CFD Modelling of Turbulent Flows through 5×5 Fuel Rod Bundles with Spacer-Grids

Krishna Podila and Yanfei Rao
Canadian Nuclear Laboratories, Chalk River, Canada, ON, K0J 1P0
Krishna.podila@cnl.ca; Yanfei.rao@cnl.ca

Abstract
In this investigation, post benchmark evaluation of the OECD/NEA-KAERI MATiS-H benchmark was performed to predict the turbulence intensities as well as velocity variation downstream of the split-vane spacer grid. A three-dimensional computational fluid dynamics (CFD) tool (STAR-CCM+) was used to model the single phase flow through a cold 5-by-5 rod bundle with a split-vane spacer grid. The effects of the domain length upstream of the split-vane spacer grid and the mesh resolution at the split-vane spacer grid regions were assessed for their impact on the predicted attenuation of turbulence intensity downstream of the spacers. Polyhedrons were used to discretize the computational domain, along with prismatic cells near the walls, with an overall mesh count that spanned from 25M to 64M cell volumes. The Realizable $k$-$\epsilon$, $k$-$\omega$ SST and Reynolds stress models were tested to assess their capability in predicting the turbulence intensities as well as the velocities measured in the experiment. The calculated turbulence intensities were found to be dependent on the choice of the turbulence model used, the mesh density on the spacers as well as on the domain length upstream of the spacers. The CFD simulations underpredicted the measured turbulence intensity at the downstream region of the split-vane spacer grid. The predicted velocities, however, were in good agreement with the measurements downstream of spacers.

Keywords
CFD, Rod bundles, MATiS-H, Turbulence, Spacers

1.0 Introduction
Analyses of flow and heat transfer in rod bundles play a crucial role in the design and safety analysis of nuclear power plants. These analyses have been traditionally performed using system and sub-channel codes that provide quick solutions but strongly rely on the use of empirical correlations. Over the past 20 years, significant improvements have been made in commercial computational fluid dynamics (CFD) codes, in computing power and in parallel-computing. These improvements have facilitated the use of CFD as a common practice in many sectors; its use in the nuclear industry is growing.

Computational Fluid Dynamics has been increasingly used to simulate single phase flows in rod bundles in recent years (Baglietto and Ninokata, 2005; Holloway et al., 2006; Gandhir and Hassan, 2011; Frank et al., 2012; Cinosi et al., 2014). Despite significant progresses made in the field, the use of CFD in predicting single-phase flows in rod bundles still faces some challenges due to difficulties in accurately predicting the turbulent structures such as secondary flows, vortex shedding, and flow pulsations that contribute to inter-channel mixing (Krauss and Meyer, 1996; Baglietto and Ninokata, 2005). In addition, it is well recognised that the results from CFD are user-dependent; i.e., they are highly sensitive to the choice of the mesh cell topologies and solver options used by an analyst. Hence, it was recommended in the best practice guidelines (ASME V&V, 2009) that wherever possible, assessment of CFD modelling results must be made against experimental data.
Canadian Nuclear Laboratories (CNL, formerly Atomic Energy of Canada Limited, or AECL) has initiated a program to develop the CFD capability for rod bundle flows. Under the R&D program CNL participated in an OECD/NEA blind benchmark exercise on 5-by-5 rod bundles, known as MATiS-H. The experimental data for the benchmark exercise was generated by KAERI using non-intrusive measurement techniques on a cold 5-by-5 rod bundle with a split-type spacer. A detailed description of the benchmark test facility as well as discussion on the adopted non-intrusive measurement techniques can be found in Chang et al., 2012. The high-quality “CFD-grade” data were provided to the benchmark participants only after they submitted their CFD results. In the synthesis report of the blind benchmark results (Lee et al., 2014), CNL predictions were deemed to be good for the velocity variation downstream of the split-vane spacer grid. However, the turbulence intensities and the circulation were underpredicted.

An assessment of the CNL results for the benchmark submission using the CFD code ANSYS Fluent v14 was presented in Podila et al. (2014). In the benchmark submission, due to the limitation of the available computational resources at the time, a coarse tetrahedral mesh with a short upstream length was used. Overall, the peaks and valleys of the experimental velocity variations downstream of the split-vane spacer grid were reasonably well predicted, whereas the turbulence intensities were significantly underpredicted. The predicted turbulence intensities immediately downstream of the split-vane spacer grid were much lower than the measurements, which were believed to be due to the use of the coarse tetrahedral mesh on the split-vane spacer grid. The present study is an extension of the simulations performed for the benchmark participation as presented in Podila et al. (2014).

The overall objective of this study was to simulate the MATiS-H 5-by-5 rod bundle with a split-vane spacer grid using the CFD code STAR-CCM+ v9.02.007 with enhanced meshes to improve predictions. The primary goal is to correctly capture the variations of turbulence intensities and velocities downstream of the split-vane spacer-grid. This was accomplished by using increasingly finer meshes in the split-vane spacer grid region with polyhedral cells.

### 2.0 CFD Simulation Setup

#### 2.1 CAD Model

The computational domain for the benchmark problem was developed in STAR-CCM by importing the CAD model of the split-spacer (Figure 1) exclusively provided to the benchmark participants. The split-spacer model was used as provided to participants and no modifications were made to optimize the split-vane spacer grid design. The rod bundles were built to the specifications provided in the benchmark instructions. The dimensions used for developing the computational domain in CAD are listed in Table 1.

![Figure 1 Split-vane spacer grid with buttons (OECD/NEA, 2012)](image-url)
Table 1 Dimensions Used for Developing the Fluid Domain of the 5-by-5 Rod Bundle with Split-Spacer (from OECD/NEA, 2012; Lee et al. 2014)

<table>
<thead>
<tr>
<th>Geometrical Parameter</th>
<th>Dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rod diameter, D</td>
<td>25.4 mm</td>
</tr>
<tr>
<td>Rod pitch, P</td>
<td>33.12 mm</td>
</tr>
<tr>
<td>Hydraulic Diameter, D_h</td>
<td>24.27 mm</td>
</tr>
<tr>
<td>Rod pitch to diameter ratio</td>
<td>1.30</td>
</tr>
<tr>
<td>Square channel length</td>
<td>170 mm × 170 mm</td>
</tr>
</tbody>
</table>

The flow problem comprises the rods and the split-vane spacer grid in a square casing, with the fluid domain being obtained by extracting the solid rods and the split-vane spacer grid assembly from the total domain. The overall fluid domain was comprised of three blocks: the upstream, spacer-grid and downstream regions (Figure 2). The flow straightener and outlet mixing plenum were not considered in the current investigation.

To facilitate the mesh extrusion, additional small sections immediately up- and downstream of the split-vane spacer grid were added to the spacer-grid block. The split-spacers with vanes and buttons result in large unsteady structures, i.e. von Kármán vortex street or vortex shedding. Therefore, the complete 5-by-5 rod assembly was used for the simulation. The simulation domain was designed to ensure fully-developed flow conditions at just upstream of the split-vane spacer grid, which resulted in testing of three geometries with the length of the upstream section set at about 2D_h, 30D_h and 60D_h. Downstream of the spacer, on the other hand, the computational domain was extruded to ~ 25D_h since the assessments in this investigation were limited to the immediate vicinity of the split-vane spacer grid, i.e. 1.0D_h downstream of the spacer.

![Figure 2 “Baseline” CAD Model with Three Regions: The Upstream, Spacer and Downstream Sections (Square Channel Not Shown)](image-url)
2.2 Mesh Generation

Appropriate mesh generation constitutes a key factor affecting the numerical stability and the accuracy of the calculation results. For a complex geometry, the time for mesh generation accounts for a large portion of the simulation time as it is an iterative or trial-and-error process, and the mesh generation has to be repeated several times to achieve a desired resolution. Moreover, generation of unstructured meshes requires larger amount of memory per core especially when the mesh routines are not parallelised. For the rod bundle problem in hand, meshing was computationally challenging as it featured split-vanes and buttons that required small cell sizes to be specified thereby resulting in increased cell counts. A 12-core workstation with a shared memory of 128 GB was used to mesh the overall computational domain serially (on a single core). The mesh generator in STAR-CCM+ is a face-based meshing tool, by which surface mesh is generated first, and then, based on the quality of the surface mesh, the volume mesh for the fluid domain is generated by adjusting the growth rate and the maximum mesh size within the fluid region. Hence, to facilitate high-quality surface mesh, the developed CAD model for the fluid domain was re-tessellated to achieve finer triangulation, and was then used for meshing.

Prism layer, polyhedral and extruder meshing schemes were used to discretize the baseline CAD model presented in Figure 2. The polyhedral cell topology was used to densely mesh the spacer region along with the small up- and downstream sections that served as the extrusion planes. Polyhedral cells typically have twelve-to-fourteen faces, so gradients can be better estimated at the split-vane spacer grid location where the flow is significantly perturbed resulting in generation of turbulence which is further propagated downstream. Also, the polyhedral cell can be stretched more, and because of more neighbouring faces it permits placement of the mesh points in tight spots such as on the vanes and buttons of the split-vane spacer grid.

The extrusion planes at the up- and downstream locations of the spacer-grid block were extruded with a cell aspect ratio of 1.0 to extend the length of the domain axially (along the Z-axis). For the extruded section at the upstream location, flow is relatively uniform compared to that at the spacer region as well as at downstream locations. Therefore there was no need for upstream section to be meshed with fine cells. On the other hand, for the downstream location, a much finer surface size was used especially in the vicinity of the split-vanes to capture the turbulence characteristics immediately downstream of the split-vane spacer grid. The downstream section used blended cells with polyhedral cells close to split-vane spacer grids and a combination of prisms and polyhedrons created during the mesh extrusion further from the split-vane spacer grid. A fully-conformal mesh is developed for the spacers and for the boundaries between regions of different topology (upstream and downstream regions of the solution domain). The sample of the resulting volume mesh generated based on this methodology is presented in Figure 3.

Two to five layers of prism cells within the boundary layer on the walls of the fuel rod surfaces were specified. The orthogonal prismatic cells next to the wall boundaries helped to improve the blending from near wall region to the far field of the split-vane spacer grid. Coarse and fine meshes were used in this study, major notable differences between coarser and finer mesh models were: (1) an independent surface cell size control was used on the split-spacers, (2) the boundary layer thickness was refined from 2.25 mm to 2.0 mm and (3) the surface growth rate was reduced from 1.3 to 1.1. The $y^+$ value on the surface of the vanes of the split spacer was
determined to range from $\approx 0.04$ to $40$. The wide range of $y^+$ was generally unavoidable due to the complicated geometry of the split-vane spacer grid and the rapid flow separation at vanes and buttons. The characteristic mesh sizes and the mesh settings that were varied during the mesh development are summarised in Table 2. The number of meshes in the current investigation spanned from $25M$ to $64M$ control volumes. The meshing time varied from eight to 12 hours depending on the surface cell size and the mesh growth rate used for developing the meshes.

![Flow direction](image)

**Figure 3 Mesh Used for the 5-By-5 Rod Bundle Simulations**

**Table 2 Details on the Mesh Parameters and Characteristics**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Domain Length</th>
<th>Cell Size on Spacer Block</th>
<th>Prism layer Thickness (mm)</th>
<th>Surface Growth Rate</th>
<th>Polyhedral Cells Growth Rate</th>
<th>Total # of Cells on Spacer-Grid (M)</th>
<th>Total # of Cells in Fluid Domain (M)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>2</td>
<td>25</td>
<td>0.175</td>
<td>0.35</td>
<td>2</td>
<td>1</td>
<td>46</td>
</tr>
<tr>
<td>B</td>
<td>30</td>
<td>25</td>
<td>0.35</td>
<td>0.35</td>
<td>2</td>
<td>1</td>
<td>18</td>
</tr>
<tr>
<td>C</td>
<td>60</td>
<td>25</td>
<td>0.35</td>
<td>0.35</td>
<td>2</td>
<td>1</td>
<td>18</td>
</tr>
<tr>
<td>D</td>
<td>30</td>
<td>25</td>
<td>0.504</td>
<td>1</td>
<td>2</td>
<td>1.3</td>
<td>8</td>
</tr>
<tr>
<td>E</td>
<td>30</td>
<td>25</td>
<td>0.175</td>
<td>0.28</td>
<td>2</td>
<td>1.3</td>
<td>39</td>
</tr>
<tr>
<td>F</td>
<td>30</td>
<td>25</td>
<td>0.35</td>
<td>0.35</td>
<td>2.25</td>
<td>1.3</td>
<td>18</td>
</tr>
<tr>
<td>G</td>
<td>30</td>
<td>25</td>
<td>0.35</td>
<td>0.35</td>
<td>2</td>
<td>1.1</td>
<td>18</td>
</tr>
</tbody>
</table>

2.3 Solution Control

The flow patterns and the turbulence field of the fluid flow were obtained by solving the three dimensional Reynolds averaged Navier-Stokes equations for an incompressible flow along with
the two-equation and Reynolds stress turbulence models. The numerical simulations were performed in steady state using segregated solver. No symmetry or periodic boundary conditions were employed to preserve the ability to resolve all possible vortex structures downstream of the split-vane spacer grid. The boundary condition at the entrance and the exit of the computational domain (resulting from extrusion) was set to mass flow inlet and pressure outlet.

The benchmark organizers provided measurements in the bare rod bundle that were meant to serve as inlet conditions. However, the measurements covered only part of the cross-section rather than the entire inlet plane which proved to be of limited use for the flow simulations. Hence, the simulations performed at CNL used uniform inlet boundary condition at the MATiS-H’s entrance region with a mass flow of 24.2 kg/s based on the benchmark specifications (OECD/NEA, 2012), which corresponds to a Reynolds number ($Re_{D_h}$) of about 50,000. A turbulence intensity of 5% was assumed at the inlet.

The no-slip wall boundary condition was imposed at the fuel rod boundaries. The second-order upwind scheme was used for the convection terms, and the second-order central difference scheme for the diffusion terms. For all the calculations, coupling of the velocity and pressure solutions was accomplished using the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) scheme. Line probes were created 1.0 $D_h$ downstream of the split-vane spacer grid at the measurement location to monitor the CFD predictions. The solution was said to be converged once the residuals had dropped by at least three orders of magnitude at completion and constancy of the axial velocity at the monitored sections (1.0 $D_h$) was achieved.

### 2.4 Turbulence Modelling

The turbulence in rod bundles is anisotropic in nature and features secondary flows that contribute to turbulent mixing between subchannels. The primary challenge in using CFD for simulating rod bundles was to select an appropriate turbulence model that can properly account for the inter-subchannel mixing. The large eddy simulation (LES) method has resulted in limited success in predicting rod bundle flows due to the vast CPU resources and excessive computation time that are required to resolve the large scale eddy motion based on which the smaller eddies are modelled. For the LES model to be used for simulating the “baseline” CAD model (Figure 2), the grid requirements for the computational model and the long duration of statistical averaging would be prohibitively expensive for an engineering analysis. Furthermore, based on the submitted results by the participants, it was reported in the synthesis report by Lee et al. (2014) that LES did not yield better results compared to that of the two-equation models.

For the current problem in hand, the mixing of the fluid immediately downstream of the split-vane spacer grid is controlled by the split-vanes and the flow is primarily geometry driven compared to that at far downstream locations. The inter-subchannel mixing is mainly controlled by advection which, the two-equation models based on linear eddy viscosity model should be able to capture. However, for the case of bare bundles which do not have spacers, the two-equation models would fail to predict the secondary flows.

Based on these observations, both the two-equation models and the Reynolds stress model (RSM) were tested in this investigation (see section 4). The decision for using these turbulence models was primarily made for shorter solution turnaround times (compared to a LES simulation) to facilitate parametric studies. Among many available two-equation models, the
Realizable two-layer k-ε and the k-ω SST model were selected for evaluating their ability to predict the turbulence intensities in rod bundles. The two-equation turbulence models use the linear eddy viscosity hypotheses and cannot correctly predict the velocity fluctuations in the three components. Hence, the turbulence intensities were estimated from equation 1 with the turbulence kinetic energy computed from equation 2:

\[ TI = \sqrt{\frac{2}{3} \times k \left(V_{\text{magnitude}}\right)} \]  
\[ k = \frac{1}{2} \left( u' u' + v' v' + w' w' \right) \]  

On the other hand, RSM solves the differential transport equation for each Reynolds stress component, and is capable of resolving anisotropic behaviour. The turbulence intensity for RSM was obtained by taking the square root of the diagonal components (normal stresses) of the Reynolds stress tensor defined as:

\[
\begin{pmatrix}
    u'u' & u'v' & u'w' \\
    v'u' & v'v' & v'w' \\
    w'u' & w'v' & w'w'
\end{pmatrix}
\]  

The flow in the 5-by-5 rod bundles is reported to have coherent fluctuations (Frank et al., 2012; Chang and Tavoluaris, 2014). These fluctuations are associated with vortex-shedding due to buttons and swirling flow due to vanes on the split-vane spacer grid (Figure 1). The measurements from KAERI were performed over a long period of time that included the additional unsteadiness arising due to the vortex shedding. However, since RANS approach was used in this work, the turbulence intensity extracted from the simulations included only the modelled part (non-coherent fluctuations) which was obtained from the turbulence model. The contribution of coherent fluctuations related to flow unsteadiness such as vortex shedding could not be properly resolved with RANS modelling approach used in this investigation.

For all the three turbulence models tested, constants were not modified and the high-\(y^+\) wall function treatment approach was imposed on wall surfaces to resolve the fluid flow in the near wall region. The use of high-\(y^+\) wall treatment was based on its extensive use in the past (Rashkovan et al., 2014; EPRI/NESTOR, 2014) in modeling 5-by-5 rod bundles using two-equation models. Also, for the current rod bundle configuration, it should be emphasized that the flow and turbulence physics are primarily influenced by split-vanes and buttons on the spacers rather than the walls of the fuel rod and shroud. Hence, it is expected that the influence of wall treatment for the current flow problem is relatively low.

### 3.0 Sensitivity Analyses

The parameters affecting the numerical accuracy including the domain length and overall mesh count were studied systematically in this section to develop the correct computational model which will be used to assess the models against the experiments from KAERI. The predicted turbulence intensity was used as a key parameter to study the sensitivity of the parameters on the predictions. All the sensitivity analyses and the assessments in this investigation were limited to a single cross-sectional plane 1.0 \(D_h\) downstream of the split-vane spacer grid, and at a height of \(Y=0.5P, 0.016\) m (Figure 4). The Realizable k-ε turbulence model with the high \(y^+\) wall treatment option was used for performing all the sensitivity analyses unless otherwise stated. For
this investigation, all the post processing and assessment of the CFD data was performed at a height of 0.5P (Figure 4).

![Diagram of subchannels]

1: Side Subchannel 1; 2: Side Subchannel 2; 3: Corner Subchannel 3

**Figure 4 Data Reporting Locations for Velocity and Turbulence Intensity on One of the Marked Lines, Y=0.5P**

### 3.1 Effect of Entrance Length (Upstream of Spacer)

The entrance length plays an important role in the current CFD simulations as it affects the conditions at the inlet of the split-vane spacer grid section. The effect of the upstream domain length, was varying from 2Dₜ to 60Dₜ, Mesh C (see Table 2), is shown in Figure 5. As can be seen, the use of a shorter upstream domain resulted in lower turbulence intensities at side subchannels 1 and 2 (see Figure 4 for location), compared to those for a longer upstream domain. As the domain length was further increased from 30Dₜ to 60Dₜ, only marginal increase in the turbulence intensity was achieved. This observation is in line with the rule of thumb of hydraulic entrance lengths of ≈ 50Dₜ in pipes, but the length of 30Dₜ is shorter than the measured values that were reported for channel flows, which is as high as 65Dₜ (Lien et al, 2004). The difference arises primarily due to the inter subchannel mixing in rod bundles that results in shortened entrance lengths to attain fully developed flow compared to flows in pipes. The present sensitivity result suggests that at least an upstream length of 30Dₜ is required to minimise the influence of entrance lengths on the turbulence intensities downstream of the split-vane spacer grid. The upstream domain length also had a strong effect on the predicted locations of the peaks and valleys.
Figure 5 Effect of Computational Domain Length Upstream of the Spacer Block on the Predicted Turbulence Intensities Using Mesh A, B and C (in Table 2)

3.2 Effect of Mesh Resolution at the Split-Vane Spacer Grid

Considering the scale of the MATiS-H geometry and degree of geometrical complexity, it would be impractical to perform a mesh refinement on the entire computational domain as per the CFD best practise guidelines (ASME V&V, 2009). Therefore, it was decided that mesh refinement would be performed only at the pertinent sections of the computational domain which would have significant impact on the turbulence generation. Hence, for the present case to capture the turbulence generated by split-vanes and buttons, the mesh refinement was limited to the spacer region. The refinement of mesh on the spacers was performed using an upstream domain length of $30D_h$.

By reducing the surface cell size (refer to Table 2 for details) in the spacer-block, three gradually refined meshes were generated with polyhedral cells at the bulk and prismatic near wall elements. This resulted in development of meshes that spanned from 8M-39M cell volumes in the spacer region, corresponding to an overall mesh count of 25M to 58M cell volumes for the entire fluid domain. As seen in Figure 6, upon approximately doubling the mesh cells from 8M to 18M in the spacer region, the predicted turbulence intensities increased by as much as 23% especially at the corner subchannel 3 (Figure 4). Further refinement of the mesh to 39M cells in the split-vane spacer grid region lead to only a slight increase in the turbulence intensity at $x/P=0.4$ and 0.61 compared to the results from the mesh with 18M cells. Since the results of the predicted turbulence intensities did not significantly change upon doubling the mesh count, the mesh for the spacer region was considered to be converged. Therefore, a mesh with 18M control volumes on the split-vane spacer grid corresponding to 35M cells for the complete fluid domain was used for further testing of the effect of BL thickness and the effect of mesh growth factor on the CFD predictions. The overall BL thickness on the wall surfaces was coarsened from 2 mm to 2.25 mm (Mesh F, Table 2). For the current rod bundle geometry, the inclined mixing vanes of the split-vane spacer grid play a dominant role in the generation of turbulence downstream, compared to the wall shear stresses associated with other boundaries (Cinosi et al., 2014). Thus,
it was reasonable to thicken the boundary layer instead of reducing it, which would have otherwise been the case if it were a bare bundle, or some other spacer configuration that does not have inclined vanes, e.g. wire wraps. Based on the results obtained in Figure 7, it can be inferred that coarsening of the boundary layer had a negligible effect on the turbulence intensity predictions.

![Figure 6 Influence of Mesh Refinement on Turbulence Intensities Using Mesh B, D and E](image)

![Figure 7 Effect of Boundary Layer Thickness on the Predicted Turbulence Intensities Using Mesh B and F](image)

The benchmark synthesis proposed new best practice guidelines for rod bundle simulations with specifics for meshing that were found to be of importance to correctly predict the flow and turbulence characteristics. It was recommended that employment of sufficiently dense control volumes in the central region of subchannels was necessary to capture the transition and decay of
vortical structures. Hence, the growth rates of the surface mesh cells were decreased from their default values (Table 2) and the results are presented in Figure 8. It was found that upon decreasing the growth factor of the surface mesh from 1.3 (default value) to 1.1, the results for the turbulence intensities remain unchanged as shown in Figure 8. Hence from this point onwards, all the assessments were made using Mesh B (see Table 2) that comprised of an upstream domain length of 30Dh along with a prism layer thickness of 2 mm and a surface growth rate of 1.3.

![Figure 8 Effect of Surface Mesh Growth Rate on the Predicted Turbulence Intensities Using Mesh B and G](image)

### 4.0 Assessment

The challenge in using CFD for simulating rod bundles with split-vane spacer grids has been to correctly predict the increase in turbulence intensity downstream of the split-vane spacer grid region. Apart from the mesh, the turbulence model selected for the simulation of rod bundles affects the predictive capabilities of CFD. Hence as discussed in section 2.4, the suitability of the selected three turbulence models was assessed for predicting turbulence characteristics; predicted turbulence intensities were compared against the experimental data provided by KAERI to BM participants after submission of their blind predictions. The KAERI data set also included the velocity measurements downstream of the spacers. Although the predicted velocities were in general not as sensitive to the turbulence model as turbulence intensities, they were compared against measurements using different turbulence models. It should be pointed out, however, that the suitability of a turbulence model for simulating rod bundles must be judged primarily on the basis of its ability to predict turbulence intensities.

### 4.1 Turbulence Intensities

Since the underlying formulation varies for different turbulence models, the results were grouped into two-equation models and the Reynolds stress model. To determine the magnitude of the increase predicted by each of the models tested, the turbulence intensity at axial locations immediately up- and downstream of the split-vane spacer grid was evaluated in Figure 9 and Figure 10 respectively. The data was extracted by creating lines at the entrance and the exit of
the split-vane spacer grid. As the flow is predominantly in the axial component, only the w-component of the stress was plotted for evaluating the increase of turbulence intensity between two locations with RSM (Figure 10). As can be seen from the results, all the three turbulence models predicted an increase in the turbulence intensity at the downstream location compared to the upstream location. However, the magnitudes of increase in turbulence intensities differed amongst the three turbulence models selected for testing. The differences in the predicted turbulence intensities are examined and the suitability of each of the turbulence models for predicting the rod bundle flow physics is further discussed.

As seen in Figure 9A at the upstream location, the k-ω SST model predicted slightly higher turbulence intensities than the Realizable k-ε model did. On the other hand, at the downstream location (Figure 9B), the Realizable k-ε model predicted (up to 30%) higher turbulence intensities than the k-ω SST model did. The vane region is associated with flow separation and re-circulation which the k-ω SST model with high y⁺ wall treatment failed to capture thereby leading to lower predicted turbulent intensities. Unlike in the k-ω SST model, the dissipation term in the Realizable k-ε model was derived from an exact equation for the transport of the mean-square vorticity fluctuation (STAR-CCM+ user manual, 2015). This positions it to better capture the flow physics downstream of spacers compared to the k-ω SST model. At the upstream region, the two models had predicted peaks and valleys at the same radial location (x/P). However, at the downstream location, the k-ω SST model had predicted considerably higher number of peaks compared to that of the Realizable k-ε model.

For the case of RSM, at the upstream location, the predicted turbulence intensities were lower compared to those by the two-equation models. At the downstream location, however, the magnitude of the predicted turbulence intensities is comparable to that by the Realizable k-ε model. Based on the above observations, it can be inferred that the Realizable k-ε model and RSM are suitable for modelling this configuration.

![Figure 9 CFD Predictions of the Turbulence Intensities Using Two–Equation Models at Locations: (A): Upstream, (B): Downstream of the Split-Vane Spacer Grid](image-url)
Figure 10 CFD Predictions of the Turbulence Intensities at Up-and-Downstream Locations to the Split-Vane Spacer Grid Using Reynolds Stress Model

The capability of these models was further assessed against measurements provided by the benchmark. It was found that the three tested turbulence models all underpredict the turbulence intensities downstream of the split-vane spacer grid (Figure 11 and Figure 12). Amongst the two equation models, the predicted turbulence intensities are considerably lower using the k-ω SST model than using the Realizable k-ε model, specifically at the corner subchannel 3.

The comparison of the velocities predicted by RSM with the measured data is presented in Figure 12 for the three velocity components. RSM accounts for the turbulence anisotropy and thus was able to predict distinctively the peaks and valleys in turbulence intensities compared to the two-equation models. In spite of using a fine mesh in the spacer region, the turbulence intensities downstream of the split-vane spacer grid were predicted much lower than those reported in the experiments. This underprediction of turbulence intensities suggests that the assumed inlet kinetic energy to the split-vane spacer grid was much lesser than it was in the real test case. This underprediction may be due to the use of a steady solution scheme to resolve the inherently unsteady vortices in rod bundles. Although, the contributions of coherent fluctuations in 5-by-5 rod bundles under the benchmark test conditions could not be quantified, few studies were undertaken in a 37-rod bundle to judge its importance in correctly predicting turbulence parameters. In the case of 37-rod bundle, Chang and Tavoularis (2007) reported coherent fluctuations contributed up to 50% of the total kinetic energy where values as high as 70% were reported by Suh and Lightstone (2004).

Apart from the use of RANS, the density of the mesh used upstream of spacers may have introduced interpolation errors between the dense mesh in the spacer region and the relatively coarse mesh in the upstream section. In the case of two-equation models, the approximation made through equation (1) to extract turbulence intensities may also be a source of error.
Figure 11 Assessment of Two Equation Turbulence Models Against Measured Turbulence Intensities at 1.0Dₜ downstream of the Split-Vane Spacer Grid

Figure 12 Assessment of Reynolds Stress Model Against Measured Turbulence Intensities at 1.0 Dₜ Downstream of the Split-Vane Spacer Grid: (A): u'; (B): v'; (C): w'

4.2 Axial Velocity

As mentioned, the split-vane spacer grid consists of the split-vanes and buttons (Figure 1) that cause the flow to be unsteady. The resolved turbulence upstream of the split-vane spacer grid comprises the interaction of the inlet turbulence and a vortex shedding in the wake of the buttons. As seen in the contour plots (Figure 13), all the three turbulence models predicted wake turbulence separation at the buttons and downstream of spacers. Although it was expected that RSM would predict vortices at the buttons and vanes more prominently than the two-equation turbulence models, the results did not differ much from each other. As discussed earlier, this is likely due to the use of a steady RANS incapable of resolving the flow unsteadiness, thereby resulting in reduced vortex-shedding downstream of the vanes of the split-vane spacer grid.
Figure 13 Contours of the Calculated Axial Velocity Component in a yz-Plane at $x = 0.014$ m; Comparison of Turbulence Model Approaches
As seen in Figure 14, the three turbulence models predicted similar trends and values for the axial velocity distribution at Y= 0.5P. The CFD predictions presented in Figure 14 show a good agreement for the locations of peaks and valleys with the measurements. However, for x/P up to 1.5, the degree of variation (peaks and valleys) was significantly underpredicted, or the drops in velocity at the valleys were overpredicted, suggesting that the use of RANS models here is likely to be insufficient. The result shows that the significant underprediction of the turbulence intensities did not affect the predicted location of velocity peaks and valleys. This is expected as these velocity characteristics are less affected by the vortex shedding.

![Figure 14 Assessment of the Turbulence Models Against Measured Axial Velocity at 1.0Dh Downstream of the Split-Vane Spacer Grid](image)

**5.0 Summary**

Canadian Nuclear Laboratories participated in an OECD/NEA blind CFD benchmark exercise on modelling of a cold 5-by-5 rod bundle with a split-vane spacer-grid. The post-benchmark analysis was carried out in this work in order to better predict the turbulence and fluid characteristics downstream of the split-vane spacer grid.

Two equation and Reynolds stress models were assessed against the measured turbulence intensities downstream of the split-vane spacer grid. The mesh was resolved at the split-vane spacer grid region in an attempt to correctly predict the increase in turbulence intensities downstream of the split-vane spacer grid region. The assessment of the CFD predictions was made with experiments from KAERI that was provided to participants of the blind benchmark exercise. The key findings are summarised below:

- The upstream length and the mesh density in the spacer region have significant effects on the predicted turbulence intensities downstream of the split-vane spacer grid.
- The three turbulence models tested all underpredicted the turbulence intensities downstream of the split-vane spacer grid, with the Realizable k-ε model predicting the highest values closer to the experiments.
Good agreement for the location of peaks and valleys between the predicted and measured axial velocities was achieved; the three turbulence models yielded predictions that were very close to each other.

The future work would include the use of URANS turbulence models to examine the individual contribution of the coherent and non-coherent fluctuations on the predicted turbulence intensities.

6.0 Notations

**Acronyms**

AECL  Atomic Energy of Canada Limited
BPG  Best Practise Guidelines
BM  Benchmark
BL  Boundary Layer
CAD  Computer Aided Drafting
CFD  Computational Fluid Dynamics
CNL  Canadian Nuclear Laboratories
CPU  Central Processing Unit
CRP  Coordinated Research Project
HPC  High Performance Computing
IAEA  International Atomic Energy Authority
KAERI  Korean Atomic Energy Research Institute
LES  Large Eddy Simulations
MATiS-H  Measurement and Analysis of Turbulent Mixing in Sub-channels—Horizontal
M  Millions
NEA  Nuclear Energy Agency
OECD  Organization for Economic Cooperation and Development
RAM  Random Access Memory
RANS  Reynolds Averaged Navier Stokes
RSM  Reynolds Stress Model
TI  Turbulence Intensity in equation 1
URANS  Unsteady Reynolds Averaged Navier Stokes

**Symbols**

D  Rod diameter (mm)
D_h  Hydraulic Diameter (mm)
k  Turbulence kinetic energy (m^2/s^2)
k  Turbulence kinetic energy (m^2 s^-2) in equation 2
P  Rod pitch (mm)
Re_{D_h}  Reynolds number based on hydraulic diameter
u  Velocity along x-axis
v  Velocity along y-axis
V  Velocity magnitude (m/s)
w  Velocity along z-axis
\( y^+ \) non-dimensional distance from wall (-) 

**Greek letters**

\( \varepsilon, \omega \) Specific dissipation rate \( (s^{-1}) \)

**Over-bar**

\( \overline{.} \) Fluctuations

### 7.0 References


