Chemical Engineering Journal 151 (2009) 39–45

Contents lists available at ScienceDirect

Chemical Engineering Journal

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

Numerical investigation of performance characteristics of a cyclone prolonged with a dipleg

Fuat Kaya*, Irfan Karagoz[1]

Uludag University, Department of Mechanical Engineering, 16059 Bursa, Turkey

ARTICLE INFO

Article history:
Received 23 May 2008
Received in revised form 19 January 2009
Accepted 26 January 2009

Keywords:
Swirling flows
Separation efficiency
Numerical simulation
Lagrangian method

ABSTRACT

Numerical modeling of a particle separation process is carried out to understand the gas-particle two-phase flow field inside a cyclone prolonged with a dipleg and results of the numerical simulations are compared with experimental data to validate the numerical results. The flow inside the cyclone separator is modeled as a three-dimensional turbulent continuous gas flow with solid particles as a discrete phase. The continuous gas flow is predicted by solving Navier–Stokes equations using the differential RSM turbulence model with nonequilibrium wall functions. The second phase is modeled based on a Lagrangian approach. Analysis of computed results shows that the length of the dipleg considerably influences the cyclone separation efficiency rather than the cyclone pressure drop, especially for lower inlet velocities in relatively short cyclones, by providing more separation space.

1. Introduction

An efficient removal of particles from two-phase flows is essential to develop advanced separation technologies. Cyclone separators are widely used for this purpose, due to their several advantages such as simple design, absence of moving parts, low manufacturing and maintenance costs. Entrance of flow into cyclone can be axial or tangential through inlet section, which can be in different shapes for each cyclone. Cyclone separators operate under the action of centrifugal forces. Fluid mixture enters the cyclone and makes a swirl motion and, due to the centrifugal forces, the particles in the flow gain a relative motion in the radial direction and are separated from the main flow. It is difficult to analyze this problem, since this swirling flow is very complex, and there are many parameters influenced this flow. The main performance characteristics of a cyclone separator are collection efficiency, fractional efficiencies and pressure losses. Many experimental and theoretical studies performed on this difficult problem provide semiempirical models ranging from simple [1–4] to more comprehensive models [5,6] for the prediction of a cyclone performance.

Due to the fact that the fluids dynamics of cyclones are complex, including highly turbulent structure and limitations in the usage of empirical models, optimization of cyclone performance by improving the cyclone efficiency and minimizing the pressure drop was essentially based on experiments rather than theoretical studies [7–14]. However, the computational fluid dynamics (CFD) of cyclones has exploded recently with advances in computer capabilities, numerical methods and software, and with the advent to the field of a large number of investigators [15–23].

Although many works have been carried out to investigate the influence of different geometric parameters such as cyclone length, inlet and outlet pipe geometries etc. on the performance of cyclones, there has been little work concerning the dust outlet geometries. Obermair et al. [24] performed cyclone tests with five different dust outlet geometries to find the influence of the dust outlet geometry on the separation process. They showed that separation efficiency can be improved significantly by changing the dust outlet geometry, and they reported that further research is needed to clarify precise effects of dust outlet geometry. The effect of a dipleg was posed and investigated by several researchers [25–27].

It is well known that in a reverse flow cyclone, the outer vortex flow weakens and changes its direction at a certain axial distance from the vortex finder. This axial magnitude has been called the “natural length” or the “vortex length” of the cyclone, and the axial position is referred to as “the end of the vortex”. The influence of the dipleg on the vortex and flow characteristics was investigated by Gil et al. [27], and they found that the vortex end was related not only to inlet and vortex finder geometry, but also to inlet gas velocity and solid loading. They also found that a small amount of “underflow” drawn through the dust exit causes the vortex end to move down the dipleg and a higher dust loading caused the end of the vortex to move up the dipleg.

Hoffmann et al. [28] studied the effect of cyclone length on separation efficiency and pressure drop, experimentally and theoretically, by varying the length of the cylindrical segment of a cylinder-on-cone cyclone. They showed that the separation efficiency improves for certain ratio of L/D. However, they found an

* Corresponding author. Tel.: +90 224 2941997; fax: +90 224 2941903.
E-mail address: fkaya@uludag.edu.tr (F. Kaya).
[1] Tel.: +90 224 2941911; fax: +90 224 2941903.

1385-8947/$ – see front matter
doi:10.1016/j.cej.2009.01.040

optimal cyclone length: lengthening the cyclone to more than $L/D = 5.65$ dramatically lower separation efficiency.

Qian et al. [29] studied, experimentally and numerically, a cyclone prolonged with a dipleg. They stated that the dipleg has important influence on separation efficiency of the cyclones. However, their work is very specific and has not exhaustively explored the effects of the dipleg. They studied only one cyclone with three different dipleg lengths. Their experimental and numerical tests were carried out for a single inlet velocity and a particle diameter, respectively. Therefore, the question can be considered still open and needs further investigation.

The aim of the present study is to give a detailed description of the flow structure in a tangential inlet cyclone with a dipleg located under the cyclone and to investigate the effects of the dipleg on the cyclone collection efficiency and pressure drop. Computational results were verified by comparing them with experimental data available in the literature. The differential Reynolds Stress Model (RSM) was employed for turbulence closure and the Lagrangian approach was used to compute discrete particle motions.

2. Theoretical bases

2.1. Modeling of airflow and turbulence

Cyclone separator consists of three main parts: inlet, separation chamber and vortex finder. Tangential inlets are preferred for the separation of particles from gases [30]. Therefore, the present work is deal with a tangential inlet cyclone whose schematic representation is given in Fig. 1.

The Reynolds-averaged equations for conservation of mass and momentum can be written in the following compact form:

$$\frac{\partial(\rho\phi)}{\partial x_t} + \frac{\partial(\rho u_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\Gamma_\varphi \frac{\partial \phi}{\partial x_j}\right] + S_\varphi \qquad (1)$$

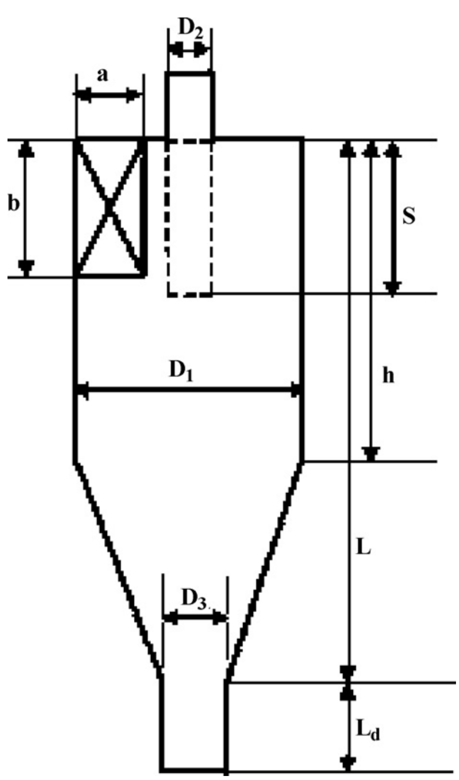

Fig. 1. Tangential inlet cyclone.

where ϕ is a universal dependent variable, Γ_φ is the diffusivity, and S_φ is the source term. For the continuity equation, ϕ is l, Γ_φ and S_φ are 0. In the momentum equations, ϕ, Γ_φ, and S_φ stand for u_i, μ and

$$-\frac{\partial P}{\partial x_i} + \rho g_i + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_i}{\partial x_j} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right) - \frac{\partial}{\partial x_i}(\rho \overline{u_i' u_j'})$$

respectively.

In order to obtain values for the Reynolds stress terms $(\rho\overline{u_i' u_j'})$, the differential Reynolds stress model (RSM) was used, due to strong anisotropy in the cyclone flow. The RSM closes the governing equations by solving transport equations for individual Reynolds stresses as

$$\frac{\partial \rho \overline{u_i u_j}}{\partial t} + \frac{\partial}{\partial x_k}\left(U_k \partial \rho \overline{u_i u_j}\right) = P_{ij} + \phi_{ij} + \frac{\partial}{\partial x_k}\left[\left(\mu + \frac{2}{3}c_s \rho \frac{k^2}{\varepsilon}\right)\frac{\partial \rho \overline{u_i u_j}}{\partial x_k}\right] - \frac{2}{3}\delta_{ij}\varepsilon\rho \qquad (2)$$

where $P_{i,j}$ is the production term, and ϕ_{ij} is the pressure–strain correlation term. The pressure–strain correlation term is responsible for the redistribution of turbulent energy amongst the six stress components and needs modeling. The linear pressure–strain model was used to model this term [31].

The turbulence dissipation (ε) appears in the individual stress equations and can be expressed as

$$\frac{\partial \rho \varepsilon}{\partial t} + \nabla \cdot (\rho U \varepsilon) = \frac{\varepsilon}{k}(c_{\varepsilon 1} P - c_{\varepsilon 2}\rho\varepsilon) + \nabla \cdot \left[\frac{1}{\sigma_{\varepsilon RS}}\left(\mu + \rho C_{\mu RS}\frac{k^2}{\varepsilon}\right)\nabla \cdot \varepsilon\right] \qquad (3)$$

The model constants in these equations are $c_s = 0.22$, $c_{\varepsilon 1} = 1.45$, $c_{\varepsilon 2} = 1.9$, $C_{\mu RS} = 0.1152$.

2.2. Modeling of particle motion

Modeling of particle motion is based on the assumptions that the second phase consists of spherical particles dispersed dilutely in the continuous phase so that particle–particle interactions and effect of the particle volume fraction on the continuous phase are negligible. In this study, the discrete phase model (DPM) was used to simulate the second phase in a Lagrangian frame of reference by defining the initial position, velocity and size of individual particles.

The trajectory of a particle was obtained by integrating the force balance on the particle. This force balance equates the particle inertia with the forces acting on the particle and can be written in the following form:

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} + F_x \qquad (4)$$

where $F_D(u - u_p)$ is the drag force per unit particle mass and can be written as [31]

$$F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D Re_r}{24} \qquad (5)$$

Re_r is the relative Reynolds number, which is defined as

$$Re_r = \frac{\rho d_p |u_p - u|}{\mu} \qquad (6)$$

where u is the fluid phase velocity, u_p is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_p and d_p are the density and the diameter of the particle, respectively. The drag coefficient C_D for spherical particles is calculated by using the correlation developed by Morsi and Alexander [32].