Available online at www.sciencedirect.com
SCIENCE DIRECT®
Chemical Engineering and Processing 44 (2005) 7–12
Chemical Engineering and Processing
www.elsevier.com/locate/cep

The influence of temperature and inlet velocity on cyclone pressure drop: a CFD study

Jolius Gimbun*, T.G. Chuah, A. Fakhru'l-Razi, Thomas S.Y. Choong

Department of Chemical and Environmental Engineering, Faculty of Engineering, Universiti Putra Malaysia 43400 UPM Serdang, Selangor D. E., Malaysia

Received 16 February 2004; received in revised form 22 March 2004; accepted 22 March 2004
Available online 18 May 2004

Abstract

This work presents a computational fluid dynamics (CFD) calculation to predict and to evaluate the effects of temperature and inlet velocity on the pressure drop of gas cyclones. The numerical solutions were carried out using spreadsheet and commercial CFD code Fluent 6.1. This paper also reviews four empirical models for the prediction of cyclone pressure drop, namely [Air pollution control: a design approach, in: C. David Cooper, F.C. Alley (Eds.), Cyclones, second ed., Woveland Press Inc., Illinois, 1939, p. 127–139] [Chem. Eng. (1983) 99] [Doctoral Thesis, Havarad University, USA, 1988], and [Chem. Eng. Progress (1993) 51]. All the predictions proved to be satisfactory when compared with the presented experimental data. The CFD simulations predict excellently the cyclone pressure drop under different temperature and inlet velocity with a maximum deviation of 3% from the experimental data. Specifically, results obtained from the computer modelling exercise have demonstrated that CFD is a best method of modelling the cyclones operating pressure drop.

Keywords: Cyclone; CFD; Pressure drop; Temperature; Inlet velocity

1. Introduction

Cyclones are devices that employ a centrifugal force generated by a spinning gas stream to separate particles from the carrier gas. Their simple design, low capital cost and nearly maintenance-free operation make them ideal for use as pre-cleaners for more expensive final control devices such as baghouses or electrostatic precipitators. Cyclones are particularly well suited for high temperature and pressure conditions because of their rugged design and flexible components materials. Cyclone collection efficiencies can reach 99% for particles bigger than 5 μm [12], and can be operated at very high dust loading. Cyclones are used for the removal of large particles for both air pollution control and process use. Application in extreme condition includes the removing of coal dust in power plant, and the use as a spray dryer or gasification reactor.

Engineers are generally interested in two parameters in order to carry out an assessment of the design and performance of a cyclone. These parameters are the collection efficiency of particle and pressure drop through the cyclone. An accurate prediction of cyclone pressure drop is very important because it relates directly to operating costs. Higher inlet velocities give higher collection efficiencies for a given cyclone, but this also increases the pressure drop across the cyclone. Therefore, a trade off must be made between higher collection efficiency and low pressure drop across the cyclone. Computational fluid dynamics (CFD) has a great potential to predict the flow field characteristics and particle trajectories inside the cyclone as well as the pressure drop [8]. The complicated swirling turbulent flow in a cyclone places great demands on the numerical techniques and the turbulence models employed in the CFD codes when modelling the cyclone pressure drop.

In this study, pressure drop calculations are performed using CFD and compared with four empirical model of Shepherd and Lapple [11], Casal and Martinez [3], Dirgo [5], and Coker [4]. These four empirical models and CFD prediction are compared with the experimental data presented in the literature. In this study, the CFD calculations are carried out using commercial finite volume code Fluent 6.1 and the empirical models are performed in Microsoft Excel spreadsheet.

* Corresponding author. Tel.: +60-19-248-9101; fax: +60-38946-7120.
E-mail address: jolius21@yahoo.co.uk (J. Gimbun).

0255-2701/$ – see front matter © 2004 Elsevier B.V. All rights reserved.
doi:10.1016/j.cep.2004.03.005

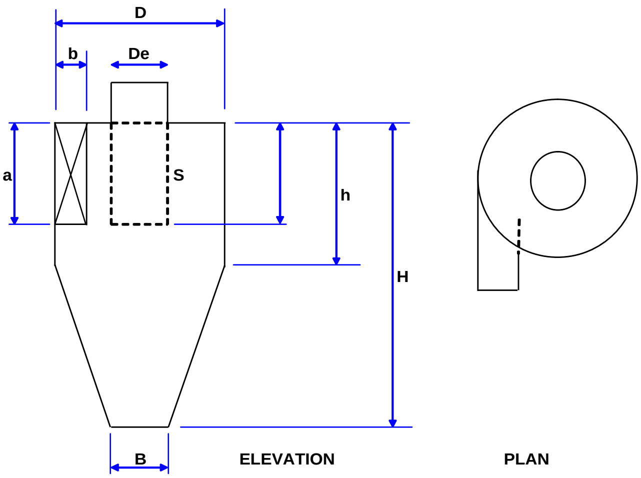

Fig. 1. Tangential cyclone configuration.

2. Cyclone design

There are a number of different forms of cyclone but the reverse flow cyclone represented in Fig. 1 is the most common design used in the industry. The cyclone consists of four main parts: the inlet, the separation chamber, the dust chamber and the vortex finder. Tangential inlets are preferred for the separation of solid particles from gases [1]. In this study, the numerical simulation deals with the standard case of reverse flow cyclone with a tangential rectangular inlet. Cyclone dimension used in this simulation are as shown in Table 1.

3. Computational fluid dynamics approach

Fluent is a commercially available CFD code which utilises the finite volume formulation to carry out coupled or segregated calculations (with reference to the conservation of mass, momentum and energy equations). It is ideally suited for incompressible to mildly compressible flows. The conservation of mass, momentum and energy in fluid flows are expressed in terms of non-linear partial differential equations which defy solution by analytical means. The solution of these equations has been made possible by the advent of powerful workstations, opening avenues towards the calculation of complicated flow fields with relative ease.

For the turbulent flow in cyclones, the key to the success of CFD lies with the accurate description of the turbulent behaviour of the flow [8]. To model the swirling turbulent flow in a cyclone separator, there are a number of turbulence models available in Fluent. These range from the standard k–ϵ model to the more complicated Reynolds stress model (RSM). The k–ϵ model involves the solution of transport equations for the kinetic energy of turbulence and its dissipation rate and the calculation of a turbulent contribution to the viscosity at each computational cell. The standard k–ϵ, RNG k–ϵ and realizable k–ϵ model was not optimized for strongly swirling flows found for example in cyclones [10,6]. Turbulence may be stabilised or destabilised in the parts of flow domain where strong streamline curvature is presence. However to reduce the computational effort the RNG k–ϵ model can be used with about 12% deviation on experimental data [8]. The numerical studies carried out by Fredriksson [7] reveal that the RNG k–ϵ model under predicts the variation of the axial velocity profile across the radial direction and also over predicts the magnitude of the tangential velocity and the cyclone pressure drop.

The Reynolds stress model requires the solution of transport equations for each of the Reynolds stress components as well as for dissipation transport without the necessity to calculate an isotropic turbulent viscosity field. The Reynolds stress turbulence model yield an accurate prediction on swirl flow pattern, axial velocity, tangential velocity and pressure drop on cyclone simulation [7,6,13,10].

The finite volume methods have been used to discretised the partial differential equations of the model using the Simple method for pressure–velocity coupling and the second order upwind scheme to interpolate the variables on the surface of the control volume. The segregated solution algorithm was selected. The Reynolds stress turbulence model was used in this model due to the anisotropic nature of the turbulence in cyclones. Standard fluent wall functions were applied and high order discretisation schemes were also used.

Under the RSM second order upwind for discretisation there is a difficulty to reach the convergence in simulation. The residuals may exhibit cyclic tendencies which mean that the transient pattern occurs. In this instance, the solver must be changed to a transient solver and makes the time step something in the region of 0.025 s or a tiny fraction of the residence time of the cyclone. The simulation is then solved with a coupling of unsteady and steady state solver in Fluent. For the simulation using RNG k–ϵ model the steady state solver is sufficient to reach the convergence. The CFD simulation was performed with a Pentium IV 2.8 GHz HP workstation XW8000 with 512 cache-memory, 1 GB RAM-memory, and 110 GB hard disc memory.

Table 1
Cyclone geometry used in this simulations

Geometry	a/D	b/D	D_e/D	S/D	h/D	H/D	B/D	D[a]
Stairmand high efficiency	0.5	0.2	0.5	0.5	1.5	4	0.375	0.305
Bohnet [2]	0.533	0.133	0.333	0.733	0.693	2.58	0.333	0.15

[a] Unit in meters.