

DESIGN OF A DECAY TANK FOR A POOL TYPE RESEARCH REACTOR WITH A CFD MODEL

Kyoungwoo Seo, Young-Chul Park, Daeyoung Chi and Juhyeon Yoon

Korea Atomic Energy Research Institute, Daejeon, Republic of Korea

Abstract

A conceptual primary cooling system (PCS) was designed for adequate cooling of the core of a research reactor. The primary coolant after passing through the reactor core contains many kinds of radio-nuclides. A decay tank provides a delayed transit time to ensure that the N-16 activity decreases enough before the coolant leaves the decay tank's shielding room. The size of the decay tank should be enlarged to provide sufficient transit time. However, there was a limitation: to minimize the tank size, it should be designed with an internal baffle, which affects the pressure loss in the system and net positive suction head (NPSH) of the PCS pump. Therefore, the decay tank should be optimized for size and the internal baffle. A vertical type decay tank was chosen to optimize the geometrical arrangement of PCS and the vertical internal baffle was installed to minimize the number of internal structures. The preliminary geometry of the tank and the internal baffle were determined to satisfy the required delayed transit time by calculating the maximum velocity and the flow path length of the circular and the annular sections of the tank. The commercially available CFD model, FLUENT, which solves the Navier-Stokes and turbulent models, was used to specifically design the decay tank with the preliminarily calculated geometry and the related flow rate. Several turbulence models, standard k- ϵ model, renormalization group (RNG) model, and realizable k- ϵ model, were conducted to isolate the root cause of these differences. By comparing the results of the velocity profile and the characteristics of each model, a detailed design study was simulated using the realizable k- ϵ model. A user-defined scalar equation was solved to estimate the delayed transit time. The size and the internal baffle that satisfy the required transit time were determined based on the CFD results.

Introduction

A conceptual primary cooling system (PCS) was designed for adequate core cooling of a research reactor, which was developed for various neutron applications. The PCS circulates demineralized water in order to remove the heat generated in the fuel, irradiation device, and reflectors. The PCS is connected to an open type pool and the reactor. The developed PCS consists of flap valves, siphon breakers, decay tanks, pumps, heat exchangers, some isolation and check valves, connection piping, and various instruments [1]. After passing through the reactor core, the primary coolant contains many kinds of radio-nuclides. In particular, N-16 is the most important nuclide for the PCS design due to its hard and high strength γ -rays. A decay tank, which provides a delayed transit time is used to sufficiently decay N-16 radioactivity before the coolant leaves the decay tank's shielding room. For the design of the decay tank, this study was performed using a preliminary and a CFD calculation. First, the decay tank size and the internal baffle location were determined with the help of the preliminary calculations. Second, the delayed transit time and the pressure loss were evaluated for a detailed design using a CFD computer model.

1. Preliminary design of a decay tank

In order to design a decay tank, the tank size should be determined based on the required flow transit time inside the tank and the inlet flow rate. Since there is a limitation in the tank size due to the research reactor capital cost, the size should be optimally minimized. Most decay tanks are installed upstream of the system pump. When an open-pool type research reactor is designed, the NPSH of the PCS pump affects the location of the pump. If the liquid pressure at a given location is reduced to the vapor pressure of the liquid, and if the required NPSH is then lower than the available NPSH, as shown in equation (1), a cavitation causing a loss in pump efficiency and structural damage can occur at the pump [2].

$$NPSH_A < NPSH_R \quad (1)$$

$$NPSH_A = \frac{p_{atm}}{\rho g} + H_s - \frac{p_v}{\rho g} - \sum h_L \quad (2)$$

To increase NPSH_A, the installation level of the pump or head loss of the pump upstream should decrease. Considering a decay tank design, the pressure loss induced in the decay tank located at the upstream of the system pump should be minimized.

First, a preliminary calculation of the decay tank size determined simply by the flow rate of 180kg/s, and the required transit time of 70seconds, was performed. FLUENT, which solves the Navier-Stokes and turbulent models was used in the 2-D axisymmetric steady state simulation.

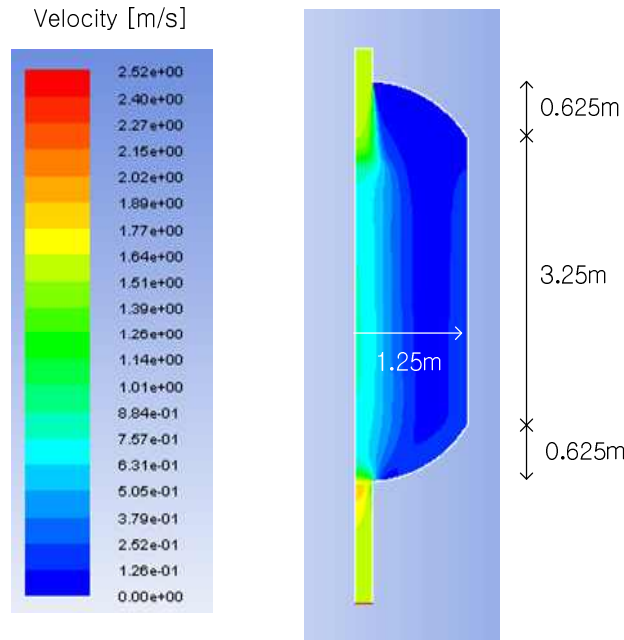


Figure 1 Preliminary calculation for the decay tank design

As shown in Figure 1, a flow stagnation zone occurred because the flow was not evenly distributed throughout the tank. Therefore, the minimum transit time of the flow in the tank is calculated as 6.3 seconds even though the decay tank was designed to be sufficiently large in order to meet the requirements.

A new decay tank equipped with internal baffles was developed to satisfy the required transit time. Since the internal design inside the tank affected the pressure loss of the system and the NPSH of the PCS pump, a vertical type baffle was chosen to minimize the flow direction inversion.

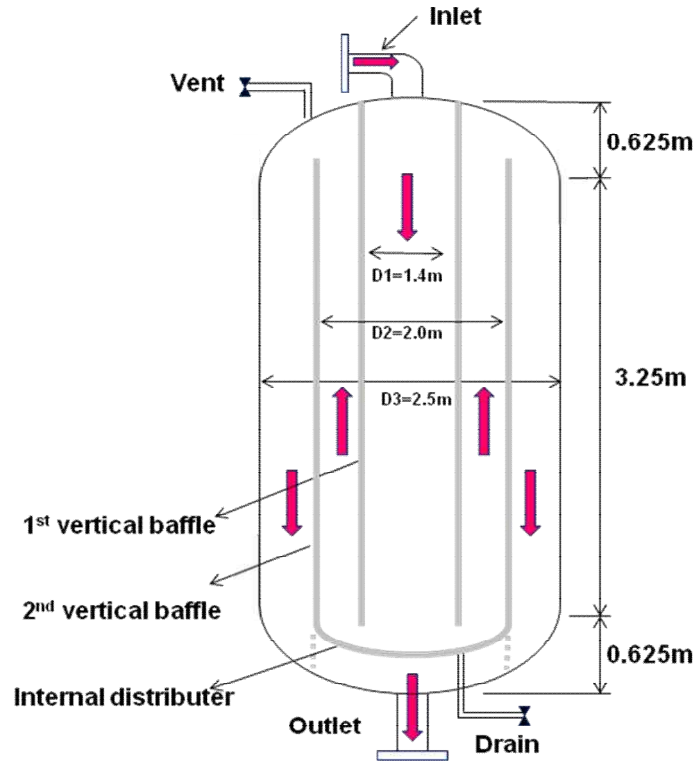


Figure 2 Decay tank with the vertical internal baffle (Geometry1)

As shown in Figure 2, after the coolant enters the inlet of the decay tank, it flows down the inside of the 1st internal baffle. The flow is changed at the internal distributor and passes the region between the 1st and 2nd internal vertical baffle. The coolant inverses the flow direction again and comes down to the tank outlet.

Since the minimum transit time of the flow through the decay tank should be satisfied with the required value, it was necessary to obtain the maximum velocity at the each region from the flow rate. For the circular pipe [3] and the annular type geometry [4], the maximum velocities are as follows:

$$v_{z,\max,circular} = 1.25\bar{v}_z \quad (3)$$

$$v_{z,\max,annular} = 1.14\bar{v}_z \quad (4)$$

where, $\bar{v}_z$ is the mean velocity obtained from the flow rate. For the decay tank design, Equation (3) was applied for all regions using a conservative approach.

As shown in Table 1, the preliminary geometry of the vertical internal baffle and the external decay tank was determined to fulfil the expected transit time by employing the maximum velocity calculated from Equation (3).

Table 1 The calculation used for the design of the internal baffle decay tank

D1 [m]	1.4
D2 [m]	2
D3 [m]	2.5
L1 [m]	0.625
L2 [m]	3.25
L3 [m]	0.625
Total flow rate [kg/s]	180
$\bar{v}_1$ [m/s]	0.12
$\bar{v}_2$ [m/s]	0.11
$\bar{v}_3$ [m/s]	0.1
$v_{1,\max}$ [m/s]	0.15
$v_{2,\max}$ [m/s]	0.14
$v_{3,\max}$ [m/s]	0.13
Total transit time [sec]	74

2. Decay tank design with numerical calculation

2.1 Delayed transit time calculation

A commercially available CFD computer model, FLUENT was used to evaluate the flow characteristics, the pressure loss, and the delayed transit time inside the designed decay tank geometry. The k- ϵ turbulent models in FLUENT, which are the simplest turbulence models that provide a reasonably accurate prediction for a wide range of fluid flows, were employed. Since the

k-ε model is only valid for a turbulent core flow, the standard wall function, which is a semi-empirical function [5], is used near the wall. This function connects the numerical solution variable at the near-wall cells and the corresponding quantities at the wall. For calculating geometry 1, three turbulent models, standard k-ε model, RNG model, and realizable k-ε model, were employed as shown in Table 2. The working fluid is water, and the inlet condition was given as 180 kg/s. The meshes are composed of a quadrilateral type. A grid size sensitivity analysis was performed, and it was determined that an acceptable convergence was reached for the number of cells of at least 138,000. The distance from the wall at the wall-adjacent cells, which the wall function was used for, must be determined by considering the range over which the law of the wall is valid. The first cell was around $y^+=30\sim80$ from the walls for these calculations. The simulation was solved using geometry 1, a steady state, and two-dimensional and axi-symmetric assumption.

Table 2 Turbulent model sensitivity analyses

Cases	Geometry	Turbulent model	Near wall treatment	Number of cells	Inlet condition
Case 1	Geometry1 (Figure 2)	Standard k-ε model	Standard wall function	138,000	180kg/s
Case 2		RNG			
Case 3		Realizable k-ε model			

To obtain the flow transit time from the simulation results, a user-defined scalar equation in FLUENT was solved additionally as follows:

$$\frac{\partial \rho \phi_k}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i} \right) = S_{\phi_k} \quad (5)$$

The first term could be removed with the steady state condition. Since the diffusivity was neglected for this study, the diffusion term could be canceled. If density was assumed to be constant and the source term was given as the same value with the density, the scalar value can be indicated as the residence time at various positions.

The simulation results are shown in Figure 3 and Figure 4. The results was decided to be converged by estimating the scaled residual ($< 10^{-5}$) of the solved equations including the user defined scalar equations and the convergence history of mass flow rate for the inlet and the outlet surface.

FLUENT has several models for turbulent viscosity, namely the standard k-ε model, RNG model, realizable k-ε model, and a Reynolds stress model for 2-D problems. The advantage of the Boussinesq hypothesis approach is the relatively low computational costs associated with the computation of the turbulent viscosity even though the Reynolds stress model is superior for anisotropy under turbulent situations. In many cases, since models based on the Boussinesq hypothesis perform sufficiently well, a sensitivity analysis was performed on the turbulence models for the standard k-ε model, RNG model, and realizable k-ε model. The standard k-ε model is typically used for a simple geometry such as a pipe flow at a high Re without a swirl or rotation. In order to compensate the shortfalls of the standard k-ε model, the RNG model [6] and the realizable k-ε model [7] have been developed. As shown in Figure 3, the velocity profile showed a similar

trend and none of the other models showed any significant improvement over the realizable k- ϵ model, which provided its superior performance for flows involving rotation, separation, and recirculation. By comparing the results of the velocity profile and the characteristic of each model, a detailed design study was simulated using the realizable k- ϵ model.

The minimum transient times at the outlet of the decay tank were calculated as 72 seconds for the standard k- ϵ model, 66 seconds for the RNG model, and 67 seconds for the realizable k- ϵ model.

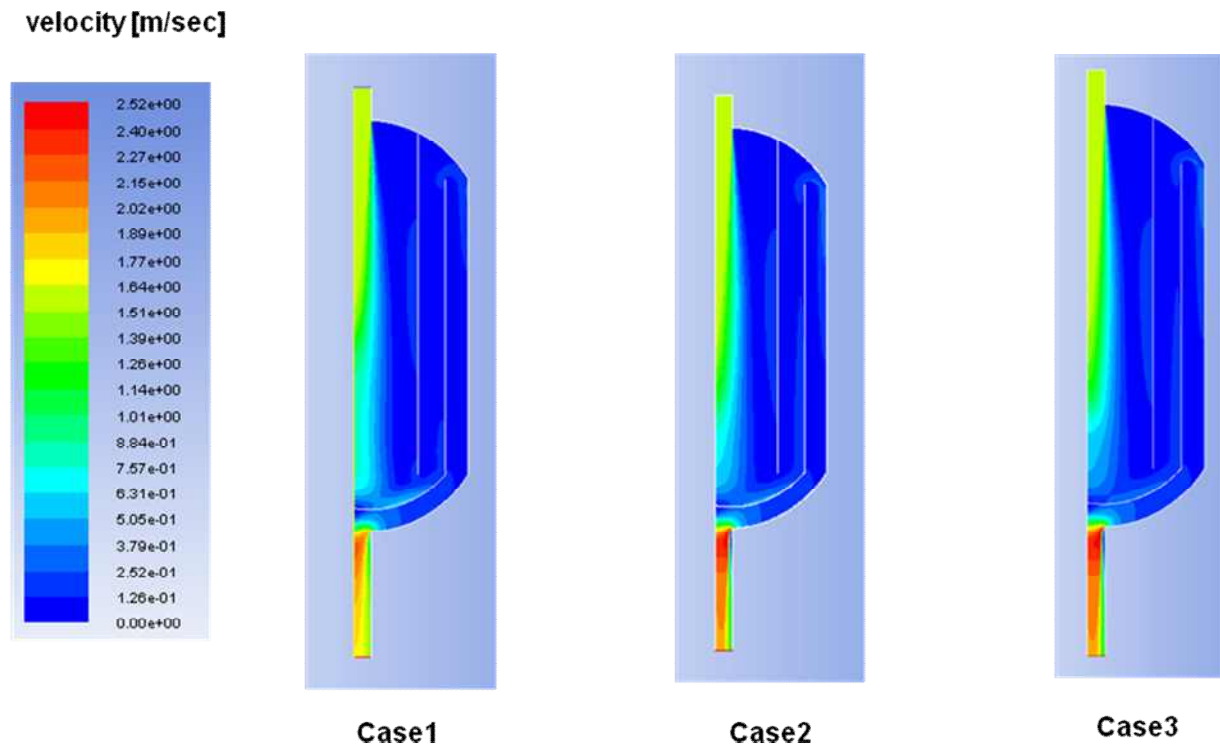


Figure 3 FLUENT simulation results (the velocity profile, geometry 1)

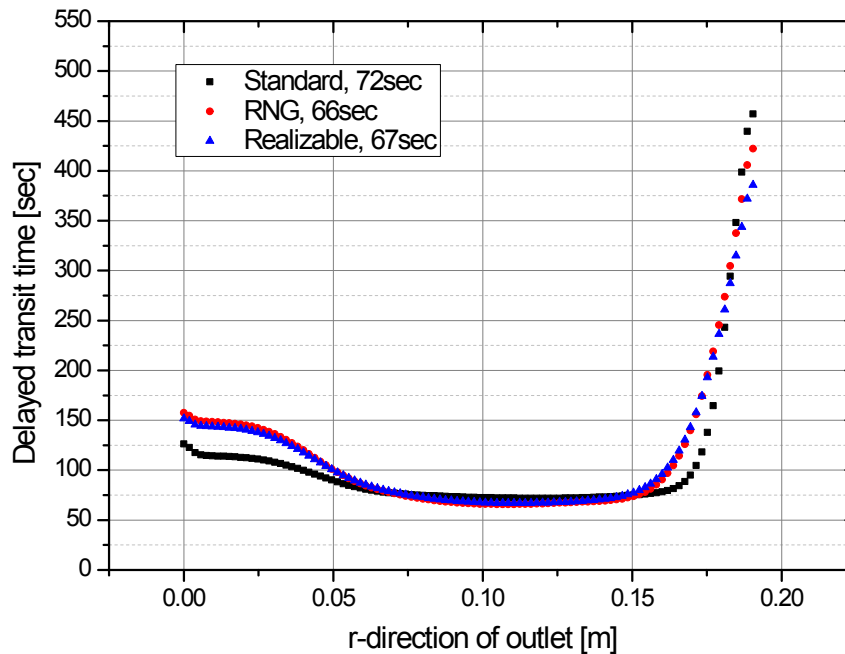


Figure 4 FLUENT simulation results (delayed transit time, geometry1)

Figure 4 shows that the calculated transit time for case 3 employing the realizable $k-\epsilon$ turbulent model did not satisfy the required time of 70 seconds. This was caused by fluid velocity discharged from the inlet in a non-uniform distribution concentrating in the central region. Since a flow stagnation zone in the 1st vertical internal baffle occurred, the expected transit time in Table 1 was not obtained.

To remove the flow stagnation zone caused by a jet-like flow discharged from the narrow inlet, the geometry of the decay tank was improved, as shown in Figure 5. Geometry 2 changed the 1st internal baffle to be identical to the inlet size as the stagnation zone occurred in the region next to the narrow center of the tank, which is connected to the inlet. The inlet flow distributor near the inlet for geometry 3 was installed in order to intentionally distribute the inlet flow. For geometry 4, similar flow area sizes of the flow paths with the inlet inside the tank were established by the four internal vertical baffles so that the flow path length increased and the stagnation zone was expected to be removed.

Figures 6 and 7 show the velocity profiles and the computed transit times for each geometry. As seen in Figure 6, the stagnation zone still occurred at the internal distributor region between the 1st and 2nd baffles even though the velocity was distributed uniformly in the center region of geometry 2. According to the simulation results, the minimum transit time at the outlet was calculated to be 66 seconds for geometry 2, which did not meet the required value. As shown in the velocity profile for geometry 3 of Figure 6, the velocity near the inlet was distributed uniformly by the inlet distributor, and therefore the fluid flow could be sufficiently retained in the entire decay tank. The minimum flow transit time was simulated to be 87 seconds. Geometry 4, shown in Figure 5, had a narrow and long flow path, and was specifically designed to ensure a smooth flow transition by keeping the same diameter from the inlet pipe through the central region of the tank to the

internal baffle regions. The velocity distribution was formed uniformly according to the flow path, and the flow transit time was calculated as 80 seconds. However, even if the calculated transit time remained within the required time, the number of vertical internal baffles increased, and this may complicate the tank manufacturing process.

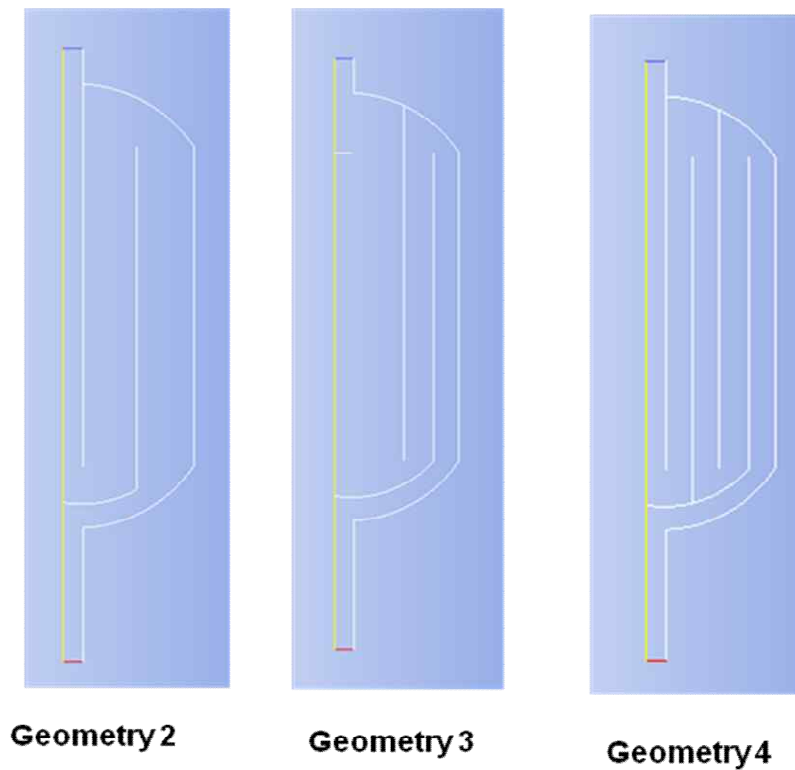


Figure 5 The simulated geometries

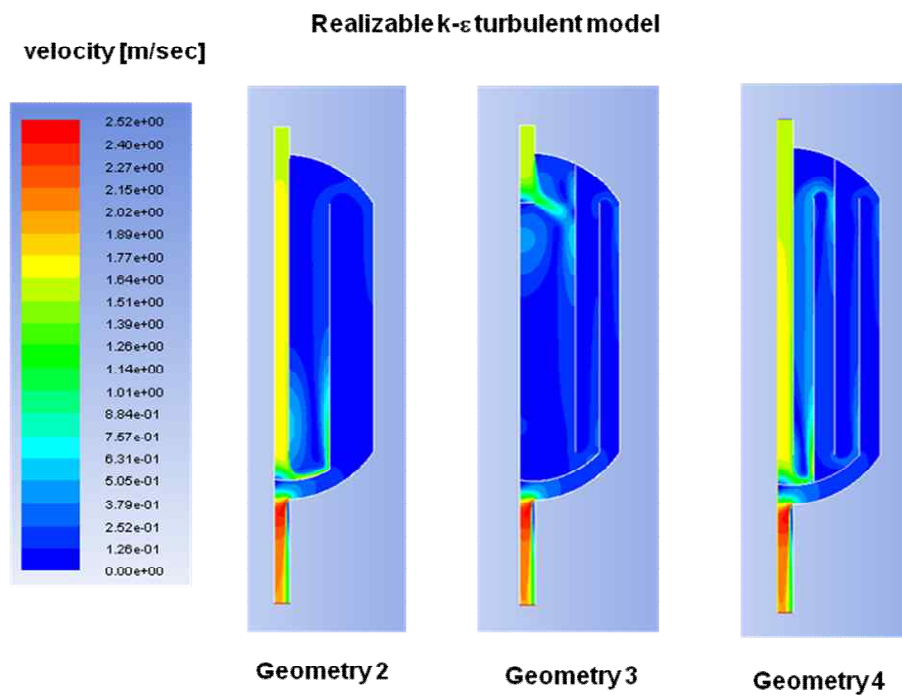


Figure 6 FLUENT simulation results (velocity distribution, geometries 2, 3, and 4)

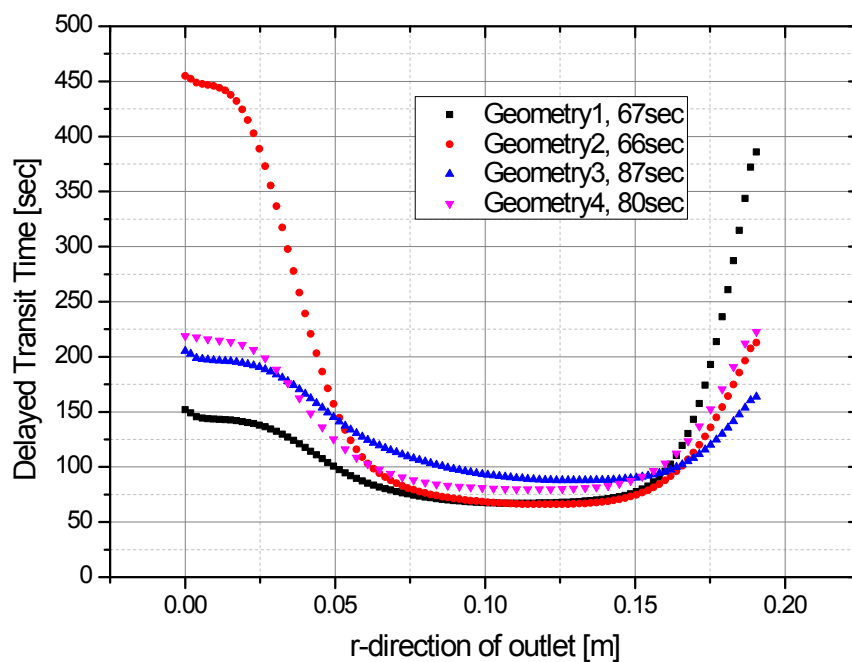


Figure 7 FLUENT simulation results with the various geometries (delayed transit time)

2.2 Pressure loss calculation

The decay tank design should also consider the pressure loss in the tank since the decay tank is installed upstream of the system pump, as described in section 1.

The pressure loss in the decay tank can be calculated based on a theoretical dimensional analysis and a head loss coefficient of the tank section. Idelchik empirical correlations [8] were used for validating the application of FLUENT in the pressure loss simulation. For a pressure loss calculation of geometry 1, the geometry could be sectioned as follows:

$$\Delta P_{Total} = f_1 \left(\frac{h_1}{D_1} \right) \frac{\rho V_1^2}{2} + k_1 \frac{\rho V_1^2}{2} + f_2 \left(\frac{h_2}{D_2} \right) \frac{\rho V_2^2}{2} \quad (6)$$

$$+ k_2 \frac{\rho V_1^2}{2} + f_3 \left(\frac{h_2}{D_3} \right) \frac{\rho V_2^2}{2} + k_3 \frac{\rho V_1^2}{2}$$

where “k” is the loss coefficient for the flow geometry variation [9], and “f” is the friction factor for the straight flow geometry. Subscripts 1, 2, and 3 denote region 1, which is the zone between the center and the 1st vertical internal baffle; region 2, which is the zone between the 1st and the 2nd baffle; and region 3, which is the zone between the 2nd and the tank wall.

The pressure loss of geometry 1 was estimated to be about 1kPa based on the Idelchik empirical correlation. The complicated structure was simplified when the loss coefficient was calculated using this correlation. The FLUENT simulation using the realizable k-ε turbulent model was performed in the same geometry and evaluated as 1.9kPa. Based on its reasonable and conservative results, FLUENT was used for estimating the pressure loss of the other decay tank geometries. The simulation calculations for the geometries are shown in Table 3. As shown in the table, the pressure loss for geometry 1 was lower than the other geometries, but it was estimated that the differences of the pressure loss values were negligible. In addition, since the pressure losses for all the geometries were satisfied with the limitation due to the pump NPSH, the vertical internal baffle type was evaluated as the good decay tank design.

Table 3 The FLUENT simulation (pressure loss for the geometries)

Cases	Characteristics	Pressure loss
Geometry 1	Base cases (2 vertical internal baffles)	1.9 kPa
Geometry 2	1 st vertical baffle modification	2.1 kPa
Geometry 3	Inlet distributor	2.1 kPa
Geometry 4	4 vertical internal baffles	2.0 kPa

3. Conclusion

A decay tank for an open pool type research reactor was designed using preliminary and numerical calculations. The decay tank size and the internal baffle location were determined based on the preliminary calculation using the maximum velocity obtained from the inlet flow rate and the required flow transit time. For a detailed design of the decay tank, the delayed transit time and the pressure loss were compared among several prospective geometries. The calculations were performed using a commercially available CFD code, FLUENT. In order to compute the flow transit time, a scalar equation in FLUENT was additionally solved. In addition, the turbulent models and cell number sensitivity analyses were performed, and the realizable k- ϵ turbulent model was selected to design the decay tank. Using the simulation results, the preliminary geometry was improved, and the delayed transit time and the pressure loss, which were compared with empirical correlations, were estimated to satisfy the required values.

4. References

- [1] K.W. Seo et al., “Estimation on the pressure loss of the conceptual primary cooling system and design of the primary cooling pump for a research reactor”, Transactions of the Korean Nuclear Society Autumn meeting, Korea, 2009
- [2] F.M. White, Fluid mechanics, 4th Edition, MacGraw-Hill Book company, Inc.
- [3] R. B. Bird et al., Transport phenomena, 2nd Edition, John Wiley & Sons, Inc.
- [4] J.G. Knudsen and D. L. Katz, Fluid dynamics, McGraw-Hill Book Company, Inc., 1958
- [5] Launder, B. E., Spalding, D. B., “The Numerical Computation of Turbulent Flows.”, Computer Methods in Applied Mechanics and Engineering, 3:269-289, 1974.
- [6] V. Yakhot and S.A. Orszag, “Renormalization group analysis of turbulence : I. Basic Theory”, Journal of Scientific Computing, 1(1): 1-51, 1986.
- [7] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu., “A New Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - k- ϵ Model Development and Validation.” *Computers Fluids*, 24(3):227-238, 1995.
- [8] I.E. Idelchik, Handbook of Hydraulic Resistance, 2nd Edition, Hemisphere Publishing Corp., (1986).
- [9] CRANE Co., Fluid of fluids through valves, fittings and pipes, 1988