Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2023

A Numerical Study on Heat Transfer Characteristics of Various Shapes of Fuel Rods in Nuclear Reactors

Hyerim Woo, Sungjin Yang, Jongrak Choi*
Korea Electronics Technology Institute, 388, Songnae-daero, Bucheon-si, Gyeonggi, 14502, Korea
**Corresponding author: jchoi@keti.re.kr*

1. Introduction

The fuel assembly inside the nuclear reactor consists of the fuel rods and coolant sub-channels for cooling generated heat. Coolant flows in the subchannels for the longitudinal direction of the fuel rod to release heat generated from the fuel rod. To reduce the size of the nuclear reactor core, the power output of fuel rods per volume has to be high, and high-cooling performance is reserved. Such related research is challenging to carry out experimentally, and in most cases, research based on numerical analysis has been conducted previously [1-5].

In the present study, numerical analyses of fuel rods and subchannels in various types of fuel rods were conducted. The heat transfer characteristics were compared considering the various cross-sectional shapes of fuel rods. In addition, the hydraulic energies of the fluid were quantitatively compared by comparing the pressure drop for case studies.

2. Numerical Analysis

2.1. Governing equations

To simulate flows and heat transfer, three governing equations of continuity, mass conservation, and energy conservation are used as follows:

$$\nabla\cdot\vec{u} = 0 \tag{1}$$

$$\nabla\cdot(\rho\vec{u}\vec{u}) = -\nabla p + \nabla\cdot\left(\nabla\vec{u} + \nabla\vec{u}^{T}\right) \tag{2}$$

$$\nabla\cdot\left[\rho\vec{u}\left(h + \frac{|\vec{u}|^2}{2}\right)\right] = \nabla\cdot(k_{\text{eff}}\nabla T) + S_h \tag{3}$$

where $\vec{u}$ is the velocity vector, ρ is the density, p is the pressure, h is the static enthalpy, k_{eff} ($= k + k_t$) is the effective thermal conductivity, T is the temperature, and S_h is the volumetric heat source. No gravitational acceleration was assumed. The incompressible flow was considered, and viscous heating was neglected. The realizable k–ε turbulence model was adopted with enhanced wall treatment to describe the turbulent effect of walls. In the near wall region, all dimensionless wall distance (y+) was applied less than 5 to use enhanced wall treatment. The conjugate heat transfer method between solid and fluid were considered.

2.2. Computational domain and boundary conditions

The computational domain and boundary conditions considered in this study is shown in Fig. 1. Since the fuel rods in the reactor core are regularly arranged in the transverse direction, the unit shape was considered, and symmetry plane boundary conditions were set in the transverse direction. The conformal meshes were set to describe the heat transfer between fluid and solid cell zones. The volume and surface mesh growth ratios were adopted to less than 1.2.

This study was conducted in two steps: flow analysis and heat transfer analysis. First, the flow analysis used periodic boundary conditions for the inlet and outlet to simulate the fuel rod and cooling channel of infinite length. After the flow analysis was performed, all physical parameters were frozen. The energy conservation equation with a volumetric heat source was solved for the conjugate heat transfer analysis. The superheated water's temperature at the inlet was set as the boundary condition.

2.3. Simulation cases

Fig. 2 shows the cross-section of the fuel rod and cooling passage considered in this study. The analysis was conducted considering four types of cross-sectional shapes. A circular case was set as a reference, and elliptical (ratio of the major and minor axis is 1.2), regular hexagonal, and square shapes were considered. The volume of the fuel rod in each case was set constant to keep total heat generation.

To compare the heat transfer performance, all rods have the same volumes because of the convective effects of coolant flow with the same mean velocity in the channel. Circle and ellipse shapes have almost the same heat transfer areas; however, regular hexagon and square cases have larger heat transfer areas of 5.0% and 12.9%, respectively. Therefore, the heat flux was considered to compare case each case.

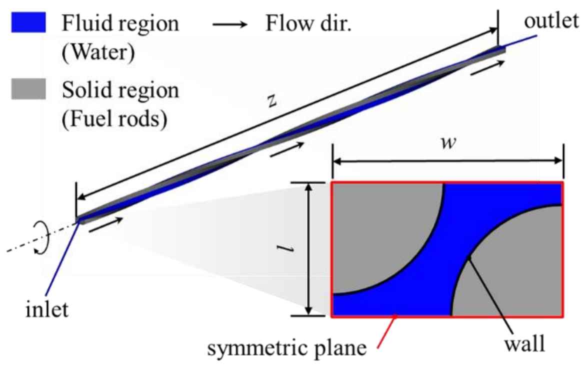

Fig. 1. Computational domain and boundary conditions for a fuel rod heat transfer analysis.

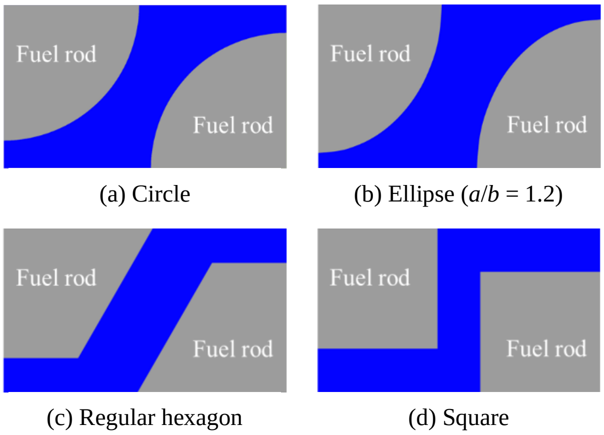

Fig. 2. Cross-sectional shapes for simulation cases.

The present study used Ansys Inc.'s Fluent® R19.1, commercial CFD software, for solving the flows and temperature distribution. The computational meshes used were approximately 490,000 in the form of square and triangular prisms. A case was taken 3 hours for flow analysis and 2 hours for heat transfer analysis with Intel® Xeon® Gold 6348@2.60GHz 56 CPUs with HPC clustering.

3. Simulation Results

Fig. 3 shows the heat fluxes and convection heat transfer coefficients on the fuel rod walls for case studies. The mean heat transfer coefficient was considered to compare quantitatively and can be calculated as follows:

$$h_{\text{avg}} = \frac{\sum_f h_i A_i}{\sum_f A_i} \qquad (4)$$

where h_{avg} is the mean heat transfer coefficient, f is the face, h_i is the local heat transfer coefficient, A_i is the face area on the wall. The local heat transfer coefficients are calculated based on the Prandtl number and near wall velocity. The heat fluxes have a similar level in the cases of circular and ellipse; however, the heat fluxes of a regular hexagon and square cases were predicted to be relatively lower than round wall surfaces. On the other hand, the convection heat transfer coefficient was the smallest in the case of the circular shape, and it became more significant in regular hexagon and square shapes. The circular and ellipse shapes have relatively uniform convection heat transfer coefficients spatially, whereas the regular hexagon and square relatively have larger convection heat transfer coefficients at the corners. The more uniform the velocity distribution of the cooling water near the wall, causes more effective the heat transfer.

Fig. 4 illustrates the pressure drops in the longitudinal length for case studies. The lower the pressure drop, the lower the pump's power loss, causing the pressure for transporting the cooling water to be smaller. The circle and/or ellipse cases with curved surfaces have better

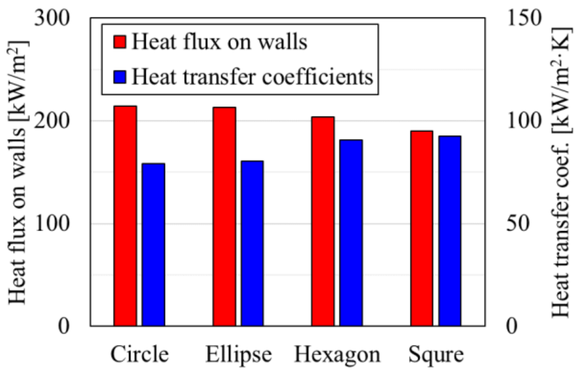

Fig. 3. Heat fluxes and heat transfer coefficients on the walls by cross-sectional fuel rod shapes.

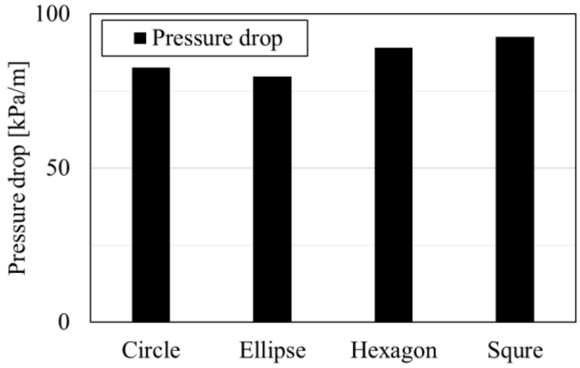

Fig. 4. Comparison of pressure drops by cases.

pressure drop characteristics compared to the angled shapes of the regular hexagon and squares. The best pressure characteristic through the case studies was the elliptical coolant channel. The ellipse showed a smaller pressure drop of approximately 4% compared to the circular shape. According to the present studies' results, the elliptical fuel rod shape is more suitable for the cross-sectional shape of the flow path than the circular shape.

4. Concluding Remarks

In the present study, the heat transfer and pressure drop characteristics based on the cross-sectional shapes of four fuel rods were numerically studied. The heat fluxes and the convection heat transfer coefficients according to the shape had reversed trends. The shape of the ellipse fuel rod was observed as the best pressure drop characteristic in this study. For further study, we plan to study the shape of the cooling channel with better performance through numerical analysis of the coolant passages for various fuel rod shapes.

REFERENCES

[1] L. M. Palomino and M. S. El-Genk, CFD and Thermal-Hydraulics Analyses of Liquid Sodium Heat Transfer in 19-Rod Hexagonal Bundles with Scalloped Walls, International

Journal of Heat and Mass Transfer, Vol.144, Paper No.118637, 2019.
[2] M. S. Song, J. H. Jeong and E. S. Kim, Numerical Investigation on Vortex Behavior in Wire-Wrapped Fuel Assembly for a Sodium Fast Reactor, Nuclear Engineering and Technology, Vol.51, No.3, pp. 665-675.
[3] Y. Wang, M. Wang, H. J
u, M. Zhao, D. Zhang, W. Tian, T. Liu, S. Qiu and G. H. Su, CFD Simulation of Flow and Heat Transfer Characteristics in a 5×5 Fuel Rod Bundles with Spacer Grids of Advanced PWR, Nuclear Engineering and Technology, Vol.52, No.7, pp.1386-1395, 2020.
[4] J.-H. Jeong and M.-S. Song, CFD Investigation of a JAEA7-Pin Fuel Assembly Experiment with Local Blockage for SFR, Nuclear Engineering and Technology, Vol.53, No.10, pp.3207-3216, 2021.
[5] J. Hou, Q. Song, H. Leng, C. Xue, Y. Yuan and Y. Zhou, A Non-Destructive Model for Thermal-Hydraulics of Wire-Wrapped Rod Bundle and Wire-Rod Contact Corner Microscopic Behavior, Progress in Nuclear Energy, Vol.154, Paper No.104469, 2022.