EVALUATION OF A NUMERIC PROCEDURE FOR FLOW SIMULATION OF A 5X5 PWR ROD BUNDLE WITH A MIXING VANE SPACER

Moysés A. Navarro¹ and André A. C. Santos²

¹ Brazilian Nuclear Energy Commission - CNEN
Av. Pres. Antônio Carlos, 6627 - Pampulha
30270-901 Belo Horizonte, Brazil
navarro@cdtn.br

² Federal University of Minas Gerais - UFMG
Mechanical Engineering Department - DEMEC
Av. Pres. Antônio Carlos, 6627 - Pampulha
31270-901 Belo Horizonte, Brazil
acampagnole@yahoo.com.br

ABSTRACT

The fuel assemblies of the Pressurized Water Reactors (PWR) are constituted of rod bundles arranged in a regular square configuration by spacer grids placed along its length. The presence of the spacer grids promote two antagonist effects on the core: a desirable increase of the local heat transfer downstream the grids and an adverse increase of the pressure drop due the constriction on the coolant flow area. Most spacer grids are designed with mixing vanes which cause a cross and swirl flow between and within the subchannels, enhancing even more the heat transfer performance in the grid vicinity. The improvement of the heat transfer increases the departure from the nucleate boiling ratio, allowing higher operating power in the reactor. Due to these important thermal and fluid dynamic features, experimental and theoretical investigations have been carried out in the past years for the development of spacer grid design. More recently, the Computational Fluid Dynamics (CFD) using three dimensional Reynolds Averaged Navier Stokes (RANS) analysis has been used efficiently for this purpose. Many computational works have been performed, but the appropriate numerical procedure for the flow in rod bundle simulations is not yet a consensus. This work presents results of flow simulations performed with the commercial code CFX 11.0 in a PWR 5x5 rod bundle segment with a split vane spacer grid. The geometrical configuration and flow conditions used in the experimental studies performed by Karoutas et al. were assumed in the simulations. To make the simulation possible with a limited computational capacity and acceptable mesh refinement, the computational domain was divided in 7 subdomains. The subdomains were simulated sequentially applying the outlet results of a previous subdomain as inlet condition for the next. In this study the k-ε turbulence model was used. The simulations were also compared with those performed by Karoutas et al. in half a subchannel and In et al. in one subchannel computational domains. Comparison between numerical and experimental results of lateral and axial velocities along of the rod bundle show good agreement for all evaluated heights downstream the spacer grid. The present numerical procedure shows better predictions than Karoutas et al. model especially further from the spacer grid were the peripheral subchannels have more influence in the average flow.

1. INTRODUCTION

The nuclear fuel assemblies of Pressurized Water Reactors (PWR) consist of rod bundles arranged in a square configuration. The constant distance between the rods is maintained by spacer grids placed along the length of the bundle. The coolant flows mainly axially in the subchannels formed between the rods. Most spacer grids are designed with mixing vanes which cause a cross and swirl flow between and within the subchannels, enhancing the local
heat transfer performance in the grid vicinity. The improvement of the heat transfer increases the departure from the nucleate boiling ratio, allowing higher operating power in the reactor. The constriction on the flow area leads, however, to an increase of the pressure drop in the grid region. Due to these important thermo-hydrodynamic features, the spacer grids are often improved aiming to obtain an optimal commitment between pressure drop and enhanced heat transfer. Several experimental and theoretical investigations of the flow characteristics downstream the spacer grids have been conducted in the past years with this objective.

In the experimental field Rehme [1] performed pressure drop measurements that resulted in correlations for several grids without mixing devices. Yao et al. [2] developed heat transfer correlations based on temperature measurements of rod bundles downstream spacer grids with and without mixing vanes. Chun and Oh [3] improved Rehme’s correlation to include spacer grids with mixing devices based on new pressure drop measurements. LDV (Laser Doppler Velocimetry) measurements performed by Shen et al. [4] between rod bundle subchannels downstream different vaned spacer grids showed the large influence of the vane angle on mixing and pressure drop. Heat transfer and pressure measurements of Holloway et al. [5, 6] lead to heat transfer correlations based on the pressure loss for spacer grids with and without vanes. Holloway et al. [7] showed that there is a great variation of heat transfer distribution along a fuel rod due to the spacer grid type. Ikeda and Hoshi [8] developed a measurement equipment to perform LDV measurements around a rod and found that the cross flow velocity is proportional to the velocity of the axial flow hitting the vanes.

Amongst the many studies performed involving CFD simulations of rod bundles with spacer grids some with the most significant contributions are: Karoutas et al. [9] and Imaizumi et al. [10] that demonstrated the usefulness of single subchannel CFD (Computational Fluid Dynamic) methodologies coupled with experimental results from LDV and pressure loss measurements on the development of fuel designs for PWR reactors; In et al. [11] that enhanced the numerical methodology of Karoutas et al. [9]; Ikeda and Hoshi [12] that performed 5x5 partial grid CFD simulations that showed comparable results to experimental pressure loss, cross-flow and DNB (Departure from Nucleate Boiling) measurements on freon flow passing through a grid with mixing vanes; Single subchannel numerical simulations of Cui and Kim [13], Kim and Seo [14, 15] and Haixiang and Ping [16] that optimized mixing vane shape and showed their theoretical effect on the flow structure; Simulations performed by Lee and Choi [17] that examined turbulence characteristics of flow downstream four different spacers and showed that the number of subchannels in a CFD model is important for accurate predictions due to the complex exchange between subchannels; And In et al. [18] that performed a series of four subchannels CFD simulations to analyze the heat transfer enhancement in a fully heated rod bundle with vaned spacers.

There are many issues concerning experimental and CFD modeling of the flow through a PWR rod bundle with spacer grids. As the cost of full scale experiments are too high it is usual that studies be performed on reduced models containing a small amount of subchannels, commonly 5 x 5 arrays, and that the obtained results be extrapolated to the real 17 x 17 rod bundle. Numerically some issues such as mesh refinement, turbulence model, wall treatment and appropriate definition of boundary conditions are fundamental and are defined for each specific physical model being limited by the computational capacity available to the simulation. Due to these computational limitations, a simulation of a complete rod bundle is not always possible and parts of the rod bundle have been used in parametric studies and optimizations. These simplifications not always lead to reliable results [17]. The results of a
simulation performed in a single subchannel can be unreliable for the analysis of the entire rod bundle [16], however, their coherence and qualitative similarity with the integral results demonstrate that these results can be used as a first approach to optimize the parametric studies for grid designs.

This paper presents the results of a CFD evaluation on water flow through a 5 x 5 rod bundle segment with one spacer grid using the commercial code CFX 11.0 [19]. The rod bundle geometry and flow conditions similar to the used in the experimental studies performed by Karoutas et al. [9] were assumed in the simulation. The objective of this study was to evaluate 5 x 5 rod bundle CFX 11.0 simulation in comparison to other CFD procedures.

2. THE NUMERICAL METHODOLOGY

The analysis was performed using the commercial CFD code CFX 11.0 [19] that is based on the finite volume method. The RANS equations for mass, momentum and turbulence model were solved. The numerical procedure used in the simulations is presented as follows. The model and flow conditions were inferred from the experimental studies performed by Karoutas et al. [9]. A previous study to verify the possibility to perform sequential simulations of 7 sub-domains instead a single simulation of the full domain was performed. In this study an optimization of the mesh was also performed.

2.1. The Model

Figure 1 shows the dimensional details of the model and of the split vane spacer grid used in the simulations.

![Figure 1. Computational domain with details of the spacer grid and the vanes (in mm).](image-url)
The flow cross section of the model represents ~1/7 of a real size fuel element, with a 5 x 5 rod bundle, 660 mm long, with a spacer grid, inside a 67.59 mm wide square housing. Each rod is 9.53 mm in diameter with a bundle pitch of 12.7 mm. The rod bundle has a total flow area of 2785.144 mm$^2$ in the bare region with hydraulic diameter ($D_h$) of 10.94 mm. The springs and dimples were disregarded to avoid computational complexity. As shown in Fig. 1 the model includes external straps 0.48 mm thick and 40 mm wide.

To make the simulation possible within our present computational capacity and acceptable mesh refinement, the model was divided in seven parts, as shown in Fig. 1. Each of the parts was simulated separately, using the outlet conditions obtained in the previous simulation as the inlet conditions for the next and so on up to the full length of the computational domain.

To evaluate the effectiveness and possible error caused by this approach the flow through a reduced cross section of the rod bundle with the split vane spacer grid, shown in Fig. 2, was simulated with and without the domain division, with the same mesh parameters described in section 2.2. These simulations were performed using the $k$-$\varepsilon$ turbulence model for reactor typical conditions of water flow at 300°C, 158 bar and average velocity of 4 m/s.

![Figure 2. Simplified computational domain.](image)

Results, exemplified by the pressure profile shown in Fig. 3, indicate that this approach does not cause appreciable differences of the main variables and is capable of representing the flow field behavior with reasonable quality for the purpose of this study.

![Figure 3. Pressure downstream the spacer grid without and with the domain division.](image)
2.2. The Mesh Parameters

Aiming to define proper mesh parameters, a mesh sensitivity study was performed with the same simplified geometry showed in Fig. 2 on the non-divided domain. Three meshes were generated applying different mesh refinements on the spacer grid but maintaining the remaining global mesh spacing the same. The global mesh spacing was defined based on the mesh sensitivity study performed by Tóth and Aszódi [20] on a triangular rod bundle. The evaluated meshes had 6.9, 8 and 9.3 million elements. Results indicate that there is little grid influence for meshes greater than 8 million elements, as demonstrated by the pressure profiles shown in Fig. 4, and therefore the parameters of this mesh were used in the simulations.

![Figure 4. Pressure profiles for different meshes.](image)

To simulate the seven parts of the 5 x 5 bundle segment, two mesh types were generated. Mesh type I, with a length of 60 mm, contained a rod bundle segment and a spacer grid placed 5 mm downstream of the inlet to the sub-domain. The other mesh, type II, with length of 100 mm, contained just a rod bundle segment.

The final meshes were defined with a global edge length of 0.5 mm. Near the walls, 6 layers of inflated (prismatic) elements with an expansion factor of 1.8 and first layer height of 0.0067 mm were used. For mesh type I, containing the spacer grid, a superficial edge length of 0.2 mm with an expansion factor of 1.4 was defined.

In the axial direction, the elements were stretched to reduce mesh size. The number of elements generated for each mesh was about 12,000,000 with 3,500,000 nodes. Figure 5 shows details of both unstructured meshes generated for the simulation.
Figure 5. Mesh cross section in the vane (a), grid (b) and bare bundle (c) regions.

2.3. The Boundary Conditions

The entrance boundary conditions assumed in the simulations correspond to the inlet conditions of the experiments performed by Karoutas et al. [9]. The temperature and the pressure were set to 26.67°C and 4.83 bar, respectively, and a mean axial velocity of 6.79 m/s was defined at the entrance of the bundle. Inlet velocity profile was assumed uniform. At the outlet of each simulated section a relative average pressure of 0 Pa was defined. The used outlet condition does not enforce directional nor gradient restrictions reducing the influence of this boundary on the important flow characteristics at this region, as shown in Fig. 3. The surfaces of the rods, housing and spacer grid were assumed hydraulically smooth.

2.4. The Numerical Simulation

As described previously, the simulation domain was divided in seven parts. The simulation sequence is described as follow. Firstly, a type II mesh was simulated by applying an entrance boundary condition as described in Section 2.3. The outlet results for velocity and turbulence were then used as inlet condition for a type I mesh simulation. Subsequently, the outlet conditions of the second simulation were used as inlet conditions for a type II mesh simulation. Hereafter, the outlet of a previous simulation was used as inlet for the next, all type II meshes, until the total length of the bundle was simulated.

The eddy viscosity model $k$-$\epsilon$ turbulence model [21] that relates the turbulence kinetic energy ($k$) to the eddy dissipation ($\epsilon$) was used in the simulations as it was shown by Ikeda et al. [22]
that this model displays fairly good agreement with measured velocity and high stability of convergence. In CFX 11.0 [18] the k-ε model uses a scalable wall-function approach to limit a lower value for the dimensionless distance from the wall used in the log-law.

The central differencing and the hybrid second order schemes were used, respectively, to discretize the diffusion and advection terms of the equations. A residual RMS target value of \(10^{-4}\) was defined for the simulations. Five parallelized PENTIUM IV HT 3.2 GHz with 3 GB of RAM personal computers were used for all 7 simulations amounting to \(\sim 35\) hours of computing time.

### 3. RESULTS AND ANALYSIS

CFX 11.0 simulation results are compared to LDV measurements performed by Karoutas et al. [9] and to CFD obtained results by Karoutas et al. [9] and In et al. [11]. The numerical simulation by Karoutas et al. [9] was performed with the Flow3D [22] code on half a subchannel using lateral periodic boundary conditions to allow the swirl flow to be obtained. The spacer grid strap widths were neglected by considering the surfaces of the grid thin walls. The simulation performed by In et al. [11] was performed with the CFX 4.2 [23] on a single subchannel also with lateral periodic boundary conditions. The straps were also simulated as thin walls. Both simulations were performed with the k-ε turbulence model and log-law wall functions as also was the CFX 11.0 simulation performed for this work.

Figure 6 compares the lateral \(\left(V_{x}/V_{bulk}\right)\) and axial velocity \(\left(V_{axial}/V_{bulk}\right)\) profiles downstream the spacer grid obtained in the simulations to the experimental and numerical values obtained by Karoutas et al. [9] and to the numerical values obtained by In et al. [11]. The subchannels analyzed are that showed in Fig. 1 (light blue rectangle). Profiles data were extracted at the center line crossing the two subchannels. The axial bulk velocities, \(V_{bulk}\), are averages calculated at each cross section planes downstream the spacer grid.

It is found in Fig. 6 that the qualitative and quantitative behaviors of the lateral velocities obtained in the CFX 11.0 simulation show reasonable agreement with the experimental profiles and previous CFD simulations [9, 11]. The lateral velocity profile obtained for the positions up to 50.8 mm by CFX 11.0 showed a good agreement to the experimental measurements. The fluctuations in the lateral velocities predicted by the simulation performed with CFX 11.0 are in general more intense than the previous CFD simulations [9, 11] at all evaluated heights, but show a good overall agreement to the experimental data.

On the other hand, Fig. 6 also highlights that the axial velocity profiles obtained by all the CFD simulations show sensible differences when compared to the experimental results. In general the simulation with CFX 11.0 shows a reasonable agreement to the measurements at positions above 101.6 mm. The predictions of the previous CFD simulations [9, 11] show an almost flat profile which is not experimentally observed for distances near the grid (up to 25.4 mm). In this region the CFX 11.0 predicts more oscillations of the profiles. For distances further downstream the grid (≥ 190.5 mm) the previously simulated CFD models [9, 11] show a more symmetrical behavior of velocity while experimental and CFX 11.0 simulation results show a more asymmetrical flow. This can be explained by the more pronounced effect of the peripheral subchannels and the walls of the test section, that are not simulated by Karoutas et al. [9] or In et al. [11], on the flow at the central subchannels further downstream the spacer grid.
Figure 6. Lateral and axial velocity profiles downstream the spacer grid.
Figure 7 shows the swirl factor calculated through Equation (1) proposed by Karoutas et al. [9] at different levels downstream the spacer grid. The comparison between the measured and CFX 11.0 predicted swirl factor indicates a good qualitative agreement. The three CFD simulations predict a similar tendency of reduction of the swirl with the distance from spacer. However, near to the spacer grid the value of swirl flow is highly underestimated by the previous CFD simulation [9, 11] while the CFX 11.0 5x5 rod bundle simulation over estimates the values in relation to the experiment.

\[ F = \frac{1}{L} \sum \left| \frac{v_x}{v_{axial}} \right| d_x \]  

(1)

Where \( L \) is the length of the path (=2 x Pitch), and \( d_x \) the distance between the mesh nodes.

![Swirl factor downstream the spacer grid.](image)

Figures 8 and 9 show the secondary flow (\( SF = \sqrt{V_x^2 + V_y^2} \)) at cross section planes 12.7 mm and 463.5 mm downstream the spacer grid for the CFX 11.0 simulation, respectively. Just 12.7 mm away from the spacer grid at the center subchannels almost no difference can be observed between the diagonally positioned subchannels flow patterns. The CFX 11.0 predicts an intense swirl flow in the 16 central subchannels as effect of the grid vanes. Little secondary flow is observed at the near wall subchannels that have no vanes. Far 463.5 mm from the spacer grid, however, the flow patterns predicted are very different with a preferential cross flow behavior at the center four subchannels. At this distance the center four diagonally positioned subchannels show an inverse behavior. These figures highlight that by simulating the complete 5 x 5 rod bundle geometry important aspects that influence flow characteristics can be observed.
Figure 8. Velocity vectors and intensity tangent to a plane placed 12.7 mm downstream the spacer grid.

Figure 9. Velocity vectors and intensity tangent to a plane placed 463.5 mm downstream the spacer grid.

Figure 10 shows the pressure loss along the axial length of the rod bundle obtained by CFX 11.0, Karoutas et al. [9] and In et al. [11] simulations and by the semi-empirical formulation of Chun and Oh [3] that has a 10% uncertainty. It is observed in the figure that the agreement obtained between the formulation and all of the CFD simulations were good for the spacer grid pressure drop. The simulation results of CFX 11.0 present a higher grid
pressure loss while the other CFD simulations predict lower pressure loss in relation to the formulation. The tendencies of the friction pressure drop in the bare region predicted by the CFD simulations are slightly different compared to the formulation, with Karoutas et al. [9] showing a more pronounced drop and CFX 11.0 a less pronounced. On the whole, all models behave well when compared to the semi-empirical formulation.

![Figure 10. Pressure loss along the rod bundle.](image)

### 4. CONCLUSIONS

A CFD methodology using a commercial code (CFX 11.0 [19]) to simulate a 5 x 5 rod bundle segment with a vaned spacer grid was analyzed. The rod bundle geometry and flow conditions similar to the used in the experimental studies performed by Karoutas et al. [9] were assumed in the simulation. The results were compared with the experimental investigation performed by Karoutas et al. [9] in which a LDV system was used to measure the axial and lateral velocities and to CFD simulations performed by Karoutas et al. [9] and In et al. [11] on half and one subchannel computational domain. To make the simulation possible with a small computational capacity and acceptable mesh refinement, the computational model was divided in seven parts. Each of the parts was simulated separately, using the outlet conditions obtained in the previous simulation as the inlet conditions for the next part and so on up to the full length of the computational domain.

The results shown a reasonable agreement between the profiles of the lateral velocities obtained in the CFX 11.0 numerical simulation and the experiments. Compared to the previous CFD simulations, the results near the spacer grid obtained by CFX 11.0 show better agreement to the measurements. On the other hand, the axial profiles shown sensible
differences related to the experimental results, and, for distances away from the grids, were observed appreciable differences between the profiles predicted by CFX 11.0 and the previous CFD simulations. In these cases the CFX 11.0 methodology shows a better qualitative agreement with the experiment.

The comparison of the measured and predicted swirl factor also indicates a good qualitative agreement. The three turbulence models predict a similar tendency of reduction of the swirl with the distance from spacer grid.

Numerical pressure loss predictions were compared to a semi-empirical formulation and showed good agreement. The tendencies of the friction pressure drop in the bare region predicted by the CFD simulations are slightly different compared to the formulation, with Karoutas et al. [9] showing a more pronounced drop and CFX 11.0 a less pronounced.

The obtained results showed that the numerical procedure using CFX 11.0 [19] of dividing the numerical domain in several parts, each one with a better quality mesh, can be considered a feasible approach for rod bundle simulation. Results also showed that by simulating the complete 5 x 5 rod bundle geometry important aspects that influence flow characteristics can be observed.

ACKNOWLEDGMENTS

Work supported by the Minas Gerais State FAPEMIG (Fundação de Amparo a Pesquisa do Estado de Minas Gerais)

REFERENCES