TURBULENCE MODEL EVALUATION IN 5X5 PWR ROD BUNDLE NUMERICAL SIMULATIONS WITH A SPLIT VANE SPACER GRID

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

André Augusto Campagnole dos Santos, acampagnole@yahoo.com.br
Federal University of Minas Gerais – UFMG, Av. Pres. Antônio Carlos, 6627 – Pampulha, Belo Horizonte, 31270-901, Brazil

Abstract. *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 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 3 turbulence models were evaluated: k-ε, BSLRS (Baseline Reynolds Stress) and SST (Shear Stress Transport k-ω). The results show that all models perform similarly up to ~100 mm downstream the spacer grid showing an reasenably accurate prediction when compared to the experimental axial and lateral velocity profiles. Away from the grid appreciable differences between the axial velocity profiles predicted by the turbulence models were observed. In these cases the SST model showed a slightly better qualitative agreement with the experiment. Measured and predicted swirl factor indicates also a good qualitative agreement. The three turbulence models predict a similar tendency of reduction of the swirl with the distance from the spacer grid. Numerical pressure loss predictions were compared to a semi-empirical formulation and showed good agreement for the total and spacer grid losses along the rod bundle for all evaluated models except the k-ε model that for the grid pressure loss predicted a value 10.9% higher than the formulation, indicating a poorer predictability of pressure behavior of this model for the spacer grid simulation. These results confirmed that the assumed numerical procedure of dividing in several parts the numerical domain with a better quality mesh is a feasible approach for rod bundle simulation. In general the results obtained with the evaluated turbulence models show little differences that indicate that SST and BSLRS models are more suited for spacer grid simulation.*

Keywords*: Computational Fluid Dynamic, nuclear fuel assembly, spacer grid, PWR type rector, turbulence model*

1. INTRODUCTION

The core of a Pressurized Water Reactor (PWR) is constituted of nuclear fuel bundles with parallel rods 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. Due to this significant fluid dynamic influence on the nuclear fuel assembly performance, 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. Experiments of Rehme (1973) resulted in pressure drop correlations for several grids without mixing devices. Yao et al. (1982) developed heat transfer correlations for spacer grids with or without mixing vanes. Chun and Oh (1998) improved Rehme's correlation to include spacer grids with mixing devices. Shen et al. (1991) performed LDV (Laser Doppler Velocimetry) measurements between subchannels of the rod bundle downstream several vaned grids and observed large influence of the vane angle on the flow characteristics. Karoutas et al. (1995) and Imaizumi et al. (1995) demonstrated that single subchannel CFD (Computational Fluid Dynamic) methodologies coupled with experimental results from LDV and pressure loss measurements are useful tools to develop fuel designs for PWR reactors. Ikeda and Hoshi (2002) 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. Heat transfer and pressure drop measurements of Holloway et al. (2004 and 2008) lead to heat transfer correlations based on the pressure loss for spacer grids with and without vanes. Holloway et al. (2005) showed that there is great variation of heat transfer distribution along a fuel rod due to the spacer grid type. Ikeda and Hoshi (2007) developed measurement equipment to perform LDV measurements around a rod and found that the crossflow velocity is proportional to the velocity of the axial flow hitting the vanes. Single subchannel numerical simulations of Cui and Kim (2003), Kim and Seo (2004 and 2005) and Haixiang and Ping (2005) research the optimized mixing vane shape and show there theoretical effect on flow structure. Simulations performed by Lee and Choi (2007) examine turbulence characteristics of flow downstream four different spacers and shows that the number of subchannels in a CFD model is important for accurate predictions due to the complex exchange between subchannels. In et al. (2008) performed a series of four subchannel CFD simulations to analyze the heat transfer enhancement in a fully heated rod bundle with mixing vane spacers and highlighted the need for further development of the numerical model towards higher order turbulence models.

Some issues such as mesh refinement, turbulence model, wall treatment and appropriate definition of boundary conditions are fundamental in a CFD simulation. These issues are defined for each specific physical model and are 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 (Lee and Choi, 2007). The results of a simulation performed in a single subchannel can be unreliable for the analysis of the entire rod bundle (Haixiang and Ping, 2005), 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 (2007). The rod bundle geometry and flow conditions similar to the used in the experimental studies performed by Karoutas et al. (1995) were assumed in the simulations. The objective of this study was to evaluate the performance of the turbulence models: *k-ε*, BSLRS (Baseline Reynolds Stress) and SST (Shear Stress Transport *k-ω*) on the flow downstream the spacer grid.

2. THE NUMERICAL METHODOLOGY

The analysis was performed using the commercial CFD code CFX 11.0 (2007) 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. (1995). 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. 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.

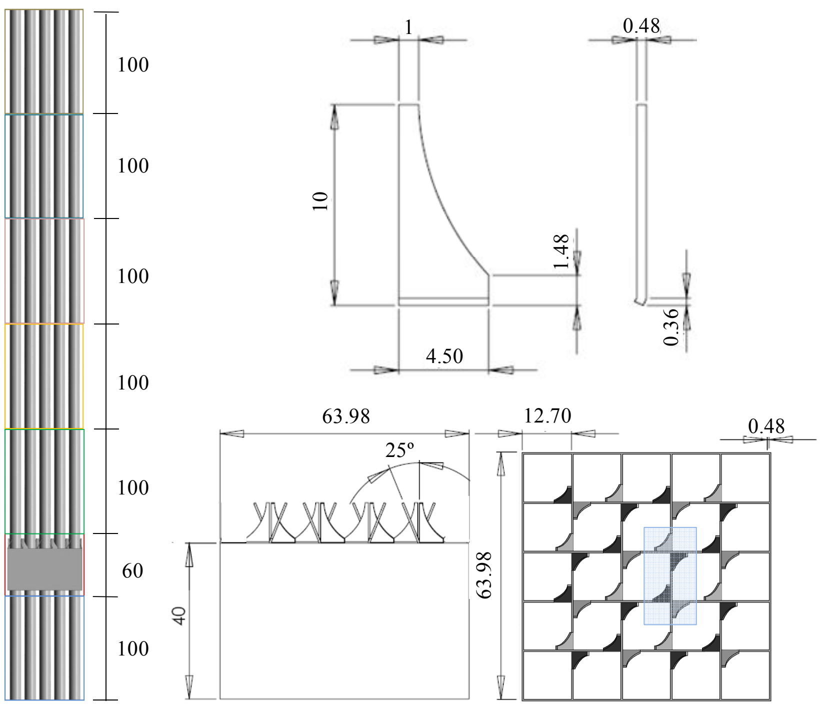

Figure 1. Computational domain of the simulations with details of the spacer grid and the vanes (units in mm)

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 part 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-ε turbulence model in the following conditions: water temperature = 300°C; pressure = 158 bar; average inlet velocity = 4 m/s.

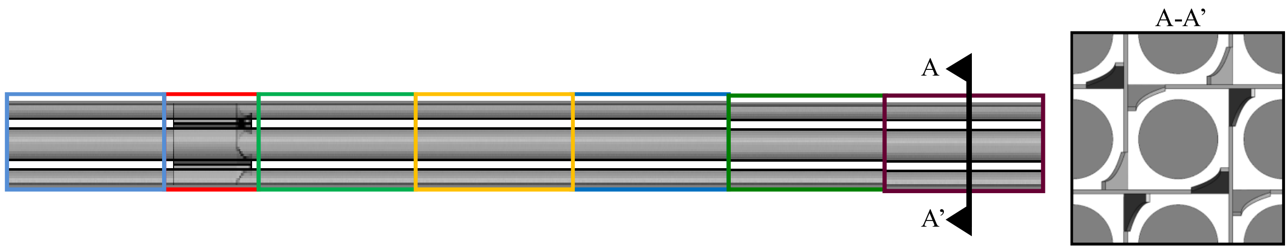

Figure 2. Simplified computational domain

Results, exemplified by the pressure profile shown in Fig. 3, indicate that this approach causes small differences of the main variables and is capable of representing the flow field behavior with reasonable quality for the purpose of this study.

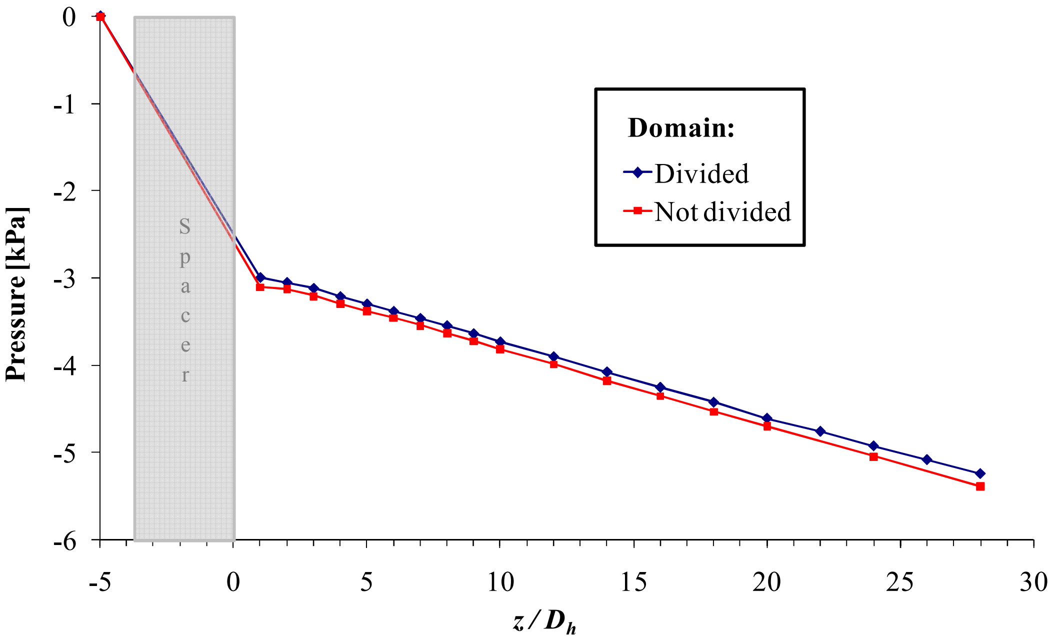

Figure 3. Pressure profiles downstream the spacer grid without and with the domain division

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 (2008) 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.

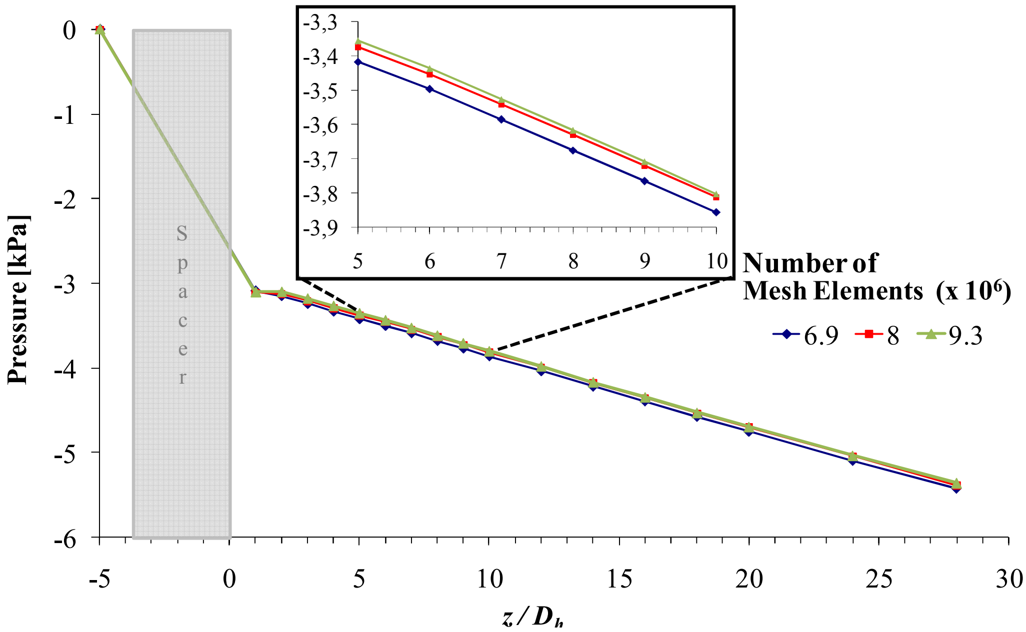

Figure 4. Pressure profiles for different meshes

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.

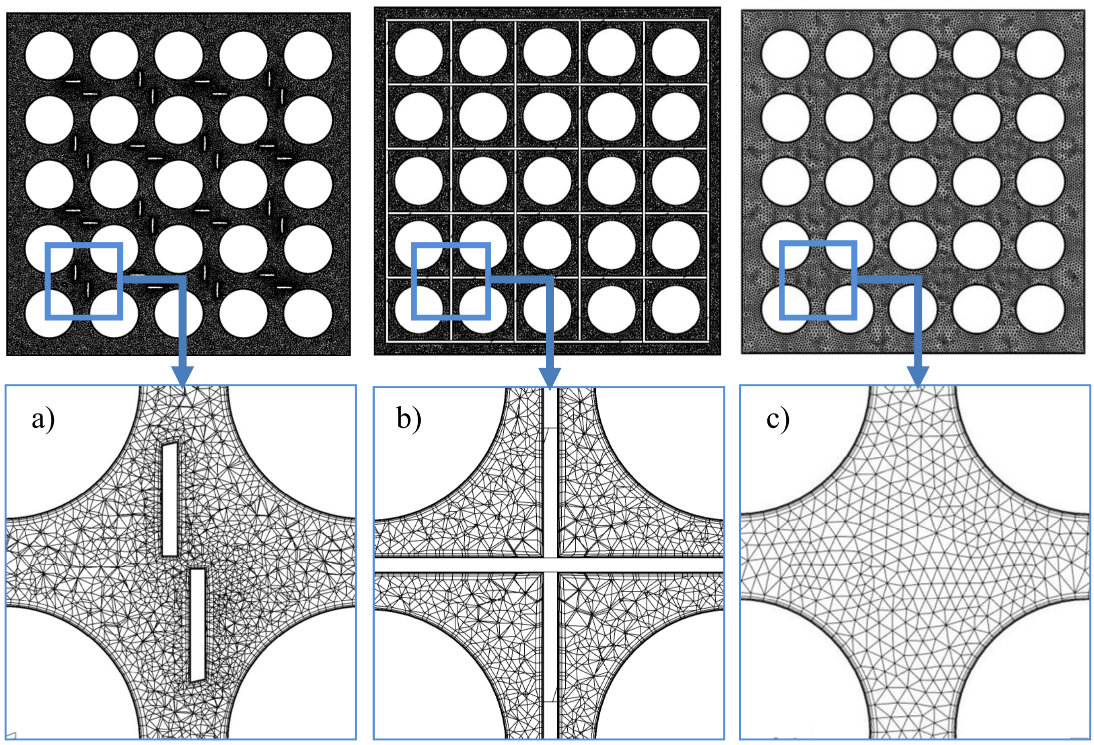

Figure 5. Mesh cross section in the vane (a), grid (b) and bare bundle (c) regions

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. (1995). 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 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 temperature, 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 described sequence was repeated for each evaluated turbulence model. The two equation k-ε and SST turbulence models and the seven equation BSLRS turbulence model were evaluated in the simulations.

The k-ε turbulence model (Launder and Spalding, 1974) assumes that the turbulence viscosity is related to the turbulence kinetic energy (k) and dissipation (ε). In CFX 11.0 (2007) 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 Shear Stress Transport-ω (SST) model (Menter, 1994) is a blend between near surface k-ω model, which relates eddy viscosity to the turbulent specific dissipation rate (ω), and core flow k-ε model. The model includes a function to modify the turbulent eddy viscosity to account for the transport of the turbulent shear stress. In CFX 11.0 (2007) the SST model uses the automatic wall treatment which switches from log-law to viscous sub–layer wall function as the mesh is refined.

The Baseline Reynolds Stress (BSLRS) model solves six transport equations, one for each of the turbulent stresses, also called Reynolds stresses, and one additional equation that is a blend between ω and ε transport equations similar to the SST model (Menter, 1994). The BSLRS model inherently accounts for the stresses anisotropies theoretically making this more suited for complex flows involving strong secondary movements such as the flow downstream a spacer grid. This model uses the same near wall treatment as the SST model in CFX 11.0 (2007).

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 21 simulations amounting to ~64 hours of computing time.

3. RESULTS AND ANALYSIS

Figure 6 compares the lateral (V_x/V_{bulk}) and axial velocity (V_{axial}/V_{bulk}) profiles downstream the spacer grid obtained in the simulations to the experimental values obtained by Karoutas et al. (1995). 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 simulations show reasonable agreement with the experimental profiles. Although the BSLRS model could be considered the most adequate for the simulation of the rod bundle with spacer grids (In and Chun, 2005) the results show little differences between the predictions of the three evaluated models. The lateral velocity profile obtained by the k-ε model, which could be considered the simplest two equation model, showed, specially for the positions further downstream the grid (≥ 190.5 mm), very similar results to the experimental data for a much smaller computational effort than the BSLRS model or even the SST model, that requires more interactions to converge. This behavior has also been observed by other authors (Cui and Kim, 2003, and Ikeda et al., 2006).

On the other hand, Fig. 6 also highlights that the axial velocity profiles show sensible differences when compared to the experimental results. For distances further downstream the grid (≥ 190.5 mm), it can be observed appreciable differences between the profiles obtained with the evaluated turbulence models. At these distances the SST model shows a better qualitative agreement with the experiment. In general the qualitative behaviors of all models for the axial profiles were acceptable compared to experimental data.

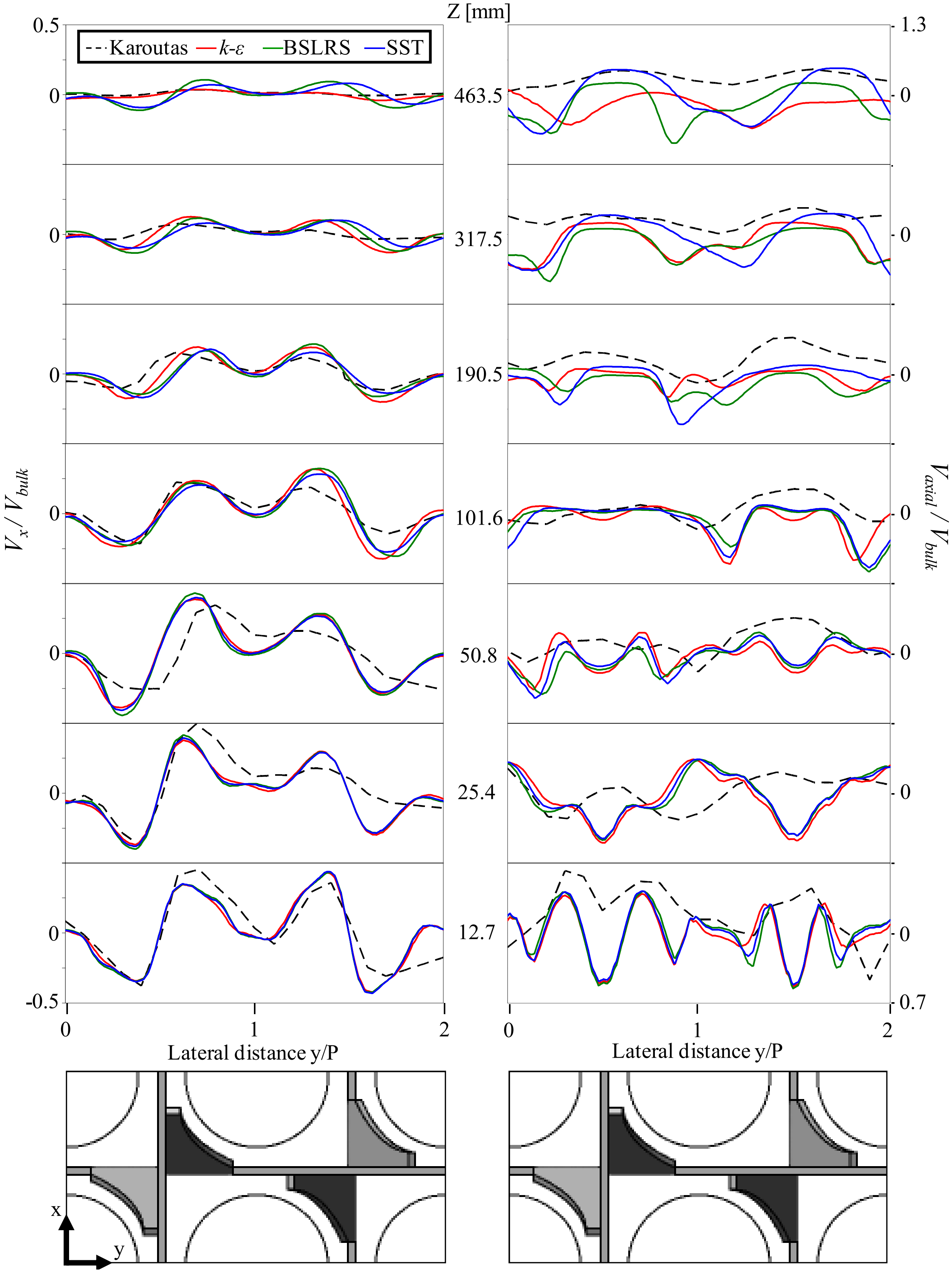

Figure 6. Lateral and axial velocity profiles downstream the spacer grid

Figure 7 shows the swirl factor calculated through the equation (1) proposed by Karoutas et al. (1995) at different levels downstream the spacer grid. The comparison of the measured and predicted swirl factor indicate a good qualitative agreement. The three turbulence models predict a similar tendency of reduction of the swirl with the distance from spacer grid until the second to last level where the *k-ε* turbulence model differs from the other models and the experimental behavior. In Fig. 7 the *k-ε* turbulence model predicts an approximately constant dissipation rate of the swirl factor after 190.5 mm of the spacer grid. This greater reduction rate does not agree well to the experimental behavior that shows a clear decrease in the dissipation rate of the swirl factor as the flow distances from the grid.

$$F = \frac{1}{L}\sum \frac{|V_x|}{V_{axial}} d_x \tag{1}$$

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

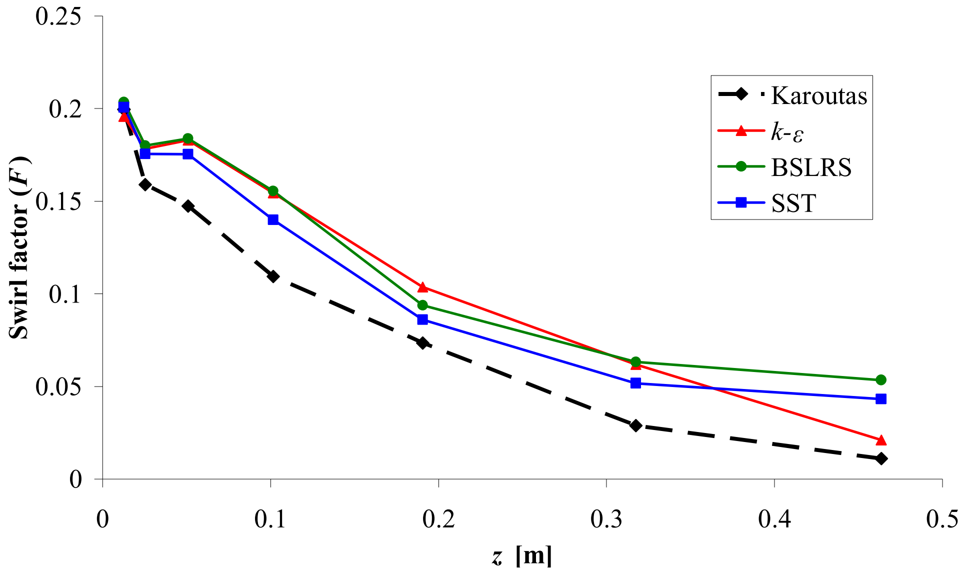

Figure 7. Swirl factor downstream the spacer grid

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, respectively. Just 12.7 mm away from the spacer grid almost no difference can be observed between the patterns obtained with the three turbulence models. Far 463.5 mm from the spacer grid, however, the patterns predicted by the turbulence models are very different.

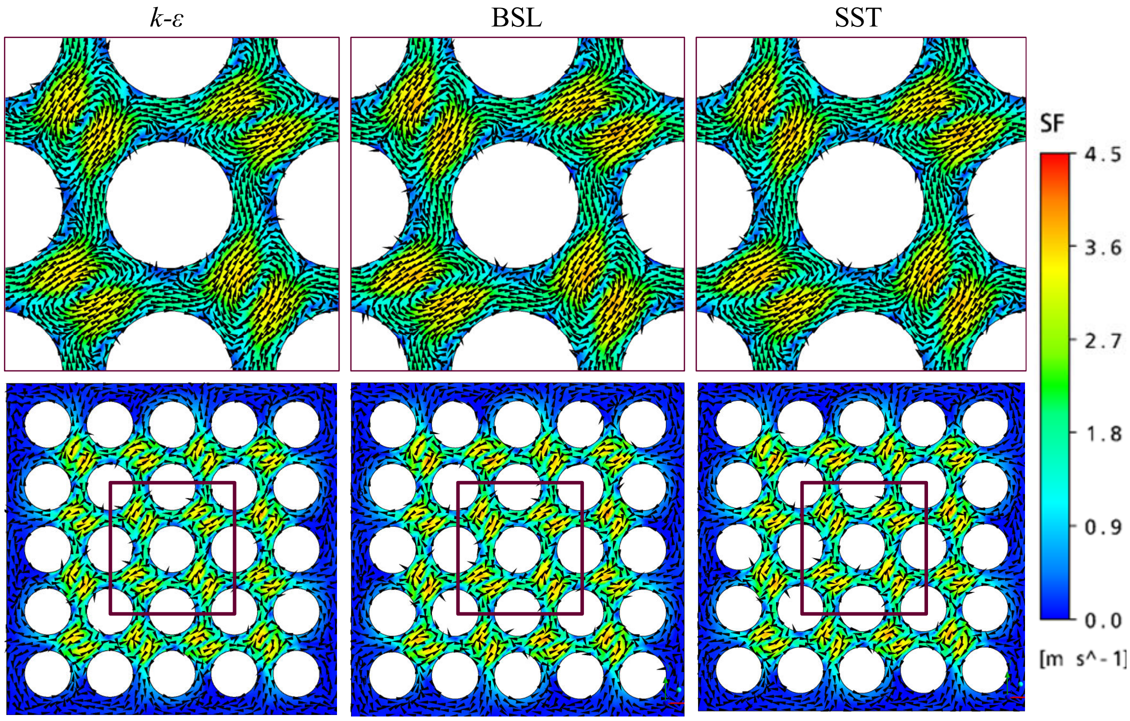

Figure 8. Velocity vectors tangent to a plane placed 12.7 mm downstream the spacer grid

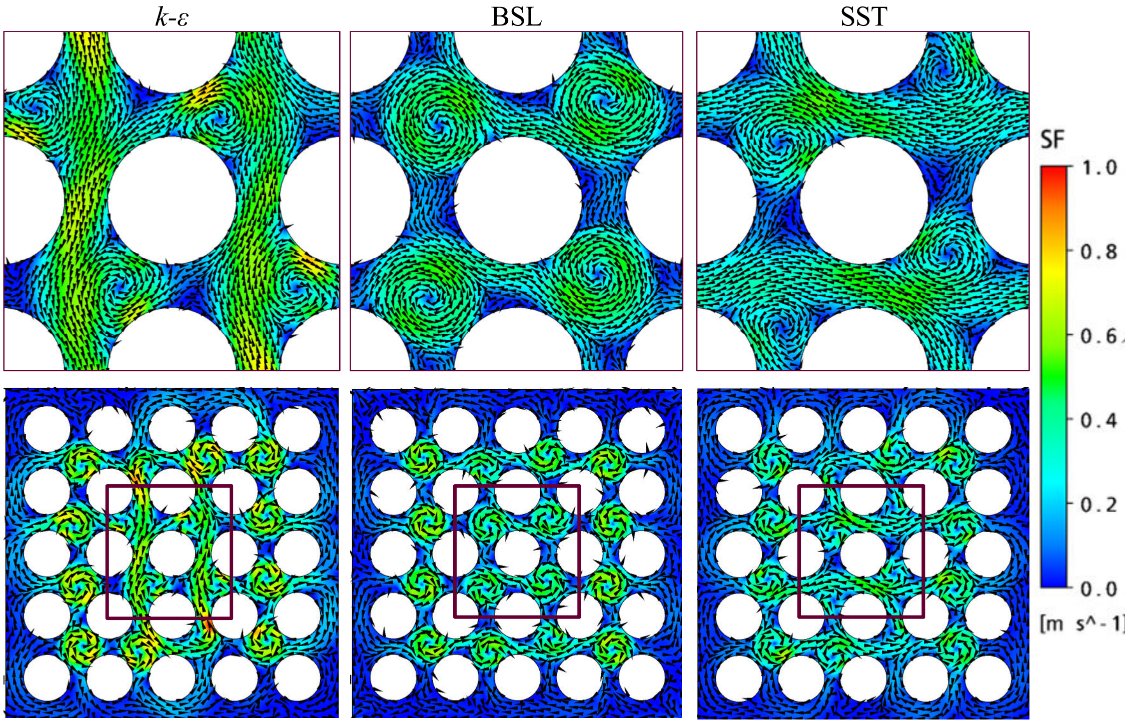

Figure 9. Velocity vectors and intensity tangent to a plane placed 463.5 mm downstream the spacer grid

Although near the grid (Fig. 8) the lateral velocity vectors are very similar, away from the spacer (Fig.9) the k-ε model predicts a preferential cross flow between the subchannels while the BSLRS model predicts a swirl flow in the subchannels and the SST model predicts a combination of the swirl and cross flow patterns.

It is visible in Fig.9 that the secondary flow (SF) was more intense for the simulation with the k-ε turbulence model than for the others. However, the swirl flow shown in Fig. 6 could lead to an opposite conclusion. This is due to the manner in which the swirl flow was calculated by Karoutas et al. (1995) (Eq. 1) taking in account only velocity values normal to one center line crossing the subchannels.

Figure 10 shows the pressure loss along the axial length of the rod bundle obtained in the simulations using the three turbulence models and by semi-empirical formulation of Chun and Oh (1998) that has a 10% uncertainty.

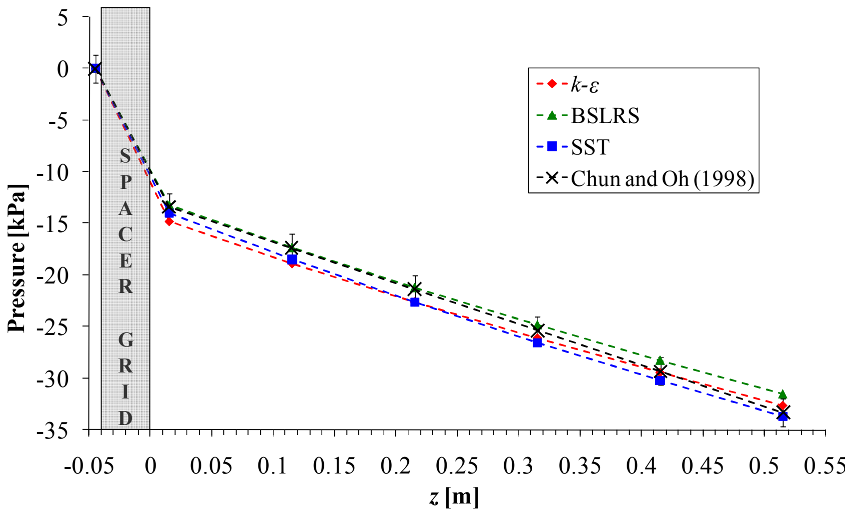

Figure 10. Pressure loss along the rod bundle

In Fig. 10 the spacer grid pressure loss obtained by the BSLRS model showed the best prediction when compared to the semi-empirical formulation with a -1.3% difference. The SST model showed a slightly worst performance but within the uncertainty limit of the formulation with a 4.6% difference. The k-ε model gave the worst prediction with a 10.9% difference, value outside of the formulations uncertainty. Although the tendencies of the friction pressure drop in the bare region predicted by the models are slightly different, on the whole, all models behave well when compared to the semi-empirical formulation. Considering the total pressure drop the BSLRS model showed the worst prediction and SST the best with -5.4% and 1.2% difference, respectively. The k-ε model showed an intermediate difference of -2%.

4. CONCLUSIONS

Simulations on a 5 x 5 rod bundle with spacer grids were analyzed with a commercial CFD code (CFX 11.0). Three turbulence models were evaluated in the simulations: k-ε, BSLRS (Baseline Reynolds Stress) and SST (Shear Stress Transport k-ω). The results were compared with the experimental investigation performed by Karoutas et al. (1995) in which a LDV system was used to measure the axial and lateral velocities. To make the simulation possible with 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 numerical simulations and the experiments. Almost no differences were observed between lateral velocities predicted by the turbulence models. 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 the turbulence models. In these cases the SST model 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 for the total and spacer grid losses along the rod bundle for all evaluated models except the k-ε model that for the grid pressure loss predicted a value 10.9% higher than the formulation, indicating a poorer predictability of pressure behavior of this model for the spacer grid simulation.

In general the results obtained with the evaluated turbulence models show little differences that indicate that SST and BSLRS models are more suited for spacer grid simulation.

The obtained results showed that the numerical procedure of dividing the numerical domain in several parts, each one with a best quality mesh, can be considered a feasible approach for rod bundle simulation.

Although good insight of turbulence model influence on spacer grid simulation results were obtained, further studies, including detailed turbulence and velocity experimental measurements and numerical simulations, are necessary to further comprehend the complex flow downstream spacer grids and properly define the best RANS turbulence model for its simulation.

5. REFERENCES

ANSYS CFX 11.0, 2007, User Manual.

Chun, T. H., Oh, D. S., 1998, “A pressure drop model for spacer grids with and without flow mixing vanes”, Journal of Nuclear Science and Technology, vol. 35, pp. 508-510.

Cui, X. Z., Kim, K. Y., 2003, “Three-dimensional analysis of turbulent heat transfer and flow through mixing vane in a subchannel of nuclear reactor”, Journal of Nuclear Science and Technology, vol. 40, pp. 719-724.

Haixiang, H., Ping, C., “An optimization method applied in development of spacer grid of fuel assembly”, 13th International Conference on Nuclear Engineering ICONE13, Beijing, China, paper no 50673.

Holloway, M. V., Beasley, D. E., Conner, M. E., 2008, “Single-phase convective heat transfer in rod bundles”, Nuclear Engineering and Design, vol. 238, pp. 848-858.

Holloway, M. V., Conover, T. A., McClusky, H. L., Beasley, D. E., Conner, M. E., 2005, “The effect of support grid design on azimuthal variation in heat transfer coefficient for rod bundles”, Journal of Heat Transfer, vol. 127, pp. 598-605.

Holloway, M. V., McClusky, H. L., Beasley, D. E., Conner, M. E., 2004, “The effect of support grid features on local, single-phase heat transfer measurements in rod bundles”, Journal of Heat Transfer, vol. 126, pp. 43-53.

Ikeda, K., Hoshi, M., 2002, “Development of Mitsubishi high thermal performance grid”, JSME International Journal, series B, vol. 45, no 3, pp. 586-591.

Ikeda, K., Hoshi, M., 2007, “Flow Characteristics in spacer grids measured by rod-embedded fiber laser Doppler velocimetry”, Journal of Nuclear Science and Technology, vol. 44, no. 2, pp. 194-200.

Ikeda, K., Makino, Y., Hoshi, M., 2006, “Single-phase CFD applicability for estimating fluid hot-spot locations in a 5 x 5 fuel rod bundle”, Nuclear Engineering and Design, vol. 236, pp. 1149-1154.

Imaizumi, M., Ichioka, T., Hoshi, M., Teshima, H., Kobayashi, H., Yokoyama, T., 1995, “Development of CFD method to evaluate 3-D flow characteristics for PWR fuel assembly”, Transactions of the 13th International Conference on Structural Mechanics in Reactor Technology SMIRT 13, Porto Alegre, Brazil, pp. 3-14.

In, W. K., Chun, T. H., 2005, “Assessment of the RANS turbulence models for a turbulent flow and heat transfer in a rod bundle”, Nuclear Technology, vol. 150, pp. 231-250.

In, W. K., Chun, T. H., Shin, C. H., Oh, D. S., 2008, “Numerical computation of heat transfer enhancement of a PWR rod bundle with mixing vane spacers”, Nuclear Technology, vol. 161, pp. 69-79.

Karoutas, Z., Gu, C., Sholin, B., 1995, “3-D Flow analyses for design of Nuclear Fuel Spacer”, Proceedings of the 7th International Meeting on Nuclear Reactor Thermal-hydraulics NURETH-7, New York, USA, vol. 1, pp. 3153-3174.

Kim, K. Y., Seo, J. W., 2004, “Shape optimization of a mixing vane in a subchannel of nuclear reactor”, Journal of Nuclear Science and Technology, vol. 41, pp. 641-644.

Kim, K. Y., Seo, J. W., 2005, “Numerical optimization for the design of a spacer grid with mixing vanes in a pressurized water reactor fuel assembly”, Nuclear Technology, vol. 149, pp. 62-70.

Launder, B. E. and Spalding, D. B., 1974, “The numerical computation of turbulent flow”, Computer Methods in Applied Mechanics and Energy, vol. 3, pp. 269-289.

Lee, C. M., Choi, Y. D., 2007, “Comparison of thermo-hydraulic performances of large scale vortex flow (LSVF) and small scale vortex flow (SSVF) mixing vanes in 17x17 nuclear rod bundle”, Nuclear Engineering and Design, vol. 237, pp. 2322-2331.

Menter, F. R., 1994, “Two-equation eddy-viscosity turbulence models for engineering applications”, AIAA-Journal, vol. 32, pp. 269-289.

Rehme, K., 1973, “Pressure drop correlations for fuel element spacers”, Nuclear Technology, vol. 17, pp. 15-23.

Shen, Y. F., Cao, Z. D., Lu, Q. G., 1991, “An investigation of crossflow mixing effect caused by grid spacer with mixing blates in a rod bundle”, Nuclear Engineering and Design, vol. 125, pp. 111-119.

Tóth, S., Aszódi, A., 2008, “CFD analysis of flow field in a triangular rod bundle”, Nuclear Engineering and Design, doi:10.1016/j.nucengdes.2008.08.020.

Yao, S. C., Hochreiter, L. E., Leech, W. J., 1982, “Heat transfer augmentation in rod bundles near grid spacers”, Journal of Heat Transfer, vol. 104, pp. 76-81.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.