Powder Technology 295 (2016) 209–217

Contents lists available at ScienceDirect

Powder Technology

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

Effects of cylindrical and conical heights on pressure and velocity fields in cyclones

Selami Demir [a,*], Aykut Karadeniz [a], Murat Aksel [b]

[a] Yıldız Technical University, Faculty of Civil Engineering, Environmental Engineering Department, Davutpaşa-İstanbul, Turkey
[b] İstanbul Kültür University, Engineering Faculty, Civil Engineering Department, Bakırköy-İstanbul, Turkey

ARTICLE INFO

Article history:
Received 26 January 2016
Received in revised form 16 March 2016
Accepted 26 March 2016
Available online 29 March 2016

Keywords:
Cyclones
Cylindrical/Conical height
CFD
Pressure fields
Velocity fields

ABSTRACT

Nine modifications of Stairmand high-efficiency type cyclone with various cylindrical and conical heights were used to investigate their effects on pressure drop and flow field within cyclones. An experimental setup was built and experiments were conducted on various cyclone geometries at inlet velocities ranging from 10 to 18.5 m/s. The body heights ranged from *D* to 2*D* (*D* being the diameter of the cyclone body), while the conical heights were between 2*D* and 3*D*. The experimental results were used to calibrate CFD (Computational Fluid Dynamics) model. Experimental results showed that the pressure drop ranged from 191 to 235 Pa and 690 to 920 Pa at the lowest and highest inlet velocities, respectively, and that pressure drop is a function of both cylindrical and conical heights with reduced pressure drops as cylindrical and/or conical heights increase. Maps showing the change in pressure drop with respect to cylindrical and conical heights were prepared and interpreted to determine optimum ratio of conical-to-cylindrical height for reduced pressure drop. Optimum ratio was found experimentally as between 1.67 and 2.5. CFD results agreed well with the experimental data and the results helped gain insights into complicated phenomena taking place in cyclones. CFD results showed that increased axial velocities at the lip of vortex finder distrub radial velocity profiles and that tangential velocities in the stabilized vortex can be as high 1.4 times the mean inlet velocity. Both findings indicate that collection efficiency of the cyclone may be influenced by the height of the cyclone.

1. Introductıon

Cyclone separators, being in widespread use in air pollution control field during 20th century, are used as a preliminary or main control device for removing particulate matter (PM) from flue gases due primarily to their easy construction and installation as well as low costs of operation and maintenance. Besides, they are preferred for separation and recycling of valuable materials in many powdered production processes [1]. Use of cyclone separators also has advantages related with their capabilities to adapt extreme operating conditions such as high temperatures and pressures, high PM loads, and high concentrations of corrosive components in flue gases [2,3].

A cyclone separator is a simple control device in which centrifugal forces are employed for separating PM from flue gases. Particle-laden gas enters the cyclone from top, and cylindrical structure of the cyclone forces the gas into a spinning motion as a result of which a vortex is formed. This vortex produces a centrifugal force depending on particle's size [4], and particulate matter, being heavier than gaseous molecules, drift toward the cyclone wall and moves in gas-wall boundary layer toward the bottom of the cyclone. The conical part connected to the bottom of the cylinder reverts the gas flow to the vortex finder, where particles are collected in a dust bin placed at the bottom of the conical part. The gas flow then forms a smaller vortex within the main, outer vortex and leaves the cyclone separator through the outlet pipe.

A conventional cyclone separator is composed of four main parts, namely the inlet part, the body, the conical part and the vortex finder. Of the inlet parts, tangential inlets are most widely used, where several gas velocities such as 6–15 m/s [5] and 15–25 m/s [6] are reported. The body sets the outer boundary for the vortex and the separation starts in this part. The conical part is produced with various slopes and heights and is responsible for forcing the gas flow toward the vortex finder while collected particles are transported to an external dust bin under the effect of gravity. A number of research papers have been published related with optimum shapes and dimensions of body and conical parts [7–9]. The vortex finder can be constructed in various shapes and dimensions with an optional flow rectifier [10].

The performance of a cyclone separator is defined by two operating parameters, namely pressure drop and collection efficiency. These parameters are intimately related with each other. Usually, cyclone's collection efficiency increases with increasing pressure drop. For this reason, a great number of research projects have been dedicated to investigation of these parameters for distinct cyclone shapes under various operating conditions [4,7,8,11–18]. Cyclone pressure drop is

* Corresponding author.
E-mail address: seldemir@yildiz.edu.tr (S. Demir).

http://dx.doi.org/10.1016/j.powtec.2016.03.049
0032-5910/

also a function of gas temperature and PM load. A number of mathematical models were suggested to estimate the effects of these operating parameters on cyclone pressure drop [11,19–20].

In spite of a great number of models for cyclone pressure drop [13, 20–21], Computational Fluid Dynamics (CFD) applications have become very popular for simulating pressure and velocity fields in cyclone separators [2,3,22–31]. Due to requirements of high computational capacity and time, CFD models are not very good for field engineers. On the other hand, these models are suitable for scientific studies since they are capable of simulating complicated phenomena that take place within cyclone separators.

This study aims at experimental investigation of the effects of cylindrical and conical dimensions on the cyclone performance. In this aspect, nine cyclone separators were built and run. Pressure drop measurements are performed and used for calibrating CFD model. Then, CFD simulations were performed to interpret pressure and velocity fields within the cyclone separators. Conclusions were drawn based on the effects of cylindrical and conical heights on cyclone pressure drop.

2. Materials and methods

2.1. Experimental setup

A laboratory-scale experimental setup was used for investigation of pressure and velocity fields within cyclones. The setup was composed of an air blower, an inlet channel, and a cyclone separator [13]. A schematic representation of the experimental setup is shown in Fig. 1. A variable speed, radial fan was used to produce the gas flow. The capacity of the fan was 1500 m^3/h with a maximum pressure of 1 m water column. The gas flowrate is adjusted at the orifice meter (ISO 5167-2:2003) on the inlet channel equipped with a flow-control valve. QA/QC procedures for flowrate measurement using orifice meter are given in detail in related standard documentation.

Nine cyclone separators were used in the experiments. These cyclones were modifications of Stairmand high efficiency type cyclones [8]. Three body (cylindrical) heights and three conical heights were experimentally investigated. The dimensions of cyclones are summarized in Table 1. Since the cyclones discharged to the atmosphere, the pressure drop was measured as the static pressure at the inlet. Pressure measurements were also performed at the extended part of the vortex finder and no significant differences were observed.

Table 1
Cyclone dimensions.

Geometry	Notation	Dimension (mm)	Ratio	Remarks
Body diameter	D	290	–	Constant for all runs
Inlet height	a	145	0.50	Constant for all runs
Inlet width	b	58	0.20	Constant for all runs
Body height	h_b	290	1.00	Three different body heights for
		435	1.50	each conical height
		580	2.00	
Conical height	h_c	580	2.00	Three different conical heights for
		725	2.50	each body height
		870	3.00	
Vortex finder length	S	145	0.50	Constant for all runs
Vortex finder diameter	D_e	145	0.50	Constant for all runs
Cone-tip diameter	B	109	0.38	Constant for all runs

2.2. CFD simulations

Computational Fluid Dynamics simulations were also performed to investigate the effects of body height and conical height on the pressure and velocity fields within the cyclone. FLUENT software version 14.5, which is a part of ANSYS Workbench Products, was used for this purpose. A CAD-based software were used to prepare the 3D solid model and fluid domain, and the fluid domain was meshed over 10^6 elements using ANSYS Meshing Module. Since CFD results were compared with experimental results, a mesh independence check was unnecessary. Inflation layers were created to minimize wall effects on core of flow domain. Mostly tetrahedral cells were used inside the mesh domain, while in inflation layers, hexahedral cells were employed to handle with the wall boundary effect.

Three boundary conditions were defined for numerical modelling. For outlet, static pressure boundary was used with a value of 101.3 kPa. At the inlet side, velocity boundary condition was used. The inlet velocity was calculated for each cyclone using the adjusted flowrate to the cyclone. Stationary wall boundary condition was used for cyclone walls and dust bin. The solver parameters for ANSYS-FLUENT are given in Table 2. Note that PISO algorithm gives more stable results in steady-state simulations and requires less computational power than SIMPLE. Thus, PISO algorithm was used in the solution. Reynolds Stress Model (RSM) is usually preferred for calculating swirling flows. This model

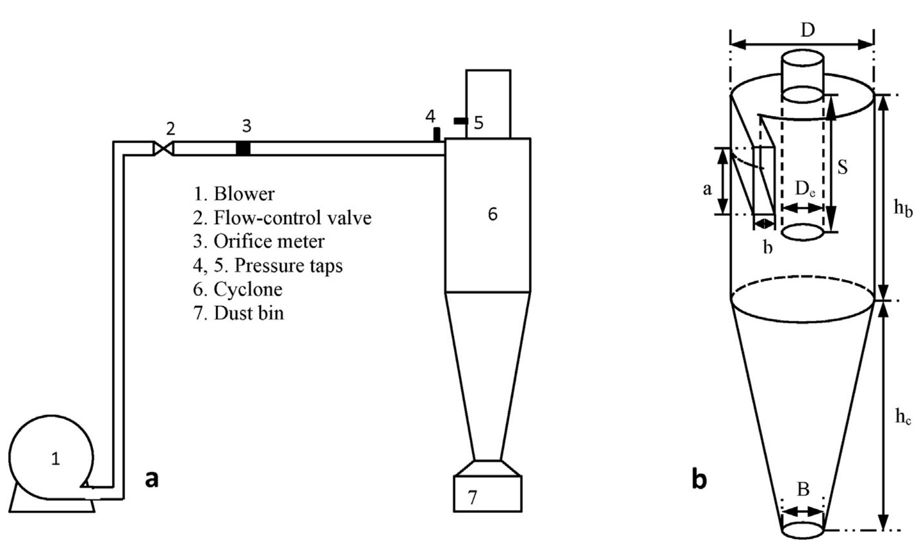

Fig. 1. Schematic representation of a. experimental setup, b. cyclone dimensions.