Estimation on the Flow Phenomena and the Pressure Loss for the Inlet Part of a Research Reactor Vessel

Kyoungwoo Seo, Jae-Min Oh, Jae-Kwang Seo, Juhyeon Yoon, Doo-Jeong Lee
a KAERI, P.O.Box 105, Yuseong, Daejeon, Korea, 305-353,seokw@kaeri.re.kr

1. Introduction

For a research reactor, a conceptual primary cooling system (PCS) was designed for an adequate cooling to the reactor core. The developed primary cooling circuit consisted of decay tanks, pumps, heat exchangers, vacuum breakers, some isolation and check valves, connection piping, and instruments. The main function of the primary cooling pumps (PCPs) of the PCS was to circulate the reactor coolant through the fuel core and the heat exchangers during a normal operation. The head according to the design flow rate which was determined by the thermal hydraulic design analysis for the core should be estimated to design the PCPs in the fluid system. The pressure loss in the PCS can be calculated by the dimensional analysis of the pipe flow and the head loss coefficient of the components [1], [2]. However, it is insufficient to estimate the pressure loss for 3-dimensional flow phenomena such as the flow path in the reactor with the theoretical dimensional analysis based on experimental data [3].

The purpose of this research is to evaluate the pressure loss of the part of a research reactor vessel. For evaluating the pressure loss, the commercially available CFD computer model, FLUENT, was employed.

First, for validating the application of FLUENT to the pressure loss, a simple case was calculated and compared with the Idelchik empirical correlation. Secondly, several cases for the inlet part of a research reactor vessel were estimated by a FLUENT 3-dimensional calculation.

2. Theoretical Dimensional analysis

As shown in Figure 1, a simple case was estimated to compare the results of the pressure loss by the Idelchik empirical correlations [4] to those by the FLUENT simulation.

. The inlet velocity was 4.6 m/s and the pressure loss by the Idelchik empirical correlation of the flow with a sudden change in velocity and flow area was obtained as follows;

$$\Delta P_{Total} = f_1\left(\frac{h_1}{D_1}\right)\frac{\rho V_1^2}{2} + k\frac{\rho V_1^2}{2} + f_2\left(\frac{h_2}{D_2}\right)\frac{\rho V_2^2}{2} \qquad (1)$$

$$f = \frac{1}{(1.8\ln \mathrm{Re} - 1.64)^2} \qquad (2)$$

$$k = \left(1 - \frac{A_1}{A_2}\right)^2 \qquad (3)$$

The results were compared with the result of 3-D the FLUENT simulation in the same boundary conditions. 2nd order implicit scheme, the standard k-ε model, and the standard wall function [5] were used. As shown in Figure 2, the number of cells was 635,000, which was determined to perform the grid sensitivity tests.

The pressure loss from the inlet to the outlet by the Idelchik empirical correlation was calculated as 5.9 kPa and the result of the FLUENT simulation was 6.4kPa. By the reasonable result which was about 8% higher than the pressure loss introduced by the Idelchik, FLUENT was used for estimating the pressure loss of the part of a research reactor vessel.

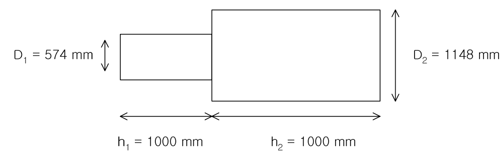

Figure 1. Geometry for the dimensional analysis

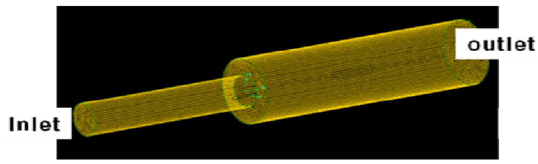

Figure 2. Geometry and cells for FLUENT calculation.

3. FLUENT simulation for research reactor design

For the design of a research reactor vessel, the FLUENT calculation for three cases which are shown in Table 1 and Figure 1 was performed with their geometries and boundary conditions. Flow distribution was obtained from the result calculated for a straight pipe using the same diameter for the inlet condition of a fully developed flow. Total pressures at the inlet, region 1, and the outlet are described Table 2. Pressure loss of Case 2 was 98% higher than that of Case1, especially in region 1 in which the value of case1 was 99.5% higher than that of Case2. However, the flow distribution after the region 1 was more uniform for Case 2. Since the pressure loss and velocity magnitude which can affect the thermal hydraulic phenomena and instability for a core and PCP performance, was relatively high for Case 2, Case 3 which had 6 x 32 holes was analyzed. Pressure and flow distribution are shown Figure 4 and 5. For Case 3, the pressure loss and velocity magnitude was relatively small and the flow distribution was uniform. However, since the holes were designed with a larger size at the top and a smaller size at the bottom, the pressure losses of all the holes layers were slightly different with each other. It may have an effect on the velocity at the holes and the core thermal hydraulic phenomena. Therefore, more studies for several cases will be performed to design a research reactor.

Since there was a significant discrepancy between the 3-dimensional FLUENT prediction of the pressure loss and the value (45kPa for case 1, 757kPa for case 2, and 44kPa for case 3) calculated by using the Idelchik empirical correlations for a part of a research reactor vessel, one should be careful when applying these correlations to a 3-dimensional structure.

Finally, the sensitivity tests for the inlet conditions and the turbulence models were performed and none of the other models showed any significant improvement over the current input data and the standard k- ε model.

Table 1. Description of conditions for cases

	Case 1	Case 2	Case 3
Geometry of flow path region 1	Annular flow path	1 x 8 holes	6 x 32 holes
Flow area of flow path region 1	2.6 m^2	0.06 m^2	0.62 m^2
Quantities of cells	108,000	1,063,000	462,000
Inlet velocity	4.6 m/s	4.6m/s	4.6 m/s
Inlet diameter	0.574 m	0.574 m	0.574 m
Outlet pressure	0 Pa	0 Pa	0 Pa

Table 2. Total pressure for cases

Location	Total pressure [kPa]		
	Case1	Case 2	Case 3
Inlet	9.9	488.8	5.3
Region 1	1.4	290.6	2.5
Outlet	0	0	0

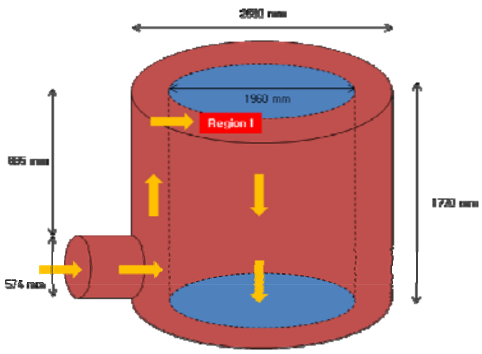

Figure 3. Geometry and flow path description for modeling the part of a research reactor vessel

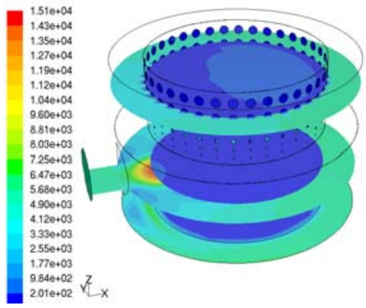

Figure 4. Static Pressure contours for case 3 [Pa]

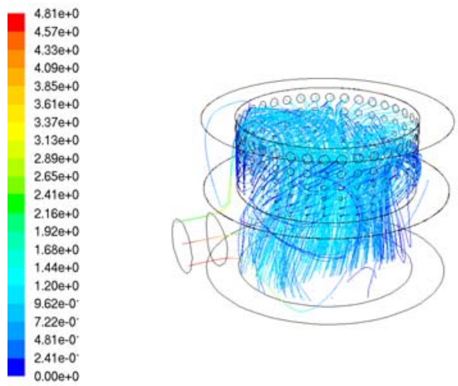

Figure 5. Flow distribution for case 3 [m/s]

4. Conclusion

Our research work sought to analyze and understand the flow phenomena and pressure loss near a research reactor vessel inlet and to be applied to the design of a research reactor.

FLUENT was validated to estimate the pressure loss by comparing the results using a dimensional analysis and a head loss coefficient to those with a FLUENT simulation for a sudden change in the velocity and flow area.

FLUENT was used to predict the pressure loss and flow phenomena for three cases with their geometries and boundary conditions. In Case 3, the pressure loss and velocity magnitude were relatively small, and the flow distribution was uniform.

REFERENCES

[1] CRANE Co., “Fluid of fluids through valves, fittings and pipes”, 1988

[2] Park, J.P., and Jeong, J.H., “The pressure loss coefficient evaluation based on CFD analysis for elementary geometries”, NUTHOS, Seoul, Korea, 2008

[3] Munson, B.R., Young, D.F., and Okiishi T.H., “ Fundamentals of fluid mechanics”, 1990

[4] Idelchik, I.E., “Handbook of Hydraulic Resistance”, 3rd Edition, 1994

[5] Launder, B. E., Spalding, D. B., “*Lectures in Mathematical Models of Turbulence*.”, Academic Press, London, England, 1972.