International Journal of Mineral Processing 104-105 (2012) 58–70

Contents lists available at SciVerse ScienceDirect

International Journal of Mineral Processing

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

Investigation of drag models in CFD modeling and comparison to experiments of liquid–solid fluidized systems

Olli Visuri [a,*], Gijsbert A. Wierink [b], Ville Alopaeus [a]

[a] Aalto University, School of Chemical Technology, Department of Biological and Chemical Engineering, PO Box 16100 FI-00076 Aalto, Finland
[b] Aalto University, School of Chemical and Technology, Department of Materials Science and Engineering, PO Box 16100 FI-00076 Aalto, Finland

ARTICLE INFO

Article history:
Received 23 May 2011
Received in revised form 22 December 2011
Accepted 27 December 2011
Available online 5 January 2012

Keywords:
Euler–Euler
Drag
Fluidization
CFD
Digital imaging

ABSTRACT

A liquid–solid fluidization system was investigated with Computational Fluid Dynamics (CFD) by using a transient Eulerian-Eulerian model. The study focused on various drag models between the phases and how the results vary when simulating the system 2-dimensional or 3-dimensional. Also the grid dependencies to the results were investigated. The simulation results were validated experimentally using a digital imaging method. The suitability of this experimental method is also investigated in this paper. The CFD results show that the outcome from different drag models vary considerably and therefore the used model has to be chosen with care.

1. Introduction

Multiphase reactors are very often used in chemical industry. These involve gas–liquid, liquid–liquid, liquid–solid and gas–liquid–solid phases. In order to optimize the reactor design in industrial scale it is important to understand the hydrodynamics of the reactor. Traditionally this has been done with experimental scale-up. However, the correlations gained this way depend on the vessel geometry and therefore the final geometry has to be specified fairly early during the process development. If several options on the vessel dimensions are investigated the increase of costs is usually high.

In recent years due to the increase of computational capacity, Computational Fluid Dynamics (CFD) has been applied extensively to model these reactors and their flow behavior. CFD in multiphase systems can be divided into Eulerian–Eulerian (E–E) or Eulerian–Lagrangian (E–L) modeling. In the E–E method both phases are described using the Eulerian conservation equations and the phases are assumed to be interpenetrating continua (Drew, 1983), (Bothe et al., 2007). These are represented as averaged conservation equations. In E–L modeling the carrier phase is still described in an Eulerian framework while the dispersed phase is described using a Lagrangian frame of reference. For CFD simulation of more dense systems where concentration profiles are a key result, the E–E method is preferred (Enwald et al., 1996).

Although CFD is a promising tool for designing the equipment, there is a need to validate the CFD results, since the phenomena inside the cell volume has to be modeled. The goal in CFD is to be independent on the system, but the closure models that are used are commonly direct derivations from certain experimental system. Therefore their consistency should be checked experimentally before using them in modeling industrial systems. Normally the models are validated with much simpler systems, using so called mock-up experiments, where the chemicals are safer and therefore the need to invest complex systems during laboratory and pilot scale is reduced.

For experimental validation for multiphase systems numerous different methods can be found from literature. Basically these can be divided into two categories, invasive and non-invasive. In invasive techniques, commonly a probe (e.g. conductivity, thermal, optical) is placed inside the system. These methods provide local data from the system, especially when considering spatial and time resolution. The drawback with these systems is that they tend to disturb the experiment. Also gaining data from multiple locations causes problems. In non-invasive techniques the analysis is made outside the system, commonly using visualization or imaging technique. The drawbacks with these techniques are usually that they are limited to low hold-ups of the dispersed phase and the need of transparent equipment. A comprehensive review of various experimental techniques is presented (Boyer et al., 2002).

In this work the main focus was to investigate how a liquid–solid fluidized bed can be modeled with CFD. The experimental system consists of rectangular pipe where the solid bed height and concentration is measured using photographic technique. The fluid flow was driven through a dense grid making the turbulence isotropic.

* Corresponding author. Tel.: +358 470 25813.
E-mail address: Olli.Visuri@tkk.fi (O. Visuri).

0301-7516/$ – see front matter © 2012 Elsevier B.V. All rights reserved.
doi:10.1016/j.minpro.2011.12.006

Total amount of particles placed into the system was measured and the particle distribution was analyzed with photographic method. This method is non-intrusive, so there is no disturbance to the flow field.

The numerical simulation was carried out using the open source CFD package OpenFOAM, version 1.6, released by OpenCFD Ltd. (OpenCFD Ltd., 2009). The system was modeled by OpenFOAM's Eulerian-Eulerian solver twoPhaseEulerFoam, based on the work of Rusche (2002).

2. Experimental

2.1. Experimental setup and procedure

2.1.1. Particles

The used particles were porous cylindrical extrudates (see Fig. 1a), which are common in fluidized beds in chemical industry. The porosity of the used particles was 0.4744. The density and size distribution was measured by photographing the particles. It was assumed that the cross sectional area is the same for every particle and the length of the particles were determined from the image. The wetted density of the particles was approximately 2173 kg/m^3, mean volume of the particles were 1.8 mm^3 and the average sphericity ($\frac{surfacearea_{equivalent}}{surfacearea_{real}}$) were approximately 0.81. The volumetric size distribution is presented in Fig. 1b.

2.1.2. Experimental apparatus

The experimental apparatus consisted of a transparent vertical channel of square cross-section that was used for the fluidization. The particles mentioned above were suspended using water. A schematic diagram of the apparatus is shown in Fig. 2.

The dimensions of the pipe were 0.1 m (width) 1.8 m (height) and 0.1 m (depth). 0.6 m from the bottom a dense metal grid (internodal distance approx. 0.7 mm) was placed and the particles were placed above this grid.

A 2 mega pixel digital video camera (JAI, CV-M2) with 0.95/25 mm lens (Schneider optics, Xenon) was used in the experiments. At the acquisition the f-number was 2.0. Shutter speed was set to 1/250 s. The camera's frame rate was 30 frames/s.

The system was illuminated with a 2 kW HQI lamp using it as backlight. A set of diffusers were used to equalize the light intensity over the whole tube.

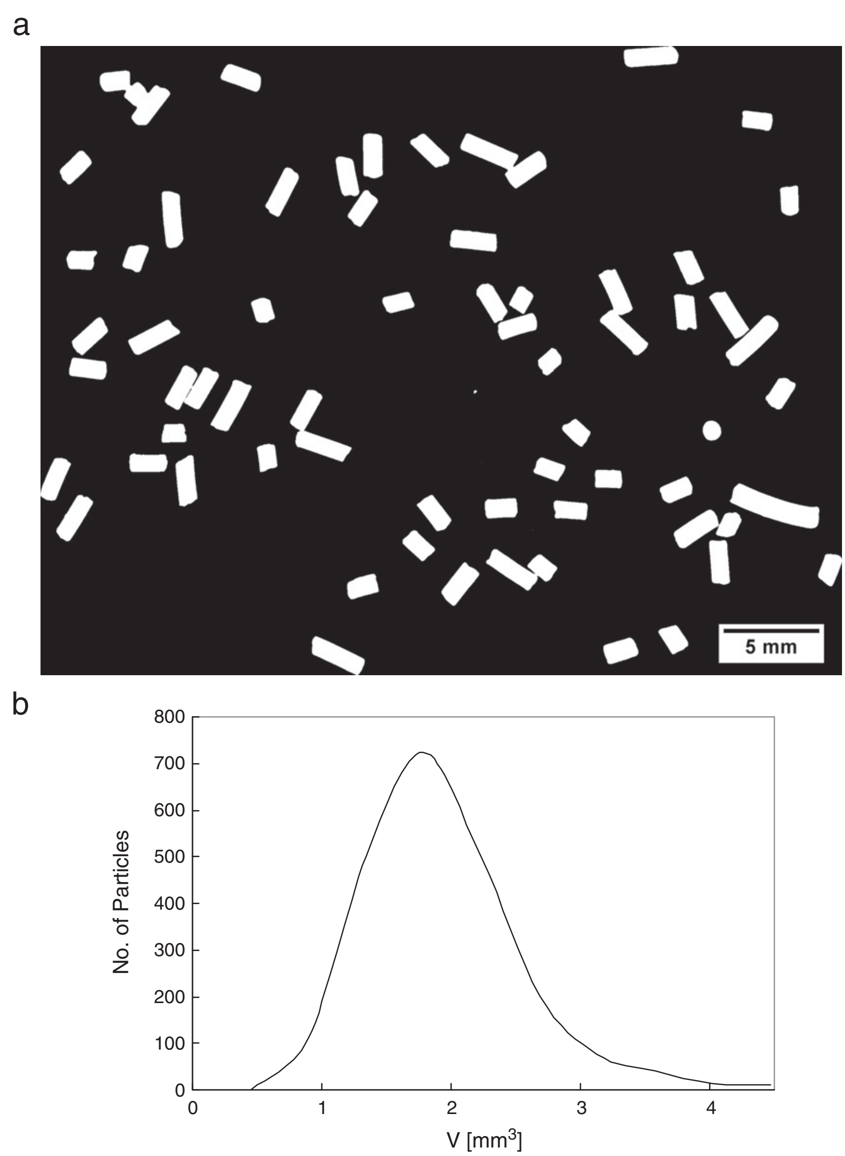

Fig. 1. An image of the particles and the volumetric distribution of the particles.