Transactions of the Korean Nuclear Society Spring Meeting
Jeju, Korea, May 7-8, 2026

Eulerian Approach to Preliminary CFD Analysis of an Aerosol Experimental Facility Using AeroSolved

Keun-sang Choi*, Jongtae Kim, Jaehoon Jung
Intelligent Accident Mitigation Research Division, Korea Atomic Energy Research Institute,
Daejeon, 34057, Rep. of Korea
**Corresponding author: ksc307@kaeri.re.kr*

Keywords : AeroSolved, radioactive material filtration system, sectional method, inertial drift

1. Introduction

A multi-institutional consortium was established to facilitate the development of a radioactive material filtration system, and as a member of this consortium, KAERI is currently conducting performance evaluation experiments. Among the target substances to be removed by this system are fission products including iodine compounds and radioactive aerosols, and the decontamination performance of the aerosol filter used in the system is being evaluated using solid aerosols.

And as a method to design performance evaluation experiments and further analyze the field application of the system, computational fluid dynamics (CFD) was chosen. In KAERI, development and improvement based on open-source CFD software package OpenFOAM are underway. As a test for development of CFD code simulating integral phenomena while the system is working, a preliminary computational analysis using AeroSolved[1,2] was conducted to evaluate the behavior of aerosols inside the test section without the aerosol filter applied. AeroSolved is a third-party open-source Eulerian aerosol analysis package based on OpenFOAM and developed by Philip Morris. And it was modified in KAERI to be compatible with OpenFOAM-v2312, which was the latest version at the time of modification.

2. Methods and Results

AeroSolved is an Eulerian aerosol simulation package based on the reactingFoam solver. It includes an Eulerian solver capable of simulating a single-phase carrier gas alongside components comprising gaseous and particulate (liquid or solid) phases—essentially simulating aerosol species—along with the necessary libraries for analyzing aerosol behavior. The solver solves the continuity, momentum, energy, and species transport equations for each component, including separate equations for each phase of the aerosol components.

For modeling the aerosol size distribution, AeroSolved offers both the log-normal moment method and the sectional method; the sectional method was employed in this study. While the log-normal moment method allows for simple simulation with low computational overhead when the target particles follow a log-normal distribution, the fixed sectional method provides the number rate [#/s] and particle velocity for representative sizes within each section of the distribution.

Sub-models governing aerosol behavior in the code include condensation, nucleation, coalescence, diffusion, Brownian motion, and inertial drift.

Since the subject is a solid aerosol and there are no additional aerosol generation mechanisms in the test section of the experimental facility, condensation and nucleation were not considered. The drift velocity model solves motions of aerosol through a partial differential equation (PDE) considering relative velocity of aerosol particles against the carrier gas and relaxation time. AeroSolved provides two drift velocity models for aerosols that Stokes drag can be applied—fullStokes and Manninen[3,4]. Specifically, the fullStokes model, an iterative solver for drift velocity PDE running independently of the PIMPLE loop, was used to simulate the aerosol drift flux for this preliminary analysis.

A computational mesh was generated based on the existing experimental apparatus, with the outlet extended by 3 meters to investigate the vertical behavior of aerosols. And six face sets were defined to monitor the number of aerosols passing per second and named as sampling 1 – 6(Fig. 2).

The sectional method is employed to model the size distribution of aerosol particles in this study. The sectional interval of particle size and particle size distribution are calculated based on particle mass, which is considered as spherical. The particle size distribution used in this simulation is described in Table I and Fig. 1, the term df/dlogd represents the number fraction of particles of corresponding size (d).

The particle size distribution employed in the simulation was based on the distribution used in the experiment, which has a count mean diameter (CMD) of 3.85 μm. As the experimental particles exhibited a log-normal size distribution, logarithmic interpolation was adopted to define the sectional boundaries.

Table I: Particle size distribution

	d (m)	df/dlogd		d (m)	df/dlogd
g1	1.84E-06	0.042769	g8	4.43E-06	1.266013
g2	2.09E-06	0.126843	g9	5.02E-06	0.919304
g3	2.37E-06	0.307651	g10	5.69E-06	0.545992
g4	2.68E-06	0.610271	g11	6.44E-06	0.265219
g5	3.04E-06	0.990105	g12	7.31E-06	0.105364
g6	3.44E-06	1.313859	g13	8.28E-06	0.034231
g7	3.90E-06	1.426046			

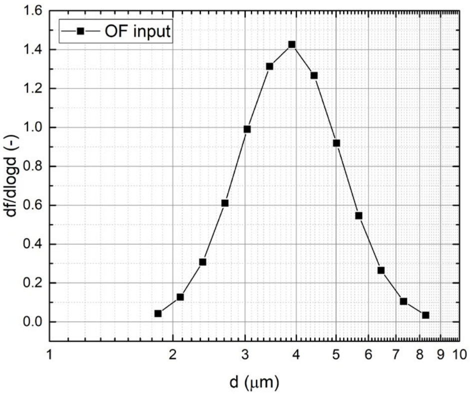
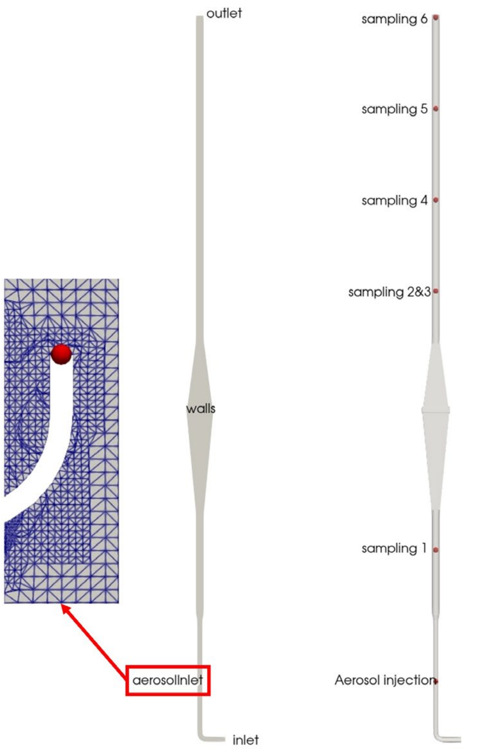


Fig. 1. Particle size distribution



<Patches> <Sampling points>

Fig. 2. Computational domain and aerosol monitoring points

CFD simulations were conducted for 100 s, and the particle number flux passing the upstream and downstream locations of the filter housing are shown in Figs. 3 to 6. Sampling point 1 in Fig. 2 corresponds to the upstream of the filter housing, whereas sampling points 2 and 3 are located downstream.

The particle number flux at the upstream location (Fig. 2) was calculated to be on the order of 10^6. The particle number flux at the downstream (Fig. 4, 5) was also calculated to be on the order of 10^6, their value were calculated larger than the value of upstream. The particle number density field near the sampling position 1 (upstream) showed higher concentration in the vicinity of the wall as shown in Fig.7. In contrast, the particle number density field in the downstream region resembled the velocity profile. In the current study, sampling flow rate was excluded to simplify the model. Since particle number flux is a vector quantity, the sampling surface must be re-modeled to ensure that only positive values are counted, accounting for the sampling flow rate and piping geometry rather than simply measuring the flux passing through a face set. Future work will incorporate these factors into refined geometry and mesh generation, followed by more detailed simulations.

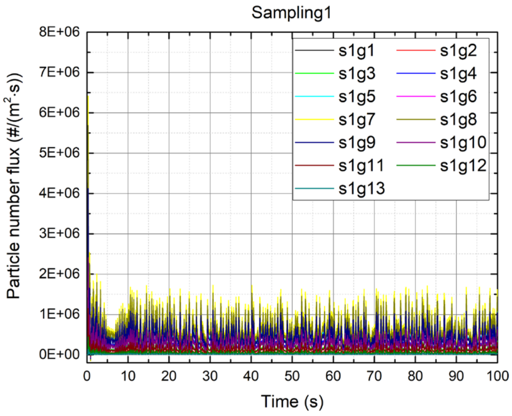


Fig. 3. Particle number flux at the sampling face below the filter housing

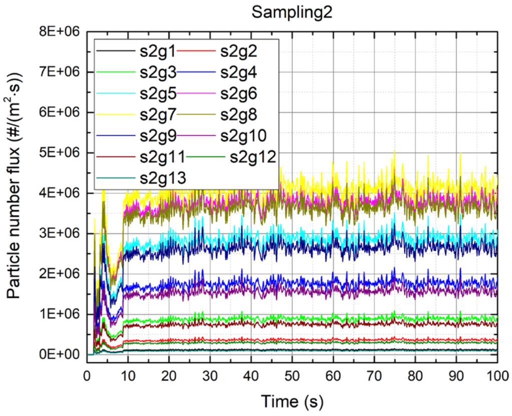


Fig. 4. Particle number flux at the sampling face above the filter housing (Sampling 2)

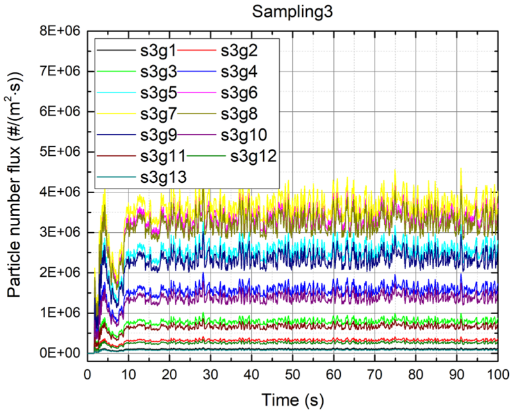


Fig. 5. Particle number flux at the sampling face above the filter housing (Sampling 3)

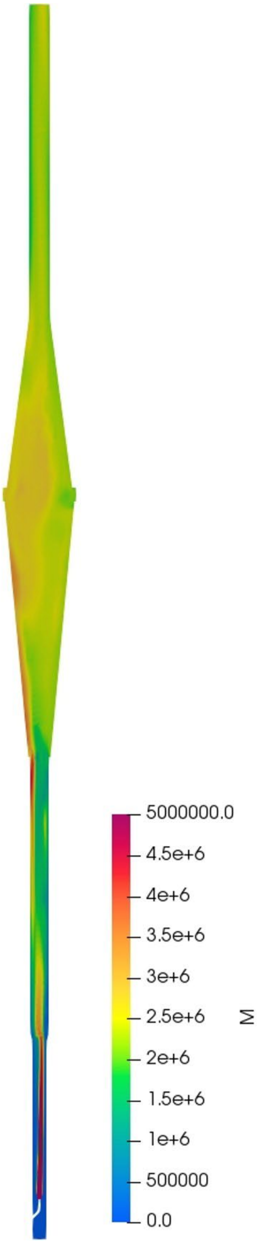


Fig. 6. Particle number density field near the filter housing

3. Conclusions

This preliminary study established a computational framework using an updated version of the AeroSolved package to simulate the behavior of solid aerosols in a test section without filter installation. The sectional method was adopted to represent the particle motion in each discrete size section. To model inertial drift, the fullStokes model was applied, which iteratively solves the partial differential equations governing aerosol motion. Although the initial results posed challenges in evaluating the mass balance across the filter, the accuracy of the simulation will be improved in future studies through refined geometric modeling and updated boundary conditions.

ACKNOWLEDGEMENT

This research was supported by the National Research Foundation of Korea (NRF) funded by the Ministry of Science and ICT.(RS-2022-00144236)

REFERENCES

[1] F. Lucci, E. Frederix, A. K. Kuczaj, AeroSolved: Computational fluid dynamics modeling of multispecies aerosol flows with sectional and moment methods, Journal of Aerosol Science, Vol.159, p. 105854, 2022.
[2] E. Frederix, Eulerian modeling of aerosol dynamics, Ph.D. Thesis, University of Twente, 2016.
[3] M. Manninen, V. Taivassalo, S. Kallio, On the mixture model for multiphase flow, VTT Publications 288, Technical Research Center of Finland, 1996.
[4] W.C. Hinds, Aerosol Technology, John Wiley & Sons, Inc., 2nd edition, 1999.