Minerals Engineering 39 (2012) 173–183

Contents lists available at SciVerse ScienceDirect

Minerals Engineering

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

CFD modeling of hydrocyclones: Prediction of particle size segregation

M. Narasimha [a,*], M.S. Brennan [b], P.N. Holtham [b]

[a] Department of Chemical Engineering, IIT Hyderabad, Ordinance Factory Estate, Yeddumailaram 502 205, Andhra Pradesh, India
[b] Julius Kruttschnitt Mineral Research Centre, The University of Queensland, Isles Road, Indooroopilly, 4068 Queensland, Australia

ARTICLE INFO

Article history:
Received 20 February 2012
Accepted 7 May 2012
Available online 5 October 2012

Keywords:
Hydrocyclone
Computational fluid dynamics
Multiphase flow
Particle separation
Sampling probe

ABSTRACT

The flow behavior in a hydrocyclone is a highly swirling and turbulent multiphase structure. A multiphase CFD model with sub modules for the air-core, turbulence, and particle classification with a suitable slurry viscosity model was used to simulate performance of hydrocyclones. The predicted velocity field from the LES, DRSM turbulence models is compared with the LDA measurements data for a 75 mm cyclone. The multiphase CFD model is used to understand the particle size segregation inside a 6 in. hydrocyclone. The predictions are validated against the Renner (1976) data, which was originally measured using high-speed sampling probe at different precisely controlled positions. The overall classification curve predicts close to the experimental data. It is observed that the predicted position sample size distributions are in good agreement with the experimental data, at most of the cyclone sampling positions. Close to the forced vortex (inner position), the predicted size distributions slightly deviate from the measured data. The discrepancy may be an effect of sampling turbulence due to probing close to the unstable forced vortex. Simulations are also carried out using two different CFD models, with and without the viscosity correction due to the fines fraction. The predictions are improved with respect to Renner's data with the fines viscosity corrected CFD model.

1. Introduction

Hydrocyclones are widely used in the mining and chemical industries for the separation of solids or droplets based on their size and density. A typical hydrocyclone consists of a cylindrical section with a central upward flow discharge tube connected to a conical section with a downward flow discharge tube. An inlet conduit is attached tangentially to the top section of the cylinder. The fluid being injected tangentially into hydrocyclone causes swirling and thus generates centrifugal force within the device. This centrifugal force field brings about a rapid classification by size of the particulates suspended within the fluid.

The flow in a hydroyclone is a multiphase structure which consists of solid particles which are dispersed throughout the fluid, generally water. In addition, an air core is present. Such multiphase flows can be studied using a number of Computational Fluid Dynamics (CFD) techniques. These include the full Eulerian Multiphase approach, simplified Eulerian approaches such as the Mixture (Manninen et al., 1996) and Volume of Fluid (VOF) models (Hirt and Nichols, 1981) and the Lagrangian approach (Crowe et al., 1998).

Most of the previous numerical studies which have adopted the Lagrangian frame were not comprehensive. They only include the drag and centrifugal forces in the calculation of the particle trajectory, with or without particle dispersion effects (Hsieh, 1988; Hsieh and Rajamani, 1991; He et al., 1999; Rajamani and Millin, 1992; Boysan et al., 1982; Griffiths and Boysan, 1996). Also, these studies are limited to very dilute particle concentrations in cyclones. The Lagrangian approach has been extended to modeling cyclones at large particle concentrations by Rajamani and Millin (1992) and Devulapalli (1996). They couple the effect of solid concentration with fluid viscosity but were limited to Prandtl-mixing turbulence models. A similar model (Rajamani and Millin, 1992) has been reinvestigated using LES turbulence model by Delgadillo and Rajamani (2005). They found that the prediction of particle classification follows closely the experimental values at low feed solids, whereas the predictions for the high feed solid concentrations overestimate the mean coarse size particle classification when compared to the experimental classification data in a larger cyclone.

There have been few studies where various cyclones have been modelled using a full Eulerian approach in conjunction to RANS turbulence models (Suasnabar, 2000; Nowakowski et al., 2004, Nowakowski and Dyakowski, 2003; Cokljat and Slack, 2003; Huang, 2005). All these studies have used the Fluent based full Eulerian models while simulating dilute solids flow in cyclones. The disadvantage of the full Eulerian multiphase modeling has been its high computational cost. Further implementations in commercial CFD codes have until recently been limited to using the *k*-epsilon/RSM models for turbulence.

* Corresponding author. Tel.: +91 40 2301 6086; fax: +91 40 2301 6003.
E-mail address: narasimha@iith.ac.in (M. Narasimha).

0892-6875/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.mineng.2012.05.010

Nomenclature

C_{lp}	lift coefficient
d_k	diameter of phase k (m)
f_{rep}	drag coefficient
g_i	i component of gravity (m/s^2)
k	turbulent kinetic energy (m^2/s^2)
p	pressure (pa)
p_k	granular pressure of phase k
r	radial position (m)
Re_p	particle Reynolds number
u_i	i component of mixture velocity (m/s)
$u_{kc,i}$	i component of velocity of phase k relative to mixture (drift velocity) (m/s)
$u_{km,i}$	i component of velocity of phase k relative to continuous phase (slip velocity) (m/s)
u'_i	i turbulent fluctuating component (m/s)
x_i	i co-ordinate
ρ_p	particle density (kg/m^3)
$\tau_{d,ij}$	drift stress tensor of mixture
$\tau_{s,ij}$	turbulent or sub grid scale stress tensor of mixture
$\tau_{\mu,ij}$	viscous stress tensor of mixture
μ_m	effective molecular viscosity of the mixture
ε	turbulent dissipation rate (m^2/s^3)
d_{50}	actual cut size (classification)
v	characteristic fluid velocity in a cyclone (subscripts: t = tangential, r = radial, z = axial)
μ	fluid viscosity (subscripts: m = slurry, c = liquid)
ρ	fluid density (subscripts: p = pulp or m = slurry or suspension, l = liquid, s = solid)

Subscripts

c	continuous phase c
i, j	components in i and j directions
k	phase k
m	mixture

Initially Pericleous and Rhodes (1986) and Davidson (1994) coupled the particle and fluid equations by modifying the mixture density and effective viscosity using an algebraic slip mixture model for 2D hydrocyclone simulations. Suasnabar (2000) and Brennan (2003) studies have adopted the mixture model (Manninen et al. (1996)) for dense medium cyclone simulations in which the mono average size of dispersive phase was considered as particulate phases and the particle classification mechanism was explained qualitatively.

There have been numerous studies where hydrocyclones have been modelled using modified algebraic slip mixture (ASM) model by the author Narasimha (2010). This CFD work has been validated by either Laser Doppler Anemometry (LDA), conducted on a water flows in clear Perspex models (Brennan, 2006; Delgadillo and Rajamani, 2005; Narasimha et al., 2006), or Gamma Ray Tomography (GRT) measurements of density profiles in a plastic cyclone (Narasimha et al., 2007). Whilst both LDA and GRT have generated useful data for validation, they are laboratory techniques which investigate the internal flow structure and have been primarily used on small cyclones. LDA and GRT do not provide any information on how the particles are distributed inside the cyclone by size. Recent numerical studies on various cyclone geometries also successfully used multi-fluid models (Rajamani et al., 2010; Hsu and Wu, 2010; Wang and Yu, 2010; Brennan et al., 2009; Davailles et al., 2012). ASM model was successfully used to model the multiphase flows in dense medium cyclones (Wang et al., 2009, 2011; Chu et al., 2009a,b, 2012), in particularly simulating the magnetite segregation levels inside the dense medium cyclone.

In this paper the multiphase CFD model validation is attempted against the Renner's (1976) data on particle size segregation inside a hydrocyclone. Renner's data was measured using a high-speed sampling probe at different precisely controlled positions (Renner, 1976; Renner and Cohan, 1978). The work compares the particle size distributions predicted by CFD model to Renner's data. This work also looks at the reliability of both the sampling probe and the multiphase CFD model for hydrocyclones.

2. Model description

2.1. Cyclone geometries and grid generation

The simulations use Fluent with 3-D body fitted grids which were generated in Gambit (a pre-processor for FLUENT). The grids typically have the size between 120,000 and 200,000 nodes. The feed port is a velocity inlet boundary condition and the overflow and underflow are pressure outlet boundary conditions. All other boundary conditions are no slip at the wall. Two geometries are selected for the CFD study, which were a Hsieh75 mm hydrocyclone, a Renner (1976) 150 mm cyclone. The key dimensions of the Hsieh and Renner cyclones, and the particle size sampling positions in the Renner cyclone are shown in Fig. 1.

2.2. Turbulence models

The equations of motion are solved with the unsteady solver for the slurry mixture using the Large Eddy Simulation (LES):

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial \rho_m u_{mi}}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho_m u_{mi}) + \frac{\partial}{\partial x_j}(\rho_m u_{mi} u_{mj}) = -\frac{\partial}{\partial x_i} p + \frac{\partial}{\partial x_j}(\tau_{\mu,ij} + \tau_{d,ij} + \tau_{t,ij}) + \rho_m g_i \tag{2}$$

In this paper, the equations use cartesian tensor notation: u_{mi} is the ith component of slurry velocity vector, $\tau_{t,ij}$ denotes the sub grid scale stresses—which are solved with the Smagorinsky (1963) SGS model, and $\tau_{d,ij}$ is the drift tensor which arises in Eq. (2) as part of the derivation of the Mixture model (Manninen et al., 1996). The drift tensor accounts for the transport of momentum as the result of segregation of the dispersed phases, and is an exact term:

$$\tau_{d,ij} = \sum_{p=1}^{n} \alpha_p \rho_p u_{pm,i} u_{pm,j} \tag{3}$$

2.3. Multiphase modeling – Mixture model with lift forces and rheological inputs

The solids are treated using the Mixture model (Manninen et al., 1996), which is a simplified Eulerian multiphase CFD methodology, where the equations of motion are solved only for the slurry mixture. The slurry is assumed to consist of a primary fluid phase (denoted by c) and a secondary dispersed particulate phases (denoted by p). The Mixture model solves the transport equations for the volume fractions of these dispersed phases α_p: