Laminar Thermal-Hydraulic Analysis for a PWR Fuel Assembly

Dong-Hyeog Yoon*, Han-Chul Kim, Kwang-Won Seul
Korea Institute of Nuclear Safety, 62 Gwahak-ro, Yuseong-gu, Daejeon, 305-338
**Corresponding author: dhyoon@kins.re.kr*

1. Introduction

Recently, in relation to severe accidents in the spent fuel pool (SFP), the possibility of ignition due to decay heat in the case of complete loss of coolant of spent fuel pool, has been suggested. After the Fukushima accident interests in this matter are visualized more clearly. When the accident of loss of coolant in the SFP occurs, nuclear fuel becomes oxidized due to decay heat, and ignition may occur because of this.

The thermal-hydraulic characteristic in the fuel assembly is a highly important element in predicting accidents by decay heat. Especially, in spent fuel pool, when an accident of complete loss coolant happens, cooling by natural convection occurs, and prediction of cladding temperature of nuclear fuel rod by this is a very important element in management of severe accidents. When an accident of complete loss of coolant happens, the flow inside the fuel assembly becomes laminar flow with low velocity by the natural convection. In this study, the characteristics of heat transfer and flow with this kind of laminar flow were analyzed. The results about the thermal-hydraulic characteristics of a fuel assembly will be useful for severe accident assessments or a supporting experiment.

2. Numerical Method

To analyze the three-dimensional incompressible steady heat transfer in the fuel assembly, continuity, energy and Navier-Stokes equations are solved using a finite volume solver [1], which employs unstructured grid system. Figure 1 shows the typical geometry of a 17*17 fuel assembly and the grid system, respectively. The dimensions of the assembly components are listed in Table 1. Except the bottom nozzle and top nozzle, all the domains are used as a computational domain (Fig. 2), while, by using the symmetry of fuel assembly, only $1/8^{th}$ of the total shape is used. About 2.7 million nodes are used for the calculation. Two different mass flow rates, 0.0067kg/s and 0.0201kg/s are considered. The Reynolds number for each mass flow rate is 150 and 450, respectively. For boundary conditions, adiabatic conditions are used for the guide tube and the storage cell wall. For the fuel rod, constant heat flux (150.8185 W/m^2) condition is used, considering decay heat of 5 kW. For each symmetry plane, the symmetry condition is used. To take into account the effect of spacer grids and flow mixers existing in the fuel assembly, a porosity

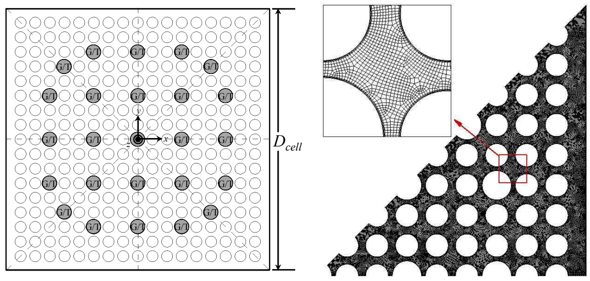

Fig. 1 Flow configuration of a typical 17*17 PWR fuel assembly and grid system.

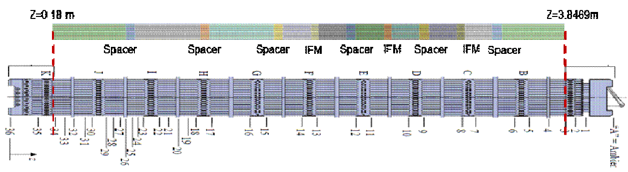

Fig. 2 Computational domain.

Table 1 Dimensions of fuel assembly components.

Number of Pins	264
Pin Diameter (mm)	9.525
Pin Pitch (mm)	12.6
Number of Guide Tubes (G/T)	25
G/T Diameter (mm)	12.2
Size of Storage Cell (mm)	217.5
Flow area, A (m^2)	0.0256
Hydraulic Diameter, D_h(m)	0.0105

scheme is used. For pressure loss in the porous model, the following loss model is used.

$$\Delta p = S_{LAM}\left(\frac{L}{D_h^2}\right)\left(\frac{W_{Avg}\cdot\mu}{2}\right)+\Sigma k\left(\frac{W_{Avg}^2\cdot\rho}{2}\right) \quad (1)$$

3. Results and Discussion

Figure 3 shows the computed pressure drop along the axial direction for Re=150 and Re=450. Figure 4 presents the temperature distribution at a height of about 3.85m. In the area where guide tubes are closely formed, the flow area is narrowed, especially because the diameter of a guide tube is relative larger than that of a fuel rod. Because of this, the flow velocity decreases, further decreasing the heat transfer efficiency as well. Figure 5 shows the contour of axial velocity magnitude. According to figures 4 and 5, it is shown that the velocity is low in the guide tube close-packed regions.

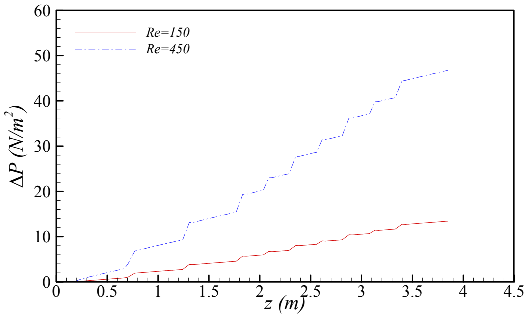

Fig. 3 Assembly pressure drop along the axial height.

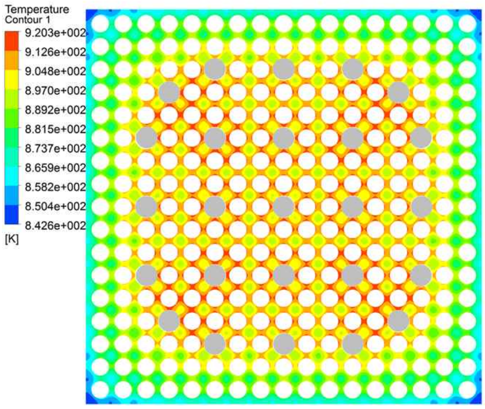

Fig. 4 Temperature field of horizontal cross section, z=3.8489m, Re=150.

Understandably, the area having high velocity shows low temperature distribution and the area having low velocity shows high temperature distribution to the contrary.

Figure 6 presents temperature variation along the axial direction at the locations having the highest cladding temperature and the lowest cladding temperature in the cross section. As shown in Fig. 4, the location having the lowest cladding temperature is rods in the corner of the assembly, and the location having the highest cladding temperature is rods in the guide tube closely-packed region. As a result, the temperature inside the assembly is distributed between the red symbol and the green symbol. As mentioned earlier, since there is no heat transfer enhancement effect made by spacer grids and flow mixers, the temperature increases along the axial direction. Besides, when the flow rate increases up to 3 times on the basis that Re equals to 150, it is observed that the maximum cladding temperature decreases from 920K even to 520K.

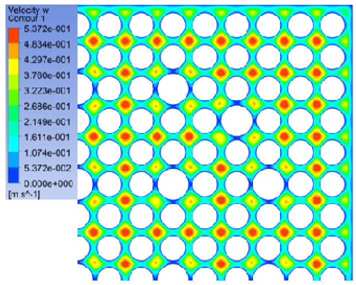

Fig. 5 Axial velocity magnitude of horizontal cross section, z=3.8489m, Re=150

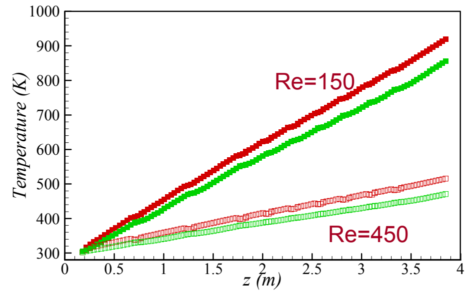

Fig. 6 Axial cladding temperature.

4. Conclusion

In this investigation, CFD analysis was performed on laminar thermal-hydraulic characteristics of a PWR spent fuel assembly as a preliminary research of the fuel ignition. Since the diameter of the guide tube is larger than the fuel rod, flow area becomes narrow in the region where guide tubes are closely formed and accordingly, the cladding temperature is estimated to be high. It is also observed that the maximum cladding temperature becomes 920K at Re of 150 and it decreases to 520K when flow rate increases three times more. The result of this study could be used as data for severe accident code verification.

Acknowledgments

This work was supported by the Long-Term Nuclear Research and Development program of Korea Ministry of Education, Science and Technology.

REFERENCES

[1] CFX-13.0 Solver Theory, Ansys Inc., 2009.