DUJASE Vol. 5(1 & 2) 37-42, 2020 (January & July)

Modelling of Flow Turbulence and Routine Analysis of a Split Type Vane Spacers Grid in a Bundle of Rods of A 5X5 PWR Using ANSYS CFX

Kazi Kamrun Naher Ononna[1], Md. Towhid Hossain[1], Md. Mahidul Haque Prodhan[1]

[1]*Department of Nuclear Engineering, University of Dhaka, Dhaka-1000, Bangladesh*

[*]*E-mail: prodhan@du.ac.bd*

Received on 05 June 2020, Accepted for publication on 10 September 2020

ABSTRACT

Spacer grids are essential for enhancing the mechanical performance of the fuel rods in the nuclear reactor core. Spacer grids are always in systematic square shape. It is significant to enhance the features of the grid with vanes since the maximum coolant mixing and less pressure drop must be permitted to the thermal hydraulic model. Earlier, researchers had used Computational Fluid Dynamics (CFD) in 3D Reynolds Averaged Navier-Stokes (RANS) and k-ϵ turbulence model. Korean Atomic Energy Research Institute (KAERI) performed CFD evaluation of a (5x5) bundle of rods experiment and their simulations were completed in OECD/NEA CFD Standard application. According to KAERI output, the turbulence features in spacer grid geometries in a PWR (pressurized water reactor) is offered and the impact of the results were evaluated with the help of ANSYS CFX in this article. A reactor stream feature and three dissimilar vane arrangements are simulated and average thermo-fluid dynamic parameters are assessed. The simulations are performed with the CFD code of CFX 15.

Keywords: Spacer grid, CFD, Vanes.

1. Introduction

A nuclear reactor is a manoeuver, intended to produce and withstand a long period, well-ordered fission chain reaction and is prepared with a carefully selected and tactically placed assortment of countless materials. Perception of atoms has occurred for thousands of years. We have recently started to comprehend the gigantic power confined in the miniature mass. In previous ages and throughout World War II, nuclear research fixated mainly on the enlargement of defence weapons. Later, scientists focused on diplomatic applications of nuclear technology. If the world is to take advantage from nuclear energy in the long term despite the impending dangers involved, it is necessary that the lessons erudite from each accident or episode are assimilated into future designs and into operator training and safety management to make remaining stations harmless.

For the safety analysis of nuclear reactor, numerous multi-dimensional thermal-hydraulic singularities cannot be predicated using one-dimensional lumped-parameter method codes to model reactor systems. The accuracy of neutronics calculations in fuel assembly designs are depended on the safety of reactor. The spacer grid shows a major role in ancillary the fuel rods horizontally and vertically. Pointing to reconcile the concept of flow features in the downstream of the spacer grids, probationary and conceptual surveys have been conducted. LDV (Laser Doppler Velocimetry) and PIV (Particle Image Velocimetry) have been used in the experiments which permit local velocities to be calculated amid the sub channels which have shown great enhancement recently still costly and lengthy. In contrast, in theoretical assessments as in Computational Fluid Dynamic (CFD), it takes short time for the outcomes where the Reynolds Averaged Navier-Stokes (RANS) model is explained, at adequately economical [1].

Typically, affluent experiments are necessary for testing of different grid geometries. Nevertheless, currently the supplementation or even replacement of experiments by CFD numerical analyses are of relevant attention, primarily owing to rapid development of progressive computer hardware that allows highly efficient parallel computing. There have been important improvements in the presentation of computers, which permit us to construct computational models of complex constructions connected with the flow in rod bundles and to achieve a converged explanation of the governing equations. Regarding to computational restrictions, the hypothesis derived from the speculation of a complete rod bundle is not constantly conceivable. Because, a simulation completed in single sub channel can be untrustworthy for the speculation of entire bundle of rods, these interpretations do not always lead to consistent outcomes. Nevertheless, to elevate the parametric studies for grid proposals, these outcomes can be used as an initial approach because of their unity and qualitative resemblance with the vital outcomes.

Exercise on mixing in a bundle of rods which is second international benchmark OECD/NEA-KAERI rod bundle CFD exercise centered on MATiSH experiments (Measurement and Analysis of Turbulent Mixing in Sub channels—Horizontal) [2-3].

Those tests can afford the data for exposing features of the mixing in sub channels. They also approve the uses of the CFD codes which can be used as tools for future in modelling of spacer-grids features as in pressure drop. Codes can be used for enumerating CHF margin dependably for regular operation situations, functioning transient scenarios and also consenting ultimately the uses of CFD codes for anticipating the DNB in accidental situations. It is important to select the greatest modelling opportunities to seizure the important characteristics of the turbulent assembly's down- stream of the spacer grid. Until recently, lessons of coolant flow properties in nuclear reactors were depending on experiments. This predestined intricate and expensive test facilities had to be fabricated in order to calculate flow properties in the rod bundles. Here, the results of a CFD evaluation on flow through a (5 x 5) bundle of rods with one spacer grid are presented using the commercial code CFX 15 [4]. Targeting to evaluate CFD methodology of spacer grid designs and evaluating the impact of vanes on the flow inside the grid of spacer are the objectives of this study.

2. Arithmetical procedure

Accurate forming of these PWRs is difficult, especially in high fuel rod to shell diameter ratios and for the huge number of fuel rods. The simplest and most regularly used method is demonstrating heat transfer by using some empirical correlations. Due to the empirical nature, this method might turn out to be highly inaccurate. This is for the reason that those correlations cannot represent the complex nature of the fluid dynamics and geometry for particular state. The most exclusive and time-consuming technique is obviously experimental study. Yet, there are major complications inherent in this method including attaining temperature profiles. The arrangement for this study may be: At first the initial design should be established. Then the flow study should be processed in detail using CFD estimation [5]. Then a prototype should be optimized theoretically. After that a practical study was set up, tentative attire had been executed and results were obtained. Finally, the design was modified and more significantly validity of the imitation by obtainment was novel, expectantly improved outcomes. The CFD code of CFX 15 based on the finite volume technique was used. Mass, turbulence, momentum and energy RANS equations were then interpreted.

2.1 Experimental Design

The channel contains a 5x5 rod bundle [6], individually the 25 rods of 22.9 mm outer diameter and 3780 mm in length, signifying the rods in an authentic fuel bundle. The rods assist as obstructions in the assessment, and are not warmed. The occupied liquid is water. The test attire is 2.6 times bigger than a reactor-grade bundle so as to afford good dimension perseverance. A spacer grid is placed in the rod bundle to improve horizontal flow collaborating. The Reynolds number was 49000, conforming to an axial velocity of 1.48ms^{-1} inside bundle part. Circumstances in the water circle were strictly maintained in the tests to control a persistent temperature of 34℃ and an ambient pressure of 1.59 bar. Thorough measurements of the velocity arena were taken at four positions of downstream in the grid of spacer: 0.5, 1.0, 4.0 and 10.0D_h, calculated from downstream corner of the grid. Flow circumstances were also calculated upstream of the grid, precisely to offer proper boundary conditions for allied CFD evaluation.

2.2 Geometry

A 3D printing method was used to made the grid which was organized to generate a flow. The geometry and mesh for the full benchmark geometry were produced using the software CATIA. In CATIA generative shape design of part design is used. Test region comprises of a square casing, setting plate, container, some supports for rods, a bundle of rods and spacer grid with vanes of a standard design. The flow plane model of this evaluation symbolizes approximately 1/7 of actual size of fuel component, with a (5 x 5) bundle of rods and a spacer grid within a housing which is square in shape and has 170 mm width. The diameter of one rod is 9.5 mm and the pitch of the bundle is 12.6 mm.

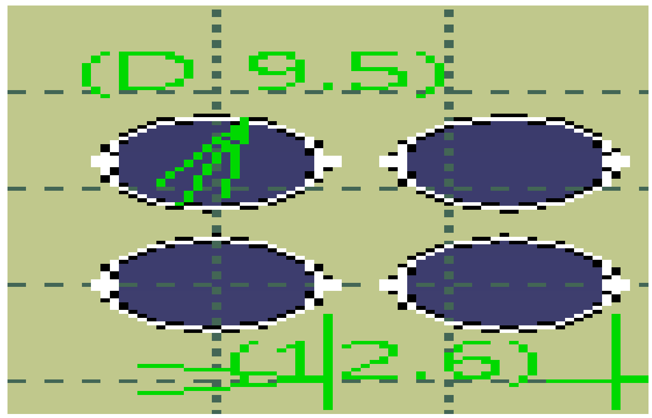

Fig. 1: The modelled sub channel and its dimensions

The bundle of rods has 14.27 mm hydraulic diameter (D_h) and an overall range of flow is 2851.95 mm^2 through bare region. The cross-sectional view unambiguously displays the vane outline and finite wideness of a grid plate and vane.

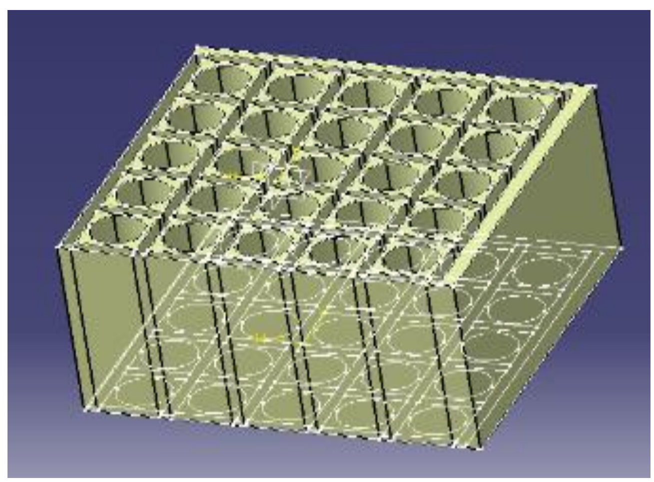

Fig. 2: Geometry of spacer grid fuel rods with no vanes

Three spacer grids were demonstrated in this analysis [7]:

1. a grid without vanes (No vanes grid),
2. a grid with the standard vane distribution (Standard grid) with peripheral vanes
3. a grid with the standard vane distribution (Standard grid) without peripheral vanes

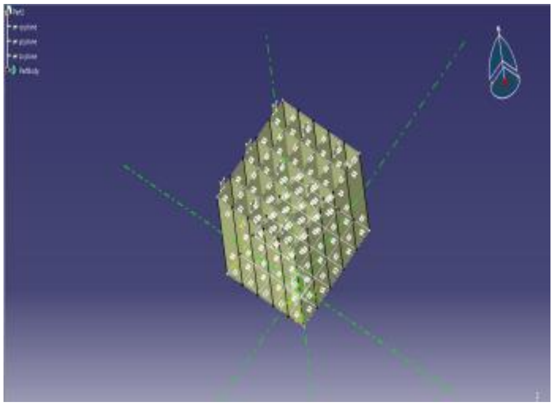

Fig. 3: CATIA geometry of standard grid split vane.

1 All appraised grid has the same geometrical proportions and appearances.

2 The pressure profiles specify that if the rod test sections are divided or not, do not cause significant differences of the variables.

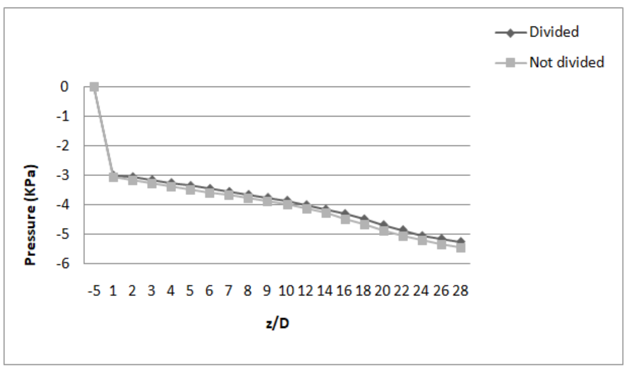

Fig. 4: Pressure downstream the spacer grid without and with the domain division

2.3 Mesh

ANSYS ICEM CFD code is used to build the geometrical model and to create different numerical meshes of the sub channel. Meshing is an essential step in such an exploration. The mesh has to be adequate for the given turbulence model and should be of high quality. Mesh was produced with scaling the geometry to decrease the cell size in the axial direction.

Table 1: The Tetrahedral mesh has the following properties

Spacer type	Split vane
Mesh type	tetrahedral
Cell elements	9.5 million
Cell length	0.31 mm 75.2 mm
2x2x2 determinant	0.018% elements have 0.06-0.1 the others have 0.01
Angles	between 10° -18°

The grid convergence assessment is accomplished using three dissimilar meshes modifications in spacer grid. 7.1million cells, 7.9 million cells, 9.5 million cells are generated. A tiny grid impacts the meshes larger than 7.9 million cells and verified by pressure outlines. Two mesh categories are created to evaluate the bundle section.

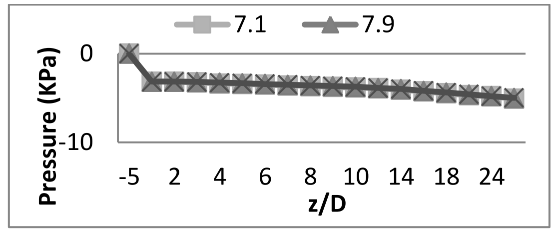

Fig. 5: Pressure profiles for different meshes.

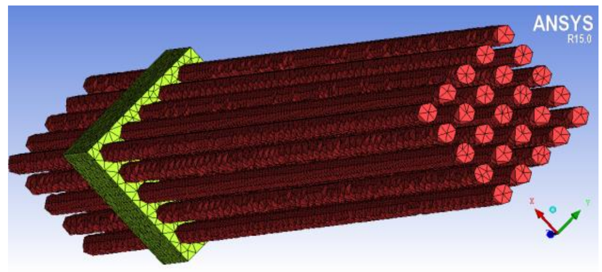

Fig. 6: Meshes of no vane spacer grid.

The un-solidified domain of the design model is meshed with a tetrahedral can be a time overriding process. Nevertheless, the simulations converge 3 to 8 times faster on assembled hexahedral mesh.

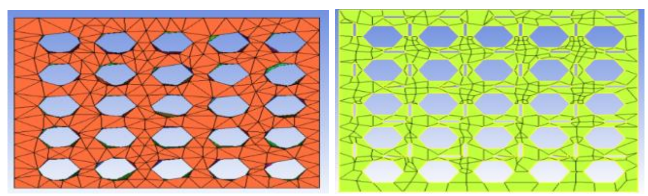

Fig. 7: Mesh cross section in the (Top) bare bundle, (Bottom) vane

2.4 Boundary Condition

Table 2: Inlet boundary condition

Mass flow rate	8.15 kg/s
Temperature	$300^{\circ C}$
Pressure	159 bar
Average velocity	4 m/s

Table 3: Outlet boundary condition

Pressure drop	0 Pa
Water temperature	35°C
Dynamic	$0.73 \times 10^{-3} \frac{Ns}{m^2}$
Kinematic viscosity	$0.73 \times 10^{-6} \frac{m^2}{s}$

3. Numerical simulation

The chief differencing and the amalgam second order systems are applied, correspondingly, to juxtapose the dispersion and the advection relations of equations. This design presents good arrangement with velocities those are being measured and have permanent convergence. A remaining RMS assessment of 10^{-5} is distinct for overall simulation. Intel core i5 CPU of 2.71 GHz with 4GB of RAM personal computer is used to simulate the outcome which result in 96-100 hours approximately. From the post CFX processor we get the calculation functions. The vectors, contours, streamlines from different velocity and directions are given

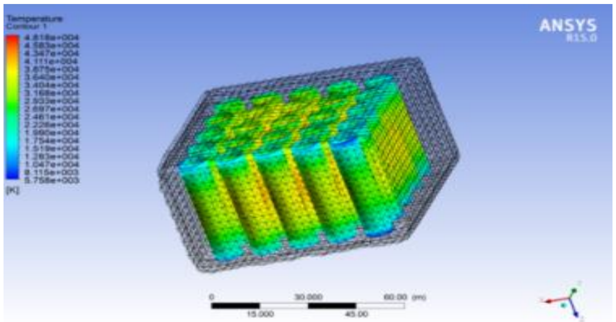

Fig. 8: temperature contour of the spacer grid with no vane

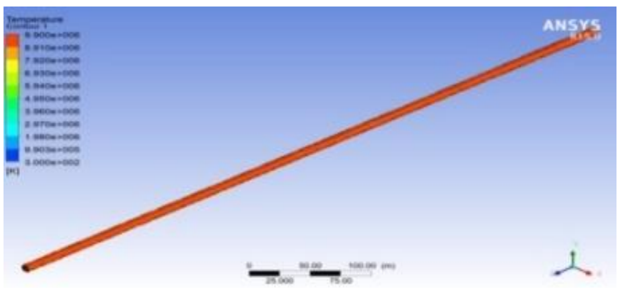

Fig. 9: Temperature contour of one rod

From the figures we can see that the turbulence flow, at the centre and near the rod is very concentrated. So the areas near the rods are considered to measure the velocity profile.

4. Results and Discussion

The swirl flow configuration declines and the twister converts into circular shape as the liquid flow develops downstream of the vanes, which could become significant convective mingling among the sub channels. The velocity connected the swirl flow is considerably greater than in the gaps between rods. Astonishingly, as the flow goes in axial path, flow in vane with channel is produced in the channel without vanes as well. A possible explanation for this might be the flow does not endure in the sub channel center and here has a robust outflow inside the gap of the rods and it could produce swirl flow in the no mixing vane sub channel [8]. So the thermal performance is increased by the addition of mixing vanes by growing inter-channel mixing, prompting swirl flow and upholding the turbulence effect in the sub channels. We can see at the time averaged values of velocity and variations of them when the simulation gets a converged situation. The time of the simulations necessary for time averaged amounts to congregate is in the sequence of seconds. Along the lines at $0.5D_h$, $1.0D_h$, $10D_h$ and $15D_h$ time averaged magnitudes were taken downstream tips of the vanes in channels among the rows of the rod. Simulation data is taken alongside the same lines after that matched to the data of the experiment. Time dependence of averaged velocities and their variations:

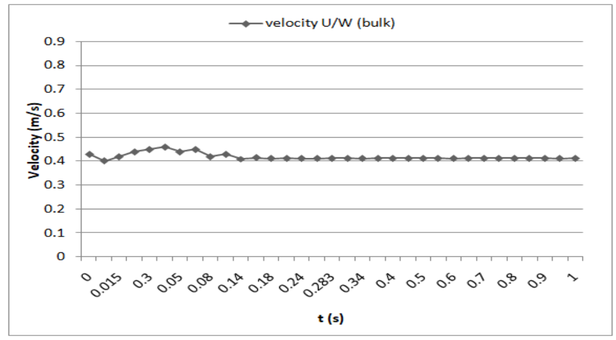

Fig. 10: Transient average velocity $\mathbf{u/w_{bulk}}$

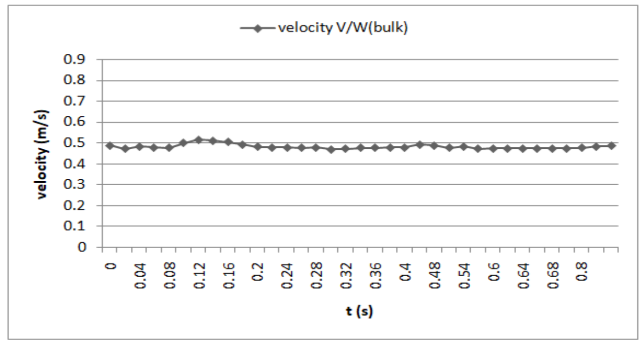

Fig. 11: Transient average velocity ($v/\mathbf{w_{bulk}}$)

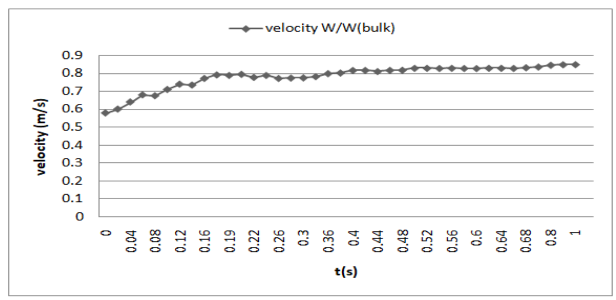

Fig. 12: Transient average velocity ($w/\mathbf{w_{bulk}}$)

When the time averaged values have converged, we are able to compare them to the measured values from the experiment [2-3] Figure shows comparison between experimental and simulated values for each component of velocity and its RMS fluctuations.

Experimental and simulated velocity components and their RMS fluctuations at $0.5D_h$:

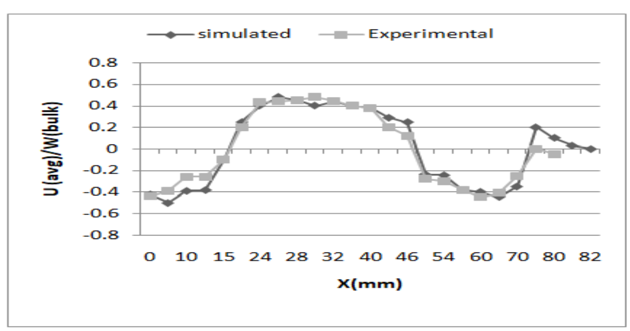

Fig. 13: Experimental and simulated variations of $\mathbf{U_{avg}}$

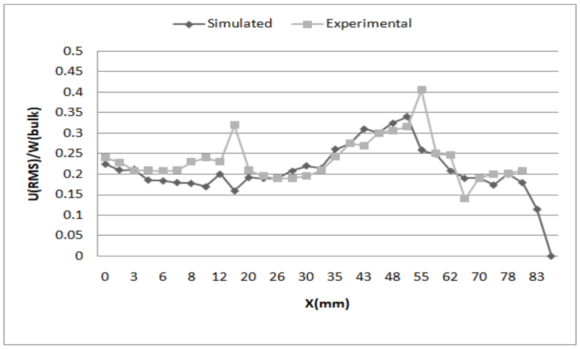

Fig. 14: Experimental and simulated variations of $\mathbf{U_{RMS}}$

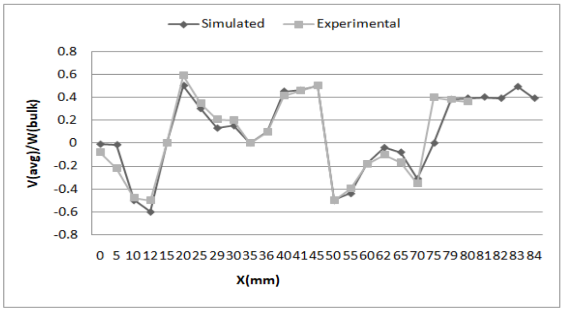

Fig. 15: Experimental and simulated variations of $\mathbf{V_{avg}}$

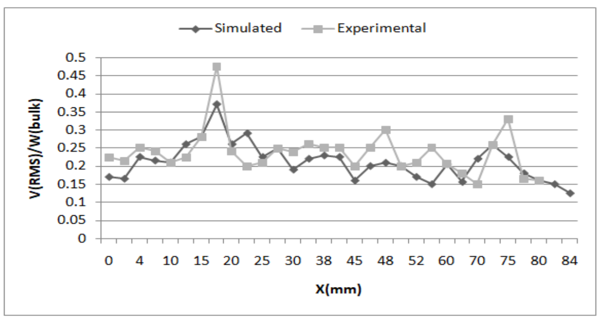

Fig. 16: Experimental and simulated variations of $\mathbf{V_{RMS}}$

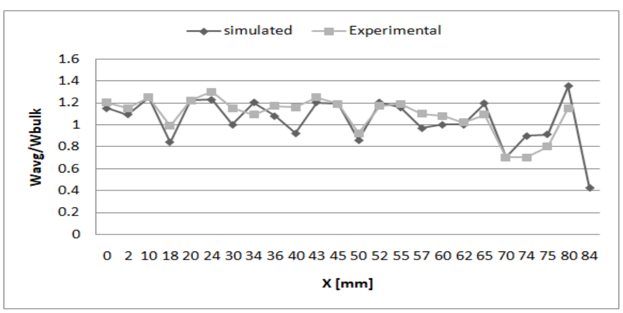

Fig. 17: Experimental and simulated variations of $\mathbf{W_{avg}}$

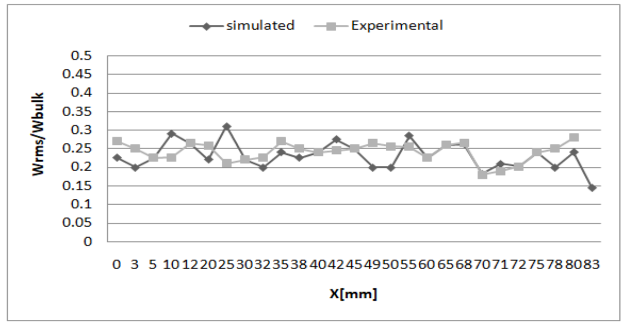

Fig. 18: Experimental and simulated variations of $\mathbf{W_{RMS}}$

Figures (13-18) show that the average velocity outlines fit the experimental velocity fine all through the calculating area and there have slight deviations. When matching the RMS variations, there show better agreement with the results of this experiment. Intensity of the peaks which are correctly predicted with the computational model in all velocity components is underestimated. The relative and absolute errors are the standard relative and absolute error.

All parameters were evaluated as averages, calculated at cross sectional planes along with the channel.

Secondary Flow (SF) equation:

$$SF = \frac{\bar{U}cross}{\bar{U}bulk} = \frac{1}{A}\sum_{i} \frac{\sqrt{(Uxi^2 + Uyi^2)}A_i}{U_{bulk}}$$

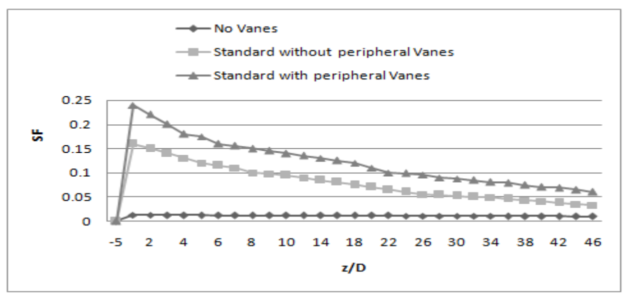

Fig. 19: Secondary flow for the no vane, standard without peripheral vanes, standard with peripheral vanes

Nusselt number (Nu_{avg}):

The average Nusselt , $Nu = \frac{\bar{h}D_h}{k}$; the average heat transfer coefficient is $\bar{h} = \frac{q''}{T_r - T_w}$

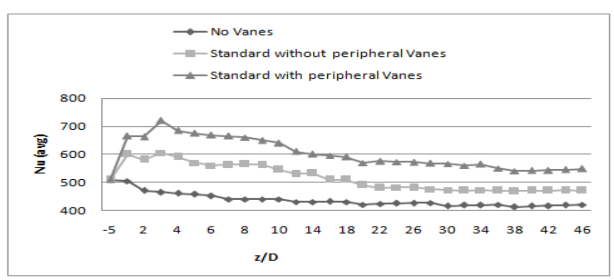

Fig. 20: Nusselt number for the no vane, standard without peripheral vanes, standard with peripheral vanes

TNU:

$$\mathrm{TNU} = \sqrt{\left(\frac{1}{\mathrm{A}}\sum(\mathrm{T_i} - \overline{\mathrm{T}})^2 A_i\right)}$$

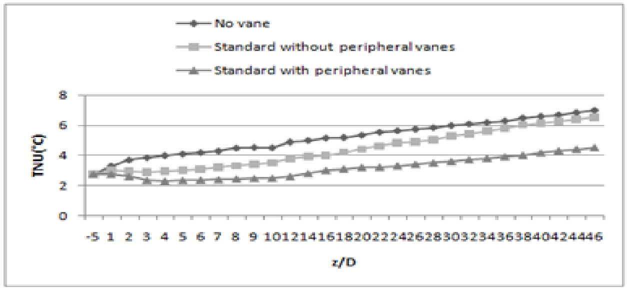

Fig. 21: TNU for the no vane, standard without peripheral vanes, standard with peripheral vanes

The figures (19-21) are showing that the presence of peripheral vanes greatly increases the SF. Secondary flow differences between the Standard grids with and without peripheral vanes are the velocity vectors in a cross sectional plane. While the grid without peripheral vanes has a defined swirl flow on each of the vane sub channels, the grid with the vanes shows a combination of swirl and cross flow between sub channels and less organized flow behavior. Thermal non-uniformity (TNU) is decreasing with the vanes. The figures show that the presence of peripheral vanes greatly increases the Nusselt number. These figures compare thermo-fluid dynamic parameters along the axial length of no vane and standard grids (without and with peripheral vanes).

5. Conclusion

Hydrodynamic safety calculation is a significant method for explaining the safety of NPP. Specification of the fluid proportion enables us to prohibit and anticipate the probable influence that connects to fluid representative modification. The grids exert great impact to the thermal-hydraulic enactment of the PWR assembly. The spacers that bolster the rods in an assembly are furnished with vanes acted as turbulence-enhancing equipment to enhance the heat transfer. The PWR fuel assembly grids geometry has a robust impact on safety and performance issues. The results of the evaluation of a (5 x 5) bundle of rods with grid analysis show that the simulated values have an agreement with the experimental [2-3] Values. Though in the experiment, LDV (Laser Doppler Velocimetry) and the PIV (Particle Image Velocimetry) are used to complete the experiment but in CFD, velocity measurements are done by in-built equations. So, there is some difficulty to compare the experiments. The trends and magnitudes of separate velocity components are correctly predicted by the computational model for improvement in the computational mesh. The RMS fluctuations are slightly underestimated. The results give a sufficient level of confidence in the computational model for giving the CFD models an ever-increasing role in designing both the components of nuclear reactors and other industrial products [9]. Peripheral vanes on (5 x 5) bundle of rods spacer grid was presented. Effects of vane arrangement were assessed. The vanes influence the grid pressure drop and secondary flow. Vanes also enhance the transfer of heat and mixing in downstream. It is assumed that the flow and heat flux conditions are similar to the reactor. In hypothetical experiments as in CFD, outcomes can be generated in a moderately low duration. Future endeavour is essential to analyses the impacts of a turbulence prototype and mesh species, especially for a remote downstream heat transfer in a thermal assortment simulation. In this prospect, the simulations should be accomplished with several vane samples (shapes and dimensions) and different configurations.

References

1. Mimouni, S. et al. "A Second Order Turbulence Model Based on a Reynolds Stress Approach for Two-Phase Boiling Flow and Application to Fuel Assembly Analysis", *Nucl. Eng. Des.*, 240, 2010, pp. 2225-2232

2. J. R. Lee, J. Kim, C. H. Song. "Synthesis of the OECD/NEA-KAERI Rod Bundle CFD Benchmark Exercise''', Proc. Conf. CFD4NRS-4, Daejeon, Korea, September 10-12, *Korea Atomic Energy Research Institute*, 2012

3. S. K. Chang, S. Kim, C. H. Song, "OECD/NEA-KAERI Rod Bundle CFD Benchmark Exercise Test", Proc. Conf. CFD4NRS-4, Daejeon, Korea September 10-12, *Korea Atomic Energy Research Institute*, 2012

4. ANSYS CFX, Release 13.0, *ANSYS CFX-Solver Theory Guide*, ANSYS.

5. Shen Y. F., Cao Z. D., Liu Q. G., An investigation of crossflow mixing effect caused by grid spacer with mixing blades in a rod bundle, *Nuclear Engineering and Design*, 125(2): 111–119, 1991.

6. McClusky H. L., Holloway M. V., Beasley D. E., Conner M. E., Development of swirling flow in a rod bundle sub channel, *Journal of Fluids Engineering*, 124(3): 747-755, 2002.

7. McClusky H. L., Holloway M. V., Conover T. A., Beasley D. E., Conner M. E., Smith L. D., Mapping of the lateral flow field in typical sub channels of a support grid with vanes, *Journal of Fluids Engineering*, 125(6): 987-996, 2004.

8. Karoutas Z., Gu C. Y., Scholin B., 3-D flow analyses for design of nuclear fuel spacer, *Proc. 7th Int. Mtg. Nuclear Reactor Thermal-Hydraulics (NURETH-7)*, New York, USA, 1995.

9. Imaizumi M., Ichioka T., Hoshi M., Teshima H., Kobayashi H., Development of CFD method to evaluate 3-D flow characteristics for PWR fuel assembly, *Transaction of 13th Int. Conf. Structural Materials in Reactor Technology (SMiRT)*, Porto Alegre, Brazil, 1995.