Minerals Engineering 21 (2008) 789–796

Contents lists available at ScienceDirect

Minerals Engineering

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

Simulation of liquid–solid flow in a coal distributor

K.J. Dong [a], B.Y. Guo [a], K.W. Chu [a], A.B. Yu [a,*], I. Brake [b]

[a] Center for Simulation and Modelling of Particulate Systems, School of Materials Science and Engineering, University of New South Wales, Sydney, NSW 2052, Australia
[b] Technical Improvement Group, BHP Billiton Mitsubishi Alliance, Moranbah, Qld 4744, Australia

ARTICLE INFO

Article history:
Received 22 December 2007
Accepted 30 May 2008
Available online 15 July 2008

Keywords:
Liquid–solid flow
Coal distributor
Discrete particle simulation
Computational fluid dynamics

ABSTRACT

This paper presents a numerical simulation of the liquid–solid flow in a 12-way coal distributor used in a coal preparation plant. The liquid flow, coupled with gas-flow, is determined by a homogeneous multiphase model by computational fluid dynamics (CFD). A strong swirling and non-uniform flow is obtained. The fluid–solid interaction is determined based on the CFD results. Then, discrete element method (DEM) is employed to calculate the solid flow in the distributor. The effects of some key variables on the distribution of both fluid outlets and solid outlets are studied through the numerical experiments. The particle–fluid interactions are analyzed in order to understand the reasons for the maldistribution of the coal particle phase. It is shown that the asymmetric layout for liquid inlets and other asymmetric factors are responsible for the asymmetric behavior of liquid, which dominates the maldistribution of the particle flow.

1. Introduction

Over the past 30 year, parallel modular operations have been used as a way to increase productivity in most coal preparation plants. Coal distributor was developed to split the raw coal among the subsequent parallel modules. The so-called "sputnik" hydraulic distributor is widely employed to mix run of mine and water in the first stage of coal separation process – almost 40% of Australia's washed coal passes through a sputnik distributor. In the operation, the coal materials carried by a conveyor belt are fed into the distributor through its top, and water is pumped in through a number of tangentially arranged water inlets near the top. Then the resultant coal–water mix is distributed through a number of outlets. Thus the properties of the outlet stream, i.e., water and solids flow rate, particle size and density distribution, are expected to be identical to each other. In practice, however, this happens to varying degrees and the device in some cases performs rather badly. Instances have been observed of mass flow rates between different outlets differing by a factor of three or more (Holtham and Kelly, 1998). Such biased distribution will bring serious problems in the downstream operations.

Few studies on the sputnik distributor have been available in open literature; most work has been confined to unpublished company internal reports (see review in: Holtham and Kelly, 1998; Kelly, 1999). A mathematical modeling and numerical experiments offer a cost-effective approach to the understanding of the internal flow and underlying mechanism for biased distribution. The earlier work of numerical work in this area should be attributed to Yang (1999). He simulated the water–air flow using the volume of fluid (VOF) method in a simple configuration of sputnik distributor (for example, evenly arranged water inlets, fewer device internals). The coal particle flow was simulated by means of Lagrangian approach using one-way coupling. Then the effect of coal particle density and turbulence was considered.

A Lagrangian approach, without considering particle interactions, will produce unrealistic results, particularly when particles accumulate in certain parts of the distributor. Instead, the so-called discrete element method (DEM) (Cundall and Strack, 1979), which considers the contact forces between particles, will overcome this problem. DEM has been proven to be reliable and effective in studying particle flows in various industrial processes (Cleary, 2000; Yu, 2004; Zhu et al., 2007). And it has also been coupled with the computational fluid dynamics (CFD) method to model the particle–fluid flow (Tsuji et al., 1993; Xu and Yu, 1997; Yu and Xu, 2003; Yu, 2004). However, to date, no such effort has been made for coal distributor, although more and more efforts are now made for more complicated flow systems (see, Chu and Yu, 2008, for example).

This paper presents a numerical model on the fluid–solid flow in a full scale coal distributor. A two-phase flow model is used to calculate the water–air flow, while a DEM model is developed to calculate the coal flow in the distributor. The liquid–solid phase interaction is included in the simulation of the solid flow. The model is applied to a typical distributor design in coal plants and the effects of several key operational parameters are analyzed.

* Corresponding author. Tel.: +61 2 93854429; fax: +61 2 93855956.
E-mail address: a.yu@unsw.edu.au (A.B. Yu).

0892-6875/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.mineng.2008.05.014

2. Model description

2.1. Geometry of the distributor

A full scale of a 12-way coal distributor at BHP Billiton Mitsubishi Alliance is studied. Fig. 1 shows schematically the internal geometry of the distributor and numbering of inlets and outlets (the detailed dimensions are not given here for confidential reasons). Coal particles are fed from the top of the upper chamber and discharged through the 12 outlets on the bottom of the lower chamber. In the upper chamber of the current equipment, there are totally eight water inlets at two different levels (A and B). Three water inlets are arranged 120° apart for level A near the top of the distributor, while the five inlets are not evenly oriented (or non-symmetrical) at level B. In the numerical experiments, cases with symmetric inlets, where only four water inlets are arranged 90° apart, are also considered. These tangential inlets will result in a swirling water flow and subsequently enhanced mixing of the coal and water in the upper chamber. In practice, it appears that a water vortex forms in the upper chamber, and the discharged coal passes through the centre of the vortex, encountering little mixing before impacting on the bottom. Hence some modifications have been made, i.e., a number of slots were opened on the orifice plate, and a table splitter was inserted below the coal feed port, so that the coal was directed into the upper chamber water vortex. These modifications have inevitably changed the water flow pattern and would affect the distribution of water flow. Therefore these variations of design should be taken into account in the model.

2.2. Model on fluid flow

Commercial CFD software ANSYS-CFX10 is used for the simulation. To solve the water flow, two phases, i.e., water and air, must be considered altogether, although the effect of the air flow on the coal is negligible. The flow is treated as steady and continuous, and the time-averaged mean velocity and other flow quantities are calculated by using a so-called homogeneous model, which assumes the same velocity for all phases but distinct volume fraction, r_α, for each phase α. The set of steady state fluid flow governing equations are as follows (CFX-user manual):

The continuity equation:

$$\nabla \cdot (r_\alpha \rho_\alpha \vec{\boldsymbol{U}}) = 0 \tag{1}$$

The momentum equation:

$$\nabla \cdot (\rho \vec{\boldsymbol{U}} \otimes \vec{\boldsymbol{U}} - (\mu + \mu_t)(\nabla \vec{\boldsymbol{U}} + (\nabla \vec{\boldsymbol{U}})^{\mathrm{T}}))) = \vec{\boldsymbol{g}} - \nabla p \tag{2}$$

where $\vec{\boldsymbol{U}}$ is the velocity vector; $\vec{\boldsymbol{g}}$ is the gravity vector and p represents the static pressure. The density and laminar viscosity are

$$\rho = \sum_{\alpha=1}^{N_{\mathrm{P}}} r_\alpha \rho_\alpha \quad \text{and} \quad \mu = \sum_{\alpha=1}^{N_{\mathrm{P}}} r_\alpha \mu_\alpha \tag{3}$$

where N_{P} is the total number of phases. A standard k–ε turbulence model is used for its computational robustness, although a second order closure model, such as the Reynolds stress model, is more accurate for such a swirling flow. A compressive differencing scheme is applied to the advection of volume fractions to keep the interface reasonably sharp. For all other flow quantities, a blended discretization scheme between the first order upwind

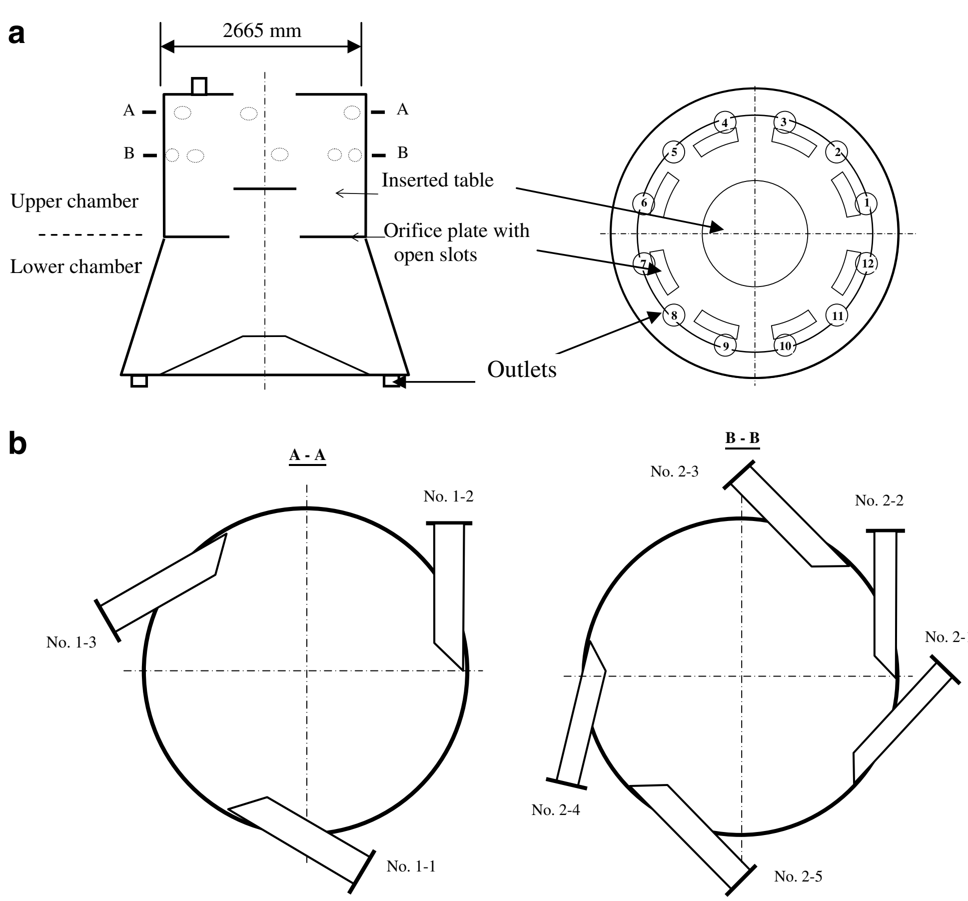

Fig. 1. Schematic illustration of the distributor considered: (a) side and plan view of geometry, internal structure and numbering of outlets; and (b) orientation and numbering of water inlets for both level.