Particuology xxx (2015) xxx–xxx

Contents lists available at ScienceDirect

Particuology

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

Experimental and numerical study of pseudo-2D circulating fluidized beds

Adam Klimanek[a,*], Wojciech Adamczyk[a], Sirpa Kallio[b], Paweł Kozołub[c], Gabriel Węcel[a], Andrzej Szlęk[a]

[a] Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland
[b] VTT Technical Research Centre of Finland Ltd, P.O. Box 1000, FI-02044 VTT, Finland
[c] Amec Foster Wheeler, Staszica 31, 41-200 Sosnowiec, Poland

A R T I C L E I N F O

Article history:
Received 3 April 2015
Received in revised form 6 August 2015
Accepted 1 September 2015
Available online xxx

Keywords:
Circulating fluidized bed
Euler–Euler approach
Gas–solid flow
Kinetic theory of granular flow
Particle size distribution
2D vs 3D

A B S T R A C T

We present experimental investigations and numerical simulations of a pseudo-2D riser. Experiments were performed for various airflow rates, particle types/diameters, and particle size distributions. Pressure distributions along the wall of the riser were measured. Additional measurements from a smaller pseudo-2D riser (Kallio et al., 2009; Shah et al., 2012) were used to analyze horizontal solids volume fraction profiles. The experimental data were compared with simulation results carried out using an Euler–Euler approach. A mesh sensitivity study was conducted for numerical simulations and effects associated with simplifying real 3D geometry to a 2D model were examined. In addition, the effect of using an algebraic equation to represent the granular temperature versus a full partial differential equation also was examined for numerical simulations. Results showed small but significant near-wall sensitivity of the flow variables to mesh size. Substantial differences in mean pressure, solids distribution, and solid velocities were obtained, when 2D and 3D simulation results were compared. Finally, applying the simplified granular temperature equation for turbulent fluidization and for dilute-phase transport can lead to incorrect predictions in models.

Introduction

Circulating fluidized beds (CFB) are frequently used as chemical reactors for combustion, gasification, and heterogeneously catalyzed synthesis and cracking. Understanding and appropriately modeling CFB hydrodynamicsis vital for predicting their transport processes, i.e., heat and mass transfer, as well as chemical reactions occurring in these reactors. This knowledge is used in multiphase flow research, and for proper design and optimization of industrial facilities. A variety of methods have been developed and used to model such hydrodynamics, using various solvers for computational fluid dynamics. These approaches can be subdivided, based on the spatial and temporal scales accounted for in the models (Myöhänen & Hyppänen, 2011). They can also be distinguished based on the way the solid phase is treated. Lagrangian or discrete particle (or phase) models track the particles or their groups (clouds). In Eulerian or two-fluid models, the solid phase is treated as continuous. Here, the particle motion is averaged out, which allows simulations to be performed on meshes much coarser than the particle diameters involved. Such models can be applied to large-scale system simulations. However, this infers that closure approximations must be provided for both fluid–solid and solid–solid interactions. Fluid–solid interactions are realized using drag coefficients (Gidaspow, 1994). For solid–solid interactions, the kinetic theory of granular flow (KTGF) (Gidaspow, 1994; Lun, Savage, Jeffrey, & Chepurniy, 1984) is applied, which allows the user to determine solid stresses arising from particle streaming and collisions. In dense regions, however, frictional stresses become important and additional closures must be provided to account for these phenomena. Thus, in Eulerian models, the solid phase is represented by its volume fraction, density, and velocity. Furthermore, solid pressure, bulk, frictional and shear viscosities, as well as a single representative diameter of the particles are assigned to the solid phase. In real systems, however, we usually deal with polydisperse particles. An Eulerian approach is frequently applied to gas–solid flow and specifically to CFB modeling for both small scale facilities (Almuttahar & Taghipour, 2008a, 2008b; Chalermsinsuwan, Piumsomboon, &

---

* Corresponding author. Tel.: +48 32 2372974; fax: +48 32 2372872.
E-mail address: adam.klimanek@polsl.pl (A. Klimanek).

http://dx.doi.org/10.1016/j.partic.2015.09.009
1674-2001/

Please cite this article in press as: Klimanek, A., et al. Experimental and numerical study of pseudo-2D circulating fluidized beds. *Particuology* (2015), http://dx.doi.org/10.1016/j.partic.2015.09.009

Gidaspow, 2009a, 2009b; Cloete, Amini, & Johansen, 2011; Hartge, Ratschow, Wischnewski, & Werther, 2009; Kallio et al., 2009; Lu et al., 2008; Wang et al., 2010; Zhang, Lu, Wang, & Li, 2008) and large industrial systems (Wischnewski, Ratschow, Hartge, & Werther, 2010; Zhang, Lu, Wang, & Li, 2010).

Here, we present application of the Eulerian approach to modeling of a pseudo-2D CFB with Geldart B and D particles. First, we present our experimental study, followed by simulations of this same setup, which are used to compare with experimental data obtained from two pseudo-2D CFBs. One of the CFBs is installed at the Silesian University of Technology (SUT) in Poland, and the other at the Åbo Akademi University in Finland. Results of the experiments and simulations carried out at the second facility are from the literature (Kallio et al., 2009; Shah, Ritvanen, Hyppänen, & Kallio, 2012). In this research, the effect of particle size and particle size distribution on the quality of results is examined. This was evaluated through comparison of simulations and experimental results of particle groups in three sizes. In our simulations, the Eulerian solid phase was characterized by a single Sauter mean diameter.

In the two-fluid model, solid–solid interactions are determined using the KTGF. This requires solution of the balance equation for granular temperature. A full partial differential equation, or its simplified form (an algebraic equation), is solved to obtain granular temperature and the solid stress tensor. The partial differential equation is complex and difficult to solve, while the simplifications introduced in the algebraic equation are only valid for higher solid volume fractions and relatively low solid velocities (van Wachem, Schouten, Krishna, & van den Bleek, 1998; van Wachem, Schouten, van den Bleek, Krishna, & Sinclair, 2001). Such conditions occur in bubbling fluidized beds. However, because of its simplicity and stable behavior during calculations, the algebraic equation is frequently used in both bubbling fluidized beds (Cloete et al., 2013; van Wachem et al., 1998) and risers (Cabezas-Gomez, Silva, & Milioli, 2006). Here, we evaluate the effect of using the algebraic versus the partial differential equation for solving turbulent fluidization.

Because of the tremendous computational resources required to perform simulations using 3D models, in many studies (Almuttahar & Taghipour, 2008a, 2008b; Benyahia, Arastoopour, Knowlton, & Massah, 2000; Cloete et al., 2011), as well as in this research, the geometries are reduced to 2D, or else the simulations are performed on coarse meshes. The effect of using 2D vs 3D computational models for a bubbling, slagging, and turbulent fluidized bed was examined in detail by Xie, Battaglia, & Pannala (2008). They concluded that the difference between their results became more pronounced, as fluidization velocity increased. Therefore, such effects are expected to influence modeling of risers. Cloete et al. (2013) presented the influence of representing a pseudo-2D facility operated in a bubbling regime by 2D and 3D geometry. They concluded that the neglected friction at the walls in the 2D simulation considerably affected the flow field. To separate out the effects of neglecting thickness in the pseudo-2D facilities at higher superficial velocities, a 3D simulation was performed for one of our cases, using the same mesh in the *XY* plane as the 2D mesh. The influence of neglecting the third dimension in this model is examined in the last section of this paper.

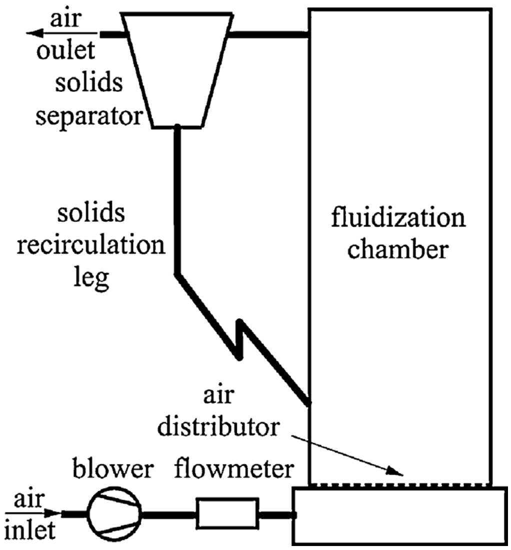

Fig. 1. Circulating fluidized bed at the Silesian University of Technology.

Experimental

The experimental facility at the SUT is shown in Fig. 1. The CFB is a 3.0-m high and 0.6-m wide pseudo-2D riser, with depth of 17 mm. The equipment consists of a blower, flowmeter, air distributor, riser, a solids separator (settling chamber), and a return leg with a loop seal. The air is supplied through 13 nozzles situated at the bottom of the riser. The fluidized solids, which leave the riser, are separated in the settling chamber, and returned back to the riser via a recirculation leg, with a fluidized loop seal. Measurements were conducted for different gas velocities. Gas pressure distribution along the right wall of the riser was measured. The pressure sensors were unevenly spaced at various heights, i.e., at 0.1, 0.3, 0.4, 0.6, 0.9, 1.65, and 2.35 m, measured from the bottom of the riser.

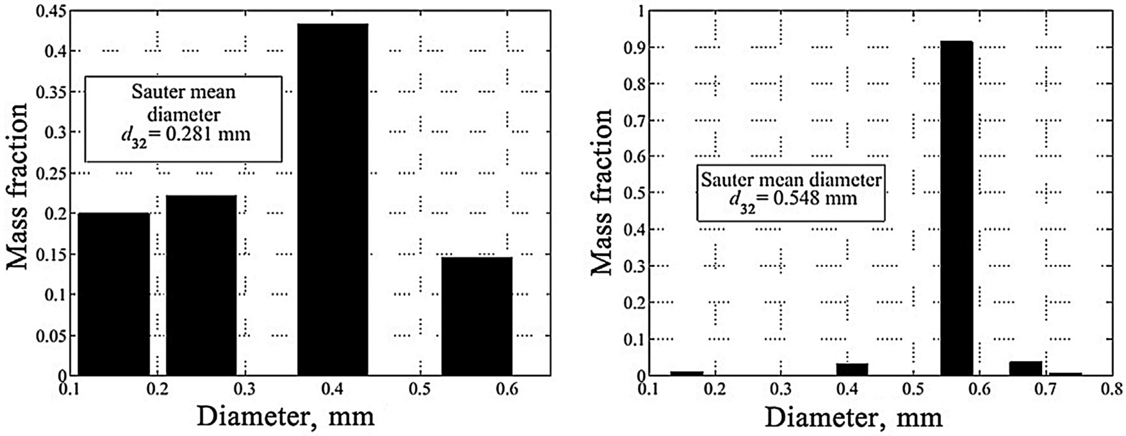

Fig. 2. Particle size distributions of type 1 (left) and type 2 (right) particles obtained from sieve analysis.

Please cite this article in press as: Klimanek, A., et al. Experimental and numerical study of pseudo-2D circulating fluidized beds. *Particuology* (2015), http://dx.doi.org/10.1016/j.partic.2015.09.009