Minerals Engineering 31 (2012) 32–41

Contents lists available at SciVerse ScienceDirect

Minerals Engineering

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

Analysis of swirling flow in hydrocyclones operating under dense regime

Aurélien Davailles [a,b], Eric Climent [a,b,*], Florent Bourgeois [b,c], Arun Kumar Majumder [d]

[a] Université de Toulouse, INPT, UPS, IMFT (Institut de Mécanique des Fluides de Toulouse), Allée Camille Soula, F-31400 Toulouse, France
[b] CNRS, Fédération de recherche FERMAT, Toulouse, France
[c] Université de Toulouse, INPT, UPS, LGC (Laboratoire de Génie Chimique), 4 Allée Emile Monso, BP 44362, 31432 Toulouse Cedex 4, France
[d] Department of Mining Engineering, Indian Institute of Technology, Kharagpur 721 302, India

ARTICLE INFO

Article history:
Available online 22 February 2012

Keywords:
Hydrocyclone
CFD
Solids concentration
Roping

ABSTRACT

There are many circumstances where hydrocyclone performance and dense flow are intertwined, such as for example when feed solids flow exceeds hydrocyclone capacity during continuous operations. The work reported here, which is part of an ongoing research effort to develop a robust CFD model for prediction of hydrocyclone performance, focuses on hydrocyclone operation under high solids concentration. The paper presents the basic physics framework that accounts for solid–liquid and solid–solid interactions under hydrocyclone's swirling flow. Operating conditions that are past the transition from spray to rope regime are deliberately chosen for this purpose. Model predictions are validated by comparison with solids split and separation curves measured on a 100 mm diameter hydrocyclone. CFD model predictions permit taking an insightful look at the inside of a hydrocyclone under extreme operating conditions, which would be difficult to achieve experimentally. Velocity profiles, G-force distribution and distribution of solids predicted by CFD are bound to lead to a better understanding of the separation that takes place inside a hydrocyclone, which may eventually help improve hydrocyclone design and performance.

1. Introduction

There are many circumstances where hydrocyclone performance and dense flow are intertwined, such as for example when feed solids flow exceeds hydrocyclone capacity during continuous operations. Moreover, under normal operating conditions, it is generally perceived that some regions inside a hydrocyclone might be very densely populated with solids while other regions might only experience dilute conditions. Dilute conditions are expected in the central part of the hydrocyclone and in the cylindrical section of the hydrocyclone, whereas dense conditions, as dense as packed beds in some instances, can be expected along the hydrocyclone walls as well as in the conical region. With the above in mind, solid–solid separation inside a hydrocyclone is bound to vary significantly throughout the body of an operating hydrocyclone. This reasoning emphasizes the need to better understand particle and water flow behavior inside a hydrocyclone under high solids concentration, whether dense regions occur locally or spread throughout the hydrocyclone body. In an ongoing research effort, therefore, an attempt has been made to develop a robust CFD model for the prediction of hydrocyclone performances at different operating conditions. The physics of particle separation behavior inside a hydrocyclone at dilute conditions and the CFD simulation have already been described elsewhere (Davailles et al., in press). The present article, however, deals with the CFD modeling of a hydrocyclone at high solids concentrations.

1.1. Experimental studies

The hydrodynamic behavior of a hydrocyclone depends on inlet solid concentration, feed flow rate and geometry of the device. Fig. 1 presents two distinct behaviors, below and above the transition to roping (Neesse et al., 2004a). Under those two operating regimes, visualization of solid–liquid flow patterns is difficult as the accumulation of particles along the wall highly complicates the optical access to the hydrocyclone's core. Usual techniques which have been applied to study single phase flow, such as Laser Doppler Velocimetry (Hsieh, 1988), are not applicable for concentrated systems. Much more complex measurement tools, such as X-ray tomography (Galvin and Smitham, 1994) or radioactive particle tracking (Chang et al., 2011), have, therefore, been tried. However, it is still very challenging to gain new experimental insights into local hydrodynamics of dense solid–liquid flows. At low feed solids concentration, the underflow discharge of a hydrocyclone has a spray profile and a stable air core is established due to the suction of air through the underflow (Fig. 1a). However, with increasing feed solids concentration the quantity of solids exiting through

* Corresponding author at: Université de Toulouse, INPT, UPS, IMFT (Institut de Mécanique des Fluides de Toulouse), Allée Camille Soula, F-31400 Toulouse, France.
E-mail addresses: aurelien.davailles@imft.fr (A. Davailles), eric.climent@imft.fr (E. Climent), florent.bourgeois@ensiacet.fr (F. Bourgeois), majumder1965@gmail.com (A.K. Majumder).

0892-6875/$ - see front matter
doi:10.1016/j.mineng.2012.01.012

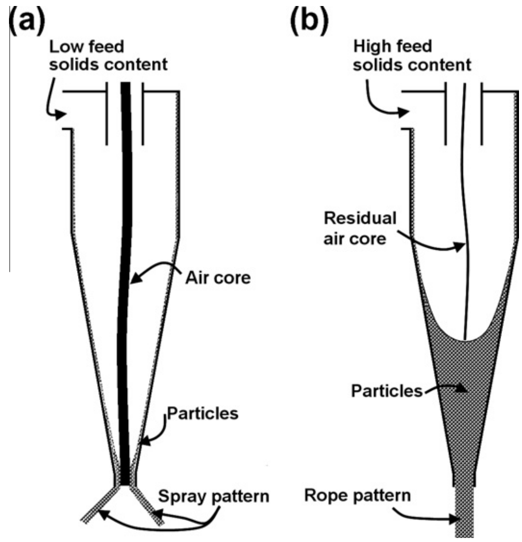

Fig. 1. Hydrocyclone: (a) dilute and (b) dense slurries.

the underflow discharge also increases and above a critical point (Neesse et al., 2004b) the rate of accumulation of solids in the conical portion exceeds the rate of discharge. As a result the spray profile collapses and the air suction is also hindered. This condition is generally termed as roping (Fig. 1b). Neesse and Dueck (2007) opined that the air core cannot totally disappear while the exit is plugged by particles. An air core remains within the hydrocyclone and vanishes on the particles bed. However, the small size of the air core and the absence of connection with the atmosphere severely limit its influence on the hydrodynamics for this operating regime.

1.2. Numerical simulations and the objectives

CFD studies on hydrocyclones generally aim at giving new insights on local flow patterns, such as velocity profiles, pressure distribution and spatial distribution of solids. These information are difficult to access through experimentation and are critical for better understanding of separation performances.

However, numerical modeling of all physical aspects still remains a challenging research topic due to the complexities associated with the swirling flow behavior of concentrated slurries within a hydrocyclone. The geometry is not an obvious one and contributes significantly to flow streams repartition between the two exits. The level of turbulence is high corresponding to a hierarchy of spatial flow structures interacting selectively with the particles. Due to strong swirling, the anisotropy of the flow couples particle centrifugation and triggers air focusing in the core of the device. The intrinsic three phase (gas–liquid–solid) flow with a large amount of particles make hydrocyclones a real challenge for CFD codes.

Large Eddy Simulation (LES) appears to be the most promising approach for modeling flow turbulence under dilute feed flow rate (Slack et al., 2000; Delgadillo and Rajamani, 2005) as it preserves turbulence spatial structure. For particle laden flows, LES can be coupled with Discrete Particle Simulation. Without momentum coupling between the solid phase and the liquid phase, this permits to evaluate separation curves under dilute regime based on trajectory analysis. Unfortunately, this technique is very time consuming and the relevance of the model for concentrated slurries needs further theoretical developments and validation tests on academic configurations. This approach is still reserved for small scale study and only thousands of particles, while the work that is presented here is concerned with the simulation of industrial size hydrocyclones and dense suspensions. This implies a degree of compromise between the accuracy of the physics and the computational cost.

Processes at both the macroscopic and local scales have some influences on hydrocyclone's behavior; hence both scales must be embedded in models for making prediction of hydrocyclone performance. Quite a few experimental and numerical studies helped build a strong understanding of single-phase flow inside hydrocyclones (Kelsall, 1952; Hsieh and Rajamani, 1991; Brennan, 2006).

In contrast, the sensitivity of separation performance to feed solids concentration possibly has not been studied so far and, therefore, remains unexplained. One reason for this situation is that the measurement of slurry velocity profiles or solids spatial distribution anywhere inside the hydrocyclone is an experimentally challenging task.

It is comparatively simple to obtain such information from CFD simulations based on the understanding of swirling flow patterns inside a hydrocyclone. The physics and the CFD modeling dealing with dilute suspensions was validated with our experimental data as well as with a set of published data (Davailles et al., in press). The present study focuses on the analysis of swirling flow in hydrocyclones operating under dense flow regimes having the following objectives:

- Development of a modeling scheme based on the strong foundation of physics of fluid flow and particle–fluid interactions.
- Simulation of the roping conditions in a 100 mm diameter hydrocyclone using an Eulerian multi fluid model with RANS turbulence modeling.
- Validation of the model predicted data with experimental data.

It is not within the scope of this paper to investigate the transition between spray and rope discharge despite the occurrence of this transition is an important issue from an operational standpoint. Indeed, the issue at stake here is that of the effect of dense suspension on swirling patterns and particle separation inside the hydrocyclone. This yields the choice of operating conditions used throughout this work.

2. Description of CFD model

The three dimensional numerical simulations were performed with NEPTUNE_CFD v1.08@Tlse (Laviéville and Simonin, 1999, 2006; Ozel et al., 2010; Galassi et al., 2009). NEPTUNE_CFD is a multiphase flow simulation software developed within the framework of a French industrial consortium.

The behavior of dispersed multiphase flow is modeled using the general Eulerian multi-fields balance equations which are obtained from the fundamental conservation laws of physics (mass conservation and momentum balance), restricted to Newtonian mechanics. Conservation laws are written under differential form that are valid at any time and location within the continuum, except across interfaces between two physical phases. At interfaces, jump conditions derived from continuous equations are written and integrated through source and sink terms in the equations. This results in a fully coupled set of equations.

2.1. Transport equations

The multiphase Eulerian model is built on a kinetic approach based on a joint fluid-particle Probability Density Function (Simonin, 1996). The set of PDF's moment transport equations (mass, momentum, particle agitation) is derived from a Boltzmann-like kinetic equation of the PDF.