Available online at www.sciencedirect.com
ScienceDirect
Trans. Nonferrous Met. Soc. China 24(2014) 2642−2649
Transactions of Nonferrous Metals Society of China
www.tnmsc.cn

Numerical and experimental studies of flow field in hydrocyclone with air core

Bao-yu CUI[1], De-zhou WEI[1], Shu-ling GAO[1], Wen-gang LIU[1], Yu-qing FENG[2]

1. College of Resources & Civil Engineering, Northeastern University, Shenyang 110819, China;
2. CSIRO Computational Informatics, VIC 3169, Australia

Received 24 July 2013; accepted 26 December 2013

Abstract: For the flow field in a *d*50 mm hydrocyclone, numerical studies based on computational fluid dynamics (CFD) simulation and experimental studies based on particle image velocimetry (PIV) measurement were carried out respectively. The results of two methods show that air core generally forms after 0.7 s, the similar characteristics of air core can be observed. Vortexes and axial velocity distributions obtained by numerical and experimental methods are also in good agreement. Studies of different parameters based on CFD simulation show that tangential velocity distribution inside the hydrocyclone can be regarded as a combined vortex. Axial and tangential velocities increase as the feed rate increases. The enlargement of cone angle and overflow outlet diameter can speed up the overflow discharge rate. The change of underflow outlet diameter has no significant effect on axial and tangential velocities.

Key words: hydrocyclone; computational fluid dynamics; particle image velocimetry; flow field; air core

1 Introduction

Hydrocyclone has been widely used in many areas of separation due to the advantages of its low operating cost and maintenance, high throughput, low floor space requirement etc [1,2]. Multiple investigations on hydrocyclones have been carried out by several researchers since the 1950s [3]. Now, a commonly accepted wisdom is that the separation characteristics of the hydrocyclone including classification efficiency and cut size depend on the flow field, which has been a hot area of hydrocyclone research in recent years [1−4].

The flow field in a hydrocyclone, which is characterized by a complex three dimensional swirling flow with an air core, is decided by various operational and structural parameters. Operational parameters mainly include feed rate and slurry concentration; whilst structural parameters are composed of feed inlet form, vortex finder form, overflow outlet diameter, underflow outlet diameter and cone angle, etc. The detailed velocity profiles in a hydrocyclone were measured first by KELSALL using a stroboscope [3]. With the development of laser and high-speed camera technologies, laser Doppler velocimetry (LDV) and particle image velocimetry (PIV) were utilized to investigate the flow field inside hydrocyclones [5,6]. DABIR and PETTY [7] measured the tangential and axial velocities in hydrocyclones using LDV as early as 1984, and the tangential and axial velocities were detected by RAJAMANI, FISHER, ELDIN et al individually using LDV or PIV [8−11]. CFD simulation becomes an important numerical method for the detailed study of the flow field of hydrocyclones due to the rapid development of computer technology and mathematical models [12,13]. The first successful CFD predicting work on hydrocyclones was reported by RODES et al in 1987 [14]. Nowadays, the CFD modeling software packages, including ANSYS/Fluent, have shown sufficient accuracy in modeling the flow field in a hydrocyclone, and the simulation results have been compared with the LDV or PIV experimental measurement with good agreement [12−15].

However, most of the related numerical and experimental studies were about the flow velocities of the hydrocyclone with constant structural parameters. The air core, as an inevitable phenomenon of the flow field inside hydrocyclones, was involved rarely in similar literatures. Additionally, there are no general rules that can be used to quantitatively predict the performance for

Foundation item: Projects (50974033, 51104035) supported by the National Natural Science Foundation of China
Corresponding author: Bao-yu CUI; Tel/Fax: +86-24-83692711; E-mail: cuibaoyu@mail.neu.edu.cn
DOI: 10.1016/S1003-6326(14)63394-X

a specific hydrocyclone. Numerical simulation and validation experiment are still required to assess the relative performance of a new designed hydrocyclone. In this work, a *d*50 mm hydrocyclone was studied. Numerical studies on the flow field with an air core inside the hydrocyclone were carried out by the use of ANSYS/Fluent with the Reynolds stress model (RSM) for the turbulence calculation and the volume of fluid (VOF) model for capturing the air-liquid interface. PIV measurements were conducted at the same condition to validate the simulation results. Numerical simulation results were compared with the experiments in terms of the forming process and characteristics of the air core, the flow velocities characteristics. Following model validation, the numerical method was extended to study the effect of different operational and structural parameters on flow field inside the hydrocyclone.

2 Model description

2.1 Hydrocyclone geometry

The geometry of the hydrocyclone used for numerical and experimental models in this study is presented in Fig. 1, which is a typical hydrocyclone type with the specific dimensions for separating mineral particles.

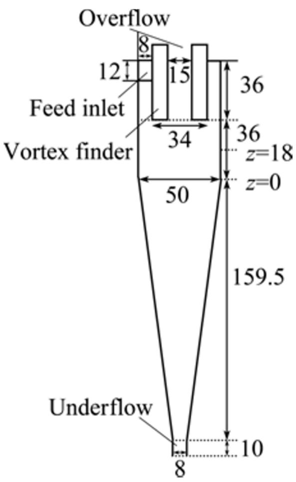

Fig. 1 Geometry of hydrocyclone of base case (unit: mm)

Table 1 lists the parameters investigated in this study, of which air core investigation and flow structures are conducted based on base case; the variable parameters, including the feed rate, cone angle, underflow outlet diameter and overflow outlet diameter, are investigated to study the influence of different parameters on flow field inside the hydrocyclone. For experimental model, due to the cost to change the structural parameters, the investigation was focused on one set of structural parameter, referred as base case. For model validation purpose and for convenience of measurement, the study was conducted using an air−water system. As an initial phase of study, no particles are injected. Influence of particles on the flow field will be discussed in the other paper.

Table 1 Parameters investigated in this study

Parameter	Base case	Variable
Feed rate/(L·min^{-1})	35	30, 40, 45
Cone angle/(°)	15	11, 19, 23
Underflow outlet diameter/mm	8	4, 6, 10
Overflow outlet diameter/mm	15	12, 18

2.2 Numerical model

Simulation of complex hydrodynamics in a hydrocyclone requires accounting for the irregular interface between air core and surrounding liquid, as well as the turbulence induced by the strong swirling flow. This has been a challenging task in the numerical modeling of multiphase flow over the past few decades.

In this study, details of the modeling approach are well documented in literature and recently widely used in the study of hydrocyclones [12−14,16]. Here, only the key governing equations are briefly provided.

For incompressible fluid, the equations for mass (or continuity) and the momentum in a general form are as follows [16]:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{1}$$

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\rho\overline{u_i u_j}\right]+\rho g_i \tag{2}$$

The RSM model uses the following transport equation for the Reynolds stress:

$$\left(\frac{\partial \overline{u_i'\partial u_j'}}{\partial t}+u_k\frac{\partial \overline{u_i' u_j'}}{\partial x_k}\right)=\frac{\partial}{\partial x_k}\left(\frac{\nu_t}{\sigma_k}\frac{\partial \overline{u_i'\partial u_j'}}{\partial x_k}\right)+P_{ij}+\phi_{ij}-\varepsilon_{ij}+R_{ij}+S_{ij}+D_{ij} \tag{3}$$

For VOF model, the governing equation for the phase *i* can be written as

$$\frac{\partial \alpha_i}{\partial t}+u\frac{\partial \alpha_i}{\partial x}=0 \tag{4}$$

Mass flow inlet is set and assumed to be constant with a specified feed rate. The volume fraction of water fed from inlet is 1. Pressure outlets are set for overflow and underflow and assumed to be at atmospheric or zero gauge pressure. No-slip boundary conditions are applied