Transactions of the Korean Nuclear Society Virtual Spring Meeting
July 9-10, 2020

CFD Simulation of Narrow Channel Pressure Drop Test including Transition Region

Hyung Min Son [a*], Jongharк 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

Many research reactors utilize plate type fuels for increased power density. In order to remove the core heat, the coolant passes through rectangular flow channels between fuel plates. For normal operating conditions, the flow inside the channel usually maintains a turbulent state. During transients such as loss-of-flow (LOF) or loss-of-coolant-accident (LOCA), the flow slows down and undergoes transition state. The transition flow condition is the state between turbulent and laminar states where both flow characteristics co-exist. This makes it hard to accurately predict the flow behavior such as friction factor and convective heat transfer coefficient which are primary information needed for system code simulation. In addition, the most of existing system analysis codes such as RELAP and MARS are developed primarily to simulate power plant system, which usually has rod-type fuel assemblies and operates at relatively high pressure condition when compared with those of research reactor. Considering the above, it is worthwhile to study transition flow characteristics of the rectangular channel geometry. In this study, computational fluid dynamic (CFD) simulation is carried out on the narrow channel and its pressure drop characteristics are compared with experimental results from literature which covers laminar to turbulent flow transition region[1].

2. Methods and Results

In this section, the pressure drop tests performed by Ma et al. (2011) is briefly described along with utilized CFD simulation methods and results.

2.1 Analysis Geometry and Simulation Method

Figure 1 shows cross section view of the channel geometry from the test which exhibits thin channel with semi-circular ends having high aspect ratio (width/thickness= 20). In the isothermal experiments, pressure drop data are obtained from pressure tabs installed along the flow direction. The tests are carried out for Reynolds numbers ranging from 1,090 to 10,200 with measurement accuracy of $\pm$30 Pa. The distance between two farthest pressure tabs is 900 mm which gives length-to-diameter ratio over 200. From the tests the pressure drop data are presented in terms of friction factors over the prescribed Reynolds numbers. In addition, extra flow length exists before and after the test section which gives room for flow development. In this study this entire flow length is analyzed using commercial computational fluid dynamics software ANSYS® Fluent which is widely utilized for engineering applications. Figure 2 shows the discretized analysis geometry where total 10,348,800 nodes are used. And the nondimensional distance of the first node from the wall (y^+) is maintained less than 2 to accurately model the near wall flows. In order to simulate the transition flow region, *k-kl-ω* model is utilized which is one of transition models recently implemented in the code[2,3]. In addition to RANS (Reynolds-Averaged Navier-Stokes) equations, the model solves three additional transport equations to obtain distribution of laminar and turbulent kinetic energy and inverse turbulent time scale to predict the transition flow. According to the literature, the model gives reasonably good predictions for flat plate and airfoil geometries, but its applicability to mini-channel flows such as one studied here is not well-known. Considering the above, the model is applied as-it-is in the simulation to check the prediction capability on the thin rectangular channel geometry. The code is run in steady state mode for different inlet velocity values covering Reynold numbers from 861.9 to 32,320.6. For simplicity, the constant water property at 25°C and 1 bar is used. The iteration is carried out until RMS (Root-Mean-Squared) error of the residuals went below 10^{-5}.

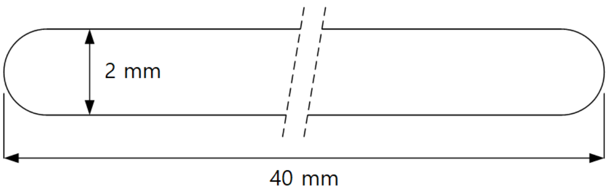

Fig. 1. Cross section view of the test section (not to scale).

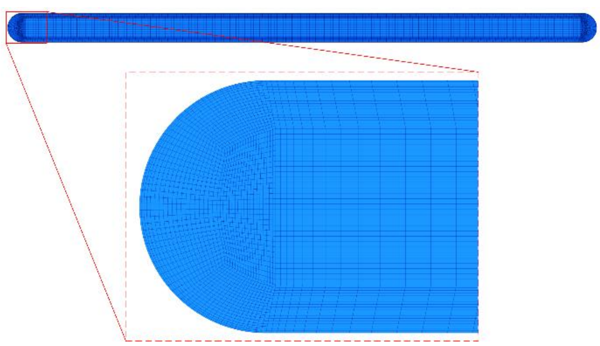

Fig. 2. Discretized analysis geometry.

2.2 Simulation Results

Although the original literature lacks raw measurement data, the best fitting friction factor correlations for laminar and turbulent regions as shown in Eq. (1) and (2) are presented in the same literature[1]. Therefore, as shown in Fig. 3, the simulation results are compared with these correlations. In addition, the friction factors in the turbulent region are compared with values estimated by Bhatti and Shah correlation as shown in Eq. (3)[4]. This correlation is developed by comparing friction factors from Techo et al. (1965) correlation with the experimental data of rectangular duct flows[5].

<u>CNNC correlation:</u>

$$\mathrm{f_{lam}} = \frac{89.3}{Re}, \text{for } 1090 \leq \mathrm{Re} \leq 2500 \qquad (1)$$

$$\mathrm{f_{turb}} = 0.0426 - 2.48 \times 10^{-6} Re^{0.9}, \text{for } 4000 < \mathrm{Re} \leq 10200 \qquad (2)$$

<u>Bhatti and Shah correlation:</u>

$$\mathrm{f_{turb}} = (1.0875 - 0.1125\alpha^{*}) f_c, \text{for } 5000 \leq \mathrm{Re} \leq 10^{7} \qquad (3)$$

where, Re, α^*, and f_c are Reynolds number [-], channel aspect ratio (thickness/width) [-], and friction factor from Techo et al. (1965)[5], respectively.

It is reported from the test that the laminar flow is maintained until Reynolds number less than 2,500 and the flow-pressure drop relationship follows Eq. (1). From Fig.3, it is seen than the simulation accurately predicts the flow behavior in the region. From the test, the flow becomes fully turbulent for Reynolds numbers higher than 4,000. However, the simulation shows that the fully turbulent region begins right after Reynolds number higher than 2,500. This gives transition region ranges much narrower than what is reported in the experiment. In the turbulent region, the simulation greatly overpredicts the pressure drop over the tested Reynolds number ranges. When compared with Bhatti and Shah (1987) correlation, the simulation results approaches the correlation line as the flow becomes higher (Re>10,000). Figure 4 depicts distribution of skin friction coefficient along the flow direction for selected inlet Reynolds numbers. It is observed that for transition Reynolds number region reported in the test (2,500~4,000), the simulation predict transition of the flow from laminar to turbulent near the inlet. The simulation shows that for most cases, the predicted transition point is located upstream of inlet pressure tab location. Because of this, the presence of the transition phenomenon could not be taken into account in friction factor estimation. Considering the above, the poor prediction of the simulation results in the transition region may come from applying wrong inlet boundary condition in terms of geometry and velocity distribution, or the adopted transition model not correctly predicting the transition onset point.

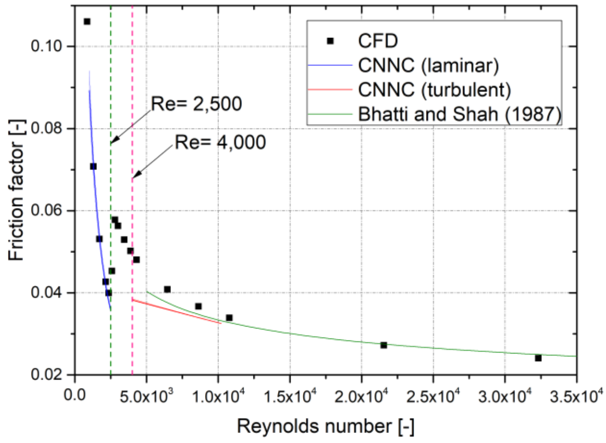

Fig. 3. Comparison between test and simulation results.

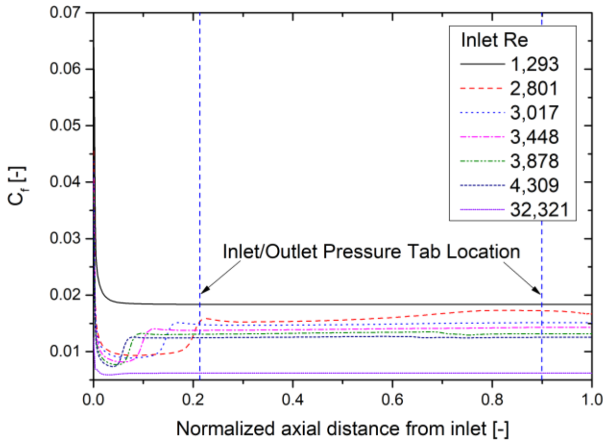

Fig. 4. Distribution of skin friction coefficient.

3. Conclusions

In this study, the computational fluid dynamic simulation is carried out on the narrow rectangular channel geometry with round corners. The *k-kl-ω* transition model was applied to capture the transition effect. The comparison with the experimental data showed that the applied transition model gave reasonable predictions in the laminar and highly turbulent flow regions. However, rather poor prediction capability is seen in the transition region. Current analysis showed possibility of applying incorrect inlet geometry and flow conditions or the transition model not correctly predicting the transition onset. In order to check these argument, the simulation need to be carried out and compared with test results from different geometries, which is left as a future study item.

ACKNOWLEDGEMENT

This work was conducted 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.

REFERENCES

[1] J. Ma, L. Li, Y. Huang, X. Liu, Experimental Studies on Single-phase Flow and Heat Transfer in a Narrow Rectangular Channel, Nuclear Engineering and Design, Vol.241, p.2865, 2011.
[2] ANSYS Fluent Theory Guide, Release 17.2, ANSYS, Inc., Canonsburg, PA, 2016.
[3] D.K. Walters, D. Cokljat, A Three-Equation Eddy-Viscosity Model for Reynolds-Averaged Navier-Stokes Simulations of Transitional Flow, Journal of Fluids Engineering, Vol.130, p.121401-1, 2008.
[4] M.S. Bhatti, R.K. Shah, Turbulent and Transition Flow Convective Heat Transfer in Ducts in: Handbook of Single-Phase Convective Heat Transfer, Ed., S. Kakac, R.K. Shah and W. Aung, John Wiley, NY, 1987.
[5] R. Techo, RR. Tickner, R.E. James, An Accurate Equation for the Computation of the Friction Factor for Smooth Pipes from the Reynolds Number, Journal of Applied Mechanics, Vol.32, p.443, 1965.