Journal of Chemical and Petroleum Engineering (JChPE)

Print ISSN: 2423-673X Online ISSN: 2423-6721

Experimental and Numerical Studies on Improving Cyclone Efficiency by Rotation of Cyclone Body

Sayed Saman Salehyar *, Ahad Ghaemi, Hossein Mashhadimoslem, Mansour Shirvani

1. School of Chemical, Petroleum and Gas Engineering, Iran University of Science and Technology, Tehran, Iran. E-mail: saman_salehyar@yahoo.com
2. School of Chemical, Petroleum and Gas Engineering, Iran University of Science and Technology, Tehran, Iran. E-mail: aghaemi@iust.ac.ir
3. School of Chemical, Petroleum and Gas Engineering, Iran University of Science and Technology, Tehran, Iran. E-mail: hmashhadimoslem@gmail.com
4. School of Chemical, Petroleum and Gas Engineering, Iran University of Science and Technology, Tehran, Iran. E-mail: shirvanim@iust.ac.ir

ARTICLE INFO

Article History:
Received: 10 November 2022
Revised: 09 January 2023
Accepted: 10 January 2023

Article type: Research

Keywords:
Silica Particles,
Cyclone,
Rotary Body,
Efficiency,
CFD

ABSTRACT

In this study, the separation of silica particles was investigated experimentally and numerically using a cyclone separator. Computational Fluid Dynamics (CFD) simulation was performed using a multi-phase Eulerian-Eulerian model for air-silica powder and a k-ε turbulent model. In the experiments, the effects of operating parameters including silica particle size, airflow rate, and rotational speed on cyclone efficiency were examined. The results showed that by increasing the particle size, the flow rate, and the body speed, the cyclone efficiency enhances. Furthermore, body rotation in the opposite direction of the inlet flow decreases cyclone efficiency by around 48%, and increasing the flow rate and rotation speed increases tangential velocity, resulting in increased centrifugal force and improved cyclone efficiency. The experimental and simulation performance maximums are about 97 percent and 90 percent, respectively. At a constant flow rate and particle size, a 1900 rpm rotating speed of the current direction of inlet flow increases performance by approximately 10-13 percent compared to a stationary body.

Introduction

Cyclone is a device used to dust and collect suspended particles in the gases. Cyclones perform separation based on centrifugal and gravitational forces. Operating parameters such as inlet gas velocity, particle size, longest radial path, rotational velocity, and temperature influence the design and cyclone efficiency [1-3]. Numerous studies on cyclones have been carried out in which the previous studies first examined the effects of cyclones geometry with considering a relative diameter for other cyclone fragments [4-6]. The findings of these studies revealed that the geometrical measurements of cyclones have a significant effect on their hydrodynamics [7-8]. In addition, some researchers looked into the influence of temperature and the amount of steam in the inlet gas on cyclone performance. [9-11]. The viscosity of the

* Corresponding Author: A. Ghaemi (E-mail address: aghaemi@iust.ac.ir)

Journal of Chemical and Petroleum Engineering, 2023, 57(1): 63-79.
Publisher: University of Tehran, College of Engineering *DOI: 10.22059/jchpe.2023.351023.1415*

gases increases as the temperature rises, and the cyclone performance reduces as a result. If the gas temperature is very high (above 400°C), the cyclone body's inner surface must be lined with refractory bricks, and the outlet must be filled with heat-resistant or ceramic covers. When the gas is at a moderate temperature, it should be at least 20 to 25 °C above the dew point of the gas, and the cyclone's surface outlet should be filled with heat insulation for this reason. Since air does not penetrate the cyclone or vice versa, gates must be installed on the outlet body to prevent air from entering. The amount of vapors in the gas has a corrosive effect on the dew point of the gas. The authors looked into the cohesion and adhesion of dust particles to the cyclone wall, as well as the frequency of dust discharge, dust corrosion properties, and cyclone wall erosion rate as effective parameters on cyclone efficiency [12-14]. The researchers looked through mathematical models with various parameters and created various models to decide the proper geometry of the cyclones, with the cyclone diameter being the most important parameter in cyclone design. The cyclones' other geometrical properties were calculated based on their particular ratio to the body's diameter [15-16]. The effects of increasing flow rate and particle size on performance were investigated, and the findings revealed that increasing flow rate and particle size improved efficiency. The results of using two symmetric inlets for feed have recently been investigated, and previous studies on increasing flow rate and particle size on single-input cyclones have been replicated for this cyclone model. The findings revealed that increasing the flow rate and particle size improves cyclone performance, and using two inlets improves cyclone efficiency in general [9, 17]. The separation performance of cyclones has been simulated using numerical approaches such as the computational fluid dynamics (CFD) methodology. Venkatesh et al. [18] reported that the separation pattern and separation efficiency were obtained using CFD simulations, and these results showed that the optimal square cyclone had the best results. In another research, the flow pattern of a cyclone that collects large particles and adjoining cyclones that collect smaller particles was studied as experimental and CFD simulation [19, 20]. As shown above, several studies have been conducted to study and investigate the effects of particle agglomeration on separation efficiency. Furthermore, only a few experimental and numerical experiments have been conducted to investigate the influence of reverse rotation on separation efficiency [21-23].

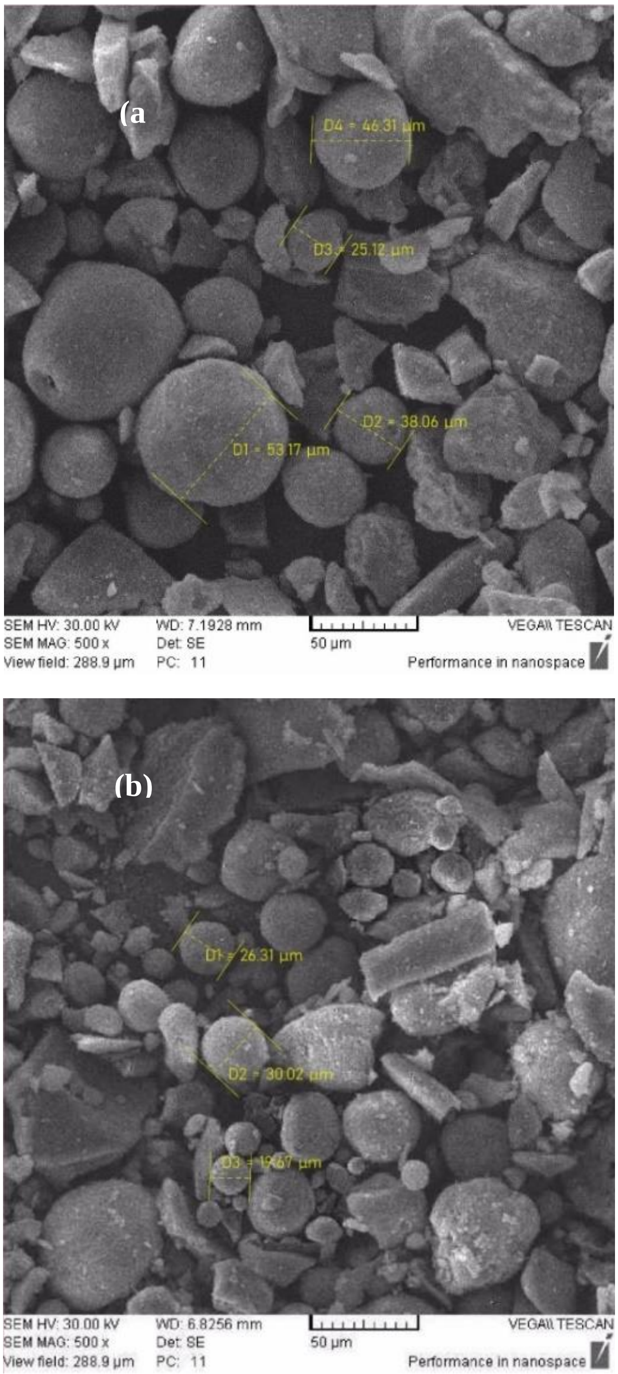
The aim of this research is to examine the effects of rotating the cyclone body on the performance of a two-inlet feed cyclone model using experimental and numerical methods. In contrast to a stationary cyclone, it was expected that moving the cyclone body in the direction of the inlet flow would help to increase performance. Furthermore, the effects of different parameters on cyclone efficiency and particle volume distribution in the cyclone are studied, including pressure drop, body rotation speed, rotation direction, and particle volume distribution in the cyclone. Due to the development of more centrifugal force and a turbulent flow, the effect of reverse rotation on cyclone performance was investigated using an experimental and numerical method.

Experiments

Material and Methods

Silica powder with a density of 2620 kg/m^3 was used in this study. These powders were supplied by Abadan Refinery Catalyst Cracking Unit, which was first milled due to its coarseness and then passed through a low-yield cyclone repeatedly to produce the final product. Thus, the dust outlet pipe at the bottom of the cyclone was removed and particles passed through the cyclone faster than allowed to enter the cyclone. Initially, the available powder was passed through a cyclone with low-yield several times and the upper and lower outputs were separated.

This is repeated several times to ensure that the output has more the same size particles. Similarly, three particles with an average size of 15, 25, and 40 μm were prepared using a low-yield cyclone to prepare silica powders of the same size. Scanning electron microscope (SEM) was used to investigate the structural properties of the powder and its results are shown in Fig. 1.



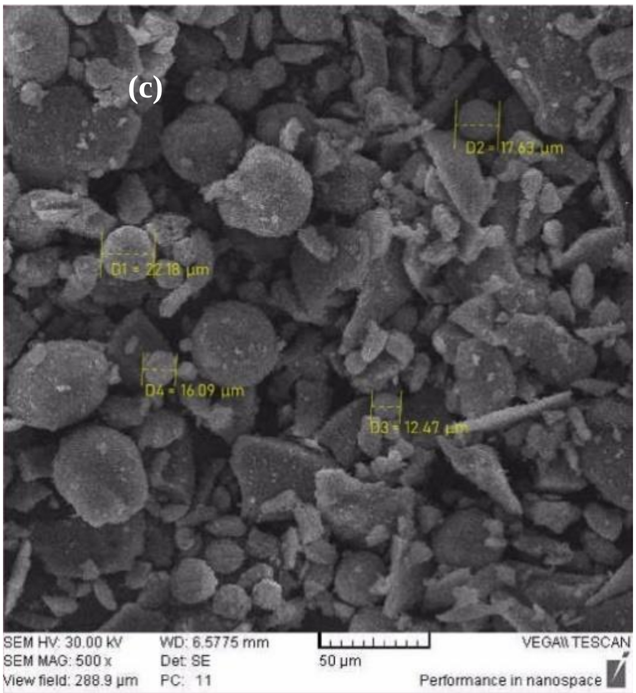


Fig. 1. SEM images of silica particles (a) 40 µm (b) 25 µm (c) 15 µm

Experimental Set-Up

A schematic view of the experimental set-up used in this study is depicted in Fig 2.

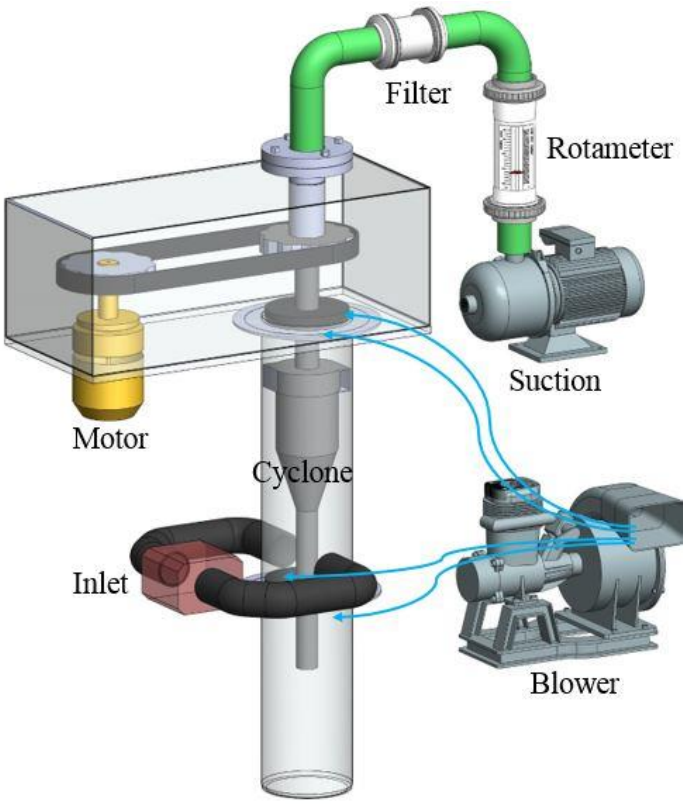


Fig. 2. Experimental set-up schema

The experimental set-up included the instruments such as a motor, stroboscope, bearing, blower, sucker, rotameter, pulp, balancer, filter, powders, and cyclone. The electrical motor with different speeds was used to rotate the cyclone body. Another advantage of this motor is that it can also be rotated counter-clockwise, which could be adjusted by rotating the chassis. The stroboscope was used to locate the device by turning the lamp on and off while the device was running. By rotating the chassis embedded behind the device, the speed of light switched on and off was raised to the point where only the object could be seen at one point. At the moment that the object was observed to be stationary at one point, the number on the chassis is the speed rate per minute. The upper and lower cyclone outlet tubes were attached to the bearing to generate the rotation. These bearings were selected as 2RS (for more resistance to soil diffusion) with two different models 6208 and 3206 for the top and bottom of the cyclone, respectively. To prevent entering the dust and sitting on the bearings, the cases were mounted on both sides of the bearing and blown into each one with a tube. The technical specifications of the blower are given in Table 1.

Table 1. Technical specifications of blower and suction pump.

Voltage (V)	Power (kW)	Current (A)	Frequency (Hz)	Suction pressure (kPa)	Pressure (kPa)	Speed (r/min)	Max flow rate (m^3/hr)
380	4	8.17	50	-28	38	2800	250

Instead of blowing, suction was used to feed into the cyclone in the tests, which changed the flow rate. One of the experimental parameters, the suction flow rate, was informed and adjusted using a rotameter. A filter is mounted at the outlet above the cyclone to collect dust escaping from the cyclone and calculate the volume of cyclone dust by measuring filter weight shifts. The cyclone inputs are built to be as close to the Stairmand model as possible in terms of proportions and scale. The Stairmand model is used to plan the dimensions and scale of the cyclone inputs. Due to the use of two inputs and similarly symmetrical inputs on both sides, the necessary surface area for the dust input is halved when Stairmand considers geometrical equations centered on a dust inlet of two inputs. Extensive analysis into the geometry of cyclones has culminated in comparatively individual components of the cyclone having a diameter that can be considered in the fabrication of cyclones to achieve high efficiency [24]. The manufactured cyclone is located within another chamber, which is attached to the container on both sides by a tube to pour powders into the container during testing and to join the chamber symmetrically from the tubes on both sides and finally into the cyclone.

Experimental Procedure

The bearing plate was first placed in the cyclone chamber, then the whole chamber was added to the trash bin, and the bushings were covered at each stage of the experiment. The cyclone was inserted into the chamber, and the motor was connected to its surface, accompanied by the gears and the strap. A suction attachment flange was mounted on the suction side and attached to the filter, as well as from the filter to the rotameter output, in the upper chamber. From the rotameter input, another tube was attached to the vacuum cleaner. The blower was installed on the outlet, and four narrow tubes were broken into two bearings and blown into the chamber, each of which was placed in its bottle. 250 g of powder was used in a steady-state blower with a continuous flow rate of suction and a specific motor rpm, and the powders slowly entered the cyclone. This procedure was repeated until all of the powder had gone through the cyclone and the separation had been completed. Finally, the variations in weight between the filter and the bin were calculated. The overall filtered and filter weight variations were recorded to provide more accurate results, and the ratio of the bin to feed weight showed the efficiency in each experiment.

Modeling

Governing Equation

The ANSYS FLUENT software was applied to solve the governing equations on the fluid flow. The continuity and Navier-Stokes equations are represented as follows [25]:

Continuity equation:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation:

$$\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = \rho g_i - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j^2} \tag{2}$$

where t is time, u is velocity, ρ is the density, μ is the viscosity, p is pressure, x_i, and x_j are the Cartesian coordinate directions, respectively.

To simulate the turbulence in the cyclone, the Eulerian-Eulerian multiphase model for air-powder flow, the k-ε model is applied which its equations are presented below [26].

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k - C_{3\varepsilon}G_b) - C_{2\rho}\frac{\varepsilon^2}{k} S_\varepsilon \tag{4}$$

Where k is the kinetic energy, ε is the dissipation rate, μ_t is the turbulent viscosity as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

The turbulence generation is determined as follows:

$$G_k = -\rho \overline{u_i' u_j'} \frac{\partial u_j}{\partial x_i} \tag{6}$$

Other constants of $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, σ_k and σ_ε in Eqs. 3 and 4 are equal to 1.44, 1.92, 0.09, 1.0 and 1.3, respectively.

Boundary Conditions and Solution Algorithm

One of the important issues in numerical solutions is to determine the boundary and initial conditions. The appropriate boundary condition is a function of the flow regime type, the information on the input and output of the flow, and the compatibility of the solver type and the numerical algorithm used with the boundary condition. In the present study, the inlet velocity boundary condition was used for the input flows and the feed volume fraction was considered in the simulation. The output of the device was assumed to be zero due to suction and the pressure difference between the top and bottom of the cyclone was calculated by a pressure gauge and considered as boundary conditions. Multiple Reference Frame (MRF) method was used to apply the body rotation. The triangle mesh structure was used to simulate the cyclone. Continuous phase equations were discretized using the finite volume method and the SIMPLE algorithm was utilized to couple continuity and momentum equations. To approximate the values in the momentum and volume fraction equations, the discrete method of the QUICK scheme was applied. The turbulent equations are discretized using the first-and second-order schemes.

Mesh Independency

The mesh independency to reduce the simulation errors was studied. To ensure the independence of the simulation results from the mesh size and experiment validation, comparing of the simulation and experimental results have been done. The detailed mesh independency results are given in Table 2.

Table 2. Comparison of mesh independency results

Processing Time (hr)	Average Error (%)	Experimental Efficiency (%)	Simulation Efficiency (%)	Mesh Number (-)
7.5	16.66	97.97	81.31	18223
15.5	13.49	97.97	84.48	35132
19.0	7.24	97.97	90.73	64985
29.0	1.03	97.97	87.94	87562
35.0	12.83	97.97	85.14	109542

According to the results, the mesh number 64985 was used for simulation. As the mesh increases, the average error decreases, but due to increasing the mesh sizes, causes little change in the results. So, the mesh size was optimized and provided a higher quality of results. The optimal mesh size causes a decrease in error in comparison to the experimental data and reduces the computational time. The final structural mesh grid is shown in Fig. 3.

Fig. 3. Structural triangle mesh grid

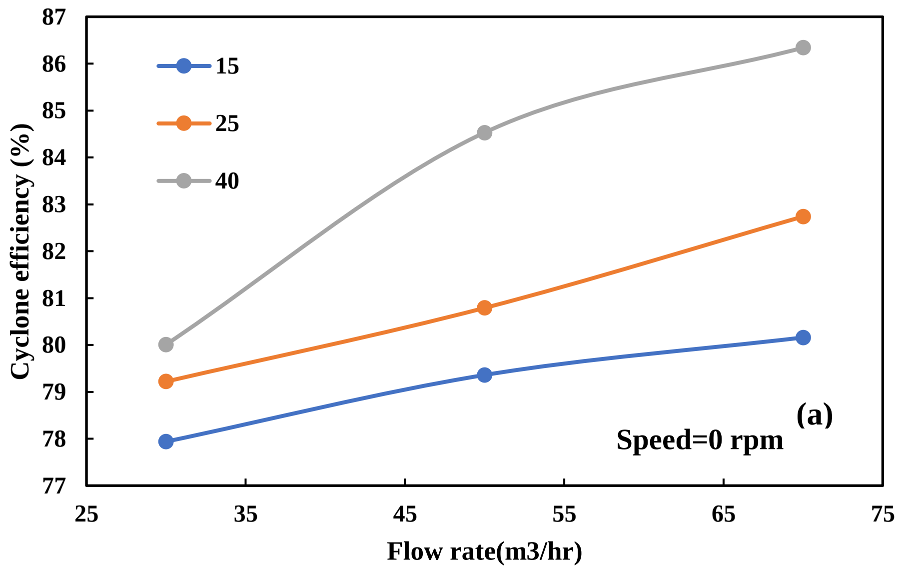
Results and Discussion

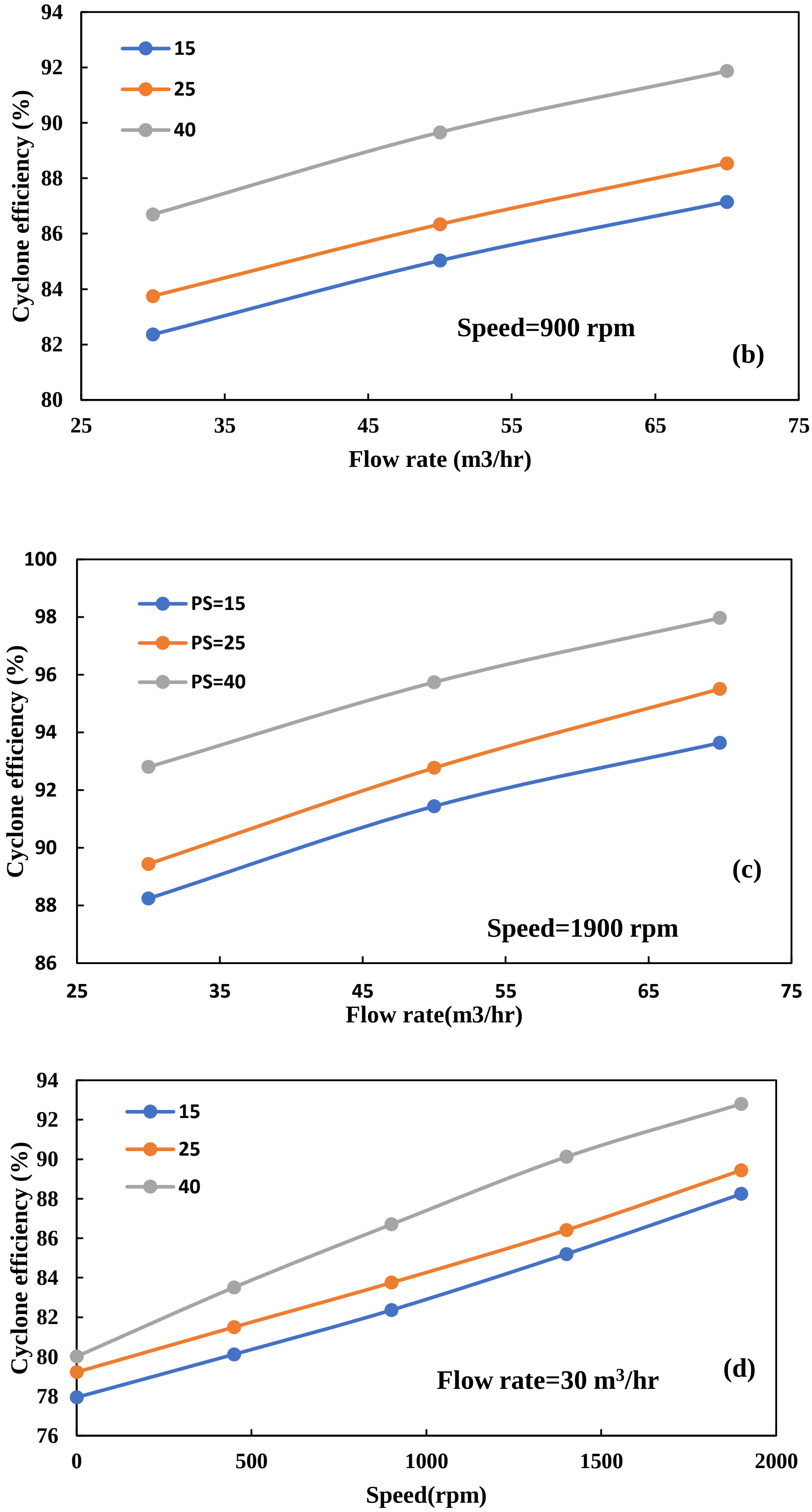
Experimental Results

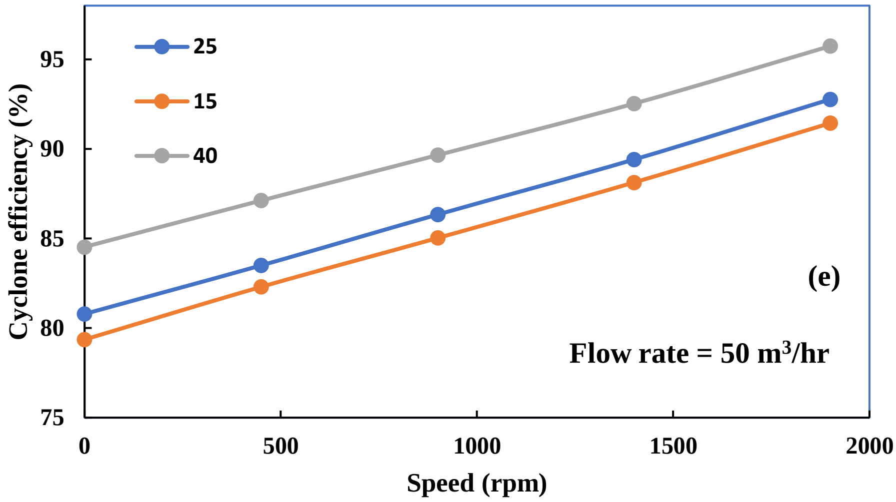
The efficiency of cyclones is highly dependent on both operating conditions and system geometry. Proper selection of these geometrical and operational variables can expect the system with good performance. On the other hand, if the particles are well separated and the accumulation of particles at the same size in each experimental sample is such that the large volume of each sample is the powders with equal size, it may be a suitable criterion to obtain accurate results.

Rotation in Inlet Flow Direction

First, the effects of cyclone body rotation speed in the inlet direction on cyclone efficiency at the same flow rate and powder size are investigated. Experiments were performed in the cyclone at a speed range of 0-1900 rpm, different particle sizes of 15, 25 40 μm, and flow rates of 30, 50 and 70 m^3/hr. As expected, the increase in rotation speed increased efficiency. As shown in Figs. 4a to 4f, the speed increases, and the centrifugal force is increased due to the increase in tangential velocity, which increases particle velocity along the wall. This causes the particles to move away from the cyclone's inlet flow, resulting in recirculation and outflow from the top of the cyclone. Another effect of the cyclone body rotation can be to make the movement more uniform, which allows for good separation. As the body rotates, particles are more likely to accumulate along the cyclone wall. Increasing the accumulation of particles along the walls indicates that the body velocity produces a good centrifugal force, which is much higher than the stationary body, resulting in an increase in efficiency.




Cyclone efficiency (%)
15
25
40
Speed=900 rpm
(b)
Flow rate (m3/hr)
PS=15
PS=25
PS=40
(c)
Speed=1900 rpm
Flow rate(m3/hr)
Flow rate=30 m3/hr
(d)
Speed(rpm)



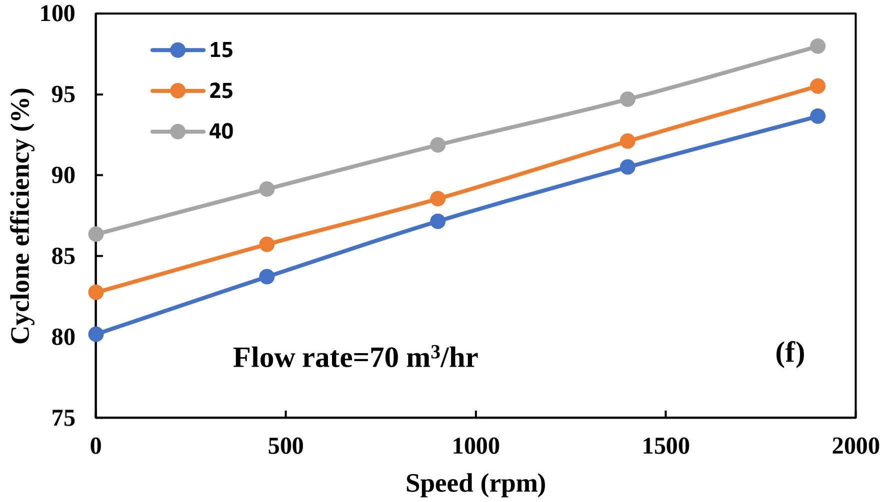


Fig. 4. Comparison of cyclone efficiencies at various rotation speeds, particle sizes, and flow rates (a-f)

The experiment was repeated at different flow rates, but the results were approximately the same, suggesting that increasing the rotation speed influenced the efficiency. The experimental results showed that increasing the rotation speed by up to 1900 rpm improves the efficiency from 10 to 13% relative to the stationary body. According to the laboratory results, the improvement of efficiency by increasing speed from zero to 900 rpm is about 4 to 5% and about 6 to 7% from 900 to 1900 rpm. To understand the efficiency changes with different parameters, the results are shown in Figs. 4a to 4f as a comparison of cyclone efficiency at different rotation speeds, particle sizes, and flow rates.

Rotation in Reverse Direction of Inlet Flow

In this section, to ensure that the cyclone body is rotated correctly, several tests on the flow rate and particle size were performed by varying the rotation speed in the opposite direction of the inlet flow. As can be deduced from Table 4, the choice of the reverse for the cyclone body rotation strongly reduced cyclone efficiency and indicates that the choice of the current direction was appropriate. In case of the body moves in the opposite direction, the turbulence within the cyclone increase. Increasing turbulence within the cyclone causes the entering dust to the middle flow of the cyclone outlet and exits the cyclone with clean air, which greatly reduces cyclone efficiency. Table 3 shows the cyclone efficiency at a volume flow rate of 70 m^3/hr, 40 μm powders, and a rotation speed range of 0-1900 rpm in the current and reverse directions of inlet flow. By rotating the cyclone body up to 1900 rpm in the reverse direction of inlet flow, the efficiency will decrease by about 48% and at 900 rpm by about 35%.

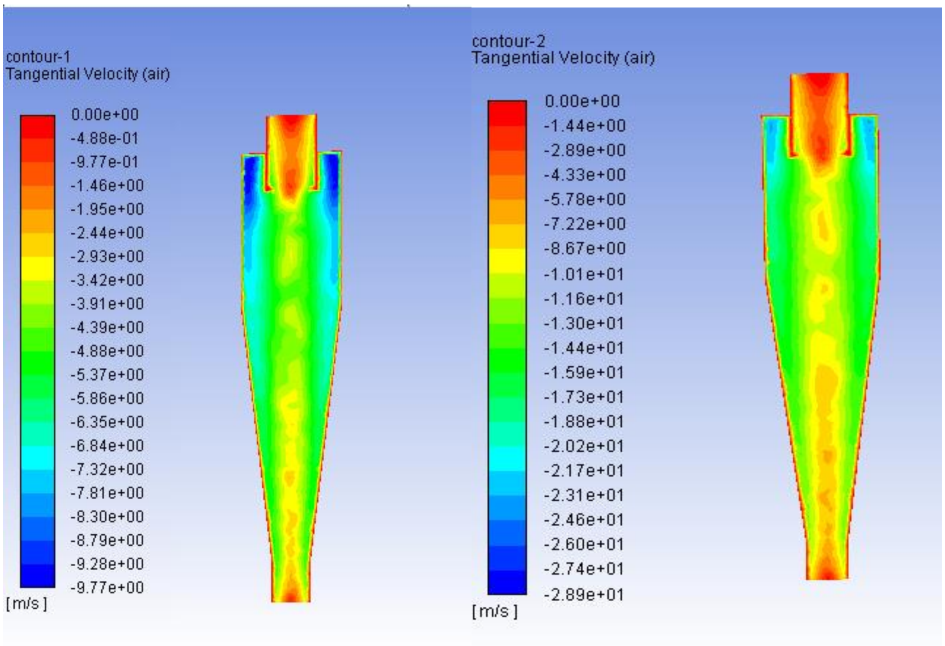
Table 3. Cyclone efficiency in the inlet flow at a volume flow rate of 70 m^3/hr with powders size of 40 μm.

Speed (rpm)	Efficiency Current direction (%)	Efficiency Reverse Direction (%)
0	86.34	86.34
450	88.95	68.47
900	91.87	57.22
1400	94.12	53.65
1900	97.97	49.87

Simulation Results

Rotation Speed Effects

By checking the tangential speed, we can check the effect of centrifugal force on the particles inside the cyclone, and as a result, the separation speed can be analyzed. The tangential velocity profile is shown in Figs. 5a and 5b at two volume flow rates of 30 and 70 m^3/hr, respectively. As shown in the figures, a certain uniformity is observed inside the cyclone for the velocity from the two sides of the cyclone, and the flow is uniformly propagated from the top to the bottom of the cyclone. As tangential velocity increases, resulting in a more centrifugal force on the particles and more particles bending to the outer wall, thereby the separation rate improves. Also, other effects of increasing velocity are the proper distribution of the solid phase at the cyclone level, which increases the cyclone efficiency and decrease the solid volume fraction in the upstream cyclone.



(a) **(b)**

Fig. 5. Tangential velocity profile in the cyclone (a) 30 and (b) 70 m^3/hr

The velocity profile in the cyclone is seen in Fig. 6 at two rotating speeds of 0 and 1900 rpm. If the tangential velocity increases, the centrifugal force increases, indicating that increasing the rotating speed increases the centrifugal force. The particle velocity along the wall increases as a result, and the particles move away from the cyclone's backward internal flow. As a result, the backward and outlet air from the top of the cyclone has fewer particles and the cyclone efficiency increases. Another effect of the cyclone body rotation speed is more uniform movement and tracing of the particles, which allows for good separation. The simulation results have an average error of around 11% as compared to the experimental data, but they support the improvement in performance for the experiment results. By viewing at the velocity contours, it can be seen that raising the body's rotation speed has a significant impact on the centrifugal force, and particles appear to collect around the cyclone wall when the body is moving. The accumulation of particles in the walls means that efficiency is improving.

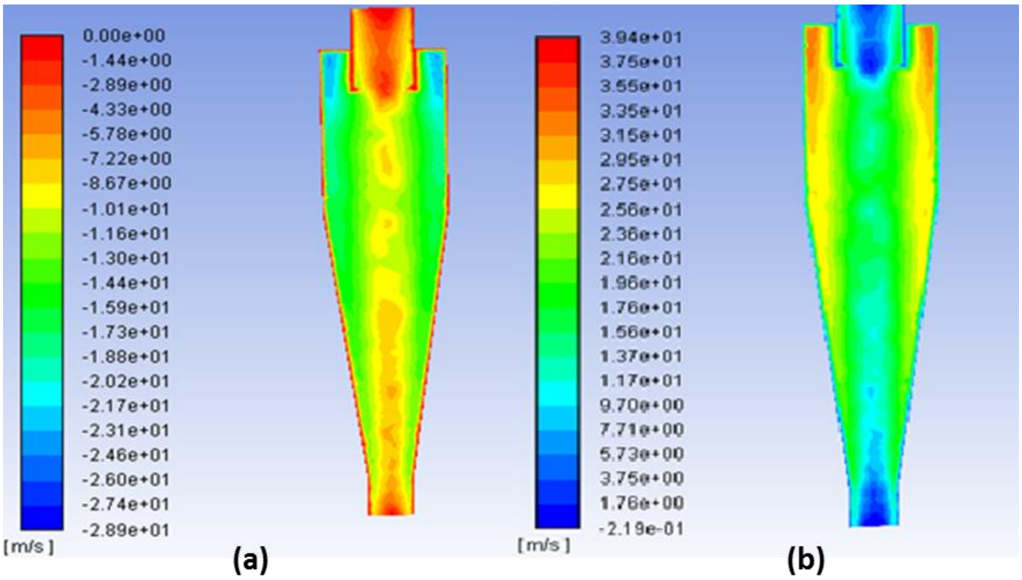


Fig. 6. Velocity profile in (a) stationary cyclone (b) cyclone with body rotation speed at 1900 rpm

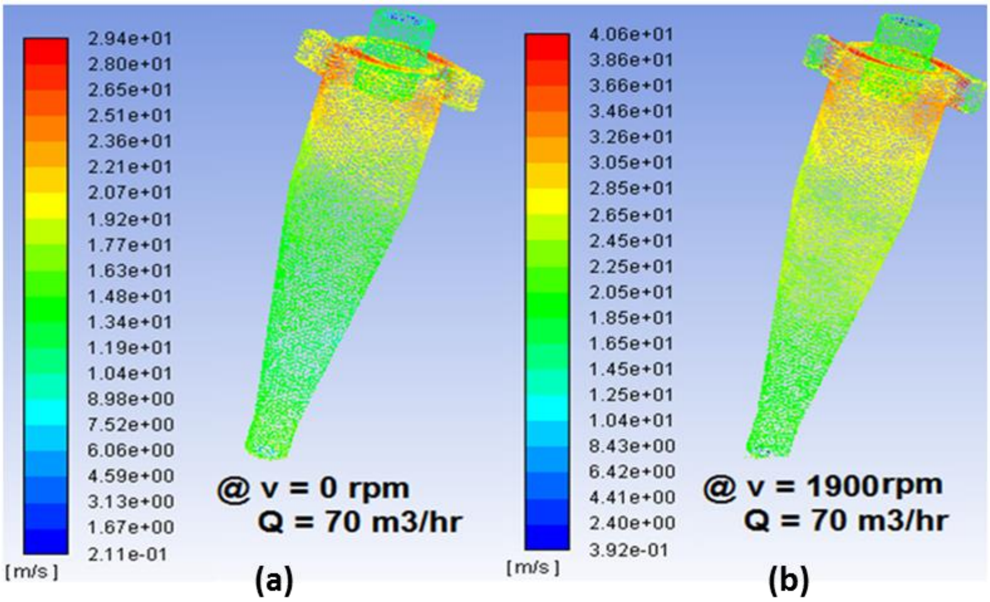


Fig. 7. Velocity vectors in cyclone at volume flow rate of (a) 30 and (b) 70 m^3/hr

The results from the velocity vectors in Fig. 7, the effects of rotation speed on generating the centrifugal force of the stationary cyclone. As can be seen in the rotational state, the velocity is increased to the lower proximity of the cyclone cone, which means that the effects of centrifugal force at the separation point are greatly increased. The results show that with the particle-free backward flow, the cyclone efficiency increase.

Volume Fraction of Particles

Fig. 8 shows the volume fraction of 40 μ particles in the stationary body relative to the cyclone body rotation speed of 1900 rpm at a specific time. As can be observed, increasing the cyclone body's rotation speed, as well as the particle uniformity in the cyclone's radial direction, reduces the size of the boundary layer. These results indicate that the particle tends to accumulate around the cyclone body more than the stationary body and the cyclone efficiency increases. Approaching the particles to the cyclone body and removing them from the outlet pipe is one of the important factors in increasing cyclone efficiency.

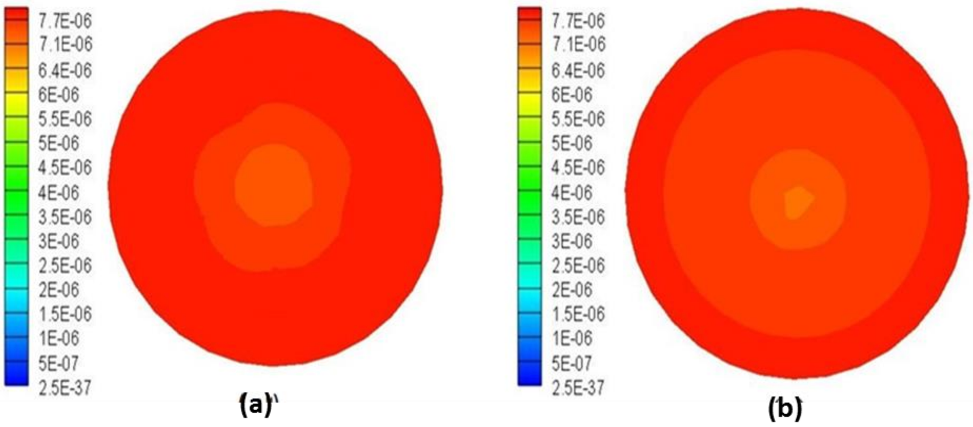


Fig. 8. Volume fraction of particles at (a) stationary body and (b) rotational body with a speed of 1900 rpm

Pressure Profile

Fig. 9 shows the pressure profile at two speeds of 0 and 1900 rpm and a flow rate of 70 m3/hr. With a flow rate of 30 m^3/hr, the pressure differential between the cyclone inlets and exits in the stationary body is approximately 280 Pa, and with a flow rate of 70 m^3/hr, it is approximately 470 Pa. This pressure differential approached 510 Pa at a flow rate of 70 m^3/hr and a maximum rotation speed of 1900 rpm.

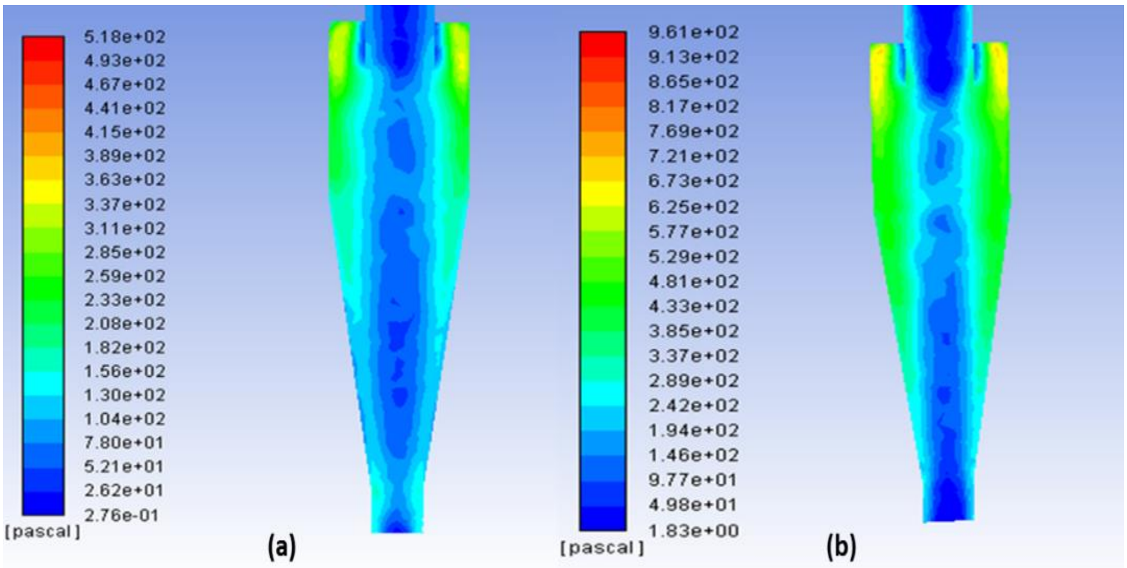


Fig. 9. Pressure profile at flow rate of 70 m^3/hr and the rotation speed of (a) 0 and (b) 199 rpm

Since the pressure differential in the stationary body was less than in the rotational situation, as seen in Fig. 9, performance is assumed to be lower. As can be observed, rotating the cyclone body uniformizes the pressure differential profile and reduces the pressure difference in the walls, resulting in a significant improvement in inefficiency. The pressure profiles also indicate that the pressure spread around the cyclone body is in a rotating state and that transferring the pressure differential to the cyclone's core improves cyclone performance.

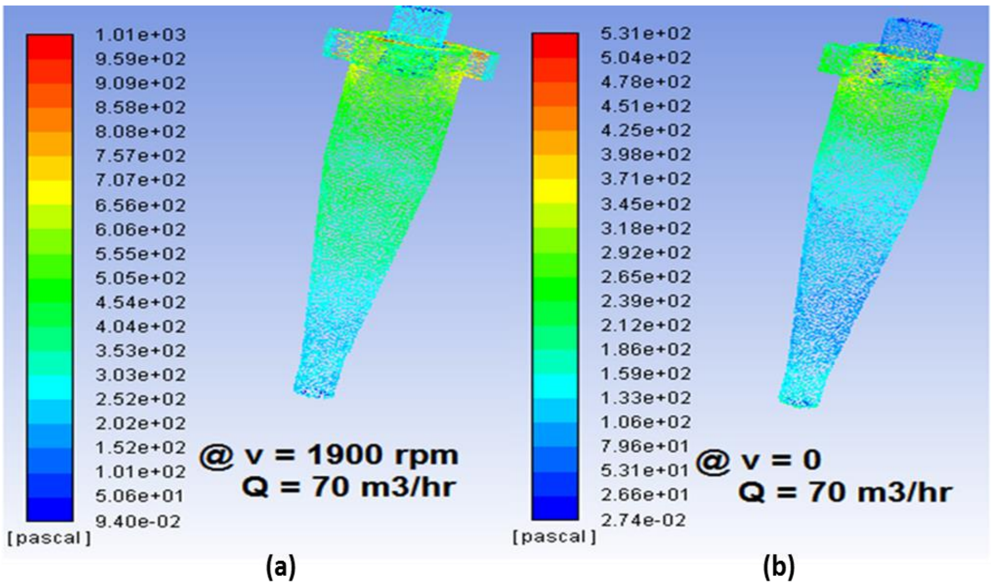


Fig. 10. Pressure vectors at a flow rate of 70 m^3/hr and the rotation speed of (a) 1900 rpm and (b) 0 rpm

Fig. 10 shows the pressure vectors at 0 and 1900 rpm rotating speeds. Although the pressure drop in the stationary body was less than in the rotary body, the pressure change at various points is so small that the pressure differential at each point and its neighboring point is negligible. However, considering the pressure vectors in the stationary cyclone body, each point's pressure differential from the next is important. Despite the higher-pressure decrease, the rotary cyclone shows an improvement in performance in comparison to the stationary one.

A comparison of simulation results and experimental data for 40 μm particles at different speeds is shown in Fig. 11. The results show that at higher velocities, the cyclone efficiency for these particles is higher than at low velocities, which also confirms the experimental results.

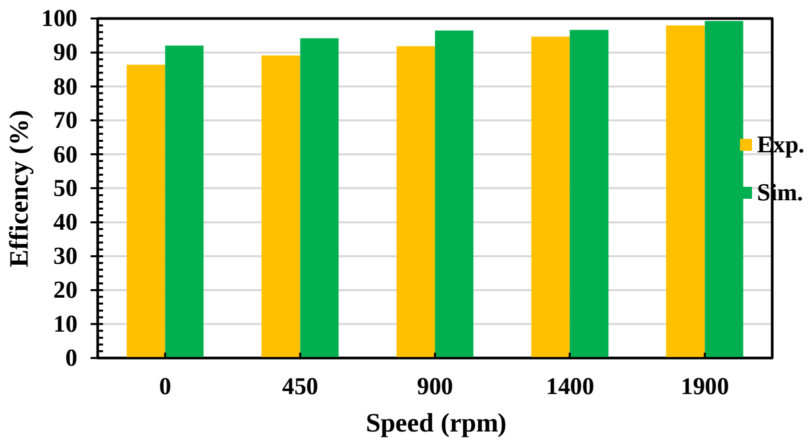


Fig. 11. Comparison of simulation and experimental results of different speed for 40 μm particle

Conclusions

This research aims to see how the rotation of the cyclone body affects its efficiency in comparison to the stationary situation, which has been studied both experimentally and numerically. Three key assumptions can be drawn from the experimental and simulation results:

- Experimental results showed that the rotation speed of 1900 rpm in the current direction of inlet flow increases the efficiency of approximately 10-13% relative to the stationary body at a constant flow rate and particle size. The higher rotation speed caused greater effects of rotation speed on the cyclone efficiency.
- As the cyclone body is rotated at the same speed of 1900 rpm in the opposite direction of the inlet flow, the cyclone's performance drops by around 48%. At the same flow rate and particle size, which is due to internal flow and higher-pressure decrease at the cyclone, higher inlet mixing air currents, and a change in centrifugal force compared to the stationary body. As a result, it can be observed that the cyclone efficiency is influenced by the rotational direction of the cyclone.
- The cyclone rotation in the inlet flow direction leads to a more uniform distribution of pressure and volume fraction of particles, creating the tangential velocity and increasing the centrifugal force, thereby increasing the tendency of the particle to be near the cyclone wall and the cyclone efficiency.

The spinning cyclone pressure drop was greater than the stationary case at the specific flow rate and particle size, according to the simulation results. The cyclone rotation causes pressure variations from the top to the bottom of the cyclone continuously and with a small variation, which reduces the cyclone's efficiency compared to a stationary cyclone. The simulation is promising for studying the conditions for increasing the efficiency of cyclones and allows for the prediction of results by modifying various parameters.

References

[1] Li, Yuan, Guoliang Qin, Zhiyi Xiong, YunFeng Ji, and Ling Fan. "The effect of particle humidity on separation efficiency for an axial cyclone separator." Advanced Powder Technology 30, no. 4 (2019): 724-731. https://doi.org/10.1016/j.apt.2019.01.002

[2] Noh, Seung-Yoon, Ji-Eun Heo, Sang-Hee Woo, Sang-Jun Kim, Myeong-Hak Ock, Young-Jin Kim, and Se-Jin Yook. "Performance improvement of a cyclone separator using multiple subsidiary cyclones." Powder Technology 338 (2018): 145-152. https://doi.org/10.1016/j.powtec.2018.07.015

[3] Zhang, Wei, Linlin Zhang, Jingxuan Yang, Xiaogang Hao, Guoqing Guan, and Zhihua Gao. "An experimental modeling of cyclone separator efficiency with PCA-PSO-SVR algorithm." Powder Technology 347 (2019): 114-124. https://doi.org/10.1016/j.powtec.2019.01.070

[4] Cristea, Eugen-Dan, and Pierangelo Conti. "CFD simulation of large dust collection cyclones positioned vertically in staggered downward cascade arrangement." In Fluids Engineering Division Summer Meeting, vol. 55546, p. V01AT03A010. American Society of Mechanical Engineers, 2013. https://doi.org/10.1115/FEDSM2013-16245

[5] Kulkarni, Sunil J., Nilesh L. Shinde, and Ajaygiri K. Goswami. "A review on ethanol production from agricultural waste raw material." International Journal of Scientific Research in Science and Technology 1, no. 4 (2015): 231-233.

[6] Woolcock, Patrick J., and Robert C. Brown. "A review of cleaning technologies for biomass-derived syngas." Biomass and bioenergy 52 (2013): 54-84. https://doi.org/10.1016/j.biombioe.2013.02.036

[7] Liu, Peiqi, Yintian Ren, Mingyu Feng, Di Wang, and Dapeng Hu. "A performance analysis of inverse two-stage dynamic cyclone separator." Powder Technology 351 (2019): 28-37. https://doi.org/10.1016/j.powtec.2019.04.002

[8] Wasilewski, Marek, and Lakhbir Singh Brar. "Effect of the inlet duct angle on the performance of cyclone separators." Separation and Purification Technology 213 (2019): 19-33. https://doi.org/10.1016/j.seppur.2018.12.023

[9] Huang, Yiqun, Man Zhang, Junfu Lyu, Zhi Liu, and Hairui Yang. "Effects of gas leakage on the separation performance of a cyclone. Part 2: Simulation." Chemical Engineering Research and Design 136 (2018): 906-915. https://doi.org/10.1016/j.cherd.2018.06.002

[10] Kosaki, Yuhei, Takayuki Hirai, Yoshinari Yamanaka, and Keishi Takeshima. "Investigation on dust collection and particle classification performance of cyclones by airflow control for design of cyclones." Powder Technology 277 (2015): 22-35. https://doi.org/10.1016/j.powtec.2015.02.051

[11] Ramachandran, Gurumurthy, David Leith, John Dirgo, and Henry Feldman. "Cyclone optimization based on a new empirical model for pressure drop." Aerosol Science and Technology 15, no. 2 (1991): 135-148. https://doi.org/10.1080/02786829108959520

[12] Akhbarifar, Sepideh, and Mansour Shirvani. "Improving cyclone efficiency for small particles." Chemical Engineering Research and Design 147 (2019): 483-492. https://doi.org/10.1016/j.cherd.2019.05.026

[13] Casal J, Martinez JM (1983) A better way to calculate cyclone pressure drop. Chemical Engineering 90(2): 90–99.

[14] Shepherd CB, CE Lapple. "Flow pattern and pressure drop in cyclone dust collector's cyclone without intel vane." Industrial & Engineering Chemistry 32, no. 9 (1940): 1246-1248. https://doi.org/10.1021/ie50369a042

[15] C Cortes, A Gil, Progress in energy and combustion Science 2007, 33, 409-452. https://doi.org/10.1016/j.pecs.2007.02.001

[16] J Gimbun, T Chuah, A Fakhru'l-Razi, TS Choong, Chemical Engineering and Processing: Process Intensification 2015, 44, 7-12. https://doi.org/10.1016/j.cep.2005.06.001

[17] Z Sun, G Sun, P Peng, Q Liu, X Yu, Chemical Engineering Research and Design 2019,145, 141-149. https://doi.org/10.1016/j.cherd.2019.03.018

[18] Venkatesh, S., M. Sakthivel, H. Saranav, N. Saravanan, M. Rathnakumar, and K. K. Santhosh. "Performance investigation of the combined series and parallel arrangement cyclone separator using experimental and CFD approach." Powder Technology 2020, 361, 1070-1080. https://doi.org/10.1016/j.powtec.2019.10.087

[19] Venkatesh, S, R. Suresh Kumar, S. P. Sivapirakasam, M. Sakthivel, D. Venkatesh, and S. Yasar Arafath. "Multi-objective optimization, experimental and CFD approach for performance analysis in square cyclone separator." Powder Technology 2020, 371, 115-129. https://doi.org/10.1016/j.powtec.2020.05.080

[20] Zhou, Faqi, Guogang Sun, Xiaopeng Han, Yong Zhang, and Wenqun Bi. "Experimental and CFD study on effects of spiral guide vanes on cyclone performance." Advanced Powder Technology 29, 2018,no. 12, 3394-3403. https://doi.org/10.1016/j.apt.2018.09.022

[21] Wang, Jiarui, Xudong Duan, Simin Wang, Jian Wen, and Jiyuan Tu. "Experimental and numerical investigation on the separation of hydrophilic fine particles using heterogeneous condensation preconditioning technique in gas cyclones." Separation and Purification Technology 2021, 259 118-126. https://doi.org/10.1016/j.seppur.2020.118126

[22] Venkatesh, S., S. P. Sivapirakasam, M. Sakthivel, S. Ganeshkumar, M. Mahendhira Prabhu, and M. Naveenkumar. "Experimental and numerical investigation in the series arrangement square cyclone separator." Powder Technology 2021, 383, 93-103. https://doi.org/10.1016/j.powtec.2021.01.031

[23] El-Emam, Mahmoud A., Ling Zhou, Wei Dong Shi, and Chen Han. "True shape modeling of bio-particulate matter flow in an aero-cyclone separator using CFD–DEM simulation." Computational Particle Mechanics 2021, 1-17. https://doi.org/10.1007/s40571-020-00383-w

[24] Zhu, Zhiping, Yongjie Na, and Qinggang Lu. "Pressure drop in cyclone separator at high pressure." Journal of thermal science 17 (2008): 275-280. https://doi.org/10.1007/s11630-008-0275-7

[25] Temam, Roger. Navier-Stokes equations: theory and numerical analysis. Vol. 343. American Mathematical Soc., 2001.

[26] S Khalid, MSH Aly, MA Wahid, MM Sies. "Numerical Simulation of Confined Vortex Flow Using a Modified k--ϵ Turbulence Model." CFD letters 1, no. 2 (2009).

How to cite: Salehyar SS, Ghaemi A, Mashhadimoslem H, Shirvani M. Experimental and Numerical Studies on Improving Cyclone Efficiency by Rotation of Cyclone Body. Journal of Chemical and Petroleum Engineering. 2023; 57(1): 63-79.