Separation and Purification Technology 201 (2018) 223–237

Contents lists available at ScienceDirect

Separation and Purification Technology

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

Numerical investigation of effects of inner cone on flow field, performance and erosion rate of cyclone separators

Farzad Parvaz[a], Seyyed Hossein Hosseini[b,*], Khairy Elsayed[c], Goodarz Ahmadi[d]

[a] Department of Mechanical Engineering, Semnan University, P.O. Box 35131-191, Semnan, Iran
[b] Department of Chemical Engineering, Ilam University, Ilam 69315–516, Iran
[c] Mechanical Power Engineering Department, Faculty of Engineering at El-Mattaria, Helwan Uinversity, Masaken El-Helmia P.O., Cairo 11718, Egypt
[d] Department of Mechanical and Aeronautical Engineering, Clarkson University, Potsdam, NY 13699-5725, USA

ARTICLE INFO

Keywords:
Gas cyclone
CFD
Eulerian-Lagrangian approach
Inner cone
Erosion rate

ABSTRACT

Gas cyclones have many industrial applications for separation of solids and liquids from gases. The geometry of the cyclone is the most influential parameter for its performance. This study investigates the effect of presence of an inner cone located at the bottom of the cyclone on the performance of the cyclone separator. Several CFD simulations in cyclones with inner cones with different diameters and heights were performed using the Reynolds stress turbulence model (RSM). The collection efficiency of the cyclone was studied using the Eulerian-Lagrangian approach. The results showed that the maximum tangential velocity is 1.6–1.7 times the inlet velocity. On the other hand, in the radial sections crossing the inner cone, the gradients of the axial and tangential velocities are zero. The maximum axial and tangential velocities occurred in the region between the top of the inner cone to the vortex finder. It was found that by increasing the inner cone height at constant diameter, the cyclone collection performance improves. An increase in the diameter of the inner cone, however, leads to a decrease in the cyclone performance. In overall, with an increase in the inner cone height and diameter, the pressure loss decreases. Finally, the erosion study was conducted using the Det Norske Veritas (DNV) erosion model. It was found that the value of coefficient of restitution affects the predicted erosion rate. In addition, the collection efficiency decreases when the erosion effect was included in the CFD model especially for higher velocities.

1. Introduction

Cyclones are widely used in different industries for separation of particles and droplets from the gas flows. The centrifugal force generated in the cyclone and gravity control the separation. Highly swirling flows are established within the cyclone in descending and ascending streams. The inertial of larger/denser particles in the descending stream are typically too high to follow the stream when it changes direction at the bottom of the cyclone. The centrifugal force moves the heavy particles towards the cone wall that get collected on the surface and then fall to the dustbin. Some particles, however, can get re-entrained in the upward flow even after being collected.

The gas flow in cyclone separators is typically unsteady and highly swirling. Using computational methods, numerous researchers investigated different geometrical and operational parameter to enhance cyclone performance [1–4]. Based on these previous studies on CFD simulation of cyclones, it is understood that the selection of proper turbulence model is vital for simulation of the complex swirling flows inside cyclones. Hoekstra [5] used three turbulence models to investigate the airflow inside a cyclone with three different swirl numbers, and recommended the Reynolds stress turbulence model RSM. Slack et al. [6] using the RSM turbulence model simulated the gas flow inside the Stairmand cyclone and found that the numerical results are well fitted to the experimental data obtained from Laser Doppler Anemometry (LDA). Gronald and Derksen [7] used the large eddy simulation (LES) methodology and two-equation RANS turbulence models to simulate the airflow inside the cyclone and compared the corresponding CFD results with the experimental LDA data. It was found that the LES model outperforms the RANS model for prediction of the fluctuating velocities; however, the LES model is time consuming due to the need for highly refined grids. Alahmadi and Nowakowski [8] used the modified shear stress with curvature correction (SSTCC), RSM and k-ε turbulence models to simulate the airflow inside the Stairmand cyclone and found that the SSTCC model gives more accurate results compared to the RSM turbulence model. However, the computational cost of the SSTCC model was considerably more than that of RSM

* Corresponding author.
E-mail address: s.h.hosseini@ilam.ac.ir (S.H. Hosseini).

https://doi.org/10.1016/j.seppur.2018.03.001
Received 1 November 2017; Received in revised form 23 February 2018; Accepted 2 March 2018
Available online 07 March 2018
1383-5866/

model.

Another important issue in the numerical simulation of flow patterns in cyclones is to take the geometrical effects into account, as emphasized by various researchers. Alexander [9] studied some of the geometrical parameters that affect the cyclone performance. Gimbun et al. [10] investigated the cone tip diameter effect on the pressure drop and performance of cyclones and found that when the cone tip diameter decreases, the total pressure drop and the cyclone performance increase. Elsayed and Lacor [11,12] studied the influence of the inlet dimensions and the cone tip diameter on the flow pattern in gas cyclones and reported that these geometrical parameters significantly affect the cyclone performance. Zhao et al. [13] compared the performance of the conventional single inlet cyclones to that of the spiral double inlet cyclones and found improved performance for the latter cyclones. Hoffmann et al. [14] conducted an experimental investigation to study the impacts of the cyclone length on the cyclone performance. The results showed that by increasing the cyclone length (up to 5.5 times the cyclone diameter), the cyclone performance significantly increases and beyond this length the efficiency dramatically reduces. Xiang and Lee [15] performed a CFD simulation of the cyclones focusing on the effect of the cyclone height on the flow field and showed that by increasing the height, the tangential velocity decreases. The effects of the barrel diameter on the flow pattern in the cyclone and its performance were numerically studied by Sharma and Brar [16]. According to their results, the tangential velocity increases nonlinearly with increasing cyclone diameter. They also found that the pressure drop and the collection efficiency increase by increasing the cyclone diameter. El-Batsh [17] investigated the effect of the outlet tube diameter and length on the performance of a gas-particle cyclone. He reported that an increase in the outlet tube diameter leads to a reduction of the pressure drop as well as the cyclone collection efficiency. Whereas, the outlet tube length does not affect the pressure drop, significantly. Safikhani and Mehrabian [18] studied the effects of length and diameter of the vortex limiter on the flow pattern and performance of a new cyclone. They found that the radial distribution of the tangential velocity for this type of cyclone was similar to that of the conventional one. They also observed that when the vortex limiter diameter decreases, the efficiency of the cyclone increases. Demir et al. [19] numerically and experimentally investigated the cyclone performance to the variation of the heights of the conical and cylindrical section of the gas-particle cyclone for different gas velocities. They observed that the pressure drop and cyclone efficiency are functions of both the cylindrical and the conical section heights. Their numerical results revealed that the increase of the axial velocity at the lip of the vortex finder disturbs the radial velocity and the tangential velocity profiles. Safikhani et al. [20] numerically compared two small square and round cyclones. According to their results, the pressure drop in small square cyclone is less than that in the small round one. In addition, the square cyclone efficiency for a given flow rate is lower than that of the round cyclone; however, by increasing the flow rate the differences in efficiency decrease.

A numerical study regarding the effects of the central channel height and diameter of an oil-gas cyclone on the flow field was performed by Gao et al. [21]. They showed that a decrease in the central channel diameter causes an increase in the pressure drop and the tangential velocity. The tangential velocity near the wall was 0.8–1 times greater than the inlet velocity, whereas the maximum tangential velocity was 1.8–2 times the inlet velocity. Using an accurate LES De Souza et al. [22,23] found that the collection efficiency in small gas-solid cyclone separators can be considerably improved using an annular tube on the overflow duct. De Souza et al. [24] studied the influence of the length and the shape of the outlet duct on the grade efficiency and the pressure drop through small cyclones. They also showed that the impact of the outlet duct length and the shape on the cyclone flow field is small.

Parvaz et al. [25] studied the effects of the vortex finder eccentricity on the gas cyclone flow pattern, as well as, the particle removal efficiency. They found that the deviation of vortex finder from the center affects the performance of the gas cyclone and the gas flow pattern. Brar et al. [26] examined the influence of variation in the cylinder and cone lengths on the Stairmand gas cyclone performance. They showed that an increase in both cylinder and the cone lengths leads to a decrease in the pressure loss, while, an increase in collection efficiency. Brar and Elsayed [27] optimized the performance of multi-inlet gas cyclones by using artificial neural networks (ANN) technique. To train the ANN, the numerical simulation results obtained by the large eddy simulation (LES) approach were used. They found that the optimally designed cyclone over-performs the conventional ones. In addition, Misiulia et al. [28] optimized the geometrical parameters of the deswirler, namely, core diameter, number of vanes, height of vanes and leading edge angle by using CFD and artificial neural network. Misiulia et al. [29] studied the inlet angle impact on the collection efficiency of a cyclone with helical-roof by using the LES method. The numerical results showed that, as the inlet angle increases, the cyclone collection efficiency reduces. They found that the optimum inlet angle is in the range of 10–15°. Raoufi et al. [30] investigated the impact of the vortex finder on the flow pattern and the cyclone performance by the Eulerian-Lagrangian method. They found that the flow rate affects the pattern of velocity vectors in the vortex finder. Decreasing the vortex finder divergence angle leads to expanding the low-pressure zone in the middle of the cyclone. In addition, by enlarging the vortex finder diameter the tangential velocity and cyclone performance decrease.

As noted in the above literature survey, almost all previous works on the CFD simulation of cyclone flows were focused on the impact of various geometrical parameters such as inlet dimensions, inlet angle, cyclone length, barrel diameter, outlet tube diameter, vortex finder eccentricity, and cone tip diameter. There are, however, limited computational investigations in the literature related to the effect of the inner cone dimensions on the cyclone performance. Only Zhao et al. [31] proposed and designed a new inner-cone hydrocyclone (gas–liquid droplet separator). They suggested that their new design would be

Table 1
Erosive model characteristics.

Erosion model of DNV

$$k = \frac{\pi^2}{2\sqrt{10}}\gamma^{\frac{5}{2}}\sqrt{\frac{1}{\rho_P}}\left[\frac{1-q^2{}_p}{Y_p}-\frac{1-q^2{}_w}{Y_w}\right]^2$$

$$f(a) = 2.4647\times10^{-3}a + 2.9284\times10^{-4}a^2 - 2.1974\times10^{-6}a^3$$

Constant

n	γ	ρ_p	q_p	q_w	Y_p	Yw
1	3.1×10^6 Pa	2740 kg/m^3	0.25	0.25	4.01×10^{11}Pa	2.9×10^9Pa

Table 2
Dimensions of the cyclone and physical properties of air and particles.

Dimension	Length (m)	Dimension ratio (Dimension/D)
Body diameter, Dcy	0.205	1
Inlet height, a	0.105	0.5
Inlet width, b	0.041	0.2
Gas outlet diameter, D_x	0.105	0.5
Gas outlet duct length, S	0.105	0.5
Cone-tip diameter, D	0.076875	0.375
Inner cone height, h	{0.205 0.3075 0.41	{1*Dcy* 1.5*Dcy* 2*Dcy*
Inner cone diameter, d	{0.03075 0.046125 0.0615	{0.4*D* 0.6*D* 0.8*D*
Inner cone small, d_x	0.1718	0.25Dx
Cyclone height, H_t	0.82	4
Duct length, L_i	0.15375	0.75
Outlet tube length, L_e	0.1025	0.5