INVESTIGATION OF SWIRLING FLOW IN ROD BUNDLE SUBCHANNELS USING COMPUTATIONAL FLUID DYNAMICS

Mary V. Holloway  
Mechanical Engineering Department  
United States Naval Academy  
590 Holloway Road  
Annapolis, MD 21402

Donald E. Beasley ¹  
Department of Mechanical Engineering  
Clemson University  
Clemson, SC 29634

Michael E. Conner  
Westinghouse Nuclear Fuel  
5801 Bluff Road  
Columbia, SC 29250

ABSTRACT

The fluid dynamics for turbulent flow through rod bundles representative of those used in pressurized water reactors is examined using computational fluid dynamics (CFD). The rod bundles of the pressurized water reactor examined in this study consist of a square array of parallel rods that are held on a constant pitch by support grids spaced axially along the rod bundle. Split-vane pair support grids are often used to create swirling flow in the rod bundle in an effort to improve the heat transfer characteristics for the rod bundle during both normal operating conditions and in accident condition scenarios.

Computational fluid dynamics simulations for a two subchannel portion of the rod bundle were used to model the flow downstream of a split-vane pair support grid. A high quality computational mesh was used to investigate the choice of turbulence model appropriate for the complex swirling flow in the rod bundle subchannels. Results document a central swirling flow structure in each of the subchannels downstream of the split-vane pairs. Strong lateral flows along the surface of the rods, as well as impingement regions of lateral flow on the rods are documented. In addition, regions of lateral flow separation and low axial velocity are documented next to the rods. Results of the CFD are compared to experimental particle image velocimetry (PIV) measurements documenting the lateral flow structures downstream of the split-vane pairs. Good agreement is found between the computational simulation and experimental measurements for locations close to the support grid.

INTRODUCTION

The core of a conventional pressurized water reactor (PWR) is constructed from an array of fuel rods. The fuel rods are grouped together into rod bundles. The typical configuration for the rod bundle examined in this study consists of a 17×17 square array of parallel rods. Support grids are spaced axially along the span of the rod bundle to provide structural support. Springs and dimples formed on the support grid straps make contact with the rods and hold the rods on a constant pitch. Split-vane pairs formed on the downstream edge of the support grids have been used to improve the fluid dynamic and heat transfer characteristics of the rod bundle. A photograph of the top portion of a 17×17 array fuel bundle is shown in Fig. 1.

Split-vane pair support grids promote higher mixing rates in the rod bundle and also create large scale lateral flow structures in the rod bundle subchannels. A subchannel is defined as the flow area between four adjacent rods arranged in

¹ Fellow, ASME
a square array. A typical subchannel downstream of a split-vane pair is shown in Fig. 2. In a split-vane pair support grid, the split-vane pairs in neighboring subchannels alternate between a down/up vane pattern (as shown in Fig. 2) and a left/right vane pattern. Each vane makes an angle of approximately 30 degrees with the axial flow direction. Two main categories of split-vane pairs have been identified in the literature: split-vane pairs with weld nugget cutouts and split-vane pairs without weld nugget cutouts [1,2]. Though differences in the flow structures occurring downstream of the two main types of split-vane pairs have been documented [2], both general categories of split-vane pairs can be used to create a lateral swirling flow structure in the rod bundle subchannels. The split-vane pair without weld-nugget cutouts, as depicted in Fig. 2, is the focus of the present investigation. The rods modeled in the present study are the same size as those commonly used in pressurized water reactors, which have an outer diameter, \( D \), of 9.5 mm and are separated on a pitch, \( P \), of 12.6 mm.

The present study investigates computational fluid dynamics (CFD) as a tool to predict the fluid flow structures and heat transfer characteristics downstream of split-vane pairs in a rod bundle. The applicability of readily available “off-the-shelf” turbulence models for predicting the swirling flow structures present downstream of split-vane pair support grids is investigated. A two-subchannel model with periodic boundary conditions is used to capture the up/down and left/right repeating vane pattern found in the split-vane pair support grid designs. Computational fluid dynamics results are evaluated based on lateral flow fields obtained experimentally using particle image velocimetry (PIV). With reliable computational simulations, insight into the single-phase flow characteristics and heat transfer downstream of split-vane pair support grids can be obtained.

**NOMENCLATURE**

- \( A_{sc} \): cross sectional flow area of a typical subchannel
- \( c_p \): constant pressure specific heat
- \( D \): rod diameter
- \( D_e \): hydraulic diameter of subchannel, \( D_e = \frac{4A_{sc}}{P_{sc}} \)
- \( F \): swirl ratio
- \( k \): turbulence kinetic energy
- \( L \): length of integral path
- \( P \): rod pitch
- \( P_{sc} \): wetted perimeter of a typical subchannel
- \( r \): radial coordinate from centroid of vorticity
- \( Re \): Reynolds number, \( Re = \frac{\rho V D}{\mu} \)
- \( Re_{re} \): turbulent Reynolds number based on wall distance
- \( TI \): turbulence intensity
- \( u \): mean velocity from Reynolds decomposition
- \( u' \): fluctuating velocity from Reynolds decomposition
- \( u' * \): friction velocity, \( u' * = \sqrt{\frac{\epsilon}{\rho}} \)
- \( V_{lat} \): lateral velocity
- \( V_z \): axial velocity
- \( V_{z,avg} \): average axial velocity
- \( x \): x-coordinate direction
- \( y \): y-coordinate direction
- \( y_w \): normal distance from rod surface
- \( y* \): dimensionless distance from the rod, \( y* = \frac{yu'}{v} \)
- \( z \): axial (streamwise) coordinate direction
- \( \delta_{ij} \): Kronecker delta
- \( \epsilon \): turbulence dissipation rate
- \( \rho \): density
Conner et al. [4] have modeled a complete grid span of a 5 lateral flow fields. Ikeda and Hoshi [9], Ikeda et al. [5], and boundary conditions eliminate wall confinement effects on the dominant effect on the lateral flow structures. The periodic subchannel models, the vane design and orientation have the periodic boundary conditions in the rod gaps [3-8]. In the two design, many researchers utilize a two subchannel model with pattern in the subchannels of a split-vane pair support grid.

In order to capture the symmetry of the alternating vane pattern in the subchannels of a split-vane pair support grid design, many researchers utilize a two subchannel model with periodic boundary conditions in the rod gaps [3-8]. In the two subchannel models, the vane design and orientation have the dominant effect on the lateral flow structures. The periodic boundary conditions eliminate wall confinement effects on the lateral flow fields. Ikeda and Hoshi [9], Ikeda et al. [5], and Conner et al. [4] have modeled a complete grid span of a 5 x 5 rod bundle. The 5 x 5 models, which represent a small portion of the larger 17 x 17 array bundles, allow for the examination of wall effects on the lateral flow fields in the subchannels. In addition, Ikeda and Hoshi [9] identify asymmetries in the lateral flow fields in subchannels downstream of identical vane orientations that are attributed to differences in the spring and dimple placement in the 5 x 5 support grid designs.

Campbell et al. [8] compares CFD simulations to experimental pressure drop and rod temperature data obtained from a 7 x 7 rod bundle array. Reynolds numbers ranging from 4750 to 26,800 are investigated for flow downstream of a mixing vane grid. The pressure drop predicted by the CFD simulations is within 2% of the pressure drop measured in the experimental facility for the high Reynolds number comparison. The pressure drop predicted for the low Reynolds number case was significantly different than the measured value due to the inapplicability of the turbulence model for the low Reynolds number flow. The lateral flow structures present downstream of the split-vane pairs are not discussed in [8].

The CFD simulations of Smith et al. [3] confirm the presence of swirling flow structures in the subchannel downstream of a split-vane pair with weld-nugget cutouts. The lateral flow structures show good agreement with the PIV measurements reported in McClusky et al. [10] for axial distances between 5 and 20 D∞. At axial locations closer to the support grid, the flow structures are qualitatively captured by the CFD. However, larger velocity magnitudes are predicted close to the grid using CFD versus PIV. Comparisons made by Karoutas et al. [11] and In [1] identify similar trends in both axial and lateral velocities between CFD and experimental laser Doppler velocimetry (LDV) measurements at axial locations near the support grid.

Karoutas et al. [11] and In [1] utilize the swirl ratio,

\[ F = \frac{1}{L} \int \frac{V_{ax}}{V_\tau} dx \]  

(1)
to compare CFD results with those obtained from the LDV reported in Karoutas et al. [11]. Results for the CFD simulation for the split-vane pair support grid design indicate a similar range of swirl ratios [1,11]. However, the computations consistently result in slightly smaller swirl ratio magnitudes. In addition, there is a noticeable difference in the decay rate of the swirl ratio in the range of 1 to 10 D∞ [11].

Split-vane pairs both with and without weld-nugget cutouts are investigated by In [1]. No differences between the flow fields downstream of the split-vane pairs with and without weld-nugget cutouts are identified from the simulations. Karoutas et al. [11] investigated both a split-vane pair and squeezed tube support grid design and documented different flow field characteristics for each support grid design. Kim and Seo [7] investigated the swirling flow and cross flow downstream of the vane design of Karoutas et al. [11] and a split-vane pair design with weld-nugget cutouts. The vane design was optimized by varying the vane angle and the width of the vane base. The CFD technique has proven to be a valuable tool to designers in optimizing the design of support grids. However, further investigation of appropriate turbulence model and meshing requirements needed to achieve grid independent solutions would extend the capabilities of CFD in rod bundle applications.

**COMPUTATIONAL METHODOLOGY**

The fluid dynamics in rod bundle subchannels downstream of split-vane pair support grids was investigated using computational fluid dynamics (CFD). Two split-vane pair support grid designs have been investigated by the authors, a split-vane pair with weld-nugget cutouts and a split-vane pair without weld nugget cutouts [2]. Results from the split-vane pair without weld-nugget cutouts are the focus of the present study. Details of the model, computational domain and mesh, grid refinement, and turbulence modeling are briefly summarized below. Further details of the computational methodology can be found in Holloway [2].

Model and Simulation Details

The computational simulations model the fluid dynamics and heat transfer for the flow of water through the rod bundle for a Reynolds number of approximately 35,000. The Reynolds number was chosen to match the Reynolds number of experimental heat transfer and fluid dynamics measurements obtained by the authors and is approximately an order of magnitude lower than that found in commercial pressurized water reactors during normal operating conditions. The flow in the rod bundle subchannels is three-dimensional and

---

The fluid dynamics in rod bundle subchannels downstream of split-vane pair support grids was investigated using computational fluid dynamics (CFD). Two split-vane pair support grid designs have been investigated by the authors, a split-vane pair with weld-nugget cutouts and a split-vane pair without weld-nugget cutouts [2]. Results from the split-vane pair without weld-nugget cutouts are the focus of the present study. Details of the model, computational domain and mesh, grid refinement, and turbulence modeling are briefly summarized below. Further details of the computational methodology can be found in Holloway [2].

Model and Simulation Details

The computational simulations model the fluid dynamics and heat transfer for the flow of water through the rod bundle for a Reynolds number of approximately 35,000. The Reynolds number was chosen to match the Reynolds number of experimental heat transfer and fluid dynamics measurements obtained by the authors and is approximately an order of magnitude lower than that found in commercial pressurized water reactors during normal operating conditions. The flow in the rod bundle subchannels is three-dimensional and

---

The fluid dynamics in rod bundle subchannels downstream of split-vane pair support grids was investigated using computational fluid dynamics (CFD). Two split-vane pair support grid designs have been investigated by the authors, a split-vane pair with weld-nugget cutouts and a split-vane pair without weld-nugget cutouts [2]. Results from the split-vane pair without weld-nugget cutouts are the focus of the present study. Details of the model, computational domain and mesh, grid refinement, and turbulence modeling are briefly summarized below. Further details of the computational methodology can be found in Holloway [2].

Model and Simulation Details

The computational simulations model the fluid dynamics and heat transfer for the flow of water through the rod bundle for a Reynolds number of approximately 35,000. The Reynolds number was chosen to match the Reynolds number of experimental heat transfer and fluid dynamics measurements obtained by the authors and is approximately an order of magnitude lower than that found in commercial pressurized water reactors during normal operating conditions. The flow in the rod bundle subchannels is three-dimensional and
incompressible. In addition, the flow is turbulent for the Reynolds number investigated in the present study. The properties of the water were modeled using a constant density of 999 m$^3$/kg and a constant specific heat of 4,184 kJ/kgK. In addition, the temperature dependence of the thermal conductivity and dynamic viscosity was modeled using linear and quadratic polynomials, respectively.

The steady-state form of the Reynolds-averaged Navier-Stokes, continuity, energy, and turbulence equations were discretized and solved using Fluent 6 software [12]. Several different turbulence closure models were investigated in the present study and are discussed later. A segregated, implicit solver was utilized to solve the governing equations. Second-order upwind discretization schemes for the convective terms in the momentum, energy, and turbulence parameters were employed. The second-order accurate discretization scheme, in conjunction with a grid independent solution, improves the accuracy of the solution and decreases numerical (artificial) diffusion. A second-order pressure interpolation scheme was used. In addition, the SIMPLE pressure-velocity coupling was implemented.

Fully converged solutions were obtained for each of the simulations investigated in the present study. The convergence of the solution was monitored in several ways. The global mass imbalance,

$$\frac{m_{in} - m_{out}}{m_{in}},$$

was less than 2×10$^{-6}$ (0.0002%) for the converged solution. and the global energy imbalance,

$$\frac{\dot{m}c_p(T_{out} - T_{in}) - \dot{q}A_x}{\dot{q}A_x},$$

was less than 0.0001 (0.01 %) for the converged solutions. In addition, the normalized residuals for all of the solution equations were monitored and decreased by 3 orders of magnitude or more before they leveled off in the converged solution. Typically, a total of 2000 to 3000 iterations were required to reach a converged solution.

A SunBlade 2000 workstation, with dual 900 MHz UltraSparc III processors and 6 GB of RAM was used for the simulations. A typical simulation required approximately 1.5 minutes of computational time per iteration. Converged solutions were obtained in approximately 3 to 4 days.

**Computational Domain and Mesh**

The computational domain chosen to document the fluid dynamics and heat transfer of swirling flow in rod bundle subchannels was carefully considered in order to incorporate as many characteristics of the physical geometry being modeled as realistically as possible. The prototype rod bundle investigated in the present study is constructed from parallel rods held in place by support grids spaced axially along the bundle. The rod diameter, pitch, and hydraulic diameter of the subchannel are 9.5 mm, 12.6 mm, and 11.8 mm respectively. The span between neighboring support grids (~ 40 $D_\infty$) remains constant throughout the bundle. The support grids disrupt the thermal and hydrodynamic boundary layer growth through the rod bundle, and the boundary layers begin to redevelop downstream of the support grids. Due to the repeating support grid assembly, a computational domain that models a full grid span downstream of a support grid was used. A uniform axial velocity of 3.24 m/s and a uniform fluid temperature of 290 K were applied at the inlet of the computational domain. An axial development length of 2.2 $D_\infty$ was located upstream of the support grid. The computational domain extends 37.8 $D_\infty$ downstream of the support grid. Experimental investigations of similar rod bundle geometries [11, 13] indicate that the flow will be fully-developed by approximately 30 to 40 $D_\infty$ downstream of a support grid; therefore, fully-developed or nearly fully-developed flow is expected at the end of the computational domain. A convective outlet with a constant static pressure was applied at the outlet of the computational domain. No-slip boundaries were applied on the surfaces of the rods and the surfaces of the support grid. In addition, a constant heat flux boundary condition of 22,000 W/m$^2$ was applied at the rods surfaces.

The symmetry of the repeating vane pattern present in the split-vane pair support grid designs was used to reduce the computational domain of the rod bundle geometry to two typical subchannels containing split-vane pairs. The split-vane pairs in the two subchannel domain are shown in Fig. 3. The alternating left/right and down/up vane pattern is captured using the two subchannel domain. Periodic boundary conditions were applied at the rod gap boundaries. The matching rod gaps for the periodic boundaries are indicated in Fig. 3 using three different colors (light blue, green, and purple). The two subchannel domain is representative of an infinite array rod bundle; therefore, wall effects present in the 5 × 5 rod bundle that was used for the experimental testing were not captured in the computational model. Springs and dimples were not included in the computational model. Instead, the straps of the support grid were modeled as flat plates. Therefore, the effect of the vane geometry on the development of the flow in the subchannels was isolated from any secondary effects from the...
springs and dimples. Modeling the grid strap using flat plates will have some impact on the predicted flow field. However, research by the authors has shown that the mixing vane is the dominant grid feature affecting the lateral flow field downstream of the grid.

The computational geometry and baseline mesh for the support grid design was created using Gambit and T-Grid software within the Fluent software package. Due to the complex geometry of the rod bundle subchannels and split-vane pair design, an unstructured, multi-block, multi-topology mesh was implemented. A conformal mesh was used for the boundaries between the blocks and for the boundaries between regions of different topology. In addition, the surface meshes for the matching periodic rod gap surfaces were identical. A boundary layer region consisting of hexahedrons was used at the rod surfaces for all of the mesh blocks. The boundary layer regions were meshed to achieve a y* value of approximately 1 in the first cell from the surfaces of the rods to allow for resolution of the viscous sublayer.

A view of the mesh used in a lateral plane of the computational domain is shown in Fig. 4. The lateral plane consists of approximately 4200 cells and is representative of the baseline lateral meshing density in the computational domain. The axial mesh is dense just downstream of the support grid (z=0.25 mm per cell) to an axial distance of 6.5 \( D_{\infty} \) in order to capture the complex flow in this region. After 6.5 \( D_{\infty} \), the axial meshing gradually transitions to a spacing of 1.27 mm which is used for the remainder of the domain. A total of approximately 700 cells are used in the axial mesh across the entire computational domain. The 3D computational domain contains approximately 3.25 million cells. A measure of the quality of the cells in the computational domain is determined by the cell skewness. The cell equiangle skewness is a comparison of the angles in the cell with the angles for an equiangular cell of the same type. A skewness of 0 indicates an equiangular cell, and a skewness of 1 indicates a degenerate cell. A high quality grid will have a low average skewness and a histogram weighted heavily towards 0 skewness. The average skewness for the cells in the computational domain is approximately 0.1 with a maximum skewness of approximately 0.85. Approximately 80% of the cells had a skewness value less than 0.2.

**Grid Refinement**

The meshing density of the baseline mesh was investigated in order to establish the grid independence of the solution. The split-vane pair support grid with weld-nugget cutouts was chosen for the grid refinement study due to the more complex features of the support grid (i.e. the weld-nugget cutouts). The baseline mesh generation technique for both the split-vane pair with weld-nugget cutouts and the split-vane pair without weld-nugget cutouts was the same, and the meshing density requirements for the split-vane pair with weld-nugget cutouts can be extended to the split-vane pair without weld-nugget cutouts. A shortened axial span (15.5 \( D_{\infty} \)) was implemented in order to allow for aggressive mesh refinement over the axial span ranging from 0 to 15.5 \( D_{\infty} \). The mesh for the shortened span is identical to that of the baseline case; however the convective outlet was moved to an axial location of 15.5 \( D_{\infty} \) as opposed to 37.8 \( D_{\infty} \). Most of the initial flow development occurs in a region between 0 to approximately 10 \( D_{\infty} \), and the choice of moving the outlet boundary closer to the inlet had little effect on the heat transfer and flow fields in the region of interest. Therefore, it is considered appropriate to use the shortened span to establish the grid independence of the solution.

The computational mesh was refined based on the gradients of the velocity and temperature fields in the region from 0 to 15.5 \( D_{\infty} \). The baseline mesh of the shortened span has 2.4 million cells in the domain. A total of 3.1 million cells were added during the adaptation, which resulted in a final mesh size of 5.5 million cells. Twenty-five percent of the cells in the region from 0 to 15.5 \( D_{\infty} \) were adapted, which approximately tripled the number of cells in the region of interest.

The resulting velocity and temperature fields obtained from the adapted and baseline cases were compared to establish the effect of the grid refinement. Results of the axial velocity profiles across the central rod gap indicated a maximum difference between the baseline and adapted case of 6%. The circumferentially averaged Nusselt numbers for the adapted case is a maximum of 7% higher than the baseline case, with a typical difference of approximately 4%. Comparison of the lateral velocity fields and axial vorticity fields indicate no significant differences in flow structure between the baseline and adapted case. Based on a comparison of the results obtained for the refined and baseline meshes, the solution variables do not change significantly. As such, it is concluded...
that the baseline mesh density is suitable for capturing the fluid dynamics and temperature distribution in the rod bundle subchannels.

**Turbulence Modeling**

Several turbulence models available in the Fluent 6 software were employed in the present study to model the Reynolds stresses in the Reynolds-averaged Navier-Stokes equations. The realizable $k$-$\varepsilon$ model (RKE), shear stress transport $k$-$\omega$ model (SST-$k\omega$), and Reynolds stress model (RSM) were investigated. The RKE and SST-$k\omega$ turbulence models are both two equation turbulence models based on the Boussinesq hypothesis. The incompressible form of the Boussinesq hypothesis is,

$$\frac{-u_i u_j}{\rho} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} k \delta_{ij}.$$  \hspace{1cm} (4)

The RKE model implemented in Fluent is based on the turbulence model of Shih et al. [14]. For the RKE model, transport equations for the turbulence kinetic energy, $k$, and the turbulence dissipation rate, $\varepsilon$, are solved in addition to the continuity, momentum, and energy equations. The SST-$k\omega$ model of Menter [15] calculates the turbulent viscosity based on solving transport equations for the turbulence kinetic energy and the specific dissipation rate, $\omega$. The RSM model [16] solves a transport equation for each of the Reynolds stresses and a transport equation for the turbulence dissipation rate. The linear pressure-strain model [17] with the included wall reflection term is used. Unlike the RKE and SST-$k\omega$ models, which are isotropic based on the Boussinesq hypothesis, the RSM model incorporates anisotropy of the Reynolds stresses.

The near-wall modeling of the turbulence is important for the flow conditions examined in the present study. For all of the turbulence models investigated, a $y^+$ value of approximately 1 on the rod surfaces was achieved and the governing equations are integrated to the wall. For the RKE and RSM models, a two-layer model was implemented. In the two-layer model, the flow is divided into two regions: the viscosity affected near wall region and the turbulent core. For both the RSM and RKE models, a turbulent Reynolds number of 200 is used to distinguish between flows in the viscosity affected region ($Re_\gamma \leq 200$) and the turbulent core ($Re_\gamma > 200$). The turbulent Reynolds number is defined as,

$$Re_\gamma = \frac{\rho y_+ \sqrt{k}}{\mu}.$$  \hspace{1cm} (5)

The SST-$k\omega$ model was developed to be valid in both the near-wall and fully turbulent regions, and a two-layer near-wall treatment is not used. Instead, the model has coefficients that provide damping to the turbulent viscosity in low Reynolds number regions (near the wall).

The RKE, RSM, and SST-$k\omega$ turbulence models were investigated for the flow downstream of the split-vane pair support grid with weld-nugget cutouts. The turbulence intensity in the subchannel at an axial location of 9 $D_\infty$ is shown in Fig 5. The turbulence intensity (TI) is defined as,
\[ TI = \frac{2k}{V_{z,\text{avg}}} \]  

(6)

As can be seen in Fig. 5, both the RKE and the RSM models lead to high turbulence intensity in the center of the subchannel. In contrast, the SST-\(k\omega\) model predicts the highest turbulence intensity near the walls and in the rod gaps, which is consistent with experimental results (for example, [18]). Examination of the development of the flow structures predicted for the RKE and RSM simulations indicates that excessive turbulence causes unnatural diffusion of the lateral flow structures. This is verified in Holloway [2] by comparison to experimental particle image velocimetry (PIV) results. The success of the SST-\(k\omega\) model for the flow conditions examined in the present study is attributed to a near wall treatment that does not rely on a one equation mixing length determination of the turbulent viscosity and a viscosity affected region determined by the turbulent Reynolds number based on the wall distance. The SST-\(k\omega\) model was chosen as the best turbulence model to use for the present study.

RESULTS

The flow downstream of a split-vane pair without weld-nugget cutouts was examined using computational fluid dynamics (CFD). Flow conditions corresponding to a Reynolds number of 35,000 were examined for a two-subchannel model. As shown in Fig. 3, the alternating left/right and down/up vane pattern used on the support grids is captured using the two-subchannel model with periodic boundary conditions at the rod gaps. Therefore, the two-subchannel model provides a model for an infinite array rod bundle. Based on the computational simulations, a single swirling flow structure is present downstream of each split-vane pair.

The development of the lateral velocity fields is shown in Figures 6 to 8. Figure 6 presents the lateral velocity vectors and axial vorticity downstream of the split-vane pairs for a lateral plane located 1.5 hydraulic diameters downstream of the grid strap (1.5 \(D_c\)). The lateral velocity vectors are normalized based on the average axial velocity through the rod bundle. In addition, contours of the normalized axial vorticity are superimposed on the velocity vectors. The axial vorticity is used to identify the swirling flow structures present in the rod bundle subchannels. The left/right vane pattern creates a vortex with a clockwise rotational sense, as indicated by the negative axial vorticity and blue coloring in Fig. 6. In contrast, the down/up vane pattern creates a vortex with a counterclockwise rotational sense, as indicated by the positive axial vorticity and the red coloring in Fig. 6. At 1.5 \(D_c\), the swirling flow structures present in the center of each subchannel are oval in shape. The alternating split-vane pair pattern also creates strong lateral flow through the rod gaps which promotes fluid exchange and mixing between neighboring subchannels. Strong lateral flows along the rod surface and regions of
impingement of the lateral flow on the rods are present at this axial location.

As the flow progresses downstream of the split-vane pairs, the strength of the swirling flow structure decreases and the vortex becomes more circular in shape. At 5 $D_\infty$, as shown in Fig. 7, the magnitudes of the lateral velocities have decreased relative to their values closer to the support grid. The shape of the swirling flow structure, as visually identified by the axial vorticity contours, is oval. In addition, the magnitudes of the axial vorticity have decreased relative to the values at 1.5 $D_\infty$. By 10 $D_\infty$ downstream of the support grid, as shown in Fig. 8, the lateral velocity magnitudes have decreased further, and the swirling flow structures are circular in shape. In addition, the lateral rod gap velocities have decreased and small recirculation regions are found in some of the rod gaps. Similar changes in the rod gap velocity profiles with increasing axial distance downstream of the support grid (also called velocity inversion) have been documented experimentally by Shen et al. [19].

A comparison of the lateral flow structures obtained from the CFD simulations and particle image velocimetry (PIV) measurements are shown for the split-vane pair support grid design. The PIV measurements were obtained downstream of a 5 × 5 rod bundle array. Since the CFD simulation models an infinite array rod bundle and the PIV measurements were obtained for a finite rod bundle array, the comparison between PIV and CFD is expected to differ somewhat due to the effects of the test section walls on the global lateral flow exchange through the test section. These wall effects increase with increasing distance downstream of the support grid in the 5 × 5 rod bundle. In addition, the CFD simulations do not include the effect of springs and dimples on the resulting flow fields. However, a comparison between PIV and CFD provides insight into the applicability of modeling the complex swirling flow through a finite rod bundle using a reduced rod bundle array.

A comparison of the lateral velocity vectors and axial vorticity contours between the PIV measurements and CFD simulations is shown in Figs. 9 and 10 for axial locations of 1.5 and 9 $D_\infty$ downstream of the split-vane pair support grid design. Results in a subchannel downstream of a left/right vane pair are shown in Fig. 11. The average tangential velocity profiles were determined for a radius of 2.5 mm from the centroid of vorticity for the PIV and CFD results. The tangential velocity, $V_\phi$, at each point within the circular region was calculated, and a fourth order polynomial curve fit was used to represent the average velocity profile. The absolute magnitude of the tangential velocities, which are negative for the vortex in this subchannel, are normalized by the bulk axial velocity and shown in Fig. 11 for axial locations of 1.5 and 9 $D_\infty$. The radial coordinate from the centroid of vorticity, $r$, is normalized by the radius of the circular region used to determine the tangential velocity profiles, $r_\phi = 2.5$ mm. The tangential velocity profiles obtained from the PIV and CFD agree well at an axial location of 1.5 $D_\infty$. The maximum tangential velocity is approximately 46 % of the bulk axial velocity for both the PIV and CFD results. However, the maximum tangential velocity occurs at a location of 0.58 $r_\phi$ for the PIV and a location of 0.68 $r_\phi$ for the CFD. Further downstream at 9 $D_\infty$, the tangential velocity profiles do not agree well between the PIV and CFD results. The tangential velocities obtained from PIV are larger at this location than...
those obtained from CFD. The maximum tangential velocity is located at 0.48 $r_o$ from the PIV measurements and at 0.89 $r_o$ from the CFD simulation. One cause for the differences in the tangential velocity profiles is the location of the vortex in the subchannel for each of the cases. In the PIV experiments, the vortex at this location has migrated toward the south rod gap, as shown in Fig. 10. Since the vortex is located close to the rod surface, it has a smaller confinement region than the vortex obtained from the CFD simulation as evidenced by the relatively small radius corresponding to the maximum tangential velocity.

Comparison of the lateral velocity fields indicates that the computational and experimental results agree well at axial locations close to the support grid strap. However, there are some significant differences in the development of the vortex structures. Vortex migration away from the geometric center of the subchannel was documented experimentally in the 5 x 5 rod bundle. In contrast, the vortex structure obtained in the computational simulations remained centered in the subchannel. This is attributed to wall effects and the effects of spring and dimple patterns that are present in the experiment and not modeled in the computations. Since the CFD simulations do not capture the wall effects that are present in the experimental test conditions, it is expected that the lateral velocities would persist longer than those documented in the 5 x 5 rod bundle. However, the opposite trend is observed and the CFD simulations predict tangential velocities with a lower magnitude 9 $D_\omega$ downstream of the grid than those measured using PIV.

A comparison of the circulation in the subchannel downstream of the left/right split-vane pair is made for the CFD and PIV. The circulation of a vortex is calculated based on the axial vorticity as,

$$\Gamma = \int \int_A \omega z dA.$$  \hspace{1cm} (7)

For the present data, a circular region with a radius of 2.5 mm and centered at the centroid of vorticity was used to calculate the circulation. Figure 12 presents the circulation downstream of the split-vane pair for the PIV and CFD results. The circulation is normalized by the circulation at 1.5 $D_\omega$, which is 183.3 cm$^2$/s for the PIV data and 219.8 cm$^2$/s for the CFD data. The initial circulation obtained from CFD is 20 % larger than that obtained from PIV. As can be seen in Fig. 12, the circulation decreases with increasing axial distance downstream of the support grid. The circulation decays more rapidly for the computational simulation than for the experimental results.

**CONCLUSIONS**

The present study utilized computational fluid dynamics (CFD) to document the fluid dynamics downstream of a split-vane pair support grid in a rod bundle subchannel. One objective of the present study was to implement several readily available turbulence models in order to determine the model best suited for the flow. The near-wall turbulence modeling was identified as a significant consideration when choosing an appropriate turbulence model for the simulations. For the flow conditions in the present study, the SST-kω turbulence model performed better than the RKE and RSM models investigated. The success of the SST-kω turbulence model is attributed to a near wall treatment which 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 RSM models.

Based on the computational results, some general conclusions regarding the development of flow downstream of split-vane pairs without weld-nugget cutouts are made. The typical lateral flow fields downstream of the split-vane pair support grid design include swirling flow structures located in the center of the subchannels. In addition, the split-vane pairs create impingement regions on the rod surfaces and strong lateral flows that follow the curvature of the rod surface.

The flow structures obtained just downstream of the split-vane pair without weld-nugget cutouts using CFD were in good agreement with those obtained from PIV. The absence of wall effects in the computational simulation is expected to cause
higher lateral velocities that persist longer than those documented for a finite rod bundle. However, an opposite trend was identified. This suggests that the turbulence model is overly diffusive. Therefore, a turbulence model that is better suited to swirling flows and curvature effects could further improve the computational predictions and more accurately capture the tangential velocities and decay of lateral flow structures downstream of the support grid. One possible improvement on the turbulence modeling efforts would be to implement the RSM model with an alternative near-wall treatment. Future work of interest includes implementing an unsteady computational solution to the flow downstream of split-vane pair support grids in a rod bundle.

ACKNOWLEDGMENTS

The authors would like to gratefully acknowledge the computational advice provided by Dr. D. Scott Holloway.

REFERENCES


