Advanced Powder Technology 24 (2013) 473–481

Contents lists available at SciVerse ScienceDirect

Advanced Powder Technology

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

Original Research Paper

CFD–DEM investigation of particle separations using a sinusoidal jigging profile

Stephen Viduka [a,b], Yuqing Feng [a,*], Karen Hapgood [b], Phil Schwarz [a]

[a] CSIRO Mathematics, Informatics, and Statistics, Box 312, Clayton South, 3169 Victoria, Australia
[b] Monash Advanced Particle Engineering Laboratory, Department of Chemical Engineering, Monash University, PO Box 36, Clayton, 3800 Victoria, Australia

ARTICLE INFO

Article history:
Received 15 June 2012
Received in revised form 29 October 2012
Accepted 8 November 2012
Available online 11 February 2013

Keywords:
Discrete element method
Computational fluid dynamics
Liquid–solid flow
Jigging
Stratification
Gravity concentration

ABSTRACT

This paper presents a numerical investigation of solid separation in jigging device. Jigging is a gravity separation method commonly used by the minerals industry to separate coal, iron ore, diamonds and other minerals on the basis of particle size and/or density. Separation is recognised as being heavily dependent on fluid motion in the jig. This study explores the effects of the inlet time dependent velocity profile in relation to a wide criterion on jigging performance. Modelling of the liquid–solid system is performed through a combination of computational fluid dynamics (CFD) to simulate liquid flow and discrete element method (DEM) to resolve particle motion. The initial packing conditions consist of a binary-density particle system of 1130 particles each 1 cm in diameter. A range of jigging profiles have been implemented in mineral processing. In this study the sinusoidal pulsation profile is selected adopting variations in both amplitude and frequency. The performance of profile variants are compared in terms of solid flow patterns, separation kinetics, energy, and mean particle position. These quantitative comparisons demonstrate significant differences in the segregation rate, energy, and solid phenomena, helping find an alternative optimum operating setting for the system. In addition, boundaries of operation are found in terms of frequency and amplitude limits and the concentration mechanics are investigated in these regions.

1. Introduction

Jigging is a gravity separation process adopted by the minerals industry to separate ore minerals on the basis of particle size and/or density [1]. Jigging units apply pulsed liquid flow dilating the particle bed where particle stratification ensues due to influences of hydrodynamic and gravity forces.

Much of the published research performed in jigging has been experimental [2–11]. Further, commercial jigs date back as far as the Neil Jig (1914) [12]. It is reasonable to assume many jigs historically were designed principally with the aid of experiments and also analytical expressions in the absence of computational capabilities. The past empirical studies develop understanding of how the feed material macroscopically responds to various operating conditions, they do not elucidate on the intricate transient behaviour of the fluid and particles, along with separation kinetics which are important to develop a full understanding of the process.

Modern investigations of jigging phenomena using numerical simulation techniques has shown to be a fast growing area. Solnordal et al. [13], applied a single phase computational fluid dynamic (CFD) technique but this was limited as it treated the slurry as a single phase. Various studies applied discrete element method (DEM) to simulate the motion of individual particles discretely coupled with simplified fluid models giving some insights into micro-mechanical processes at the particulate level [3–5,14–17]. These modelling techniques assume a uniform fluid field and do not account for the effect of non-uniform fluid velocity on the particle drag forces. The Euler–Lagrange (DEM–CFD) model, first proposed by Tsuji et al. [18], remains the most attractive technique because of its superior computational convenience as compared to Direct Numerical Simulation-DEM, or Lattice Boltzmann-DEM models, and the capability to capture the particle physics as compared to DEM-simplified fluid models. The model has been increasingly used to study a wide range of particle fluid systems [19]. The liquid phase flow is solved using the Navier–Stokes and continuity equations, while the motion of individual particles is obtained by solving Newton's second law of motion, with the liquid-particle coupling treated using Newton's third law of motion. This approach can generate detailed information about the trajectories of particles and the transient forces between two particles and between particles and fluid. Other notable models used to investigate jigging include Potential energy [20–22], Potential energy-Monte Carlo [21], Artificial Neural Network (ANN) [23,24], Statistical [25], and Unsteady-fluidisation [26].

* Corresponding author. Tel.: +61 3 9545 8982; fax: +61 3 9545 8669.
E-mail address: Yuqing.Feng@csiro.au (Y. Feng).

0921-8831/$ - see front matter © 2013 The Society of Powder Technology Japan. Published by Elsevier B.V. and The Society of Powder Technology Japan. All rights reserved.
http://dx.doi.org/10.1016/j.apt.2012.11.012

Only a few jigging studies have adopted the DEM–CFD approach [27–30]. Both the studies by Asakura et al. [27] and Xia and Peng [28] are two-way coupled and consider drag on each particle individually, but do not consider porosity. Xia and Peng [28,30] used a 2D column model and implemented forces including virtual mass force, Magnus force [31], and Saffman force [32,33]. One study analysed the importance of different forces acting on a particle in jigging and was performed for multi-sized and binary-sized particles in a sinusoidal pulsion. Additionally, the authors studied the hindered settling velocity as a function of particle densities and sizes, and the effect of sinusoidal pulsation, amplitude and frequency on the particle separation and fluid flows [28]. A separate study used the same model and highlighted that the fluid is highly dynamic and influenced by the presence of particles confirming the simplified idealised flow behaviour as assumed in DEM-simplified fluid models does not exist [30]. Asakura et al. [27] went a step further including the Basset force [34] and a 3D column model which studied the trajectory and response time of a single particle in a jig. Dong et al. [29] applied a one-way coupled 3D model to a close-to realistic geometry Inline Pressure jig. The study considered that fluid flow is the dominant factor in the jig, and implemented a saw-tooth-forward leaning jigging profile investigating vibration frequency and amplitude, and the size and density of ragging particles on the flow separation. However, one-way coupling does not account for the influence of the local particles on the fluid.

Previous studies using the DEM–CFD model have used a sinusoidal pulsation profile with the exception of Dong et al. [29], who used a forward leaning saw tooth cycle. No numerical investigations (including all various modelling techniques) have studied what effect the sinusoidal profile has on concentration mechanics by using two-way coupling in conjunction with a porous drag force model. Further, these studies have not investigated jigging aspects such as separation time, energy, and profile optimisation. The aim of this study is to elucidate how the profile induces segregation, and how variations of frequency and amplitudes affect performance based on a range of criteria.

2. Simulation method

2.1. Governing equations

The DEM–CFD model has been well documented in the literature. For brevity, only the outline of the model structure is described below. The solid phase is treated as a discrete phase and solved using DEM. The translational and rotational motions of a particle at any time, t, in the bed are determined by Newton's second law of motion. These can be written as:

$$m_i \frac{d\mathbf{v}_i}{dt} = \mathbf{f}_{f,i} + \sum_{j=1}^{k_i} (\mathbf{f}_{c,ij} + \mathbf{f}_{d,ij}) + \mathbf{f}_{g,i} \tag{1}$$

and

$$I_i \frac{d\boldsymbol{\omega}_i}{dt} = \sum_{j=1}^{k_i} T_{ij} \tag{2}$$

where m_i, I_i, k_i, $\mathbf{v}_i$ and $\boldsymbol{\omega}_i$ are, respectively, the mass, moment of inertia, number of contacting particles, translational and rotational velocities of particle i, and $\mathbf{f}_{f,i}$, $\mathbf{f}_{g,i}$ are fluid drag force, gravitational force respectively. $\mathbf{f}_{c,ij}$, $\mathbf{f}_{d,ij}$ and $\mathbf{T}_{i,j}$ are the contact force, viscous contact damping force and torque between particles i and j. These inter-particle forces and torques are summed over the k_i particles in contact with particle i.

The particle–particle and particle–wall contact force is based on the soft-sphere method. The particle fluid interaction force is calculated using the Di Felice drag force correlation [35], and Model B formulation is adopted [36]. The liquid phase is treated as a continuous phase moving through a porous medium created by the particles, and is modelled similarly to conventional two fluid models in which porosity (or liquid volume fraction) modifies the standard single phase Navier–Stokes equations. The governing equations are then the conservations of mass and momentum in terms of the local mean variables over a computational cell, given by:

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = 0 \tag{3}$$

and

$$\frac{\partial (\rho_f \varepsilon \mathbf{u})}{\partial t} + \nabla \cdot (\rho_f \varepsilon \mathbf{u}\mathbf{u}) = -\nabla P - \frac{\sum_{i=1}^{k_c} \mathbf{f}_{f,i}}{\Delta V} + \nabla(\varepsilon \tau) + \rho_f \varepsilon \mathbf{g} \tag{4}$$

where ρ_f, $\mathbf{u}$ and P are, respectively, the fluid density, velocity and pressure; τ, ε and ΔV are the fluid viscous stress tensor, porosity and volume of a computational cell.

The particle flow is solved numerically using an in-house DEM code [36] with an explicit time integration method and established geometrical and flow boundary conditions. The continuous liquid phase is readily solved using a commercial CFD software package (ANSYS CFX 10.0). The coupling between DEM and CFD is achieved as follows. At each time step DEM will give information of positions and velocities of individual particles for the evaluation of porosity and volumetric fluid drag force in a computational cell. CFD will then use this data to determine the fluid flow field, which in turn is used to determine the fluid drag forces acting on individual particles. Incorporating the resulting forces into DEM will produce information about the motion of individual particles for the next time step. The fluid drag force acting on an individual particle will react on the fluid phase from the particles, so that Newton's third law of motion is satisfied.

2.2. Simulation conditions

The model consists of a rectangular domain filled with a binary-density spherical particle system and liquid. The particles were divided into 565 light particles and 565 heavy particles with respective densities of 2540 kg/m^3 (glass) and 4630 kg/m^3 (ceramic), the liquid used was water 1000 kg/m^3 in density. Detailed model settings are shown in Table 1. The side walls were treated with no-slip boundary conditions. The bottom was considered as a wall for the particle phase, so they cannot fall through, but as an inlet for liquid. The top exit was treated with a zero normal gradient opening condition. Periodic boundary conditions were applied to the front and rear surfaces of the flow domain effectively creating infinite thickness and economically reducing the number of particles required to produce three dimensional (3D) results. The liquid flow was considered in two dimensions (2D) using only one cell in the thickness direction and hence does not resolve detailed flow fields in this direction, while DEM modelling of the particles was in 3D, with a bed thickness equal to five particle diameters. As all pulsation profiles are studied with a two dimensional model, the results remain acceptable for comparison.

Uniform liquid flow was injected through the inlet and the flowrate varied with time according to the pulsation profile simulated. The inlet flow for the sinusoidal pulsation profile was established using a sinusoidal function. The pulsation profiles are compared by holding the shape of the profile constant and using three variations of period (T) and volumetric water input/exhaust (A). These are 1, 2, and 3 s periods (or 60, 30, and 20 cycles/min), and, 1.5, 2.25 and 3 L water amplitudes. The amplitudes are represented in litres not distance as the water/air free surface is not resolved (i.e. the domain at anytime is completely filled with water), otherwise the amplitudes would be equivalent to 0.2, 0.3 and