Prediction for the flow distribution and the pressure drop of a plate type fuel assembly

Jonghark Park [a*], Deasung Jo [a], Heetaek Chae [a], Byungchul Lee [a]
[a] *Korea Atomic Energy Research Institute, Daedeokdaero 1045, Yuseong Daejeon, Korea*
[*] *Corresponding author: pjh@kaeri.re.kr*

1. Introduction

A plate type fuel assembly widely used in many research reactors does not allow the coolant to mix with neighboring fuel channels due to the completely separated flow channels. If there is a serious inequality of coolant distribution among channels, it can reduce thermal-hydraulic safety margin, as well as it can cause a deformation of fuel plates by the pressure difference between neighboring channels, thus the flow uniformity in the fuel assembly should be confirmed. When designing a primary cooling system (PCS), the pressure drop through a reactor core is a dominant value to determine the PCS pump size. The major portion of reactor core pressure drop is caused by the fuel assemblies. However it is not easy to get a reasonable estimation of pressure drop due to the geometric complexity of the fuel assembly and the thin gaps between fuel assemblies. The flow rate through the gap is important part to determine the total flow rate of PCS, so it should be estimated as reasonable as possible. It requires complex and difficult jobs to get useful data[1,2].

In this study CFD analysis to predict the flow distribution and the pressure drop were conducted on the plate type fuel assembly, which results would be used to be preliminary data to determine the PCS flow rate and to improve the design of a fuel assembly.

2. Modeling

A plate type fuel assembly has 21 fuel plates arranged with uniform space, 2.35mm, which make 22 coolant channels. Among the channels, the outermost channels of both ends are shared with 1mm flow gaps between neighboring fuel assemblies.

The geometric model of a fuel assembly in the Fig. 1 shows flow passages through a fuel assembly. The symmetric shape of fuel assembly allows the computational model to be established with only a half part. This CFD model simulates the whole flow passage of fuel assembly installed in the core as close to real geometry as possible.

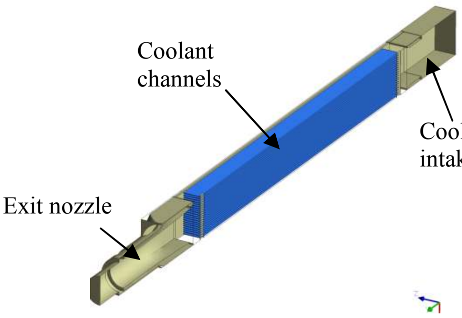


Fig. 1. Geometric model for flow analysis.

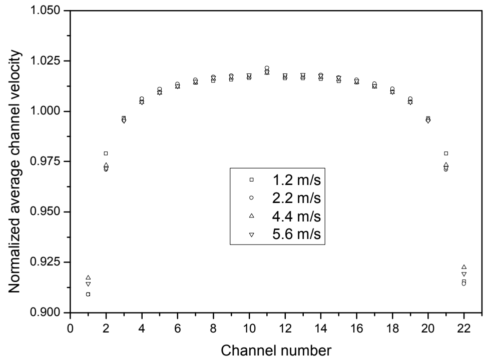
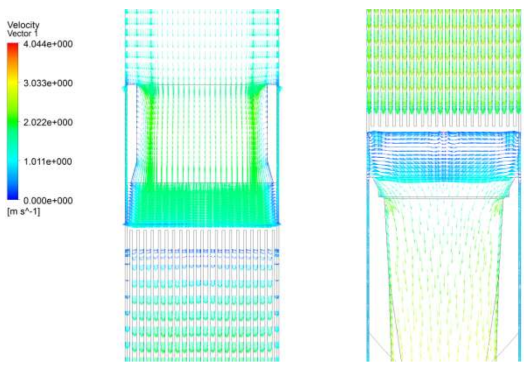


Fig. 2. Normalized average velocity distribution at the plate type fuel assembly.

3. Results and Discussions

From the CFD analysis results, the channel velocity distribution of a fuel assembly is predicted as shown in Fig. 2, which is the distribution of normalized channel average velocity (channel average velocity ratio to fuel average velocity). It seems evenly distributed except the both channels at the outermost. But average velocities at the outermost channels are at least 92% of fuel average velocity. This lower velocity near the outermost channels is caused by not only neighboring wall effect but the separation and recirculation after the end pieces at the intake region of fuel assembly as shown in Fig. 3. These flow phenomena also cause an additional pressure drop. If the end pieces of intake region can be



(a) intake region (b) exit nozzle region

Fig. 3. Velocity vectors at intake region and exit nozzle region of a fuel assembly.

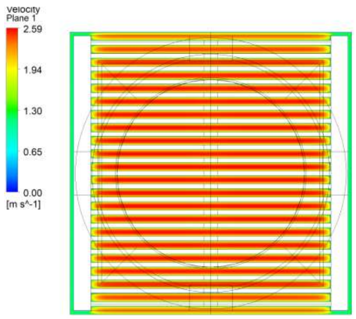


Fig. 4. Coolant velocity distribution at the channels

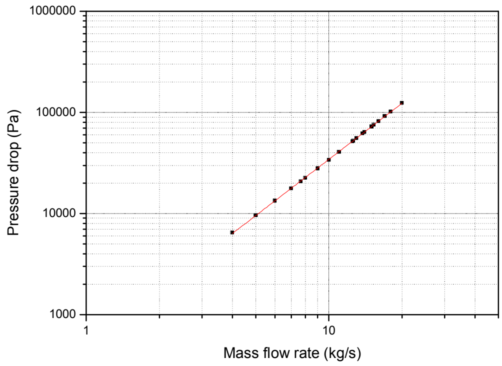


Fig. 5. Relation of pressure drop with mass flow rates.

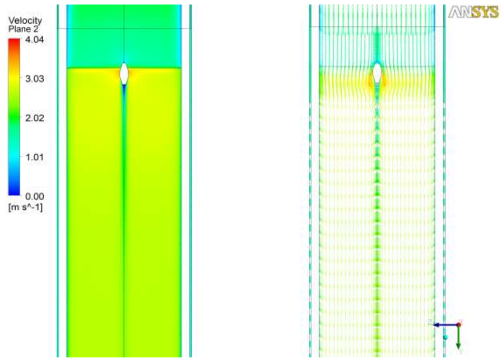
removed, the flow near the inlet area of the outermost channels would be more stable, so that this inequality of velocity distribution seems to be improved to be more even, as well as the pressure drop could be reduced.

Velocity distribution at the cross-sectional plane of a fuel assembly is shown in Fig. 4. In this figure the velocity at the gap area is lower than the fuel coolant channels due to the thinner gap size. The estimated value of average velocity at the gap is about 50% of fuel average velocity.

In order to establish the relation between pressure drop and mass flow rate, CFD predictions are carried out in range of 4 kg/s to 20 kg/s, and their results are plotted as shown in Fig. 5. The relation between flow rate and pressure drop was obtained by a curve fitted equation as follows;

$$\Delta P = 493.174 \ \dot{m}^{1.843} \qquad (1)$$

In order to enhance the resistance against the lateral force due to the serious flow difference between the neighboring channels which may cause collapsing the coolant channel gaps, combs are equipped at the top and bottom leading edges of fuel plate. The comb at channel inlet side affects the flow in the channels. Wake occurring after the comb distorts velocity distribution as shown in Fig. 6. The velocity defect by the wake maintains along the downstream of the comb. However it does not maintain until long after the wake occurs. According to Fig. 7, the most of velocity defect is recovered quickly at about 200 mm away from the leading edge of fuel plate.



(a) velocity contour (b) velocity vector

Fig. 6. Effect of a comb

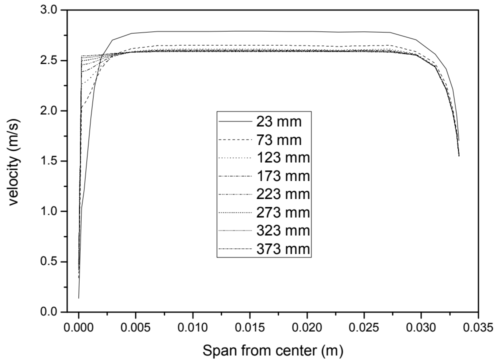


Fig. 7. Recovery of the velocity defect by a comb.

4. Conclusions

CFD analyses were conducted to find out the flow distribution and pressure drop in a plate type fuel assembly. The fuel assembly has a quite even velocity distribution in every coolant channel except the both outermost channels. The average velocity of them is at least 92% of that of fuel assembly. This is caused by the wake flow of the end pieces attached at the intake region of the fuel assembly as well as the velocity decrease near the walls. The inequality of velocity distribution seems to be improved more even by removing the end pieces. The velocity at the gap surrounding the fuel assembly is estimated about 50% of fuel average velocity. The characteristics of pressure drop through the fuel assembly are also found out by CFD calculations for several flow rate conditions. Although the comb at the leading edge of fuel plates disturbs the velocity distribution in a coolant channel, its effect disappears quickly as the flow goes away downstream.

REFERENCES

[1] W.M. Torres, P.E. Umbehaun, D. A. Andrade and J.A.B. Souza, A MTR Fuel Element Flow Distribution Measurement Preliminary Results, Proceedings of 2003 RERTR, pp. 405-410, 2003.

[2] T. Ha, W. J. Garland, Hydraulic study of turbulent flow in MTR-type nuclear fuel assembly, Nuclear Eng. & Design, Vol. 236, pp 975-984, 2006.