Japan Society of Material Cycles and Waste Management

Numerical Simulation of Pilot-scale Circulating Fluidized Bed Combustor

○ Hoon Chae Park, Hang Seok Choi*, Yong Chil Seo, Ha Na Jang, Seung Ki Baek, Jin Ho Sung
Department of Environmental Engineering, Yonsei University, Wonju, Republic of Korea, 220-710

1. Introduction
Waste biomass usually contains negligible sulfur, low nitrogen and heavy metals, and is considered neutral with respect to greenhouse gases. In recent years, circulating fluidized bed (CFB) combustion of waste biomass has received a lot of attention due to its fuel flexibility, high combustion efficiency, and low NOx and SO2 emissions [1]. For the optimal design of the CFB combustor, the solid circulation rate which influences the residence time of particles and the solid holdup are very important. In this study, simulation of a CFB combustor was carried out to investigate the hydrodynamic characteristics. For the simulation of CFB combustor, computational particle fluid dynamic (CPFD) approach is applied.

2. Computational method
Computational particle fluid dynamic (CPFD) is emerging technology for the modeling of gas-solid system. CPFD numerical methodology was proposed as a multi-phase particle-in-cell (MP-PIC) method. The gas phase is described by Eulerian mass and momentum conservation equations with strong coupling to the particle phase. The particles momentum follows the MP-PIC numerical method which provides Lagrangian description of particle motion coupled with the gas by ordinary differential equations. The particles are implicitly coupled to the gas phase through the inter-phase drag. The particles move freely within the whole computational domain and are tracked by the Lagrangian approach [2]. The large-scale systems including billions of real particles can be simulated with millions of numerical particles.

In the present study, the three-dimensional simulation of CFB combustor was performed by using CPFD method. Fig.1 shows the schematic diagram of CFB combustor. CFB combustor consists of riser, cyclone and loop seal. The CFB combustor has a total height 6m, with 0.15m of inner diameter. A dirichlet boundary condition is used for the fluid phase at gas inlet of the CFB which is located at the bottom of the CFB riser. A constant pressure boundary condition is employed at the outlet of the cyclone. Partial slip of particles and no slip of gas are assumed for the wall boundary conditions. In CFB combustor, sand particles are initially piled in the riser and fluidized by air. The sand particle has the density of 2,520kg/m³ and mean diameter of 290 µm. According to the Geldart classification, the sand belongs to the Geldart B category.

3. Results and Discussion
In the operation and design of a CFB waste biomass combustor, CPFD is very useful to predict flow parameters in the CFB riser such as the axial solid fraction, solid circulation rate and pressure drop, etc. Fig.2 shows the distribution of time-averaged solid volume fraction and pressure at the superficial gas velocity of 2.35 m/s. The solid volume fraction in the CFB riser varies with height. The solid volume fraction is higher at the bottom of riser.
and decreases toward the top of riser. The pressure is high at the lower part of the riser, which is the region of higher solid volume fraction and the pressure is low at the upper part of the riser, which is region of lower solid volume fraction.

Fig. 3 shows the particle flows colored by axial velocity and velocity magnitude. The magnitude of axial velocity is higher close to the center of the CFB riser. However, near the walls, solid particles represent negative axial velocity which indicates downward solid flow. In particular, the solid flow mainly influences its chemical reaction hence to calculate accurately the characteristics of solid flow are essential to predict the local reaction rate of the solid fuel. Also, comparing with Fig. 2 (a) and Fig. 3 (b), the higher region of solid volume fraction and velocity magnitude are located at the bottom of the CFB. Also, another higher region of velocity magnitude is generated near the inlet of cyclone. In these regions, the erosion rate of the wall surface may be increased because the erosion rate is the function of solid velocity, solid volume fraction, impact angle, etc.

4. Conclusions

In this study, CPFD was applied to investigate the hydrodynamics characteristics of CFB waste biomass combustor. The lower part of the CFB riser shows higher solid volume fraction and pressure. The axial particle velocity is higher close to the center of the CFB riser and solid particles represent negative axial velocity near the wall. These characteristics of solid flow affect the reaction rate of the solid fuel as well as the erosion rate of the surface wall. The CPFD simulation results can be further applied for sensitivity analysis during scale-up design.

Acknowledgement

“This work is financially supported by Korea Ministry of Environment (MOE) as ‘Knowledge-based environmental service (Waste to energy & recycling) Human resource development Project’.”

References