Investigative the effect of feed flow using CFD-DSMC method in a gas centrifuge

Ghazanfari V.[1*], Shadman M.M.[1], Mansourzade F.[1]

[1]Nuclear Fuel Cycle Research School, Nuclear Science and Technology Research Institute, AEOI, P.O.Box:11365-8486, Tehran, Iran

[*] *Email: vghazanfary@aeoi.org.ir*

Abstract
In the rotor of the centrifuge, the feed inlet is positioned in the rarefied area. The continuum hypothesis is not valid in the rarefied area; therefore, it desires to be analyzed by probabilistic methods like Direct Simulation Monte Carlo (DSMC). In the present study, Computational Fluid Dynamic (CFD) method is used to simulate the continuum area, and the DSMC method is employed in rarefied area. The results showed that the value of separation power obtained from pure CFD and CFD-DSMC solution is 10.5% different.
Keywords: Feed flow, Gas centifuge, CFD-DSMC, Separation power

Introduction
The gas flow regime inside the rotor covers the total range from free molecular (near the axis) to the continuum (near the rotor wall). CFD is employed to analyze the continuum region and DSMC is one of the best methods for flow analysis in the rarefied region. So the analysis of gas flow fields in the rotor desires progressive methods such as a coupled CFD-DSMC.
Numerous studies in the fields of analysis flow, including rarefied and continuum areas, have used hybrid CFD-DSMC methods, e.g. a supersonic flow over a quasi-2-D wedge [1], and a hypersonic flow over a 2-D cylinder [2].
So far, many studies have been performed to analyze the gas flow inside the rotor, in which mainly the continuum area inside the rotor is simulated, while the rarefied area is not accurately modeled [3,4].
In this study, the effect of feed flow using the CFD-DSMC method in a gas centrifuge is investigated. The radial velocity obtained from CFD and DSMC methods is compared in the feed zone. Finally, the consequence of rarefied areas on separation efficiency is evaluated.

Method
The UF_6 gas flow in the rotor is governed by a full set of hydrodynamic equations, including the conservation laws of mass, momentum, and energy [5]. To calculate the separation power, it is necessary to find the distribution of concentration in the rotor of centrifuge. For this purpose, the convection-diffusion equation is solved [5]. To solve the hydrodynamic equations using the CFD method, we used an Implicit Coupled Density-Based solver that has been created in the OpenFOAM framework. In the DSMC method, the gas flow is studied at the molecular level. The flow's macroscopic properties, including temperature, pressure, and velocity, are obtained using particle velocity. In this study, the dsmcFoam solver was utilized in OpenFOAM to analyze the rarefied areas. In dsmcFoam solver, the wall interaction and binary collision models were selected based on the Diffuse model and Larsen-Borgnake Variable Hard Sphere model, respectively [6]. The number of cells in the rarefied region was 110000. The number of particles per cell was considered 20.
In the CFD-DSMC solution, the Local Knudsen number has been employed to specify the location of interface for the rarefied and continuum areas. The state-based method was employed to transfer data between the continuum and rarefied regions [5]. The particle data in the DSMC zone is averaged to calculate macroscopic characteristics as CFD boundary conditions; At the same time, in DSMC buffer cells, it produces particles based on the macroscopic state of the continuum region [5,6]. The present rotor is considered in axisymmetric and steady states with the radius of 0.1 m, the length of 1 m, the angular velocity of 5500 rad/s, the cut of 0.45, and the wall temperature gradient of 20 K. To apply the inlet flow in the rotor through the feed entrance boundary, the inlet mass flow rate is used. The size of the cells near the rotor wall and axis is 1.0 μm and 10 μm, respectively.

Results and discussion
At the boundary with a radius of 0.086 m, the value of the Knudsen number is equal to 0.05, which is selected as the boundary between the continuum and rarefied area. Here, due to the importance of the feed and its effect on the behavior of the gas inside the rotor, the radial velocity distributions calculated by CFD and DSMC methods in the feed injection are presented in Figure 1 (a) and (b), respectively. To better observe the radial velocity, this diagram is drawn in the height range from -0.1 m to 0.1 m with the radius range from 0.025

m to 0.08 m. The height of feed injection point is considered at -0.004 m.

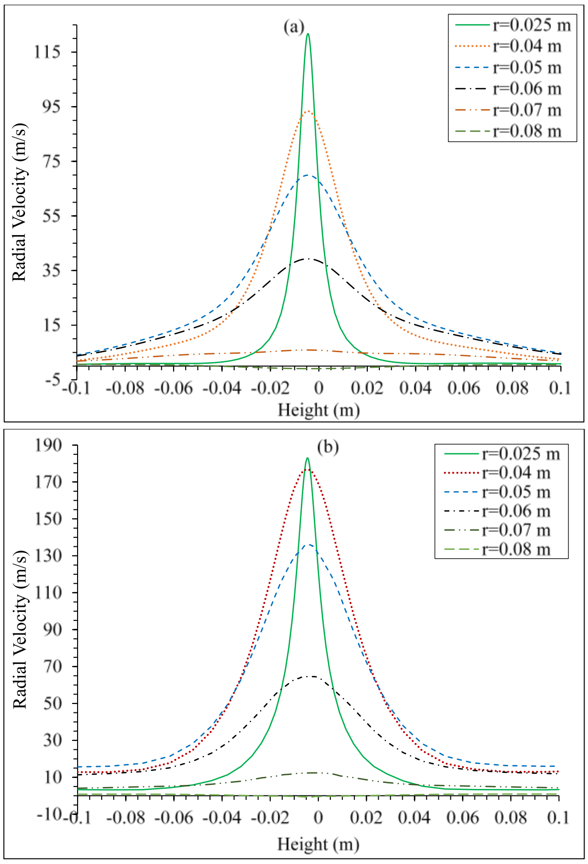

Figure 1. The radial velocity around the feed inlet obtained from CFD (a) and DSMC (b) methods

As can be seen, the UF_6 gas expands in front of the feed region because there is a high pressure difference between the inlet feed and vacuum region near the axis. With the expansion of the gas, the radial velocity increases to 120 m/s and 180 m/s calculated by CFD and DSMC method, respectively. In addition, As it moves away from the feed injection point, the radial velocity is damped so that the maximum value is the closest to the feed injection (r=0.025 m). The solution of Navier-Stokes equations using the CFD method in the rarefied zone is not sufficiently valid, so the distribution of radial velocities near the feed zone obtained from the DSMC solution is more accurate.

Using the results of the gas flow solution, the convective-diffusion equation is solved, and the concentration inside the rotor is obtained. Figure 2. shows the axial changes of $^{235}UF_6$ gas in the axial direction obtained by the pure CFD and CFD-DSMC methods in radius 0.096 m. As can be seen, $^{235}UF_6$ gas is enriched to about 0.0085 and depleted to about 0.006. In addition, although the difference in gas concentration distributions obtained from pure CFD and CFD-DSMC methods is small, for accurate evaluation it is necessary to calculate the value of separation power obtained from the two methods.

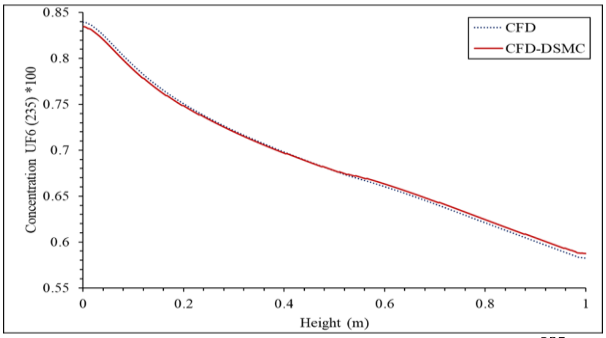

Figure 2. Comparison of the concentration of $^{235}UF_6$ calculated by CFD-DSMC and CFD methods

Based on the gas concentration distribution, the value of the separation power calculated by pure CFD and CFD-DSMC methods is 8.950 and 8.101 Kg (UF_6) SWU/year, respectively. The computational cost of CFD and DSMC methods is 24 and 50 hours, respectively.

Conclusions

This study showed due to the invalidity of Navier-Stokes equations in the rarefied area, the results of the CFD method are different from the CFD-DSMC method. The results showed that although the use of the CFD-DSMC method has more complexity and computational cost than the pure CFD, to accurately and correctly evaluate the rotor separation performance, it is necessary to use the CFD-DSMC method.

References

[1] J. Wu, et. al, *Development and verification of a coupled DSMC–NS scheme using unstructured mesh*, Journal of Computational Physics, vol. 219, pp. 579-607, (2006)

[2] G. Abbate, et. al, *Coupled Navier-Stokes/DSMC method for transient and steady-state gas flows*, *Springer-Verlag Berlin Heidelberg*, Vols. Part I, LNCS 4487, pp. 842-849, (2007)

[3] V. Borisevich, et al., *Numerical simulation of bellows effect on flow and separation*, Nuclear Instruments and Methods in Physics Research, vol. 455, pp. 487-494, (2000)

[4] D. Jiang and S. Zeng, *CFD simulation of 3D flowfield in a gas centrifuge*, International Conference on Nuclear Engineering, (2006)

[5] V. Ghazanfari, et. Al, *Investigation of the continuum-rarefied flow and isotope separation using a hybrid CFD-DSMC simulation for UF6 in a gas centrifuge*, Annals of Nuclear Energy, vol. 152, p. 107985, (2020)

[6] M. Darbandi and E. Roohi, *Applying a hybrid DSMC/Navier–Stokes frame to explore the effect of splitter catalyst plates in micro/nanopropulsion systems*, Sensors and Actuators A: Physical, vol. 189, pp. 409-419, (2013)