Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 18-19, 2017

A Study of Residence Time Distributions Using Various Numerical Methods

Minkyu Jung [a*], Kyoungwoo Seo [a], Daeyoung Chi [a], Seonghoon Kim [a], and Cheol Park[a]
[a]*Korea Atomic Energy Research Institute, Daejeon 34057, Korea*
[*]*Corresponding author: minkyujung@kaeri.re.kr*

1. Introduction

This paper presents the numerical studies of several parts of decay tank from a viewpoint of residence time. The main objective of decay tank installation is to decrease the N-16 activity by increasing the residence time. To assess the design adequacy of the tank, the residence time should be predicted [1-3].

Figure 1 shows the residence time distributions according to different types of flow. For the plug flow, the response of pulse injection is defined as a function of flow velocity and length. All tracer is exited to outlet boundary with same residence time. In completely mixed flow, tracer of the flow is monitored in a manner of Fig. 1 (B). And, most real flow passing the pipes, orifices, and tanks has similar time distribution with Fig. 1 (C). The dead zone inside the device is represented to long tail of time distribution.

To obtain the accurate time distribution, several numerical methods are compared by simulating the simple geometric cases.

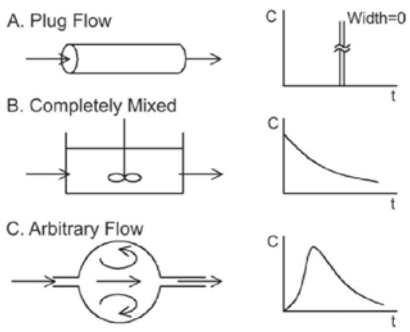


Fig. 1 Concentration versus time plots in response to different types of flow [4]

2. Residence Time Calculation Methods

A commercial computational fluid dynamics (CFD) software, ANSYS Fluent is utilized for the calculation. The fluid motion is modeled by incompressible Reynolds-averaged Navier-Stokes equations. The numerical domain is discretized using cell-centered finite volume method.

2.1 Streamline method

Streamlines are lines drawn in the flow field so that at a given time t_0 they are tangent to the direction of flow at every point in the flow field. These lines are computed by integrating the ordinary differential equations below in pseudo-time t [5].

$$\frac{d}{d\tau}\vec{x}(\tau) = \vec{v}(\vec{x}(\tau), t_0)$$

After the flow is sufficiently converged, the streamlines are generated without additional computation by using the flow variables. Then, the travelling time for each streamline is calculated using the velocity magnitude of control volumes.

2.2 Particle tracking method

Like the streamline method, the residence time is calculated by using the flow path particles. This method is different from the streamline method that the unsteady flow manner is considered. The residence time could be computed by using the multiphase model, Lagrangian Discrete Phase Model (DPM). The trajectory of a discrete phase particle could be predicted by integrating the force balance on the particle like below the equation [1, 5].

$$\frac{d}{dt}u_p = F_D(u - u_p) + \frac{g_x(\rho_p - \rho)}{\rho_p} + F_x$$

After the flow field is converged, the DPM field also needs several iterations. The particle tracking method tracks the motion of individual particles by computing the force balancing equation, thus this approach is clear and physically simple.

2.3 User Defined Scalar transport method

To predict the residence time by the experimental approach, tracer is injected for a very short time at the inlet boundary, then the concentration of tracer is measured at the outlet boundary. From the measured concentration, the residence time distribution could be obtained by dividing the injected amount at the inlet boundary. User defined scalar (UDS) transport method could be utilized to estimate the residence time by the tracer. For an arbitrary scalar ϕ_k could be solved by the UDS transport equations [5]

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

where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term for the scalar equations. To obtain the time distribution, the transient flow simulation should be conducted.

2.4 Multi-component model method

Multi-component method is also adopted to obtain the residence time distribution (RTD) using the tracer. The flow quantities of tracer are defined as those of background fluid, then the tracer could be tracked without flow disturbance. In the present study, the mixture model of multiphase flow is utilized for multi-component model. The flow simulations are conducted with the same way of UDS method.

3. Result and Discussion

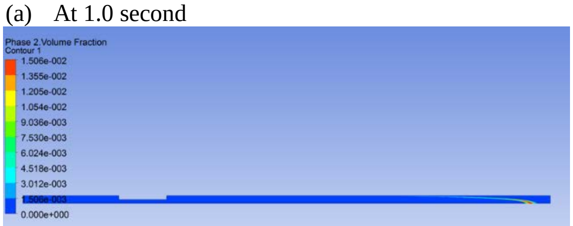
3.1 Orifice flow (coarse mesh)

The flow simulation is conducted to the orifice flow case using axisymmetric calculation method. The contraction ratio at throat is 2:1 in consideration of the perforated plate design. Uniformly distributed inflow is injected at inlet boundary with 1m/s velocity and 10% turbulence intensity. The steady state simulation is conducted until the solution converges to 10^{-8} residual. The residence time is computed by streamline method and particle tracking method from the steady state solution.

Then, the transient flow simulation is applied to obtain the RTD using the UDS and mixture model. The tracer is injected for 0.01 seconds with 10% concentration. The snapshots of tracer concentration contour of the UDS are shown in Fig. 2. It is shown that the tracer is propagated to outlet boundary on an axis. It is also observed that the maximum concentration value is decreased and the concentration spreads to the longitudinal direction due to the viscous effects.

In Fig. 3, the comparison of the RTD is made by four numerical methods. It is observed that the RTDs have arbitrary flow concentration pattern, previously explained in Fig. 1. It is shown that overall RTDs are similar except for peak magnitude and irregular oscillation after 5 seconds. Particle methods, streamline and DPM, and tracer methods, mixture and UDS, have very similar time distributions, respectively. Unphysical oscillation is represented after 4.5 seconds, because the low level of concentration could not be resolved by lack of particle number. And, by same reason, higher peak percentages are shown for particle methods.

(a) At 1.0 second



(b) At 4.0 seconds

Fig. 2 Tracer concentration contour for 2:1 orifice flow using mixture model

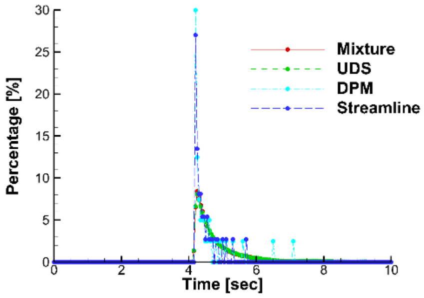


Fig. 3 Residence time distribution at outlet boundary for 2:1 orifice flow using coarse mesh

Scalar (particle) ratio and weighted averaged-time for an expansion case are presented in Table I. Mass (particle) ratio is compared to identify the tracer conservation or check the escaped particle ratio at outlet boundary. All methods show the scalar (particle) conservation over than 98% except for the streamline method. It may be inferred that several particles near the wall are terminated by impingement on the wall for the streamline method. And, weighted averaged-time of tracer methods show longer than particle methods by resolution of long tail dead zone.

Table I: Scalar (particle) ratio and weighted averaged-time for 2:1 orifice flow using coarse mesh

	Scalar (particle) ratio	Weighted averaged time
Mixture	98.7%	4.836 sec
UDS	98.4%	4.850 sec
DPM	100% (40/40)	4.573 sec
Streamline	92.5% (37/40)	4.465 sec

3.2 Orifice flow (fine mesh)

Fine mesh simulation for the orifice flow is conducted to check the grid independent test and enhance the resolution of long tail dead zone using particle methods. The mesh size is decreased to $\Delta x_{min}/h = \Delta y_{min}/h = 0.00625$ using two level refinement. Steady flow features, such as velocity and pressure contours, and unsteady concentration snapshots are almost same with coarse mesh results. Thus, in this chapter, the time distributions are discussed for fine meshes.

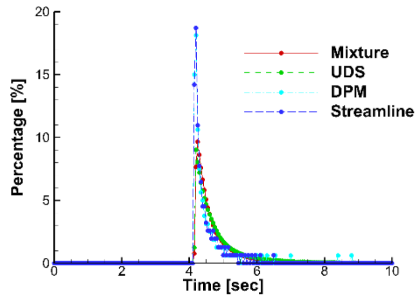


Fig. 4 Residence time distribution at outlet boundary for 2:1 orifice flow using fine mesh

The comparison of the RTD is shown using fine meshes in Fig. 4. Overall time distributions are similar with those of coarse mesh previously explained in Fig. 3. It is shown that RTDs of particle methods are getting smoother by adopting fine meshes and peak percentages are also decreased. For tracer methods, peak values of concentration are slightly increased by diminishing the numerical dissipation.

In Table II, Scalar (particle) ratio and weighted averaged-time for an expansion case are compared for fine mesh simulations by numerical methods. Mixture model infringes the scalar conservation although the immiscible model is utilized to prevent phase change. Other three methods shows scalar (particle) ratio of over 95%. The weighted averaged-time for tracer methods is shortened about 0.05 seconds while that of particle methods has little change.

Table II: Scalar (particle) ratio and weighted averaged-time for 2:1 orifice flow using fine mesh

	Scalar (particle) ratio	Weighted averaged time
Mixture	150.4%	4.744 sec
UDS	98.8%	4.809 sec
DPM	100% (160/160)	4.564 sec
Streamline	96.9% (155/160)	4.491 sec

4. Conclusions

This paper describes a comparison of residence time distributions using various numerical methods. To assess the design adequacy of decay tank, an accurate residence time calculation is required. In the present study, two particle methods, DPM and streamline, and two tracer methods, mixture and UDS, are utilized and compared by simulating the validation case. DPM shows higher exit particle ratio and accurate time estimation than streamline method. UDS methods shows better performance than mixture model for the scalar conservation. Therefore, it is concluded that the DPM methods is more proper at the stage of initial design. And at detailed design phase, UDS method could be utilized to assess the accurate tank performance. This study will be applied to design the decay tank and internal flow devices.

ACKNOWLEDMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIP) (No. NRF-2012M2C1A1026909)

REFERENCES

[1] N. G. Jeong, K. W. Seo, D. Y. Chi, and J. H. Yoon, Estimation of flow residence time in a decay tank for a pool type research reactor using CFD, Nuclear Engineering and Design, Vol. 255, pp. 162-168, 2013.

[2] N. G. Jeong, K. H. Roh, S. H. Kim, and J. H. Yoon, Design evaluation of decay tank for a pool-type research reactor from the required minimum flow residence time point of view, Journal of Nuclear Science and Technology, Vol. 51, pp. 1064-1072, 2014.

[3] G. Verma, S. Sengupta, V. Veluiri, S. Mammen, and S. Bhattacharya, Proc. 42nd National Conference on Fluid Mechanics and Fluid Power (FMFP 2015).

[4] D. Egarr, M. Faram, T. O'Doherty, D. Phipps, and N. Syred, Proc. Institution of Mechanical Engineering, Part E, 53-67, 2005.

[5] ANSYS FLUENT theory guide, ANSYS Inc., 2016.