Transactions of the Korean Nuclear Society Spring Meeting
Taebaek, Korea, May 26-27, 2011

CFD Analysis of Hydraulic Characteristics for a PWR Fuel Assembly

Dong-Hyeog Yoon*, Han-Chul Kim, Kwang-Won Seul
Korea Institute of Nuclear Safety, 19 Guseong-dong Yuweong-gu Daejeon 305-338
**Corresponding author: dhyoon@kins.re.kr*

1. Introduction

Hydraulic phenomena in the nuclear fuel assembly have been a long-lasting research subject because it is concerned chiefly with designing the fuel assembly and maintaining the spent fuels. Flow direction in the fuel assembly which is composed of rod bundle is in parallel with the rod. Therefore, pressure loss across the rod bundle and the heat transfer characteristics are very important factors in the design of nuclear fuel. With such importance, there have been many researches conducted on the pressure drop and heat transfer characteristics by rod bundle sub-channel shape change.[1-2]

Recently, the importance of the thermo-hydraulic and ignition behavior of PWR spent fuel assemblies in the spent fuel pool (SFP) during a hypothetic loss-of-coolant-accident has been increasingly recognized. Accordingly, efforts for enhancement of fuel safety and improvement of severe accident codes have been made and attention on thermal hydraulic and ignition phenomena has been increasing.

As a preliminary study for the thermo-hydraulic and ignition behavior regarding severe accidents in the SFP, we performed numerical research on hydraulic characteristics within a fuel assembly. In particular, since flow passing a fuel assembly is affected by the walls of the cask storage cell, three different sized storage cells were used to study such effects. Our results obtained from numerical simulation should be useful in the safety analysis of severe accidents and the validation of severe accident codes.

2. Numerical Method

To analyze the three-dimensional incompressible steady flow in the fuel assembly, continuity and Reynolds-averaged Navier-Stokes equations are solved using a finite volume solver [3], which employs unstructured grid system. Figure 1 shows geometry of the 17*17 fuel assembly and computational domain respectively. The flow is homogeneous in the axial direction when the spacer grids are not considered. The dimensions of the assembly components are listed in Table 1. This study used 1/8th of the total shape as computational domain utilizing the symmetry of the geometry while symmetry boundary condition was used for symmetry planes and the respective wall conditions were used for rod bundle and storage cell. A periodic boundary condition with a constant mass flow rate was specified in the homogeneous axial direction. The shear

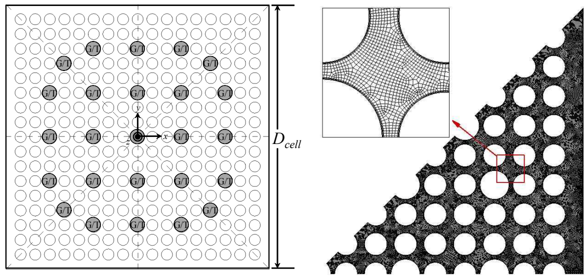


Fig. 1 Flow configuration of a typical 17*17 PWR assembly and computational domain.

Table 1 Dimensions of fuel assembly.

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

Table 2 Flow area and hydraulic diameters for the three different storage cells.

D_{cell} (mm)	217.5	221.8	226.6
Flow area, A (m^2)	0.0256	0.0275	0.0296
D_h (m)	0.0105	0.0113	0.0121

stress transport (SST) model [4] was used for the accurate prediction of turbulent flows. Three different storage cells characterized by the cell size D_{cell} were studied. The cross-sectional areas and hydraulic diameters for each storage cell are presented in Table 2.

3. Results and Discussion

As mentioned in the previous section, the accurate prediction of the pressure loss across the rod bundle is an important subject. For the laminar region, the equation for friction factors regarding the pressure drop can be simply formulated as

$$f = K_l / Re \quad (1)$$

Here, K_l is the loss coefficient determined by the flow geometry and changes depending on the size of the storage cell. Reynolds number is defined as

$$Re = \frac{W_{avg} D_h}{\nu} \quad (2)$$

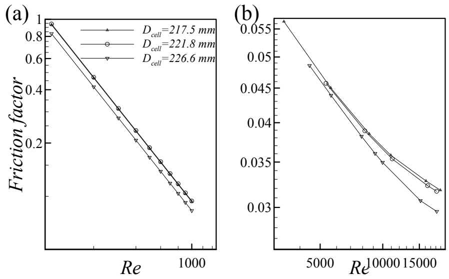
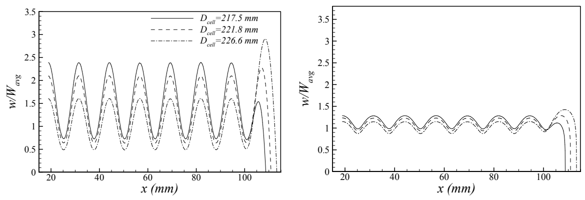
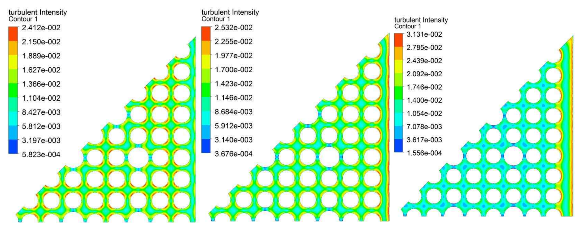


Fig. 2 Variation of the friction factor with *Re*; (a) laminar, (b) turbulent flow.



(a) (b)

Fig. 3 Normalized velocity as a function of position inside the assembly at y=18.9mm; (a) laminar flow, (b) turbulent flow, Re ≈ 18,000.



(a) (b) (c)

Fig. 4 Contour plots of turbulence intensity, Re ≈ 18,000; (a) D_{cell}=217.5 mm, (b) D_{cell}=221.8 mm, (c) D_{cell}=226.6 mm.

Examining laminar friction factor in Fig. 2(a) confirms that as D_{cell} increases, coefficient of loss, K_l, decreases to 94, 93 and 84. As the cell size increases, annular flow effect becomes stronger than rod bundle interior flow, and K_l decreases. As D_{cell} increases even more, K_l is expected to approach asymptotically the value of 57 for a square duct. [5] Friction factor of turbulence flow at Fig. 2(b) also indicates decrease with the increase of D_{cell} as it was the case with laminar flow. Especially with D_{cell}=226.6mm, friction factor shows a steep decrease with the increase of Re.

Fig. 3 shows the velocity distribution normalized with the axial average velocity. The velocity variation from the location of y=18.9mm to the x direction is indicated. In laminar flow of Fig. 3(a), as D_{cell} decreases, it is seen that the average velocity inside the rod bundle significantly increases while annular flow velocity gradually decreases. As D_{cell} increases, in the annular region, the flow area increases with a constant frictional area, yielding increased annular velocity. As a result, the average velocity inside the rod bundle gradually decreases. This means that as previously observed in the variation of friction factor, annular flow becomes gradually dominant as D_{cell} increases.

Fig. 3(b) shows the velocity distribution of turbulent flow at Re≈ 18,000. For turbulent flow, as D_{cell} increases, annular flow velocity can be seen increasing gradually also. In the case of turbulent flow, thickness of boundary layer becomes thinner than laminar flow and the effect of the boundary layer becomes smaller than laminar flow. As a result, unlike laminar flow, velocity change inside the rod bundle does not significantly change.

Fig. 4 shows the turbulence intensity according to variation in D_{cell} at Re≈ 18,000. As mentioned previously, when D_{cell} increases, annular flow becomes relatively stronger; accordingly, turbulence intensity increases relatively in the annular area. As observed in Fig. 4, when D_{cell}=217.5 mm, turbulent flow inside the rod bundle is relatively stronger and as D_{cell} increases, turbulence intensity of the annular area can be seen growing larger.

3. Conclusions

In this investigation, numerical analysis was performed on hydraulic characteristics of spent fuel assembly as a preliminary research step of the fuel ignition that is possible with major SFP accidents. In particular, the effect of the wall of the storage cell on the fuel assembly was observed and changes in pressure loss, turbulent flow characteristic, and velocity profile depending on the cell size were investigated.

When D_{cell} increases, annular flow becomes relatively stronger compared to the flow inside the rod bundle and friction factor decreases due to this effect while turbulence intensity becomes stronger in the annular area. The results of this study can not only be used as reference materials for the validation of severe accident codes but also utilized in nuclear fuel design.

Acknowledgments

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

REFERENCES

[1] K. Rheme, Simple Method of Prediction Friction Factors of Turbulent Flow in Non-Circular Channels, Int. J. Heat Mass Transfer, Vol.16, pp.933-950, 1973.

[2] S. Cheng, and N. E., Todreas, Hydrodynamic models and correlations for bare and wire-wrapped hexagonal rod bundles – bundle friction factors, subchannel friction factors and mixing parameters, Nuclear Engineering and Design, Vol.92, pp.227-251, 1986.

[3] CFX-12.0 Solver Theory, Ansys Inc., 2009.

[4] F. Menter, and T. Esch, Elements of Industrial Heat Transfer Prediction, 16th Brazilian Congress of Mechanical Engineering, 2001.

[5] W. M. Kays, and M.E. Crawford, Convective Heat and Mass Transfer, Second edition, McGraw-Hill, New York, 1980.