Powder Technology 311 (2017) 465–483

Contents lists available at ScienceDirect

Powder Technology

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

Analysis and optimization of multi-inlet gas cyclones using large eddy simulation and artificial neural network

Lakhbir Singh Brar [a,*], Khairy Elsayed [b,c]

[a] Mechanical Engineering Department, Birla Institute of Technology, Mesra, Ranchi 835215, India
[b] Vrije Universiteit Brussel, Department of Mechanical Engineering, Research Group Fluid Mechanics and Thermodynamics, Pleinlaan 2, 1050, Brussels, Belgium
[c] Mechanical Power Engineering Department, Faculty of Engineering at El-Mattaria, Helwan University, Masaken El-Helmia P.O., Cairo 11718, Egypt

ARTICLE INFO

Article history:
Received 15 October 2016
Received in revised form 26 January 2017
Accepted 2 February 2017
Available online 6 February 2017

Keywords:
Cyclone separators
Multi-inlet
Large eddy simulation (LES)
Precessing vortex core (PVC)
Artificial neural network
Surrogate-based optimization (SBO)

ABSTRACT

The present study is aimed at optimizing the performance of multi-inlet gas cyclones. The current contribution is threefold. First, a design of experiments (DoE) has been conducted for three variables viz. the flow rate through the secondary inlet, the (square) cross-sectional area of the secondary inlet and the location of the top of the main inlet from cyclone roof. Second, the numerical simulations are performed using large eddy simulation (LES) to predict the Euler number, cut-off size and the collection efficiency for different combinations of the independent variables. The CFD simulation results are used to train an artificial neural network for three responses, namely the Euler number, the cut-off diameter and the overall collection efficiency. Moreover, the simulation results explain how the variations of the design variables affect the flow pattern and performance. Furthermore, the fitted surrogate model demonstrates that the most significant factors are the ratio of flow rates and the area ratio. Third, single-objective and multi-objective optimization studies are carried out using artificial neural network. The optimum design results in better performance than the conventional cyclones.

1. Introduction

The process gas streams in industries contain suspended particles of different sizes that require faster means of separation. Among various equipment (e.g. gravity settlers, bag filters, etc.) cyclones are preferred to separate out the dispersed phase (dust) from the continuous phase (air) due to their simple design, low maintenance cost, and high rates of particle separation (to name a few). In general, cyclones are used as: (a) an end device to efficiently remove the suspended particles from the carrier gas or as an intermediate device to separate out a large fraction of dispersed particles to reduce the load on bag filters, (b) as classifiers to classify the particles according to their sizes, (c) for phase separation (e.g. for catalyst recovery in circulating fluidized beds). They find wide applications in power stations, food processing plants, crushing industries, separation plants, chemical industries, etc. [1].

In cyclones, process gas acquires swirl due to the confined geometry, and follows a helical path down to the bottom from where it reverses its direction and exits through the outlet. Therefore, cyclonic flows constitute double vortex: outer (free) vortex swirling downwards and inner (forced) vortex swirling upwards. Owing to the strong swirl, radial accelerations are set up that generates a strong centrifugal force field inside cyclones useful for separating the dense phase from the lighter one [2,3]. Therefore, the dense matter tends to move out towards the wall where the particles lose their inertia in the vicinity of the boundary layer. Under the action of axial velocity and gravity (the former being more dominant than the latter), these particles are transported to the collection bin attached to the cyclone bottom. Although the working principle sounds simple, the actual flow physics is quite complicated and not yet fully understood (even with the modern sophisticated equipment) [4,5].The performance parameters of a cyclone include the pressure drop and the collection efficiency (or cut-off diameter): the former indicates the energy consumption and the latter quantifies the separation capability of a cyclone. These two parameters form the objective functions to be optimized with an intention to decrease the pressure drop and increase the collection efficiency (or reduce the cut-off diameter). Therefore, the problems related to the optimization of cyclones are essentially multi-objective in nature.

Each cyclone is designed to meet a specific task and is associated with a unique flow physics that is greatly influenced by its geometry. Earlier studies have demonstrated that the cyclone performance strongly varies with the vortex finder diameter [2,6–8], cyclone length [9] (that include cylinder and cone lengths), and inlet area [10–12]. On the other hand, the vortex finder insertion length [2,6,7] and cone tip diameter [13,14] are reported less responsive to the cyclone performance.

Over the time, different designs to generate swirl were proposed; this includes a tangential inlet, scroll inlet, axial inlet with guide vanes,

---

[*] Corresponding author.
E-mail addresses: brarlsb@gmail.com, brarls@bitmesra.ac.in (L.S. Brar).

http://dx.doi.org/10.1016/j.powtec.2017.02.004

etc. [1]. Among these, a tangential inlet is a most popular and widely accepted configuration due to the simplicity in manufacturing and hassle-free maintenance. Studies based on inclined cyclone inlet revealed that increasing the inlet angle up to 45° increases the collection efficiency, and a further increase in angle resulted in the loss of efficiency [15–17]. Based on the tangential inlet design, cyclones with multi-inlet configurations were also investigated [18–21]. Compared with a conventional single inlet, the multi-inlet cyclones provide an additional advantage of procuring high dust loaded gas streams with a similar cut-off diameter. Secondly, for a given cut-off diameter, multi-inlet cyclones can operate at lower aerosol flow rates than single inlet cyclones (provided the total inlet area is the same) [22]. Some correlations [23] and semi-empirical models [22] were also proposed based on multi-inlet cyclone geometries to predict the pressure drops and cut-off diameters. Although less expensive than the experimental setup or numerical simulations, such models lack in accuracy when applied to other cyclone geometries [24].

Another class of multi-inlet configuration that has demonstrated an outstanding performance over the conventional equi-area multi-inlets is by making use of a secondary inlet of a very small cross-section (cf. Fig. 1). These inlets introduce an additional fluid stream at a high velocity into the cyclone. Such a cyclone configuration was first proposed by Yoshida et al. [25] that served as a classifier. The purpose of providing secondary inlet was to control local tangential velocity in the lower conical part of the cyclone, and near the upper boundary layer region where tangential velocity decreases due to increasing viscosity effects. The particles reaching the top cover of cyclone keep circulating (especially large particles) that causes a loss in the efficiency, and may also result in high rates of surface wear. Numerous changes were made later in the design to further reduce the particle cut-off size [26–28].

The variations in the flow field by introducing an additional high-velocity fluid stream, that affects the cyclone performance, have motivated the authors to extend the work and explore additional possibilities. For this, design guidelines proposed by Yoshida et al. [25,26] have been chosen as a base-line model to carry out the multi-objective optimization. With a priori knowledge of an optimum angle that resulted in a minimum cut-off diameter, the secondary inlet angle is fixed at 180° [25]. Since the cut-off diameter is affected by the location of the primary inlet and the flow rate through the secondary inlet [25], therefore the same has also been accounted for optimization. Two different cases based on the location of top surface of the primary inlet are considered (cf. Fig. 1) viz. the one above the bottom surface of the secondary inlet ($y_m < L_s$) and the other below it ($y_m > L_s$), in accordance with [26], are evaluated. Secondly, the (square) cross-sectional area of the secondary inlet is also subjected to optimization.

Early research techniques involved adjusting one parameter at a time while holding all others fixed till optimum results were achieved [29]. Such procedures suffered serious drawbacks due to large methodical times, and the optimized datasets were also not globalized. To overcome such difficulties many techniques were proposed over the time that were associated with some advantages and disadvantages. With the increasing number of independent variables, large data are to be procured before carrying out the optimization. Usually, three means of generating such data have been adopted in most of the researches viz. (a) the theoretical and semi-empirical models, (b) statistical models and (c) computational fluid dynamics (CFD). The theoretical and semi-empirical models (e.g.: [30–32]) were formulated from the physical descriptions and mathematical equations, that demand very detailed understanding of the flow field and energy dissipation mechanisms [29]. However, due to simplifying assumptions, the predictions from these models differ significantly from the measured results (especially for different cyclone designs). For cyclones, statistical models (e.g.: [33,34]) were used as an alternative in the 1980s to predict cyclone pressure drop. Although convenient to use, the accuracy of these models depends largely on the correlation function used to fit the experimental data. As an alternative to such models, CFD is a well-established numerical technique that is capable of modeling almost any type of practical fluid flow.

Many optimization studies on gas cyclone geometry are available in the literature such as [24,29,35–37] but all of these studies are related to the single-inlet gas cyclones.

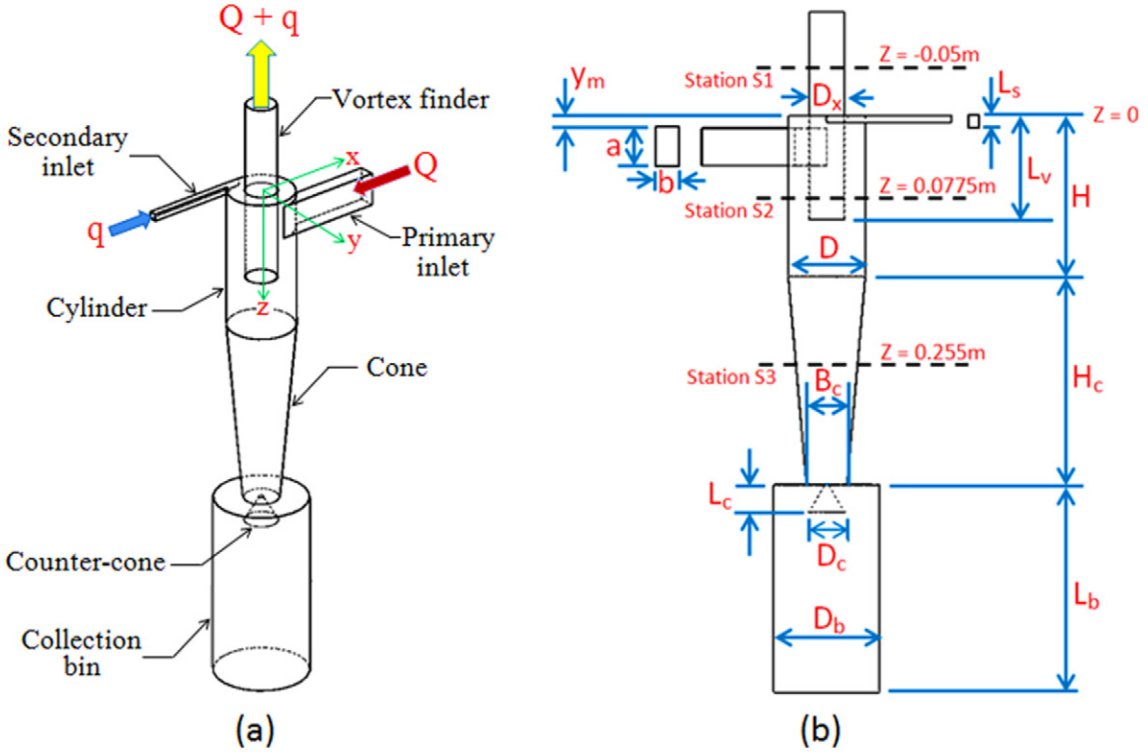

Fig. 1. Details of the cyclone geometry.