ICIT 2015 The 7th International Conference on Information Technology
doi:10.15849/icit.2015.0088

Numerical Simulation of Axial Coolant Flow in Rod Bundles of a Nuclear Reactor

Y. Rihan
Nuclear Fuel Tech. Dep.
Atomic Energy Authority, Hot Lab. Center,
Egypt.
yarihan159@yahoo.com

I. Salama
Nuclear Fuel Tech. Dep.
Atomic Energy Authority, Hot Lab. Center,
Egypt.

Abstract— **Numerical simulation of coolant flow inside the rod bundle of a nuclear reactor is of great engineering interest. In the design of innovative core solutions, such as high conversion tight lattice cores for Light Water Reactors (LWR), as flow distribution cannot be calculated with exact analytical methods, numerical modeling plays a vital role. A computational fluid dynamics (CFD) methodology is proposed to investigate the thermal–hydraulic characteristics in a rod bundle. Using a three dimensional numerical solution, the characteristics of an isotropic k-epsilon turbulence model for use in modeling turbulent interchange mixing within rod arrays was investigated. The model used to predict the radial component of turbulent eddy viscosity and wall shear. Existing data of Nusselt number distributions in the axial direction obtained by different authors have been employed to validate the CFD model.**

Keywords*— *Coolant flow; Rod bundles; Nuclear; Turbulent; Modeling

I. INTRODUCTION

Understanding the physical behavior of the coolant as it flows through the fuel bundles is of interest to those analyzing reactor operation and safety. An important aspect of understanding this behavior is the mixing of coolant momentum and heat. Without adequate coolant mixing models, the heat removal capabilities and safety margins of the reactor cannot be accurately established nor predicted. 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.

Many nuclear subchannel analysis codes adopt the lumped parameter approach, where many empirical correlations are used to simplify the complex exchange phenomena between subchannels. Therefore, the prediction capability of a subchannel analysis code depends thoroughly on the pertinent usage of the models and correlations [1]. A subchannel is defined as the flow area bounded by a cluster of fuel rods. In most rod bundle configurations, two basic subchannel geometries are found: square and triangular as shown in Fig. 1.

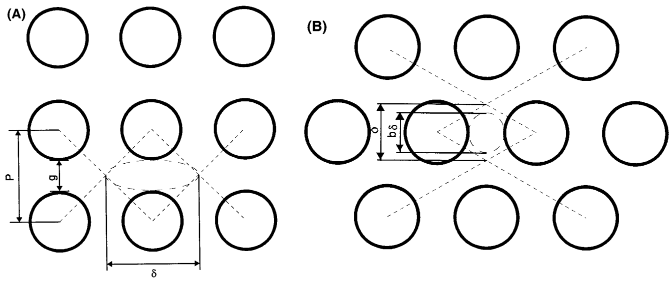

Fig. 1. Schematic view of rod bundle geometry of (a) square array; (b) triangular array

There have been several studies on flow mixing and heat transfer enhancement caused by a coolant flow in rod bundle geometry. Amongst the many studies performed involving CFD (Computational Fluid Dynamic) simulations of rod bundles with spacer grids some with the most significant contributions are: Karoutas et al. [2] and Imaizumi et al. [3] that demonstrated the usefulness of single subchannel CFD methodologies coupled with experimental results from LDV (Laser Doppler Velocimetry) and pressure loss measurements on the development of fuel designs for PWR reactors. Navarro

doi:10.15849/icit.2015.0088 © ICIT 2015 (http://icit.zuj.edu.jo/ICIT15)

et al. [4] used the *k*-*ε* model that presents results of flow simulations performed with the CFD code in a PWR 5×5 rod bundle segment with a split-vane spacer grid. Holloway et al. [5] showed that there is a great variation of heat transfer distribution along a fuel rod due to the spacer grid type. Wu and Trupp [6] clearly demonstrated that flow conditions inside the fuel bundles are very different from those in the typical pipes. The near-wall turbulence anisotropy results in the formation of secondary vortices inside the channel, causing the coolant to spiral through the bundle. Liu et al. [7] presented the results of numerical issues such as mesh refinement, wall treatment and appropriate definition of boundary conditions, which exert great influence on the results of a CFD simulation.

Recently, simulation studies were performed to investigate the thermal–hydraulic phenomena within a rod bundle [8-15]. Ga'bor [16] demonstrated that the Reynolds stress model (RSM) could be a good candidate for the accurate modeling of rod bundles. Baglietto and Ninokata [17] show that a quadratic k–e model with adjusted coefficients can reproduce the wall shear stress and the velocity distributions a fully developed flow in a triangular lattice bundle. A CFD model was developed by Lin et al. [18] to investigate the flow characteristics in the rod bundle with the different pressure–strain models in RSM, including linear pressure-strain (LPS), Quadratic Pressure–Strain (QPS) and low-Re stress-omega (LROS) models using ANSYS FLUENT solver. Subchannel model was developed and the study of mesh sensitivity was performed initially. Most of previous CFD studies for rod bundles were focused on the hydraulic simulation. In addition, previous simulation works neglect the thickness of vane-pair spacer grids for modeling simplification. This simplification would increase about 15% flow area in the grid region and lower the flow velocity, which may result in different flow and heat transfer characteristics.

This paper introduces a mathematical model to understand the physical behavior of coolant mixing within a nuclear fuel bundle and assess its applicability to this study.

II. The Mathematical Model

A mathematical model was developed in this paper to investigate the flow characteristics in rod bundles subchannels. This simulation mathematical model includes the continuity equation, momentum equation, energy equation, and *k*-*ε* turbulence model.

Continuity equation:

$$\frac{\partial(u_i)}{\partial x_i}=0 \tag{1}$$

Momentum equation:

$$\rho\frac{\partial u_i u_j}{x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\rho u_i' u_j'\right]-\rho g_i \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial x_i}\left(u_i(\rho E+P)\right)=\frac{\partial}{\partial x_i}\left(K_{eff}\frac{\partial T}{\partial x_i}+u_j\left(\tau_{ij}\right)_{eff}\right) \tag{3}$$

k-*ε* turbulence model:

The transport of turbulent kinetic energy per unit mass in high Reynolds number form can be provided by the following equation [19].

$$\rho\frac{\partial k}{\partial t}+\rho u_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+P_k-\rho\varepsilon+\frac{\rho\mu_t}{\mathrm{Pr}_t}\beta\left(g_i\frac{\partial T}{\partial x_i}\right) \tag{4}$$

Where P_k is the volumetric production of k and can be expressed as follow:

$$P_k=\left[\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]\frac{\partial u_i}{\partial x_j} \tag{5}$$

The high Reynolds number form of the transport equation for the turbulence dissipation rate is given by the following.

$$\rho\frac{\partial \varepsilon}{\partial t}+\rho u_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+\frac{\varepsilon}{k}\left(c_{\varepsilon1}P_k+c_{\varepsilon2}\rho\varepsilon\right) \tag{6}$$

The empirical constants used in the *k* and *ε* equations are summarized in Table I.

TABLE I. *K*-*ε* MODEL CONSTANTS

c_μ	$c_{\varepsilon1}$	$c_{\varepsilon2}$	σ_k	σ_ε	Pr_t
0.09	1.44	1.92	1.0	1.3	0.9

A. Boundary conditions

The wall boundary condition must account for the influence of the three layers (laminar, buffer, and logarithmic) on momentum transport to the wall. This is achieved by using a law of the wall for the momentum equation's wall boundary condition.

$$U^+=\frac{u}{u_\tau}=\frac{\ln y^+}{0.41}+5.2 \tag{7}$$

The dimensionless distance from the wall, y^+, is defined in terms of the shear velocity, u_τ, and wall distance, *y*.

where,

$$y^+=\frac{\rho u_\tau y}{\mu} \tag{8}$$

$$u_\tau=\sqrt{\tau_w/\rho} \tag{9}$$

A similar law of the wall is used as a boundary condition for the energy equation in the turbulent flow. For a specified wall temperature, T_w, the wall heat flux, q_w, is given by the following.

$$q_w=\frac{u_\tau}{T^+}\left(T_w-T\right)$$

doi:10.15849/icit.2015.0088 © ICIT 2015 (http://icit.zuj.edu.jo/ICIT15)

(10)

The dimensionless temperature can be evaluated as following [20].

$$T^{+} = \Pr y^{+} e^{-\Lambda} + \left(2.12 \ln y^{+} + \beta^{*}\right) e^{-1/\Lambda} \tag{11}$$

$$\Lambda = \frac{0.01\left(\Pr y^{+}\right)^{4}}{1 + 5\Pr^{3} y^{+}} \tag{12}$$

$$\Pr = \frac{\mu c_p}{k} \tag{13}$$

$$\beta^{*} = \left(3.85 \Pr^{1/3} - 1.3\right)^{2} + 2.12 \ln \Pr \tag{14}$$

The wall boundary condition for the k and ε equations is based upon assuming the production of turbulence equals dissipation, constant shear layer, and the velocity gradient normal to the wall is much greater than the gradient along the wall.

$$\varepsilon = \frac{u_\tau^{\ 3}}{0.41 y} \tag{15}$$

The production of turbulent kinetic energy, P_k, can be calculated as follow:

$$P_k = \frac{\tau_w^{\ 2}}{\mu} \frac{dU^{+}}{dy^{+}} \tag{16}$$

The inlet conditions for the k and ε equations utilize a specified turbulence intensity, T_u, and eddy length scale, L_ε.

$$k = \frac{3}{2} T_u^{\ 2} U_b^{\ 2} \tag{17}$$

$$\varepsilon = \frac{k^{2/3}}{L_\varepsilon} \tag{18}$$

The turbulence intensity is usually specified as 0.05, while the eddy length scale is specified as equal to a domain characteristic length, such as the radius for flow in a circular pipe.

The average dimensionless Nusselt number (Nuave) offers an insight on convective heat transfer that occurs at the surface of the rods. The average Nusselt number is defined by the following equation:

$$Nu_{avg} = \frac{hD_h}{k} \tag{19}$$

The local Nusselt number and normalized Nusselt number around the circumference of the rod bundle at each axial location are given by the expressions,

$$Nu = \frac{h(z) D_h}{k} \tag{20}$$

and

$$\frac{Nu(z) - Nu_{avg}(z)}{Nu_{avg}(z)} \tag{21}$$

The conservation and transport equations are solved on a discretized domain by integrating the differential equations over discrete control volumes. Flux terms are solved at integration points which ensure a strongly conservative solution. The weak coupling between pressure and velocity is treated by splitting the numerical evaluation of velocity into mass-flow and momentum-flow terms. This allows the use of collocated grids. A Gauss-Seidel solver is used which has the property that high-frequency error components are eliminated faster than low-frequency components.

III. Results and Discussion

The measured data of Nu number for a rod bundle obtained by Holloway et al. [21] are used to validate the present model. Figure 2 shows the comparison of normalized Nu number along the axial location of the rod bundle between the measured data and the model predictions. The coolant fluid was the air in this experiment. The normalized Nu number is the average Nu number (Nuavg) divided by Nuavg,∞. The predicted Nuavg number is obtained by averaging the local Nu number around the azimuthal angle. As clearly revealed in Fig. 2, the predicted distribution of normalized Nu number agrees well with the measured data. The comparison reveals that k-ε turbulence model is suitable to be applied in simulating the flow and heat transfer in the rod bundles.

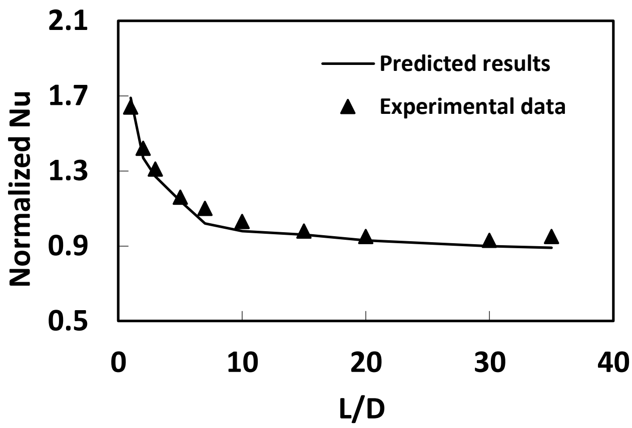

Fig. 2. Comparison between the predicted normalized Nu and the experimental data of Holloway et al. [21]

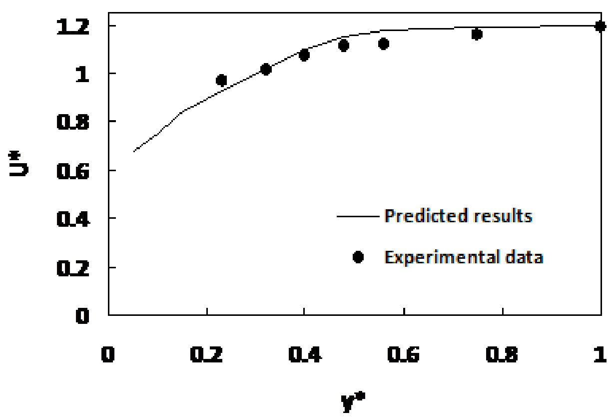

Fig. 3. Comparison between the predicted and the experimental data of Trupp et al. for normalized axial velocity [22]

A comparison of normalized axial velocity distributions (U^*) along the normalized distance from the wall ($y^* = y/L$) between the measurements data of Trupp et al. [22] and the predictions is shown in Fig. 3. As shown in this figure, the predicted distributions agree well with the measured data. This figure also reveals that the velocity distribution predicted by k-ε turbulence model is suitable to be applied in simulating the flow in the rod bundles.

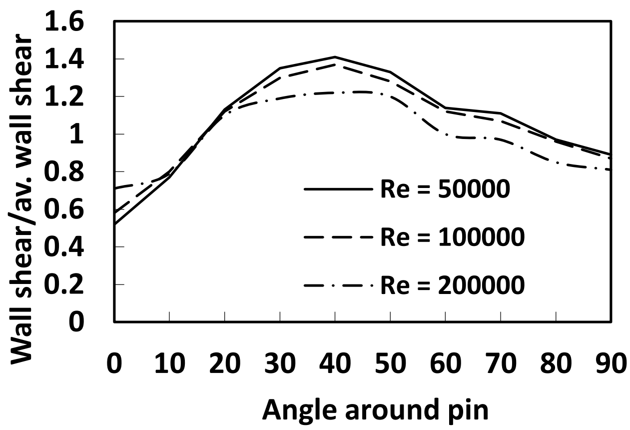

Fig. 4. The wall shear around rod, $\gamma_p = 1.1$

The mathematical model predicted variation in wall shear around the rod surface with a pitch-to-diameter ratio (γ_p) of 1.1 and different Reynolds numbers is shown in Fig. 4. The variation in wall shear near the gap is promotional to the variation in turbulence kinetic energy. The predicted maximum wall shear stress occurs approximately at angle 40 degrees from the gap and is constant with respect to Reynolds number. The effect of γ_p on the wall shear at constant Reynolds number is shown in Fig. 5. The wall shear distribution is more flat as γ_p increases and there is a definite movement of the location of peak wall shear towards the gap ($\theta = 0$). The wall shear is proportional to the radial gradient of axial velocity, this leads to the movement of the position of maximum wall shear into the gap for larger values of γ_p.

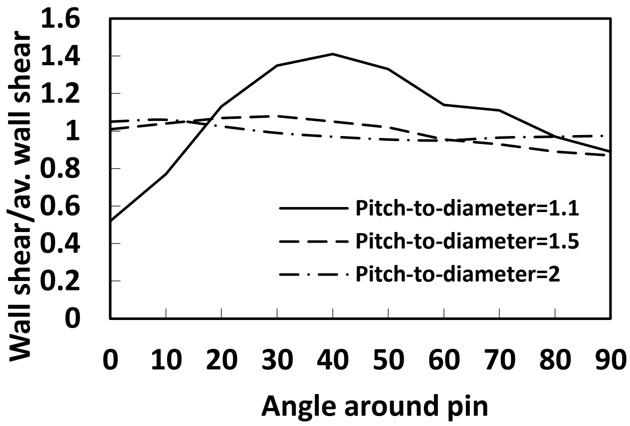

Fig. 5. The wall shear around rod, Re = 50000

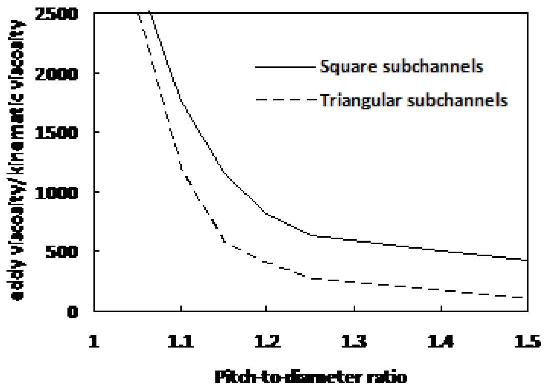

Fig. 6. Effect of pitch-to-diameter ratio on eddy viscosity, Re = 100000

The turbulent mixing increases with decreasing the gap size as shown in Fig. 6. The observed increase in mixing as γ_p decreases may be due to an increase in turbulence intensity in the gap. The predicted variation of turbulent eddy viscosity across the gap at different Re and for each of the pitch-to-diameter ratios are provided in Fig. 7 and Fig. 8.

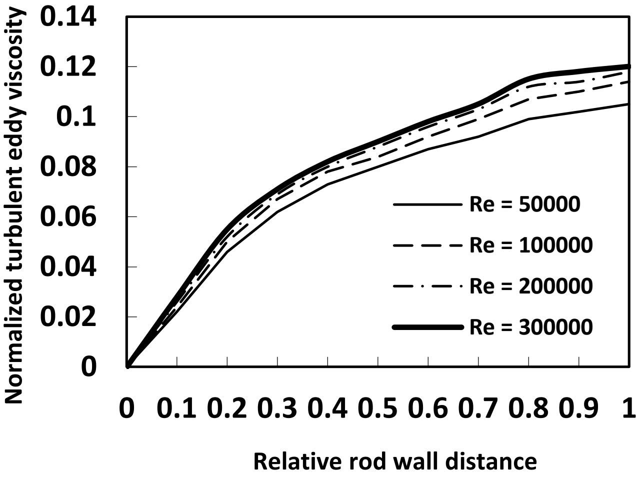

Fig. 7. Normalized eddy viscosity at different Re, γ_p = 1.1

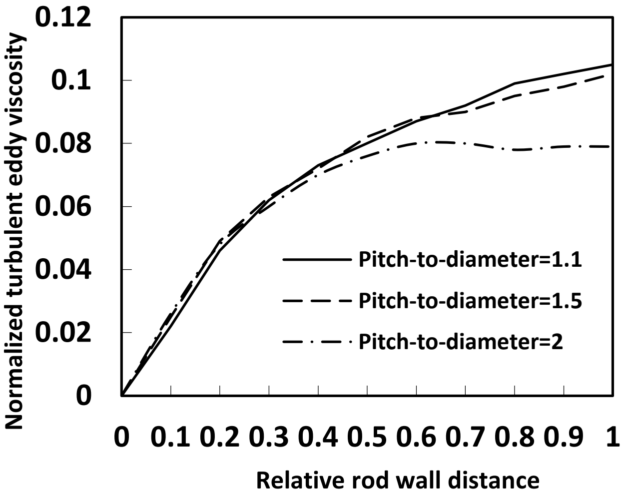

Fig. 8. Normalized eddy viscosity at different pitch-to-diameter ratios, Re = 50000

IV. Conclusions

A mathematical model of steady, three-dimensional turbulent fluid flow was presented in conjunction with a numerical solution procedure based upon finite volumes. Turbulence was modeled using an isotropic, k-ε eddy viscosity model. Existing data of Nusselt number distributions in the axial direction obtained by different authors have been employed to validate the CFD model. Compared with the measured Nu distributions for different authors the present predicted results show good agreement. These comparisons reveal that the k-ε turbulence model can be applied to reasonably simulate the flow and heat transfer behaviors for the rod bundle.

Nomenclature

c_p	heat capacity at constant temperature, J/kg.K
D_h	equilibrium diameter, m
E	modeling constant
k	turbulence kinetic energy, m^2/s^2
P	pressure, N/m^2
Pr_t	turbulent Prandtl number
T	temperature, K
u_i	velocity vector, m/s
u'_i	turbulent fluctuating quantity
U_b	axial bulk velocity, m/s
x_i	coordinate vector, m

Greek letter

β	fluid thermal expansion coefficient, K^{-1}
ε	turbulence dissipation rate, m^2/s^3
μ	dynamic viscosity, kg/m.s
ρ	density, kg/m^3
τ_{ij}	Reynolds stress tensor, Pa
τ_w	wall shear, Pa

References

[1] S. Kim, and B. Chung, “A scale analysis of the turbulent mixing rate for various Prandtl number flow fields in rod bundles,” Nuclear Engineering and Design, vol. 205, pp. 281–294, 2001.

[2] Z. Karoutas, C. Gu, and B. Sholin, “3-D flow analyses for design of nuclear fuel spacer,” In: *Proceedings of the 7th International Meeting on Nuclear Reactor Thermal-hydraulics NURETH-7*, New York, USA, 1995, pp. 3153-3174..

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

[4] M.A. Navarro, and A.C. Santos, “Evaluation of a numeric procedure for flow simulation of a 5×5 PWR rod bundle with a mixing vane spacer,” Prog. Nucl. Energy, 2011.

[5] M.V. Holloway, D.E. Beasley, and M.E. Conner, “Single-phase convective heat transfer in rod bundles,” Nucl. Eng. Des., vol. 238, pp. 848-858, 2008.

[6] X. Wu, and A. C., “Trupp, Experimental study on the unusual turbulence intensity distribution in rod-to-wall gap regions,” Exp. Therm. Fluid Sci., vol. 6 (4), pp. 360-370, 1993.

[7] C.C. Liu, Y. M. Ferng, and C.K. Shih, “CFD evaluation of turbulence models for flow simulation of the fuel rod bundle with a spacer assembly,” Applied Thermal Engineering, vol. 40, pp. 389-396, 2012.

[8] M.A. Navarro, and A. Santos, “Evaluation of a numeric procedure for flow simulation of a 5×5 PWR rod bundle with a mixing vane spacer,” Progress in Nuclear Energy, vol. 53, pp. 1190-1196, 2011.

[9] Y.S. Tseng, Y.M. Ferng, and C.H. Lin, “Investigating flow and heat transfer characteristics in a fuel bundle with split-vane pair grids by CFD methodology,” Annals of Nuclear Energy, vol. 64, pp. 93–99, 2014.

doi:10.15849/icit.2015.0088 © ICIT 2015 (http://icit.zuj.edu.jo/ICIT15)

[10] X.J. Liu, T. Yang, and X. Cheng, "Thermal-hydraulic analysis of flow blockage in a supercritical water-cooled fuel bundle with sub-channel code ," Annals of Nuclear Energy, vol. 59, pp. 194–203, 2013.

[11] S. Chen, Y. Liu, T. Hibiki, M. Ishii, Y. Yoshida, I. Kinoshita, M. Murase, and K. Mishima, "One-dimensional drift-flux model for two-phase flow in pool rod bundle systems," International Journal of Multiphase Flow, vol. 40, pp. 166–177, 2012.

[12] X. Li, and Y. Gao, "Methods of simulating large-scale rod bundle and application to a 17×17 fuel assembly with mixing vane spacer grid," Nuclear Engineering and Design, vol. 267, pp. 10–22, 2014.

[13] A.K. Chauhan, B.V. Prasad, and B.S. Patnaik, "Thermal hydraulics of rod bundles: The effect of eccentricity," Nuclear Engineering and Design, vol. 263, pp. 218–240, 2013.

[14] N.D. Patil, P.K. Das, S. Bhattacharyya, and S.K. Sahu, "An experimental assessment of cooling of a 54-rod bundle by in-bundle injection," Nuclear Engineering and Design, vol. 250, pp. 500–511, 2012.

[15] X. Zhang, and S.D. Yu, "Large eddy simulation of turbulent flow surrounding two simulated CANDU fuel bundles," Nuclear Engineering and Design, vol. 241, pp. 3553–3572, 2011.

[16] H. Ga'bor, "On turbulence models for rod bundle flow computations," Annals of Nuclear Energy, vol. 32, pp. 755–761, 2005.

[17] E. Baglietto, and H. Ninokata, "A turbulence model study for simulating flow inside tight lattice rod bundles," Nuclear Engineering and Design, vol. 235, pp. 773–784, 2005.

[18] C. Lin, C. Yen, and Y. Ferng, "CFD investigating the flow characteristics in a triangular-pitch rod bundle using Reynolds stress turbulence model," Annals of Nuclear Energy, vol. 65, pp. 357–364, 2014.

[19] A.D. Young, "Boundary layers," *BSP Professional Books*, Oxford, 1989.

[20] B.A. Kader, "Temperature and concentration profiles in fully turbulent boundary layers," Int. J. Heat Mass Transfer, vol. 29 (9), pp. 1541-1544, 1981.

[21] M.V. Holloway, T.A. Conover, H.L. McClusky, and D.E. Beasley, "The effect of support grid design on azimuthal variation in heat transfer coefficient for rod bundles," J. Heat Transfer, vol. 127, pp. 598–605, 2005.

[22] A.C. Trupp, and R.S. Azad, "The structure of turbulent flow in triangular array rod bundles," Nuclear Engineering and Design, vol. 32, pp. 47–84, 1975.