Preliminary Evaluation of Pressure Drop Characteristics of Model Grid Plate for Rod Bundle Natural Convection Test Facility

Hyung Min Son[a*], Jonghark Park[a]
[a]*Korea Atomic Energy Research Institute, 989-111 Daedeok Daero, Yuseong Gu, Daejeon, 305-353, Korea*
[*]*Corresponding author: hyungmson@kaeri.re.kr*

1. Introduction

Korea Atomic Energy Research Institute (KAERI) has been developing hybrid-low power research reactor (H-LPRR) for various purposes including education and neutron applications [1]. The heat from the reactor (50 kW) is passively cooled by natural convection of the water flowing between fuel rod bundles (20 in total). The natural convection test facility is being designed at KAERI to understand its thermal-hydraulic characteristics. The facility is designed to utilize real-scale in-core simulating components such as fuel assembly and reflector. The height of the core and constituting components (including core box and grid plate) are maintained, which is crucial in simulating natural convective behavior similar to the reactor core (‘reference’). The facility’s core box and grid plate (‘model’) are reduced in the radial direction so that only 3 by 3 arrangement (assembly-wise) of in-core components are allowed. In this study, a computational fluid dynamic (CFD) analysis has been performed on the model grid plate and its pressure drop characteristics are compared with that of the reference geometry.

2. Methods and Results

2.1 Analysis Methods

Figure 1 shows the model grid plate. The model grid plate is designed to have same height and plate thickness with respect to the original geometry. Compared to the original, the model has only 3 by 3 portion of the grid plate core block (one at the top with multiple holes). The distance from the core block to the side wall has been maintained to that of the original, but the side hole size has been modified to give relative area ratio (sum of hole area/plenum area) close to that of the original (~0.11). In addition, a model without round edges is also prepared and analyzed.

In this analysis, *k*-*ω* based Shear Stress Transport (SST) turbulence model developed by Menter (1994) is solved using ANSYS® CFX code [2]. Figure 2 shows the fluid domain for the CFD analysis. In order to obtain the pressure drop only in the grid plate region, free slip boundaries are applied to the walls before/after the grid plate. For the model and the original grid plate, about 1.2e6 and 3.1e6 elements are allocated respectively to numerically discretize the analysis region. The analysis is performed on the steady-state mode, and the code is run until the solution is converged (RMS of residuals below 5E-5). For reference run cases (original: 0.5 kg/s, model: 0.167 kg/s; temperature at 35°C), non-dimensionalized first node distance from the wall is kept below 10 for both cases [3].

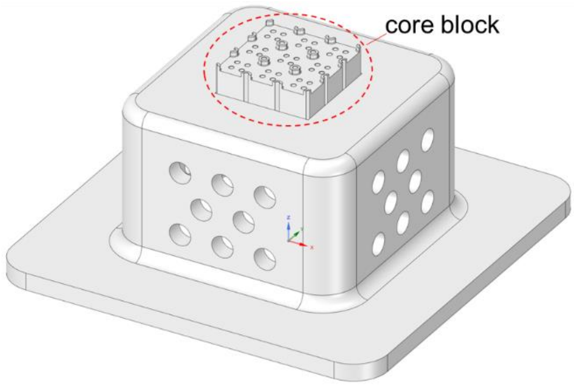

Fig. 1. Model Grid Plate Geometry

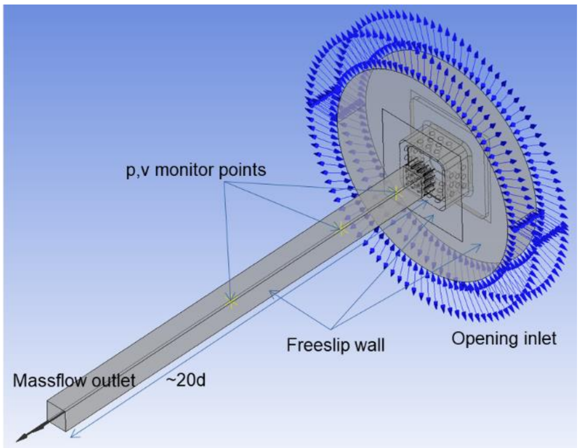

Fig. 2. Model Grid Plate Geometry

2.2 Analysis Results

In this study, two kinds of model (with/without round edges) and the original geometries have been analyzed, and their pressure drops are compared to each other. Figure 3 summarizes distribution of pressure loss coefficient (*K*) as defined by Eq. (1), where the flow area is based on the coolant hole area in the core block region.

$$K = \frac{\Delta P \cdot 2\rho A_C^2}{\dot{m}^2} \quad (1)$$

where, ΔP, ρ, A_c, and means pressure drop [Pa], coolant density [kg/m^3], and cross section area of hole [m^2], respectively.

The analysis data show that for the tested flow rate range (between 50~200% of the reference condition), the relative difference of the loss coefficient between the selected model and the original geometry is less than 5%. The analysis results also illustrate that the difference in the pressure drop characteristics between the two models is insignificant which shows that the effect of rounded/sharp edge is minor. This is because there is almost no secondary flow at the edges or corners, which is due to low velocity (~0.01 m/s) at the plenum region. Considering the majority of the pressure drop occurs in the fuel assembly region at the downstream of the flow, the designed model grid plate is expected to simulate the flow behavior of the original geometry well.

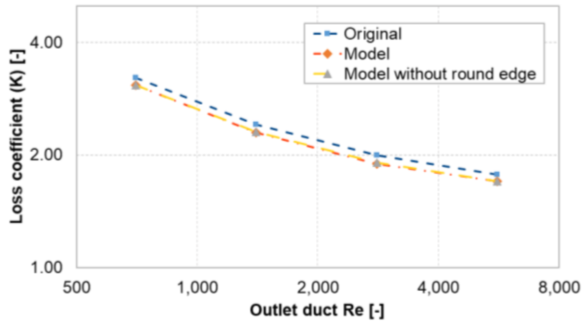

Fig. 3. Loss Coefficient Evaluation Results

3. Summary

In this study, CFD analysis is carried out on model grid plate being designed for natural convection test facility. Size and configuration of side holes are adjusted to give similar pressure drop characteristics to that of original geometry. The analysis showed that the generated models seem to give reasonable flow behavior as intended. In addition, it was shown that there would be no major problem even when designing a test grid plate without round edges, which can contribute to reducing manufacturing cost of the device. These results will be utilized in generating manufacturing design of the actual model grid plate for test facility.

ACKNOWLEDGEMENT

This work was supported as a part of the Technology Development and Enhancement for Supporting the Export of Research Reactor Systems project sponsored by the Ministry of Science and ICT of the Korean government (2020M2D5A1078126).

REFERENCES

[1] K.-O. Kim, G. Roh, B. Lee, "Neutronic Performance of the Hybrid-Low Power Research Reactor for Education and Neutron Applications," Annals of Nuclear Energy, Vol. 158, ID: 108272, 2021.
[2] ANSYS CFX-Solver Theory Guide, Release 19.1, ANSYS, Inc., Canonsburg, PA, 2018.
[3] H.M. Son, J. Park, "Automated Critical Power Limit Estimation for Natural Convection-cooled Research Reactor Core," Annals of Nuclear Energy, Vol. 141, ID: 107332, 2020.