Powder Technology 312 (2017) 113–123

Contents lists available at ScienceDirect

Powder Technology

journal homepage: www.elsevier.com/locate/powtec

Uneven distribution of particle flow in RFCC reactor riser

Hyungtae Cho [a], Junghwan Kim [b], Chanho Park [a], Kwanghee Lee [a], Myungjun Kim [a], Il Moon [a,*]

[a] Department of Chemical and Biomolecular Engineering, Yonsei University, 50 Yonsei-ro, Seodaemun-gu, Seoul 03722, Republic of Korea
[b] Green Manufacturing 3Rs R&D Group, Korea Institute of Industrial Technology, 55 Jongga-ro, Jung-gu, Ulsan 44413, Republic of Korea

ARTICLE INFO

Article history:
Received 22 June 2016
Received in revised form 10 November 2016
Accepted 6 January 2017
Available online 01 February 2017

Keywords:
Residue fluidized catalytic cracking (RFCC)
Riser
Feed injection
Uneven distribution
Computational particle fluid dynamics (CPFD)

ABSTRACT

The uneven distribution of particle flow, i.e., a different particle mass flow rate in each outlet of the riser in residue fluidized catalytic cracking (RFCC) processes, is one major problem associated with commercial RFCC processes. This problem affects the formation of carbonaceous deposits in the secondary reactor cyclone, which incurs serious catalyst carryover in the fractionators. This study analyzes particle-fluid flow patterns in the riser, and diagnoses the uneven distribution of particle flow using a computational particle fluid dynamics (CPFD) method to solve this real industrial problem. Through this analysis, the effect of the number of feed injectors is investigated. The CPFD method, which has been developed to complement the Eulerian-Eulerian and Eulerian-Lagrangian methods, applies the Navier-Stokes equation for fluid phase and multi-phase-particle-in-cell (MP-PIC) models for particle phase. The particle flow distribution was found to vary by 15.5–18.7% at different outlets in the 1 injector case, which implies that the solid loading ratio in each cyclone is different, thereby affecting the separation efficiency of the cyclone and the formation of carbonaceous deposits. The uneven distribution of particle flow phenomena was identified, and the standard deviations of particle mass flow rates were evaluated for the cases of 1, 2, 4, 6, 8 and 12 injectors, and were found to be 7.52, 4.07, 2.66, 1.78, 2.85 and 3.82, respectively. From these results, the 6 injectors case was found to have a largely even particle flow distribution.

1. Introduction

Residue fluidized catalytic cracking (RFCC) is a key unit in both refineries and typical heavy oil upgrading (HOU) processes [1]. In this process, the residue from atmospheric distillation towers are converted into highly valuable light hydrocarbons [2,3].

An RFCC unit is composed of a riser, a reactor, and a regenerator. The cracking reaction occurs in the riser, and the product gas and catalysts are then separated in a reactor cyclone. The spent catalysts are subsequently regenerated in the regenerator. Over the past ten years, the RFCC process has rapidly evolved as a major means of upgrading heavy oil, because of the quality of the crude oil [4]. With an increase in the much heavier fraction in the residue, carbonaceous deposits are formed in this unit; the buildup of carbonaceous deposits is a common problem in RFCC units [5].

One of the main problems affecting the operation period is the formation of deposits in the cyclone dipleg and on the outer surface of the gas outlet tube. In one case, this deposit grew to eventually plug the dipleg, causing significant catalyst carryover into the main fractionators. In another instance [6], a lump of heavy coke deposit broke free from the dipleg, which can either block the flapper valve at the end of the dipleg or jam the valve open, causing a malfunction of the cyclone. This malfunction resulted in a sudden and extreme loss of catalyst inventory, and the unit had to be shut down.

The formation of deposits in the dipleg or on the gas outlet tube of the cyclone in commercial RFCC units is governed by the gas-solid flow [7]. The gas-solid flow pattern in the reactor cyclone is determined by the hydrodynamic behavior in the riser. During the inspection of the cyclones after the shutdown of a commercial RFCC, Kim et al. [6] reported that deposits were found in all six diplegs of the secondary cyclones. However, the deposit sizes varied depending on the location of the secondary cyclones and the operation period. It could thus be inferred that an uneven distribution of particle flow had occurred, which affected the cyclone flow pattern, resulting in the formation of deposits. Hence, it became clear that knowledge of the hydrodynamic behavior in the riser is of considerable importance for reducing the deposit formation. In previous studies [8,9], it was also found that the feed injector influences the high velocity, temperature, and concentration gradients, thereby affecting the overall performance of the reactor.

The hydrodynamic behavior in the feed injection zone was found to be very complex. Importantly, experimental studies [10–14] on RFCCs or FCC riser reactors have typically focused on this hydrodynamic behavior in the riser. For instance, Fan et al. [10] investigated the FCC

Abbreviations: RFCC, residue fluidized catalytic cracking; CPFD, computational particle fluid dynamics; MP-PIC, multiphase-particle-in-cell; HOU, heavy oil upgrading; CFD, computational fluid dynamics; TFM, two-fluid model; DEM, discrete element method; KTGF, kinetic theory of granular flow; PSD, particle size distribution.

* Corresponding author.
E-mail address: ilmoon@yonsei.ac.kr (I. Moon).

http://dx.doi.org/10.1016/j.powtec.2017.01.025
0032-5910/© 2017 Published by Elsevier B.V.

riser performance in a large cold-riser model, and also experimentally studied the flow pattern of the feed spray and catalysts in the feed injection zone [12,13]. Xu et al. [14] then experimentally evaluated the distribution of pressure and particle concentration in the feed injection zone in a cold-riser model; however, their paper does not completely describe the flow pattern in this zone.

Conducting experiments on the riser flow under reactive conditions is difficult because of the high temperatures involved; this is a limitation of experimental studies. Therefore, to investigate the hydrodynamic behavior in the riser under commercial operating conditions such as high temperature and pressure, a simulation model of the riser is needed [15]. Several computational fluid dynamics (CFD) models have been developed to overcome the limitations of experimental studies [16–18]. A simulation model was used to analyze the gas-solid hydrodynamic pattern and optimize the feed injector in terms of the number of injectors required, in addition to the structure, position, and condition of the injectors [19–32].

To predict the performance of FCC riser reactors, Gao et al. [8] developed a three-dimensional two-phase CFD model with 13-lump reaction kinetics. Their simulation results revealed that the gas-solid turbulence reacting in the flow regime in the riser reactor is very complex, owing to the effect of the feed injector. Recently, Li et al. [29] developed a riser reactor model that combined the catalyst population balance model with a 14-lump reaction model to simulate the turbulent gas-solid flow and resultant reactions in a polydisperse FCC riser reactor. Furthermore, Li et al. [30] investigated the influences of nozzle position and angle in the riser on the performance in the feedstock injection zone under reactive conditions, using a comprehensive 3D heterogeneous reactor model. Note that these works applied a two-fluid model (TFM) or discrete element model (DEM) to simulate the multiphase flow system.

However, the TFM has limitations in dealing with the distribution of particles of different types and sizes owing to the methodology used in the treatment of the solid phase, due to the kinetic theory of granular flow (KTGF). On the other hand, the DEM has limitations in simulating commercial plant-scale multiphase flow systems, because of its high computational cost [33]. Recently, Snider [34] used computational particle fluid dynamics (CPFD) to develop a new Eulerian-Lagrangian multiphase flow model. Using a personal computer at an affordable computational cost, the CPFD model can simulate a commercial-scale fluidized bed reactor having billions of particles. Hence, the multiphase-particle-in-cell method can be applied to the CPFD numerical methodology in order to calculate particle flows [34,35].

Several papers have been published in which the CPFD model has been used to simulate commercial-scale risers and the RFCC process [36–40]. For example, Cho et al. [36] successfully used the CPFD model to analyze the gas-solid multiphase flow in the reactor cyclone of an RFCC. They predicted the location of deposit formation and proposed an advanced design that minimizes its impact. Chen et al. [37] and Liang et al. [38] then evaluated the applicability of CPFD numerical schemes by comparing CPFD schemes with experimental data and other schemes. Shi et al. investigated the effects of particle size distribution [39] and riser exit geometry [40] on solid back-mixing using CPFD schemes.

However, the above-mentioned studies did not deal with the effect of feed injector on the hydrodynamic behavior and the uneven distribution of particle flow in the RFCC riser, perhaps not realizing the importance of this issue. The uneven distribution phenomenon is a significant factor affecting the deposit formation and shutdown problem. Overcoming this oversight, we develop here a CPFD model to describe the gas-solid hydrodynamic behavior and the uneven distribution of particle flow in an RFCC riser.

The present study investigates the effect of the number of injectors on the uneven distribution of particle flow, in order to minimize the carbonaceous deposits and maximize the operation period. Here, we will first describe the gas-solid hydrodynamic behavior, and then discuss the uneven distribution phenomenon. The CPFD model, simulated using commercial operating conditions, is then validated by comparing the simulation results with commercial plant data, and the uneven distribution of particle flow, which affects cyclone performance and deposit formation, is subsequently confirmed by the simulation results. Finally, the influence of the number of feed injectors on the uneven distribution of particle flow is numerically evaluated, making it possible to select an appropriate number of feed injectors for a commercial scale plant.

2. CPFD governing equations

The CPFD model applies the averaged Navier-Stokes equation with strong coupling to the discrete particles for fluid dynamics and multiphase-particle-in-cell (MP-PIC) [34,35] formulation, with the addition of a relaxation-to-the-mean term to represent the damping of particle velocity fluctuation due to particle collisions [41] for particle momentum [42]. The model aims to solve problems associated with the commercial process scale, which are physically large three-dimensional systems [34]. In the CPFD model, a numerical particle is defined, where particles with the same properties are grouped, because it is a numerical approximation similar to the numerical control volume of the fluid. The problems with commercial process scaling containing billions of particles can thus be solved using millions of numerical particles [43].

The volume-averaged two-phase incompressible continuity equations with no interphase mass transfer and the momentum equation for the fluid are as follows [44].

$$\frac{\partial \theta_f \rho_f}{\partial t} + \nabla \cdot \left(\theta_f \rho_f \mathbf{u}_f \right) = \delta \dot{m}_p \tag{1}$$

and

$$\frac{\partial \left(\theta_f \rho_f \mathbf{u}_f \right)}{\partial t} + \nabla \cdot \left(\theta_f \rho_f \mathbf{u}_f \mathbf{u}_f \right) = -\nabla p + \mathbf{F} + \theta_f \rho_f \mathbf{g} + \nabla \cdot \left(\theta_f \boldsymbol{\tau}_f \right) \tag{2}$$

where θ_f is the fluid volume fraction, ρ_f is the fluid density, and $\mathbf{u}_f$ is the fluid velocity. The gas mass production rate per unit volume from particle-gas chemistry, $\delta \dot{m}_p$, is zero, because there is no reaction. Here, p is the fluid pressure, $\mathbf{F}$ is the rate of momentum exchange per unit volume between the fluid and the particles, $\mathbf{g}$ is the gravitational acceleration, and $\boldsymbol{\tau}_f$ is the macroscopic fluid stress tensor.

The constitutive equation for the non-hydrostatic part of the stress, $\boldsymbol{\tau}_f$, in index notation, is

$$\boldsymbol{\tau}_{f,ij} = \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \mu \boldsymbol{\delta}_{ij} \left(\frac{\partial u_k}{\partial u_k} \right) \tag{3}$$

where μ is the shear viscosity, which is the sum of the laminar shear viscosity and the turbulence viscosity from the Smagorinsky turbulence model [45]. Large eddies are calculated, and the unresolved sub-grid turbulence can be modeled using the eddy-viscosity, given by

$$\mu_t = C \rho_f \Delta^2 \sqrt{\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)^2} \tag{4}$$

where D is the sub-grid length scale, which is the cube root of the sum of the products of the distances across a calculation cell in the three orthogonal directions [42].

The mass and momentum are then solved using the mass fractions of the gas species, and the equation of the state for an ideal gas is used in the CPFD methodology.