Separation and Purification Technology 184 (2017) 54–65

Contents lists available at ScienceDirect

Separation and Purification Technology

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

An alternative for the collection of small particles in cyclones: Experimental analysis and CFD modeling

E. Balestrin, R.K. Decker, D. Noriler, J.C.S.C. Bastos, H.F. Meier [*]

Chemical Engineering Department, University of Blumenau, Blumenau, Santa Catarina, Brazil

ARTICLE INFO

Article history:
Received 9 November 2016
Received in revised form 15 March 2017
Accepted 18 April 2017
Available online 21 April 2017

Keywords:
Cyclone
PIV measurements
Computational fluid dynamics
Small particles collection

ABSTRACT

The collection efficiency of gas-solid flows in cyclones applied to fine particles is of fundamental importance. In this context, the effects of a reduction in the cross section of the vortex finder outlet duct together with a stretched cylindrical body on the flow pattern and performance of a conventional cyclone was investigated. A test facility and CFD-based modeling using an Eulerian-Lagrangian approach with the Reynolds stress turbulence model (RSM) were employed in experimental and numerical studies, respectively. Based on the results, an alternative design for a cyclone aimed at capturing fine particles (<5 μm) is proposed, specifically, a reduced area at the end of the vortex finder acting as a post vortex finder device. Experimental results obtained by stereoscopy particle image velocimetry (stereo PIV) for the velocity fields of particles with small Stokes numbers were used for the model validation. Simulations were performed by computational fluid dynamics (CFD) for different inlet velocities (10.5 and 12.25 m/s) and flow configurations, and a comparison between experimental data for the pressure drop and velocity profiles indicated good agreement, validating the CFD modeling. The application of Eulerian-Lagrangian modeling to particles of less than 5 μm in diameter showed that the collection efficiency increases with a secondary swirling flow promoted by a reduction in the cross section of the vortex finder. Thus, this design represents a promising alternative for new cyclone applications.

1. Introduction

The cyclone is a classic industrial apparatus used to separate the streams of a gas-solid mixture applying a swirling turbulent flow (centrifugal force). The main advantages are low cost, simplicity in construction and adaptability to a wide range of operating conditions [1–5]. Thus, many investigations have been carried out in order to understand the effect of different geometrical and operating parameters on the performance and fluid dynamics of cyclones [4].

There has been renewed interest in performance studies in response to demands for higher efficiency regarding industrial emissions, which has led to a need to achieve higher particle separation efficiency [6].

Although the cyclone geometry is apparently simple, the swirling flow is not. Thus, as new applications have arisen in modern plants, cyclone separators have moved from low to medium/high-technology equipment [1].

Consequently, complex gas-solid flow patterns have been the subject of many experimental and theoretical studies [1] and a great number of correlations to predict the efficiency and pressure drop of cyclone separators have been proposed [7]. Ogawa [8], for instance, published a review on the conventional cyclones and, in particular, discussed mechanical separation, flow patterns and the significant effects of the vortex finder and parameter dimensions, as mentioned by Elsayed [1].

In order to achieve design and structural improvements, quantitative analysis of the cyclone parameters is required. Once the three-dimensional flow is fully understood, the cyclone performance, in terms of the pressure drop and collection efficiency, can be improved by optimizing the cyclone design [9]. Experimental work has been carried out in order to explain the flow characteristics based on the data obtained and theoretical models have advanced the prediction of the basic features of the flow field, mostly on a semi-empirical basis [9]. Computational advances have allowed the simulation of the complex internal flow in cyclones without resorting to experimental studies. However, experimental data are still needed to validate the numerical models; recently the emphasis has been on specific parameters as well as general performance [10].

Pressure fields, velocity and turbulence, are essentials factors to be considered in these numerical models and solutions are only

---

[*] Corresponding author.
E-mail address: meier@furb.br (H.F. Meier).

http://dx.doi.org/10.1016/j.seppur.2017.04.023
1383-5866/© 2017 Elsevier B.V. All rights reserved.

Nomenclature

d	diameter [m]
r	refinement rate
q	algorithm accuracy formal order
F_s	safety factor
p	pressure [kg/ms^2]
k	turbulent kinetic energy [m^2/s^2]
$\mathbf{v}$	velocity [m/s]
t	time [s]
$\mathbf{x}$	position vector [m]
$\mathbf{g}$	gravitational acceleration [m/s^2]
Re	Reynolds number
C_D	drag coefficient
$\mathbf{D_T}$	turbulent diffusion tensor [kg/ms^3]
$\mathbf{D_M}$	molecular diffusion tensor [kg/ms^3]
C_μ	turbulence model constant
$C_{\varepsilon 1}$	turbulence model constant
$C_{\varepsilon 2}$	turbulence model constant
L20	measurement position, 20 mm below vortex finder
L40	measurement position, 40 mm below vortex finder
L60	measurement position, 60 mm below vortex finder.

Greek letters

Δp	pressure drop [Pa]
ν	molecular kinematic viscosity [m^2/s]
ν^t	turbulent kinematic viscosity [m^2/s]
μ	molecular viscosity [kg m/s]
μ^t	turbulent viscosity [kg m/s]
ρ	density [kg/m^3]
ε	dissipation rate of turbulent kinetic energy [m^2/s^3]
Υ^*	turbulence production [m^2/s^3]
$\boldsymbol{\delta}$	unit tensor
$\mathbf{T}^{eff}$	effective tensor [kg/ms^2]
$\mathbf{T}^t$	turbulent Reynolds tensor [kg/ms^2]
$\mathbf{T}$	molecular tensor [kg/ms^2]
$\boldsymbol{\Upsilon}$	production rate tensor [kg/ms^3]
$\boldsymbol{\Pi}$	deformation rate due to pressure [kg/ms^3]

Subscripts

g	refers to gas phase
p	refers to solid phase

Superscripts

T	refers to transposed
t	refers to turbulent

obtained through a complete analysis of the mass and momentum balance equations, subject to a rigorous turbulence model. Computational fluid dynamics (CFD) is applied to acquire entire numerical solutions, which allow the prediction of these phenomena [11]. Based on a pioneering study by Boysan et al. [11], several other authors [1–7,11–13,14,15,17–23,27–38] have improved the velocity field representation through the modification of turbulence models. The standard turbulence models k-ε and RNG k-ε (renormalization group), which are both isotropic, and the DSM (differential stress model), LES (large eddy simulation) and RSM (Reynolds stress model), which are anisotropic, are typically used to model cyclone flows. The results obtained with the isotropic models are not reliable for corroborating data obtained experimentally; however, the RSM can qualitatively reproduce the tangential velocity distributions [1–13,14–24,27–42].

Experimental and numerical investigations have demonstrated the effects of the inlet gas velocity, inlet width, and vortex finder length and diameter on the cyclone performance [12,23–25]. These studies report that the vortex finder length has a direct effect on the overall performance and the diameter has a strong effect on some of the parameters. Nevertheless, most investigations on cyclone separators did not include a joint analysis, focusing only on the geometrical dimensions and not on the shape.

This paper reports a study on the downstream and upstream effects of a reduction in the cross section at the end of the upper outlet duct (vortex finder extension) on the flow pattern and performance of a cyclone. An alternative design based on experimental data and simulations for small particle collection is then proposed. The objective of these geometric variations is to improve the efficiency in terms of the ability to capture fine particles. It was sought *a priori* to enable, through direct design modifications in the conventional behavior of the flow in cyclones, a second opportunity to capture the solids, through the return of the particles to the swirling flow, thus improving the equipment performance. The second downward swirling flow generated with these design modifications was then described numerically.

2. Mathematical modeling

In this study, the 3D gas-solid swirling flow was described using mathematical modeling based on Reynolds-Average-Navier-Stokes equations (RANS) employing a transient Eulerian-Lagrangian approach in a one-way coupling and the RSM for the turbulent effects. Turbulent and molecular diffusion was modeled as described by Lien and Leschziner [40]. The strain (deformation) rate was defined in two different ways: quadratic modeling proposed by Speziale et al. [41] and linear modeling suggested by Gibson and Launder [42] and Launder [43]. For the spherical particle law a drag coefficient proposed by Morsi and Alexander [44] was used. The random-walk model was used to include turbulent dispersion effects on the particles [45]. The mathematical modeling employed is shown in Table 1.

3. Methodology

3.1. Experiment

The experiments were performed in a test facility (Fig. 1) composed of a tracer feeder (01), particle feeder (02), laser (03), pressure probe (04), plexiglass cyclone (05), exhauster (06), solid collector (07), PIV (08), filter (09) and a Pitot tube (10). Basically, this system is comprised of a gas admission pipeline, cyclone, exhauster and bag filter.

In Fig. 1 the cyclone dimensions used for the acquisition of the experimental and numerical data, where the parameters *Ds* and *Lc* correspond to the vortex finder diameter and the cylindrical body height, respectively.

For the calibration procedure and data acquisition, the cameras are mounted on the same side of the light sheet. This means that both cameras are on the same side of the two-level calibration plate in a backward and forward scattering, respectively. As both cameras are looking at the same point from different angles (after