Available online at www.sciencedirect.com
SCIENCE DIRECT
ELSEVIER
Powder Technology 165 (2006) 30 – 38
POWDER TECHNOLOGY
www.elsevier.com/locate/powtec

Effect of the cylinder shape of a long-coned cyclone on the stable flow-field establishment

Jin W. Lee *, Hoe J. Yang, Dong Y. Lee

Division of Mechanical and Industrial Engineering, Pohang University of Science and Technology (POSTECH), Hyoja 31, Pohang, Kyungbuk, 790-784, South Korea

Received 23 July 2004; received in revised form 15 November 2005; accepted 22 March 2006
Available online 15 April 2006

Abstract

The flow characteristics and particle collection efficiencies of a long-cone cyclone with twice as long a conical section as that of the Stairmand cyclone were analyzed numerically to see the effect of the cylinder shape on the flow and collection performance. The three-dimensional flow field is obtained using the commercial package, FLUENT, and particle paths are calculated with the Lagrangean integration of the particle equation of motion. A number of turbulence models were tested, and it was confirmed that only the second-order Reynolds stress model gave reasonable results for the flow velocity profile.

It is also shown that the shape of the core-annulus interface is important for the overall flow and collection characteristics, and the interface shape is in turn strongly affected by the distance from the bottom of the inlet to the cylinder-cone junction. By adjusting the diameters of the cylinder and the view finder with the mass flux and inlet area kept constant, the shape of the core-annulus interface could be changed. As a result, the short-circuiting of the inlet flow to the view finder and the total pressure drop could be reduced by properly adjusting the cylinder body diameter.

Keywords: Long-cone cyclone; CFD; RSM; Short circuiting flow; Overall efficiency

1. Introduction

Cyclones are widely used for separating particles from dust-laden gas flows in many industries, since their shape and operation are simple, not influenced much by the differing environmental conditions. Though the operating principle is very simple, flow and collection characteristics of a cyclone are very complicated, and the performance characteristics may change sensitively to the design or operating parameters. The Stairmand cyclone with a particular geometric shape is considered the most optimized design, where the velocity profile is well maintained similar all over the axial stations and the collection efficiency very high [1].

In industrial processes, it is not always possible to accommodate such an optimized design, and a case-by-case design optimization has to be tried. In this study, a cyclone whose shape is much different from the Stairmand design is studied for a possible optimization. It is installed inside a fluidized-bed reactor to collect rather coarse particles, and due to the constraints coming from the reactor design, the cyclone had to be designed to have a very long cone. Then the remaining problem to be solved is to find out optimal dimensions for the cylindrical body and the inlet. The long-cone cyclone of interest has the length-to-diameter ratio of about 6.0, implying that it has about twice as long a conical section as the Stairmand design having the length-to-diameter ratio of 4.0.

The three-dimensional flow field and particle collection efficiencies were analyzed with the commercial package, FLUENT, a numerical solver. The flow field in a cyclone is very complex due to various factors, such as re-circulation, highly swirling velocity profile and turbulent eddies, etc. To verify the appropriateness of the numerical methods for analyzing the complex swirling turbulent flow, a Stairmand cyclone was used as a test model, and the calculated results for various velocity components were compared with experimental results obtained with the backscattering Laser Doppler

* Corresponding author. Tel.: +82 54 279 2967; fax: +82 54 279 3199.
E-mail address: jwlee@postech.ac.kr (J.W. Lee).

0032-5910/$ - see front matter © 2006 Elsevier B.V. All rights reserved.
doi:10.1016/j.powtec.2006.03.011

Nomenclature

a	cyclone inlet height, m
b	cyclone inlet width, m
B	cyclone dust outlet diameter, m
D	cyclone body diameter, m
D_e	cyclone gas outlet diameter, m
d_p	particle diameter, μm
d_{50}	cut diameter of particle which is collected with 50% efficiency
H	cyclone height
h	height of cylindrical body of cyclone
h_e	length of outlet duct
k_p	turbulence kinetic energy at point p
r	radial dimension, m
r_o	cyclone body radius, m
Re	Reynolds number, ud/μ
Re_p	particle Reynolds number, $\rho_f \lvert \vec{u}_p - \vec{V} \rvert d_p / \mu$
T	absolute temperature, K
u_p	particle velocity, m/s
V_a	axial component of gas velocity in cyclone, m/s
V_{in}	inlet gas velocity, m/s
V_t	tangential component of gas velocity in cyclone, m/s
V_{max}	maximum tangential component of gas velocity in cyclone, m/s
$\vec{V}$	air velocity, m/s
y_p	distance from point p to the wall

Greek letters

η	collection efficiency
μ	dynamic viscosity
ν	kinematic viscosity
κ	turbulence energy
ρ_f	gas density
ρ_p	particle density

Anemometry [2]. To obtain reasonable results for the complex flow field in a cyclone, an analysis of the turbulence is important [3–9] and also a good model for considering the turbulent boundary-layer characteristics of cyclones has been developed [10,11]. A variety of turbulence models were tested, and the second order Reynolds stress turbulent model only was confirmed to give reasonable results. Particle trajectories were calculated by the Lagrangean integration using the fourth order Runge-Kutta method. The collection efficiencies calculated with these methods were compared with the existing experimental data [12] and numerical results [13,4], showing a reasonable agreement.

When the verified numerical techniques were applied to the analysis of the flow fields of the long-cone cyclone, the shape of the core-annulus interface clearly showed that the long-cone cyclone has an unstable flow field structure, which severely affected the overall collection efficiency. Through a variety of trials it became evident that the key factor for the unstable flow structure is the distance between the bottom of the inlet and the cylinder-cone junction, and ways of attaining a stable flow field was sought with no extra pressure drop.

2. CFD approach

2.1. Numerical method

Numerical simulations were conducted with FLUENT, a commercial package, which provides comprehensive modeling capabilities for wide range of incompressible and compressible, laminar and turbulent fluid flow problems. FLUENT solves conservation equations for mass and momentum simultaneously. Additional transport equations are also solved when the flow is turbulent. The PRESTO! (PREssure STaggering Option) scheme, which uses the discrete continuity balance for a “staggered” control volume about the face to compute the “staggered” pressure, is used for discretization of pressure. Pressure-velocity coupling is achieved by using SIMPLEC (Semi-Implicit Method Pressure-Linked Equations Consistent)