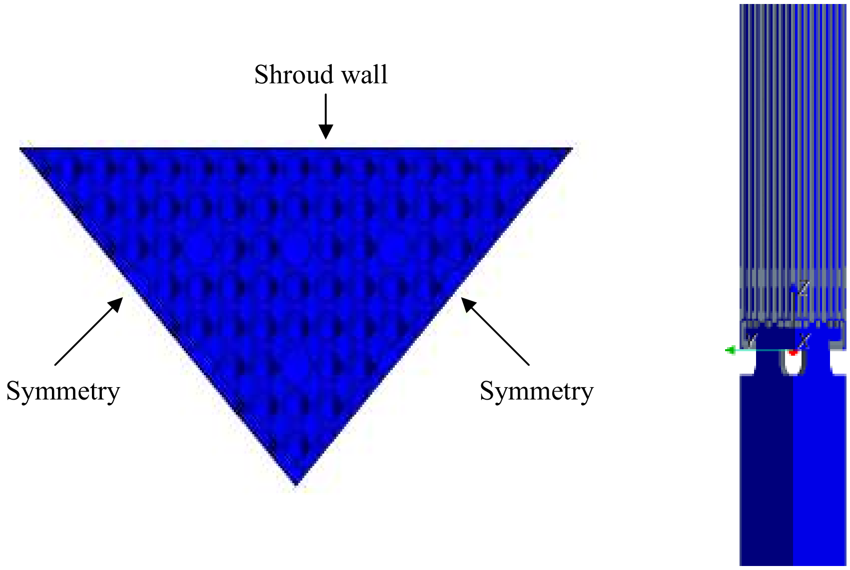


Figure 3-1: Fuel assembly inlet computational domain

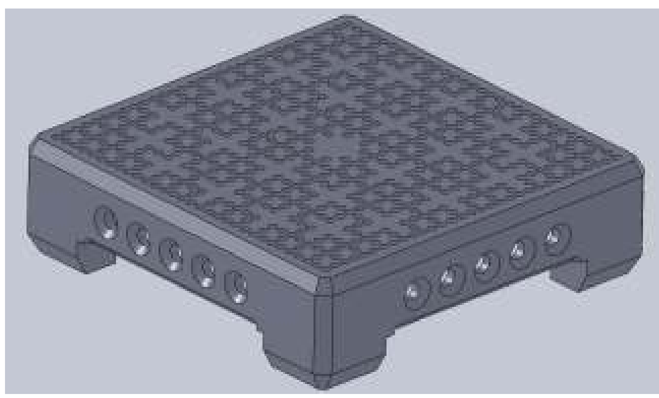


Figure 3-2: Solid model of the bottom nozzle

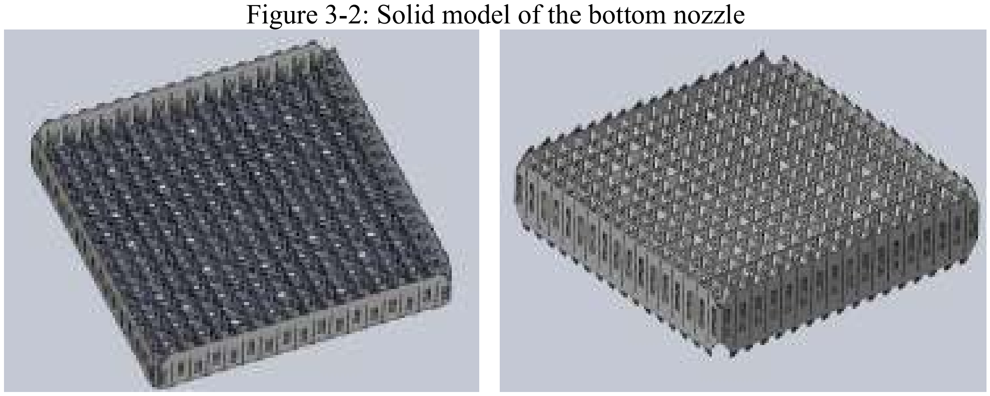


Figure 3-3: Solid model of the P-Grid (left) and Bottom Grid (right)

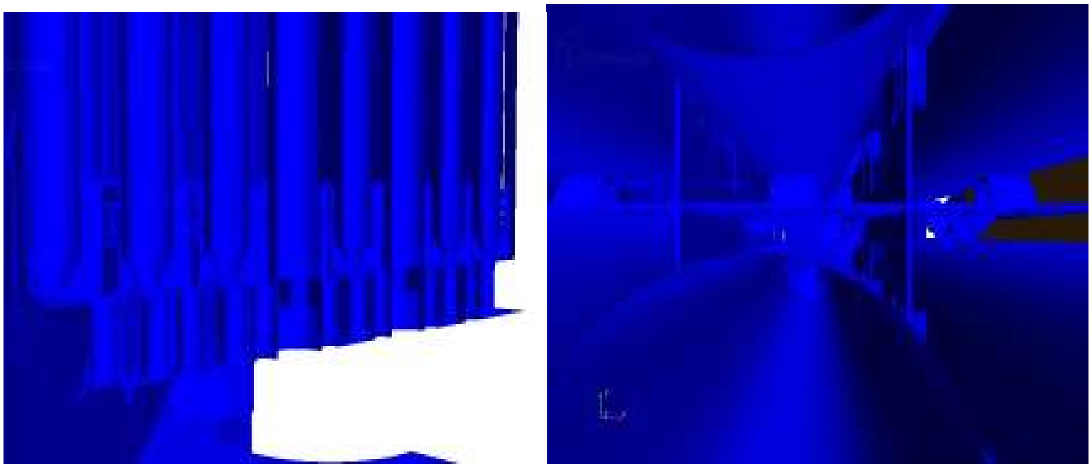


Figure 3-4: Solid model of the P-Grid (left) and Bottom Grid (right)

3.2 Computational mesh and mesh sensitivity study

The fluid domain in parasolid format was imported into StarCCM+. The boundary faces such as the inlet, outlet, wall, and symmetry, were identified and defined. The trimmer, surface remesher, and prism layer mesher were used to generate hexahedral cells. Due to the complexity and dimension difference in the model, local refinement zones were defined to capture the intricate details of the bottom nozzle holes, grid springs, and grid dimples. Different mesh resolutions were attempted for the grid sensitivity study. The differences between the CFD results and the test data were listed in Table 1. The errors in the Table 1 were calculated based on the pressure drop measured at D19 in the test. As the mesh density increases, the CFD results get closer to the test data. The mesh sensitivity study also uncovered that the results are sensitive to the geometry of the flow holes in the bottom nozzle. This finding was also supported by results from reference [6]. The bottom nozzle has hundreds of small flow holes that are chamfered at the inlet and outlet. The CFD results show that these chamfers affect the pressure drop across the bottom nozzle. The chamfer dimension and angle needed to be resolved by increasing the mesh density for the CFD results to be closer to the test results. This requirement also revealed the challenge in simulating the fuel assembly inlet region: The overall dimension of the fuel assembly and the bottom nozzle hole chamfers are at a difference of 1000:1. In order to get accurate results, the mesh has to be refined further leading to a larger mesh size. Even with local refinements, a typical mesh that resolves the geometry details contains about 60 million cells. It requires at least 8 hours to generate this kind of mesh on a single CPU with sufficient memory. A large mesh size also demands more computer resources in running the simulation and post-processing the results. Fig 3-5 shows a typical mesh on the surface of the fuel rods and the bottom nozzle holes. The meshes used in the mesh sensitivity study are displayed in Fig 3-6.

Table 1: Mesh sensitivity study.

	Initial mesh (7M cells)	*Refined mesh (16M cells)*	*Refined mesh 2 (28M cells)*	*Refined mesh 3 (50M cells)*	*Refined mesh 4 (60M cells)*	*Test results*
Error	41.9%	17.3%	14.8%	10.4%	8.7%	N/A

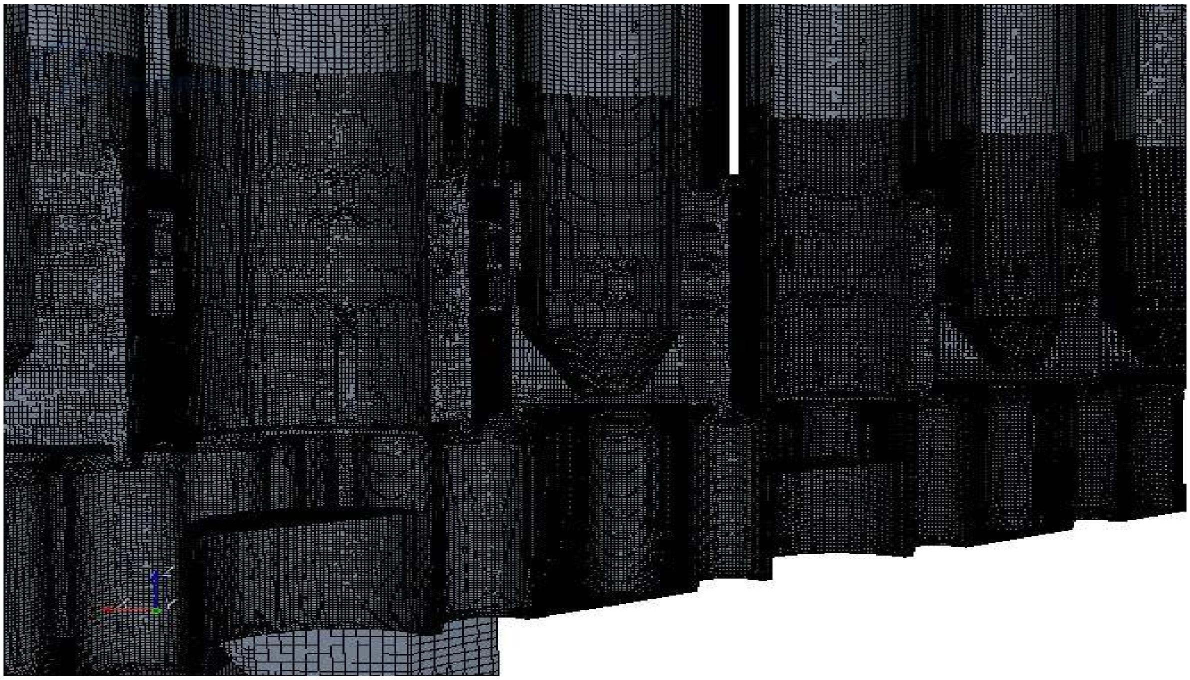
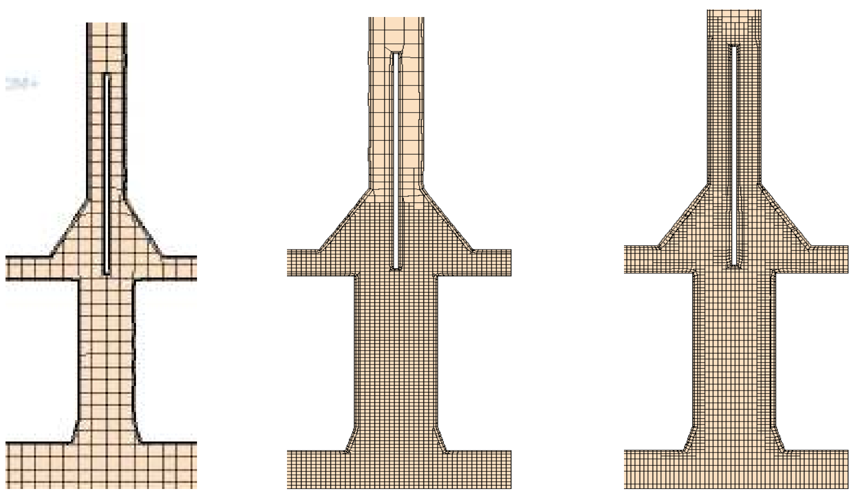


Figure 3-5: Computational mesh



7M cells 16M cells 60M cells

Figure 3-6 Mesh sensitivity study

3.3 CFD Model Setup

The FACTS test was carried out without heat transfer. The iso-thermal segregated solver in StarCCM+ was used. The segregated solver employs the SIMPLE scheme. It solves the momentum and pressure sequentially. It is suitable for general flow without strong body force. Water with constant density and viscosity was used as working fluid. Different turbulence models, i.e., realizable $k-\varepsilon$, $k-\omega$, and Reynolds Stress Model were tested with the coarse mesh. The Reynolds Stress Model failed to converge. The results from realizable $k-\varepsilon$ model gave the best comparisons to the test data. Therefore to study the different mesh refinement levels, a realizable $k-\varepsilon$ model was used. Second order upwind schemes were used for momentum and turbulence. At the inlet of the computational domain, the mass flow inlet was used. A number of flow rates were investigated as shown in Table 2. The outlet was set as the pressure boundary. Since only a quarter of the geometry was included in the model, a symmetric boundary condition was defined at the two geometrical symmetry planes.

Table 2: Flow rate at inlet

Reynolds number	60144	65815	79573	92161	105527	123707
Flow rate (GPM)	998.5	1095.6	1301.1	1501.7	1704.4	2000.9

3.4 CFD results & discussions

The calculations were carried out on 80 parallel processors. Each flow rate case took around 8 hours and 1500 iterations to reach full convergence.

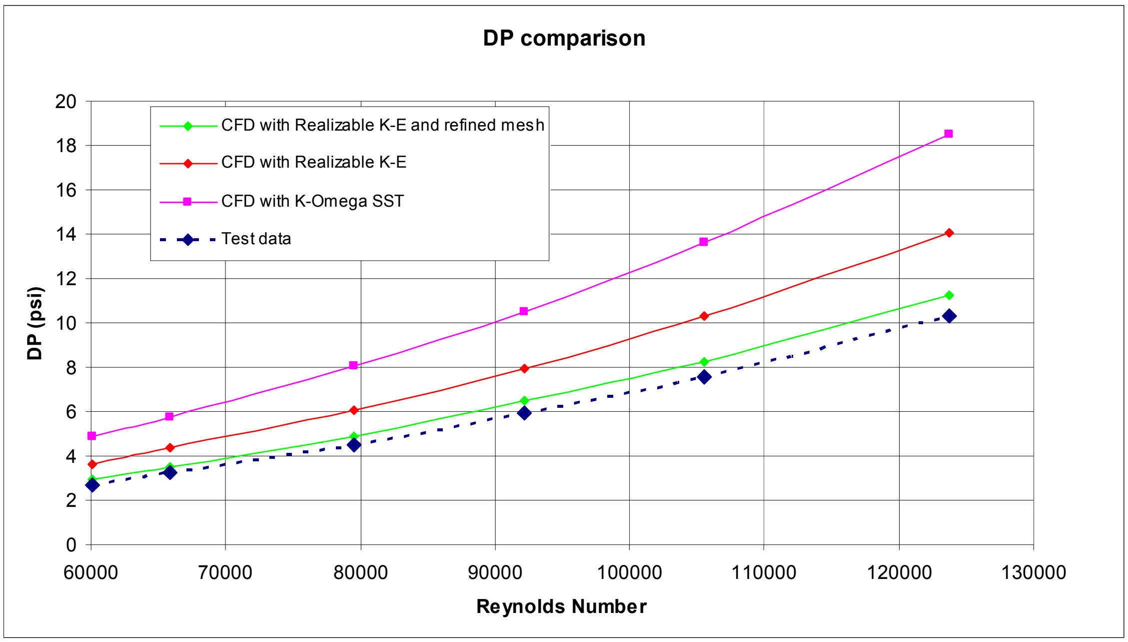
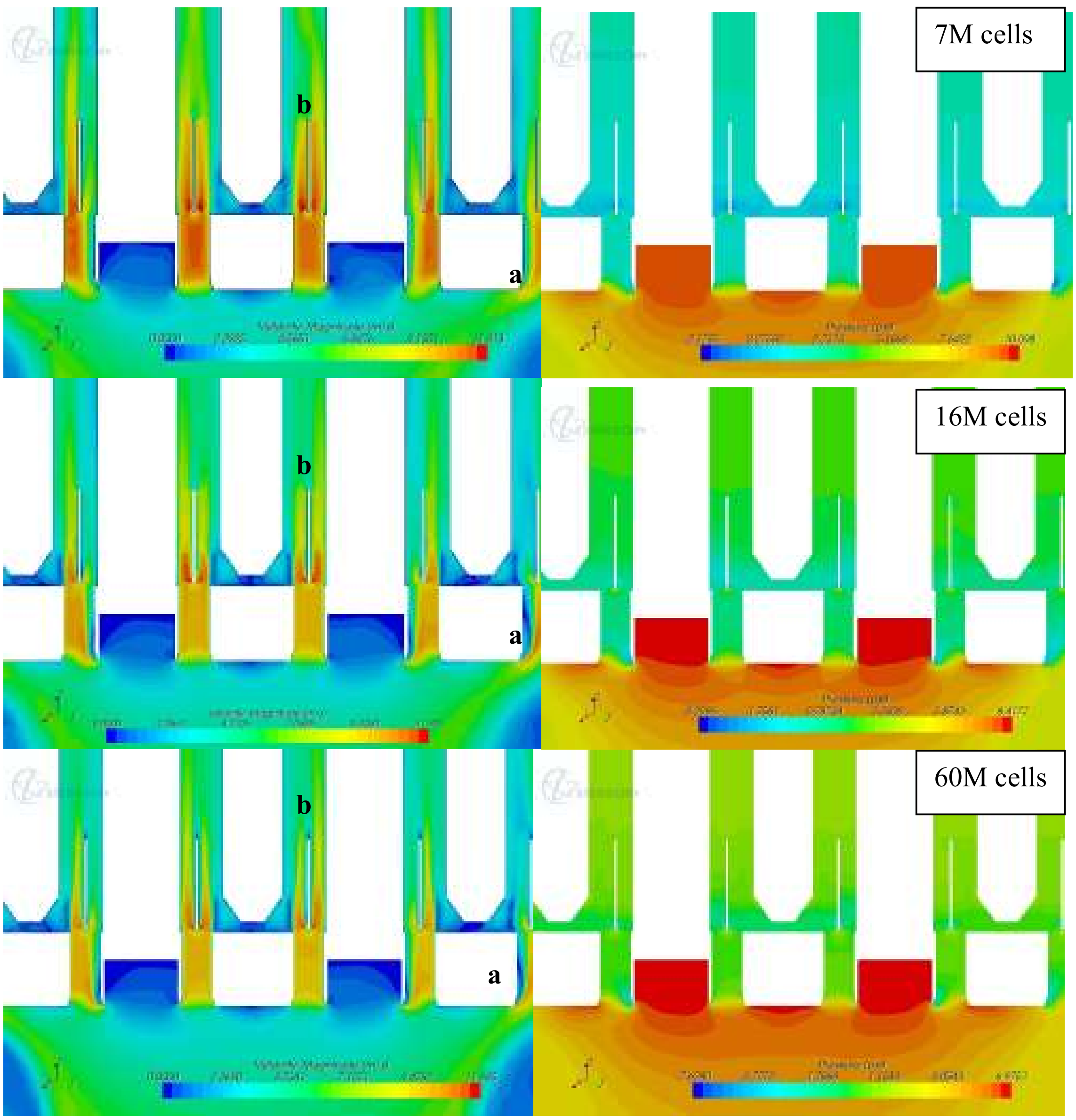


Figure 3-7 CFD results comparison with the test data

In the test, the pressure was measured at elevations of -7.25 inches and 17.58 inches, where the top of lower core plate surface (bottom of the bottom nozzle) was defined as zero as shown in Figure 2-3. There were 2 pressure taps at each elevation on the opposite side of the housing. These two pressure measurements were averaged to get the final reading. The pressure difference between the two above elevations was named as D19 as shown in Figure 2-3. The CFD results were post-processed at the same elevation by averaging the pressure at the elevation planes. The averaged pressure was then compared to the test results in Fig 3-7. It was found that the initial coarse mesh gives a much higher pressure drop compared to the test although it shows a similar trend. The $k-\omega$ SST model in StarCCM+ gave a much higher pressure drop compared to the realizable $k-\varepsilon$ turbulence model. It was then decided that the realizable $k-\varepsilon$ turbulence model should be used for the study.

After refining the mesh to around 60 M cells, the predicted pressure drop from CFD is much closer to the test results with an averaged difference of 8.7%. Further refinement of the mesh was attempted, but the simulation results were hardly changed. Considering the 5% uncertainty in the test data, it was concluded that the results from CFD are acceptable and the future CFD simulations should use the same approach.

In order to understand the effect of the mesh density, the flow field from the three different meshes shown in Figure 3-6 was investigated. Figure 3-8 displays the velocity distribution across a plane for all meshes. The results from the 60M cells model showed separation bubbles at the inlets of the bottom nozzle holes (location “a” in the figure). The low velocity region behind the trailing edge of the P-Grid (location “b” in the figure) was captured. The 16M cells model captured the separation bubbles at the same location. However, the low speed region behind the trailing edge of the P-Grid was not captured. In the 7M model, it did not show separation bubbles at the low velocity region downstream of the trailing edge of the P-Grid. The pressure contour plots of 7M cells model illustrated a low pressure region at the exits of the bottom nozzle hole. The model with the finer mesh showed a smoother pressure transition through the bottom nozzle holes. It demonstrates that the unique structure of the bottom nozzle requires a mesh with adequate refinement to be able to resolve the velocity and pressure gradients in this region.



Velocity Pressure

Figure 3-8 Velocity and pressure contours at the symmetry plane of the fuel assembly

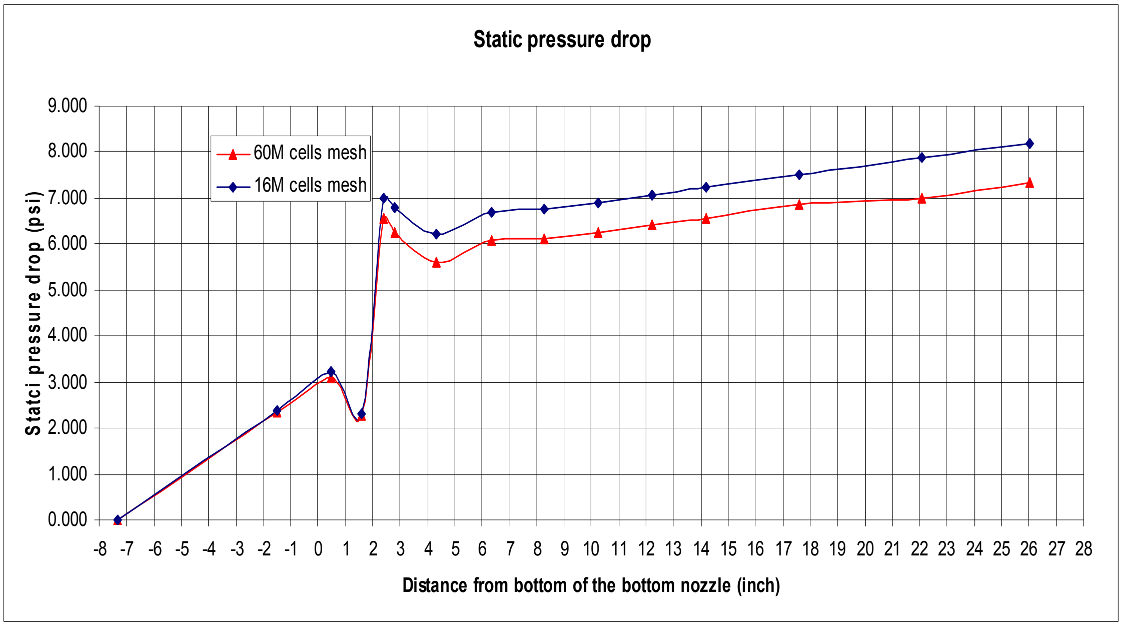


Figure 3-9 Area-averaged pressure drop at difference elevations in the fuel assembly

In order to understand the behaviour of flow, the area-averaged pressure drop is plotted at the different elevations in the fuel assembly in Figure 3-9. As the flow goes through the lower core plate holes, it accelerates and the pressure drop increases. At the 0 inch elevation, the flow starts entering the bottom nozzle. The flow expands before it accelerates into the bottom nozzle holes. It causes a pressure drop decrease and then a rapid increase between the 1.5 inch and 2.5 inch elevations, corresponding to the bottom nozzle flow hole inlet and outlet locations. Once the flow exits the bottom nozzle flow holes, it decelerates and causes a pressure drop decrease. At the bottom nozzle flow hole inlets and outlets, the velocity gradient is very high as shown in Figure 3-8. The high velocity gradient requires a fine mesh to resolve it. The majority of the mesh refinement in the 60M cells mesh was at the bottom nozzle flow hole locations. Figure 3-9 shows that the only significant difference in the pressure predictions from the two meshes happens across the bottom nozzle. This demonstrates the significance of the computational mesh in the PWR fuel inlet calculation.

4. Conclusions

A fuel inlet CFD model has been developed. Extensive investigation on the effect of the turbulence models and the mesh density has been carried out. The investigation revealed that a mesh sensitivity study is necessary to develop an accurate CFD model. The challenges in the fuel assembly CFD modelling have been discussed. The ability to capture the exact details of the bottom nozzle hole including the chamfer dimensions has a significant impact on the pressure drop predictions. The nature of the PWR fuel bottom nozzle design proves to be very sensitive to the pressure drop across the bottom nozzle. Something as minor as a manufacture tolerance might have an impact on the CFD results. The existence of areas with a high velocity gradient makes it necessary to have a fine mesh in the CFD model in order to achieve results that closely match the test results.

REFERENCES

[1] B. Liu, L. D. Smith, 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", 13th International Conference on Nuclear Engineering, Beijing, China, May 16-20, ICONE13-50924, 2005.

[2] H. Anglart, O. Nylund, N. Kurul, M. Z. Podowski, “CFD prediction of flow and phase distribution in fuel assemblies with spacers”, Nuclear Engineering and Design, 177, 215–228, (1997).

[3] K. Ikeda, Y. Makino, M. Hoshi, “Single-phase CFD Applicability for Estimating Fluid Hot-spot Locations in a 5 X 5 Fuel Bundle”, Nuclear Engineering and Design, 236, 1149-1154 (2006).

[4] B. Liu, M. B. Dzodzo, D. V. Paramonov, L. D. Smith, M. E. Conner, M. Y. Young, “Application of CFD in the Design Process for PWR Spacer Grid Mixing Vanes”, Proceedings of the 2004 International Meeting on LWR Fuel Performance, Orlando, FL, USA.

[5] B. S. Shin, S. H. Chang, “CHF Experiment and CFD Analysis in a 2 X 3 Rod Bundle with Mixing Vane”, Nuclear Engineering and Design, 239, 899-912 (2009).

[6] J. B. Filho, M. A. Navarro, A. A. C. dos Santos, “Experimental and Calculated Pressure Loss through Perforated Plates Similar to the Ones of the Bottom End Pieces of Reactor Fuel Elements”, 13th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH -13) Kanazawa, Ishikawa Prefecture, Japan, September 27-October 2, 2009.