NURETH14- XXX

RANS MODELING FOR FLOW IN NUCLEAR FUEL BUNDLE IN PRESSURIZED WATER REACTORS (PWR)

A. Gandhir[1], Y. Hassan[1]
[1] Texas A&M University, College Station TX, USA

Abstract

This paper presents use of Reynolds-Averaged Navier-Stokes (RANS) based turbulence model for single-phase CFD analysis of flow in Pressurized Water Reactor (PWR) Assemblies. An open source code called OpenFoam was used for computational fluid dynamics (CFD) study using computational meshes generated using Shari Harpoon. The PWR assembly design used in this analysis represents a 5 x 5 pin design including structural grid equipped with mixing vanes. The design specifications used in this study were obtained from the experimental setup at Texas A&M University and the results obtained are used to validate the CFD software, algorithm, and the turbulence model used in this analysis.

1. Introduction

A traditional PWR consists of 17 x 17 nuclear fuel assemblies that are supported by structural grids over the length of each particular assembly. The thermal energy produced from the nuclear reaction of the fuel is extracted from these assemblies using forced convection with water as the active coolant. Under normal operating conditions, PWRs are designed to have primarily single-phase flow inside the core. However, localized sub-cooled nucleate boiling does occur during normal operation. This localized sub-cooled boiling may cause crud build-up which is undesired from a reactor safety viewpoint [1].

Although, the phenomenon of interest from a safety perspective is of two-phase flow in nature, understanding single-phase flow inside these assemblies under normal operation is just as important. It will shed light on the flow characteristics in such assemblies, the effect of the structural grids and mixing vanes on the flow structure and their effect on overall heat transfer in the rod bundle assemblies. For this study, a 5 x 5 rod bundle test geometry is used with a concept structural grid with mixing vanes. CFD analysis using RANS based realizable k-epsilon (RKE) and shear stress transport k-omega (SST κ-ω) model was performed under isothermal conditions at a Reynolds number of 23,000.

2. Computational Design

As mentioned above, the geometry used for this analysis is a 5 x 5 fuel bundle design with a concept mixing vane support grid. Fig. 1 shows a schematic of a mixing vane grid. The mixing vanes in the support grid enhance turbulence mixing downstream of the grid spacer, which increases heat transfer in the fuel assembly.

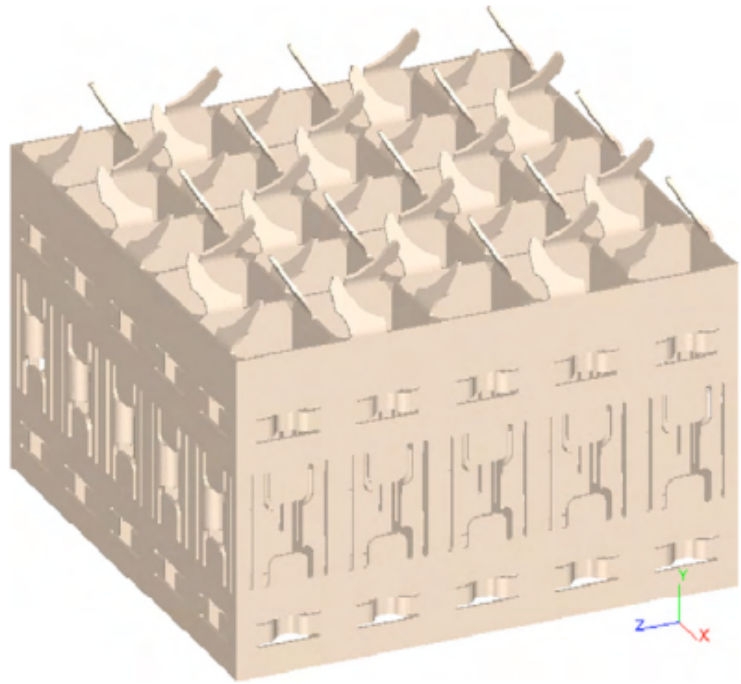

Figure 1: Structural grid with mixing vanes [1]

Fig. 2 shows the lateral cross-sectional view of the flow region around the grid spacer in the test section. The bypass used in this study is of 2.6 mm and all other sides are of 1.0 mm.

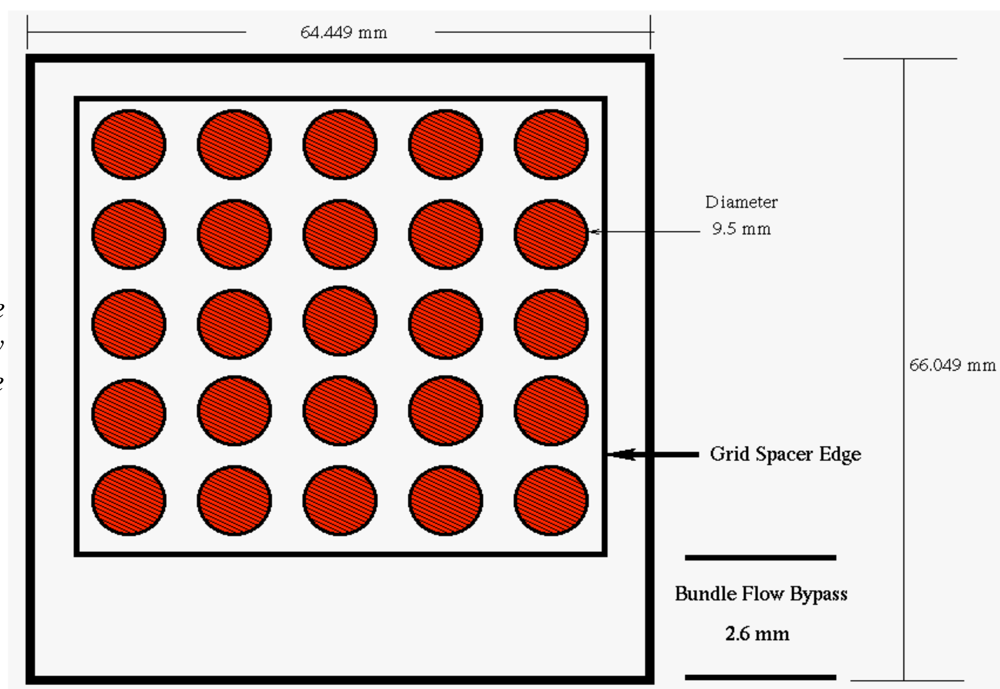

Figure 2: Two dimensional schematic of the bypass and flow area around the grid

The purpose of this particular study is to examine the effect of the grid spacer on the flow structure downstream of the mixing vanes. All the cases considered in the calculations were isothermal using Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) algorithm and realizable k-epsilon model using traditional constants for the closure modelling. Table 1 shows the boundary conditions and other important parameters used for the CFD analysis.

Table 1: Boundary Specifications

Parameter	Value
Hydraulic Diameter	10.5 mm
Inlet Area	2.6E-3 m^2
Reynolds Number	23000
Volumetric Flow Rate	5.8E-3 m^3/s

3. Mesh Sensitivity

For mesh numerical sensitivity, three meshes of different base size were used ranging from 16 million to 120 million. All the meshes were hybrid hex-dominant meshes. Table 2 presents the important parameters of each mesh. Fig. 3 shows cut cross-section of all meshes. It can be seen from the mesh images that no prism layers for wall-adjacent refinement were used for any mesh. However, refinement close to the wall was improved in each mesh refinement process.

Table 2: Mesh Specifications

Mesh Size	Average Base Size
16 Million	0.43 mm
63 Million	0.28 mm
120 Million	0.22 mm

The ratio of mesh refinement is an important parameter in calculating grid independence. The ratio between consecutive meshes is shown in Table 3.

Table 3: Mesh Refinement Ratio

Ratio	Value
r_{21}	1.56
r_{32}	1.24

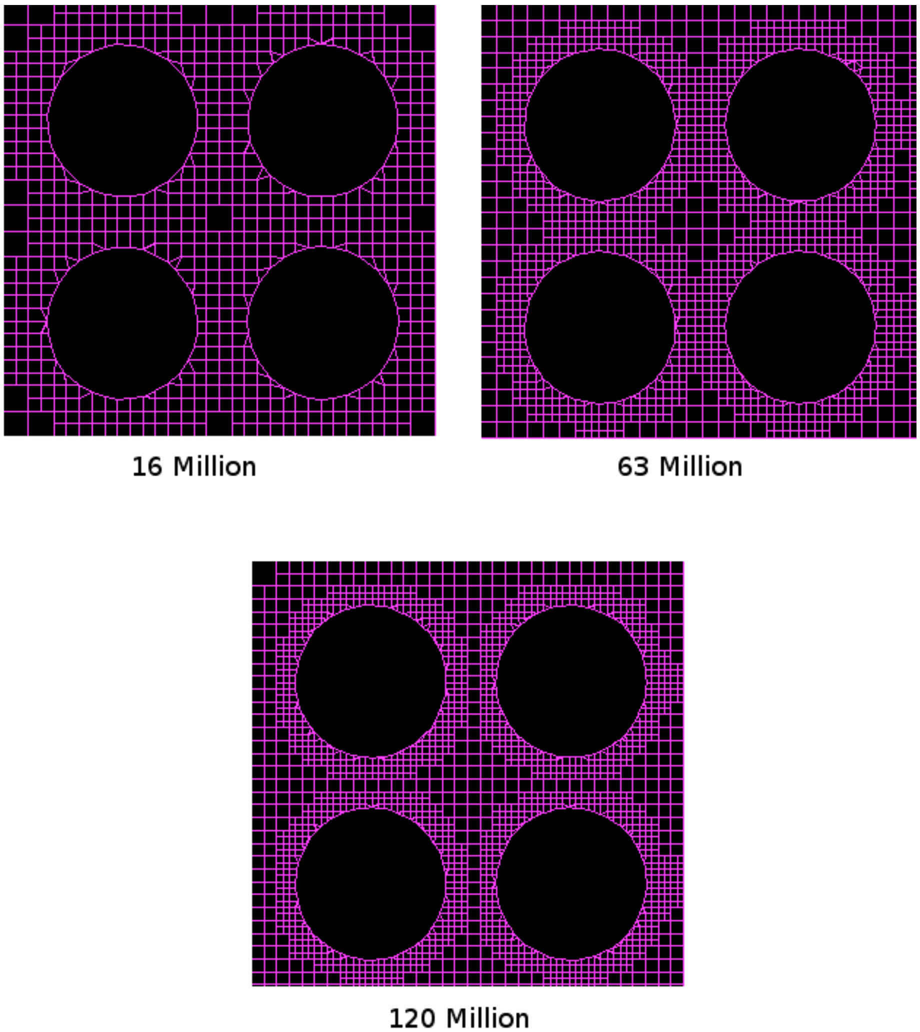

Figure 3: Cross-sectional cut section of the 5 x 5 bundle geometry

Using the data shown below, Richardson extrapolation method was used to quantify numerical uncertainty in the numerical results [2]. Pressure and velocity were taken as the sensitivity parameters. The result of uncertainty analysis is shown in Table 4. The apparent order represents the order of convergence from the lowest to highest mesh size and numerical uncertainty is defined with the Grid Convergence Index (GCI). The parameters defined for sensitivity analysis came from pressure probe data at eight different locations in the mesh. One probe was placed upstream of the grid spacer while the other seven were downstream of the grid spacer.

Table 3: Mesh Sensitivity Reporting Parameters

Mean(Parameter)	Value
Apparent Order, p	6.67
GCI_{21}	0.94%
GCI_{32}	2.66%

4. Data Analysis

Flow field downstream of the grid spacer was analysed at a Reynolds number of 23,000. The CFD simulations used in this analysis were steady state calculations at isothermal conditions. Velocity and pressure at seven different planes were analysed. The location of these planes is presented in Table 4.

Table 4: Plane Location Downstream of Grid Spacer

Plane Number	Location Above the Grid Spacer
1	$0.19*D_H$
2	$0.57*D_H$
3	$5.34*D_H$
4	$9.63*D_H$
5	$13.9*D_H$
6	$18.69*D_H$
7	$23.45*D_H$
8	$28.22*D_H$

Both the turbulence models show different local flow structure in sub-channels. However, in regards to the full 5 x 5 assembly, both the models show similar cross flow structure.

Fig. 4 shows lateral velocity magnitude for both models at plane 1, location at $0.19*D_H$ from the top of the mixing vanes. RKE model yields one main vortex in a single sub-channel, whereas, the SST κ-ω model shows two inner vortices surrounded by sub-channel cross flow circulation. Another important observation from Fig. 4 is the magnitude of the lateral flow using RKE is much lower than one obtained from SST κ-ω.

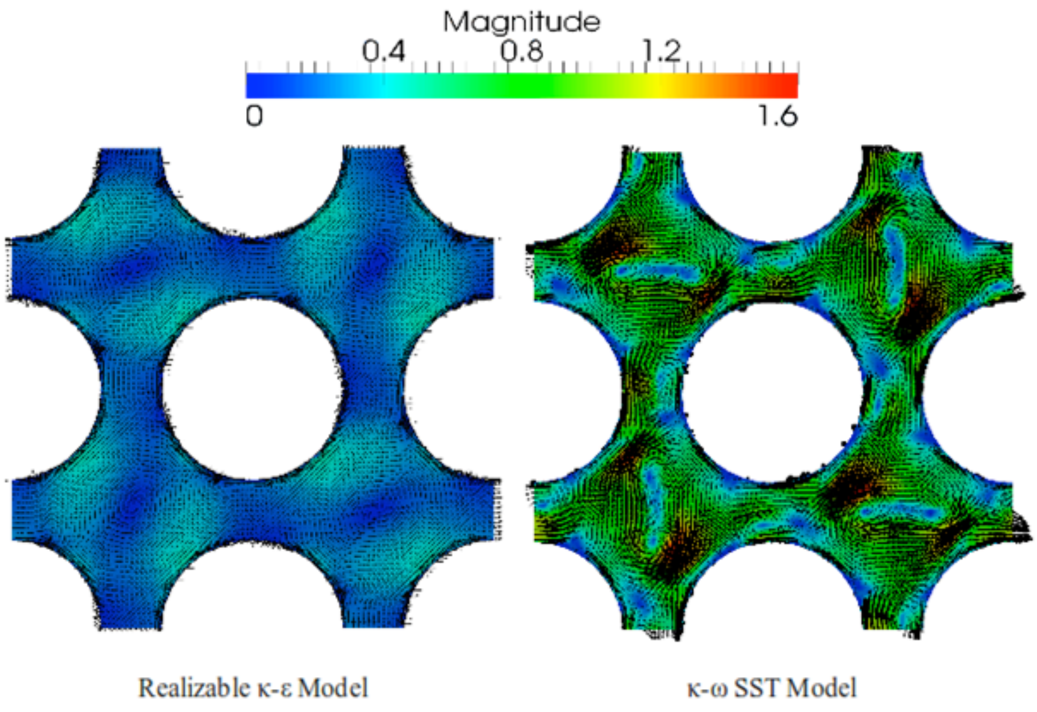

Figure 4: Lateral Velocity with Flow Vectors

Fig. 5 shows the planes downstream of the structural grid spacer located at $0.57D_H$ (left picture) and 5.34 D_H (right picture). The induced cross flow pattern from the mixing vane is clear from both images shown in Fig. 5.

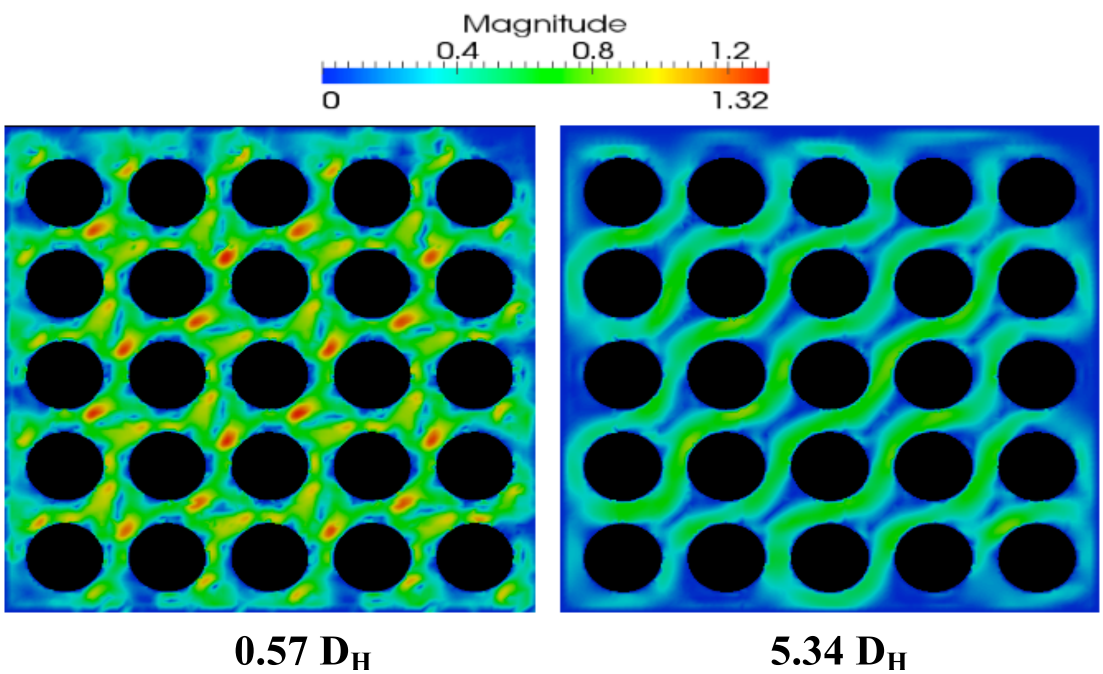

Figure 5: Flow Pattern Downstream of the Grid Spacer from SST κ-ω Model

Fig. 5 represents the cross flow in the fuel bundle that is obtained from SST κ-ω turbulence model. This is crucial because this cross flow induces vibration in the fuel rods, another important phenomenon that needs to be analysed. Another significant observation from Fig. 5 is that the magnitude of the cross flow decreases further away from the structural grid tip, which was expected. The cross flow effect of the mixing vanes essentially diminishes between 15 and 20 D_H for both turbulence models, as shown in Figs. 6 and 7. Fig. 6 shows the flow structure observed from RKE in four planes after and below the mixing vane structural grid. Fig. 7 presents the same result as Fig. 6, but for SST κ-ω turbulence model.

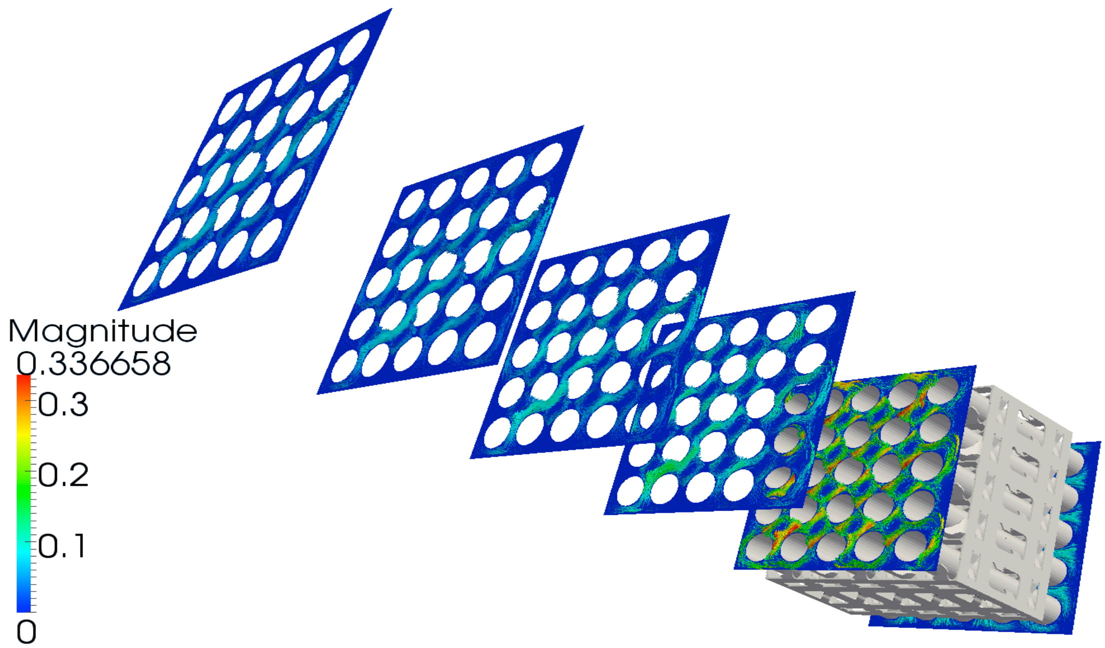

Figure 6: Flow Pattern from RKE Model

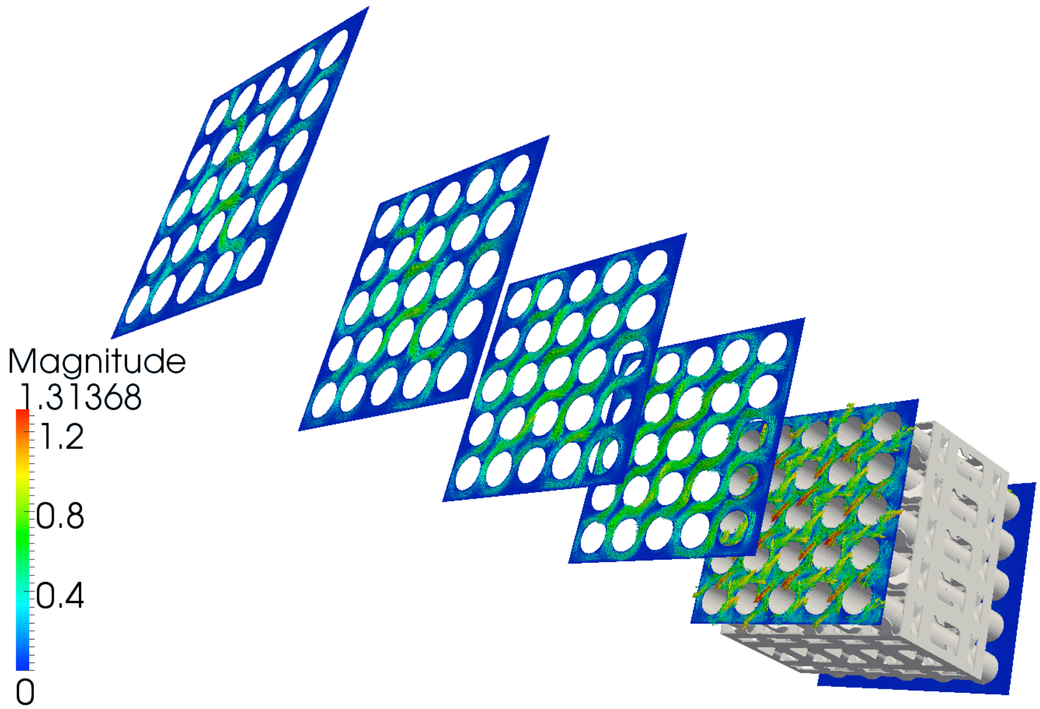

Figure 7: Flow Pattern from SST κ-ω Model

Previous work has shown that SST κ-ω has better results than RKE model for such type of flows. The success of the SST κ-ω turbulence model in capturing the flow behaviour is attributed to a near-wall treatment. This treatment relies on damping of the turbulent viscosity in low Reynolds numbers regions rather than a two-layer near-wall treatment as implemented for the RKE and Reynolds Stress models [3].

Table 5 below compares the measured pressure drop in the experiment with the two turbulence models.

Table 5: Pressure Drop Across the Structural Grid Spacer

Experimental	Realizable k-epsilon	SST κ-ω
4688.21 Pa	9123.15 Pa	3638.18 Pa

Realizable k-epsilon greatly overestimates the pressure drop across the grid. On the other hand, the pressure drop from the SST κ-ω is close to the experimental value. In fact, the SST κ-ω would have been even closer to the experimental value if the dimensions of both the experimental and CFD calculations were identical in all crucial parameters. In this study, excluding the bypass, the difference in the experimental and CFD geometry was the flow area that surrounds the grid spacer, shown in Fig. 2. The size of that small bypass in the experimental set-up at Texas A&M university is 0.5 mm, whereas, for the geometry used in the CFD analysis is 1 mm. This is may be the cause of the 22% discrepancy between the experimental and SST κ-ω result. Due to this large difference in the results, another case with the smaller bypass of 0.5 mm was simulated. The pressure drop obtained from that geometry with SST κ-ω model was 4672.57 Pa, which only yields a difference of about 0.33%. Other CFD simulations are being conducted to analyse the effect of this gap using the exact dimensions as the experimental set-up.

5. Conclusion

When using the RANS approach for estimation of turbulence effects and flow characteristics for nuclear fuel bundle with mixing vane structural grids, the choice of an appropriate turbulence model is extremely crucial. In this analysis, SST κ-ω proved more accurate than RKE in the comparison with the experimental pressure drop results. The flow profiles obtained from the models have a reasonable similarity in the sub channels. Given the agreement of the SST κ-ω model with the experimental results, it is shown that the SST κ-ω model is appropriate choice in the predictions to achieve sub-channel flow patterns seen in previous works [1] and [3].

6. Acknowledgements

We acknowledge the use of EOS computing facility at Texas A&M University for their support and computing hours for this study and the fruitful discussion with Mike Connor.

7. References

1. M. E. CONNOR, E. BAGLIETTO, A. M. ELMAHDI, “CFD methodology and validation for single-phase flow in PWR fuel assemblies,” Nuclear Engineering and Design, pp: 2088-2095, 2010.

2. I. B. CELIK, U. GHIA, P. J. ROACHE, C. J. FREITAS, H. COLEMAN, and P. E. RAAD, “Procedure of estimation and reporting of uncertainty due to discretization in CFD applications,” Journal of Fluids Engineering, vol. 130, 2008.

3. M. V. HOLLOWAY, D. E. BEASLEY, and M. E. CONNOR, “Investigation of swirling flow in rod bundle sub channels using computational fluid dynamics,” Proceedings of ICONE14 International Conference on Nuclear Engineering, July 17-20, Miami, FL, USA, 2006.