Particuology xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Particuology

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

Influence of drag laws on pressure and bed material recirculation rate in a cold flow model of an 8 MW dual fluidized bed system by means of CPFD

Stephan Kraft[a,b,*], Friedrich Kirnbauer[a], Hermann Hofbauer[b]

[a] Bioenergy 2020+ GmbH, Wiener Strasse 49, A-7540 Güssing, Austria
[b] TU Wien, Institute of Chemical Engineering, Getreidemarkt 9/166, Vienna, Austria

ARTICLE INFO

Article history:
Received 10 October 2016
Received in revised form 28 March 2017
Accepted 6 April 2017
Available online xxx

Keywords:
Cold flow modeling
Fluidization
Computational particle fluid dynamics (CPFD) simulation
Dual fluidized bed
Computational fluid dynamics

ABSTRACT

A cold flow model of an 8 MW dual fluidized bed (DFB) system is simulated using the commercial computational particle fluid dynamics (CPFD) software package Barracuda. The DFB system comprises a bubbling bed connected to a fast fluidized bed with the bed material circulating between them. As the hydrodynamics in hot DFB plants are complex because of high temperatures and many chemical reaction processes, cold flow models are used. Performing numerical simulations of cold flows enables a focus on the hydrodynamics as the chemistry and heat and mass transfer processes can be put aside. The drag law has a major influence on the hydrodynamics, and therefore its influence on pressure, particle distribution, and bed material recirculation rate is calculated using Barracuda and its results are compared with experimental results. The drag laws used were energy-minimization multiscale (EMMS), Ganser, Turton–Levenspiel, and a combination of Wen–Yu/Ergun. Eleven operating points were chosen for that study and each was calculated with the aforementioned drag laws. The EMMS drag law best predicted the pressure and distribution of the bed material in the different parts of the DFB system. For predicting the bed material recirculation rate, the Ganser drag law showed the best results. However, the drag laws often were not able to predict the experimentally found trends of the bed material recirculation rate. Indeed, the drag law significantly influences the hydrodynamic outcomes in a DFB system and must be chosen carefully to obtain meaningful simulation results. More research may enable recommendations as to which drag law is useful in simulations of a DFB system with CPFD.

Introduction

At the Technische Universität (TU) Wien, a process for biomass gasification has been developed which operates on the dual fluidized bed (DFB) principle. The DFB reactor system consists of two connected fluidized bed reactors: a gasification reactor and a combustion reactor. Between these two reactors, bed material circulates. Biomass is fed into the gasification reactor, which is operated in a bubbling bed regime and fluidized with steam. The biomass particles are dried and the volatiles are released. The product gas consists mainly of H_2, CO, CO_2, CH_4, and H_2O, and leaves the gasification reactor at the top. The product gas is further used for various applications, such as the generation of electricity and heat (Hofbauer, Rauch, Bosch, Koch, & Aichernig, 2001), hydrogen separation (Kraussler, Binder, & Hofbauer, 2016), and production of synthetic natural gas (Rehling, Hofbauer, Rauch, & Aichernig, 2011), and synthetic fuels like alcohol (Weber, Di Giuliano, Rauch, & Hofbauer, 2016) or Fischer–Tropsch diesel (Sauciuc et al., 2012).

The use of cold flow models for investigating hydrodynamics of hot plants is a practical tool because a cold flow operates at ambient conditions. Dimensional analysis is applied to keep the relevant dimensionless numbers constant between hot and cold models to ensure hydrodynamic similarity. Many approaches are available in the literature (Rüdisüli, Schildhauer, Biollaz, & van Ommen, 2012). At the TU Wien, cold flow modeling has a long tradition. Various laboratory and industrial plants have been designed based on cold flow model investigations, such as the DFB plant in Güssing (Kreuzeder, Pfeifer, & Hofbauer, 2007), a chemical looping combustion (CLC) plant (Pröll, Rupanovits, Kolbitsch, Bolhàr-Nordenkampf, & Hofbauer, 2009), and laboratories studying the conceptually new

* Corresponding author at: TU Wien, Institute of Chemical Engineering, Getreidemarkt 9/166, Vienna, Austria.
E-mail address: stephan.kraft@bioenergy2020.eu (S. Kraft).

http://dx.doi.org/10.1016/j.partic.2017.04.009

Nomenclature

a, *b*, *c*, *d*, and *e*	Constants in Eq. (5)
$\vec{a}_p$	Acceleration of particle phase (m/s)
A_q	Cross section area (m^2)
C_d	Drag coefficient
CV	Coefficient of variation
d_p	Diameter (m)
D_p	Drag function (s^{-1})
f_e	Coefficient for EMMS drag law
$\vec{g}$	Gravity (m/s^2)
h	Height (m)
K_1, K_2	Constants in Eq. (6)
m	Mass (kg)
$\dot{m}$	Mass flow rate (kg/s)
p	Pressure (Pa)
$\bar{p}$	Average pressure (Pa)
P_s	Solid pressure (Pa)
t	Time (s)
T	Temperature (K)
$\vec{u}$	Velocity (m/s)
$\dot{V}$	Volume flow rate (m^3/s)
V_p	Volume of particle phase (m^3)

Greek letters

α	Constant in Eq. (4)
β	Constant in Eq. (4)
ϵ	Volume fraction
ρ	Density (kg/m^3)
σ	Standard deviation
τ	Interparticle stress (Pa)
φ_p	Sphericity
ω	Function for the EMMS drag law

Subscripts

cp	Close pack
CR	Combustion reactor
ER	Ergun drag law
g	Gas phase
GA	Ganser drag law
GR	Gasification reactor
ls	Loop seal
mf	Minimum fluidization
tot	Total
p	Particle
prim	Primary air
rec	Recirculation
sec	Secondary air
TL	Turton–Levenspiel drag law
WY	Wen–Yu drag law
WE	Wen–Yu/Ergun drag law

Dimensionless numbers

$Ar = \frac{\rho_g(\rho_p - \rho_g) g d_p^3}{\mu_g^2}$ Archimedes number

$Re_p = \frac{u_0 d_p \rho_g}{\mu_g}$ Particle Reynolds number

$d_p^* = Ar^{1/3}$ Dimensionless particle diameter

$U^* = Re_p / Ar^{1/3}$ Dimensionless superficial velocity

Acronyms

CFB	Circulating fluidized bed
CFD	Computational fluid dynamics
CHP	Combined heat and power
CLC	Chemical looping combustion
DFB	Dual fluidized bed
PSD	Particle size distribution

DFB technology (Schmid, Pröll, Kitzler, Pfeifer, & Hofbauer, 2012). Shrestha, Ali, and Hamid (2016) have recently reviewed many studies on cold flow modeling of DFB devices.

Furthermore, in recent years, computational fluid dynamics (CFD) simulations of gas–particle flows have made great progress. There are various ways to simulate gas–particle flows. The Euler–Euler approach has been widely used for modeling gas–solid fluidized beds. The gas and particle phases are described as interpenetrating fluids. For the solid and gas phase, the continuity and momentum equations have to be solved (Hernández, 2008). The use of one average particle diameter instead of the particle-size distribution (PSD) could cause errors in the calculation. With a PSD, every class of particle size needs to be implemented as an additional phase (Gidaspow, 1994), which increases the computational effort (Hjertager, Solberg, Ibsen, & Hansen, 2004). The Euler–Lagrangian approach is more detailed at the particle level: the motion of each particle is defined by classical Newtonian mechanics. The Newtonian equations of motion are calculated for every particle, including particle–particle collisions (Van Wachem & Almstedt, 2003). Its disadvantage is its high computational cost because every particle has to be treated individually.

The multi-phase particle-in-cell (MP-PIC) approach offers a tradeoff between these two approaches (Andrews & O'Rourke, 1996; Snider, 2001). The fluid phase is described by the Navier–Stokes equations, and for the particles, a particle distribution function is found. Not every particle has to be treated individually; partitioning particles into groups reduces the computational cost.

In fluidized beds, the drag exerted by the fluid on the particles in the direction of the current is locally and temporarily greater than or equal to gravity, and causes a liquid-like motion of the particles. Other forces exerted on the particles include the static pressure gradient and the interparticle stress. Modeling these forces is crucial in predicting fluidized bed behavior with computational methods. Van Wachem, Schouten, Van den Bleek, Krishna, and Sinclair (2001) concluded from CFD simulations of fluidized beds that drag and gravity are the dominant forces. Mueller and Reh (1993) found that the formation of strands in the accelerated flow decreased drag. Furthermore, the PSD has a great influence on the hydrodynamics in a fluidized bed like bubbles or bed expansion (Gauthier, Zerguerras, & Flamant, 1999; Grace & Sun, 1991) and the correct PSD has to be employed in the simulations.

Barracuda has been used in various cold flow model studies to study fluidized bed hydrodynamics. Lim et al. (2016) investigated the influence of the restitution coefficient and the solid pressure. Investigations have been performed in recent studies with bubbling beds (Fotovat, Abbasi, Spiteri, de Lasa, & Chaouki, 2015; Liang, Zhang, Li, & Lu, 2014; Weber, Layfield, Van Essendelft, & Mei, 2013), spouting beds (Zhang, Wang, Wang, Qin, & Xu, 2016), internally circulating fluidized beds (Solnordal et al., 2015), and risers (Chen, Werther, Heinrich, Qi, & Hartge, 2013; Rodrigues, Forret, Montjovet, Lance, & Gauthier, 2015; Shi, Lan, Liu, Zhang, & Gao, 2014; Shi, Sun et al., 2015; Shi, Wu, Lan, Liu, & Gao, 2015; Wang et al., 2015), and parts of circulating fluidized bed combustors (Qiu, Ye, & Wang, 2015).

Furthermore, full-loop circulating fluidized bed systems have been reported in the literature. Clark, Snider, and Spenik (2013) simulated a full carbon capture bench scale unit consisting of two fluidized beds, two loop seals, a riser, and a cyclone with a total