



Numerical Study of Flow Field in New Design Dynamic Cyclone Separators

H. Safikhani*, F. Esmaeili, S. Salehfard

Department of Mechanical Engineering, Faculty of Engineering, Arak University, Arak, Iran

PAPER INFO

Paper history:

Received 04 September 2019

Received in revised form 16 October 2019

Accepted 08 November 2019

Keywords:

Dynamic Cyclone

New Design Cyclone

Collection Efficiency

CFD

Discrete Random Walk

ABSTRACT

In this paper the numerical simulation of flow field in the new design dynamic cyclone separator is investigated. The effects of different geometrical parameters such as impeller diameter, number of vanes, length and the location of impeller is investigated. The Reynolds averaged Navier–Stokes equations with Reynolds stress turbulence model (RSM) were solved. The Eulerian-Lagrangian computational procedure is used to predict particles tracking in the cyclones. The velocity fluctuations are simulated using the Discrete Random Walk (DRW). Results show that in new design dynamic cyclone, the collection efficiency is higher and the pressure drop is lower in compare with other studied on cyclones. Installing impeller at the lower positions leads to a decrease in efficiency. With increase in impeller diameter, pressure drop in dynamic new design decreases. Finally, with increase in impeller length, efficiency increases and pressure drop decreases.

doi: 10.5829/ije.2020.33.02b.22

1. INTRODUCTION

Separation of solid particles from gas-solid compound is very prominent in industries. Two important factors in design of separators are collection efficiency and pressure drop through the separators. Nowadays, cyclones are ubiquitous as pre-filtering in different industries like food industries, cement industries, heating, ventilating, and air conditioning systems, refineries and city gas stations. Therefore, it was already the focus of many studies.

Alexander [1] investigated conventional cyclone performance enhancement through analyzing the effect of different geometrical and process parameters. Stairmand [2] suggested proper height and length of the outer pipe to be 1.5 and 0.5 times of cyclone body diameter in order to achieve an efficient cyclone. Elsayed and Lacor [3] consider the effect of the input dimension of the cyclone on its performance and flow filed pattern was computationally studied. They found that change in input width is more effective on efficiency than the height of the input. Cortes and Gil [4] and Avci et al. [5] found that input entrance is an important parameter effective on

natural length of vortex. Zhao et al. [6] compared cyclones with two different types of input including conventional single entrance and helical double entrance. Numerical results proved that cyclone with helical double entrance can improve the symmetry of gas flow pattern, and consequently it has better collection efficiency. The effect of the angle of entrance was investigated by Qian and Zhang [7], Qian and Wu [8]. Furthermore, Erdal and Shirazi [9] studied three different geometries for entrance namely tilted single entrance, tilted double entrance, conical entrance and their effect on flow behavior inside the cyclone. Zhao et al. [10] compared three different entrance geometries namely single tangent entrance, direct symmetrical helical entrance and converging symmetrical helical entrance using a function of particle size and flow rate. Xiang and Lee [11], Chuah et al. [12] and Xiang et al. [13] studied on the effect of cone dimension on cyclone performance and flow behavior. They reported that when the cone entrance diameter is bigger than gas output diameter, the collection efficiency decreases without noticeable change in pressure drop. Yoshida et al. [14] studied different cone vertex angle for the entrance, the dust box

*Corresponding Author Email: h-safikhani@araku.ac.ir (H. Safikhani)

of the cyclone. Qian et al. [15] and Kaya and Karagoz [16] stated that the cone vertex angle effect decreases when initial velocity to the cyclone is high. The effect of a long cone with a dip-leg in the separation performance of a cyclone was studied. Yoshida et al. [17] and Kepa [18] investigated the negative effect of cone in the bottom of the cyclone on cyclone performance. Xiang and Lee [19] showed that an increase in the tangent velocity by e.g. decreasing output pipe diameter can enhance the capability of the separation and pressure drop decreases with increasing output pipe diameter. Chen and Liu [20] by using numerical simulation studied collection efficiency and pressure drop in a modified cyclone with new type of output in different depth and slope direction and slope angle. Kepa [21] numerically simulated the flow inside a cyclone when an internal vortex was inserted inside the cyclone. Hoffman et al. [22] and Safikhani et al. [23] concentrated on study of the height of the cyclones. Furthermore, the effect of holes in the cyclone and hem of the internal walls of the cyclone was studied by Wang et al. [24] and Farahani et al. [25] respectively. Su and Mao [26], Safikhani et al. [27] and Su et al. [28] investigated the performance of the rectangular cyclone. Kim et al. [29] studied different shapes of the cyclone body including plain surface, body with helical guide, body with circumferential slot and body with vertical slot.

Karagoz et al. [30] initiated a new design static cyclone model. New design cyclone body no longer consists of a cylinder and a cone. It was made of two cylinders and a vortex limiter. This design was tested again cement-air compound and results suggested that conventional cyclones and introduced one with the same size cannot be logically compared in terms of pressure drop. The decrease in friction surface in the new design cyclone can intensify the intensity and length of the vortex and therefore increasing pressure drop. The collection efficiency of the new design cyclone is higher than that in a conventional cyclone. Safikhani and Mehrabian [31] numerically studied the flow field in separation of the new cyclones. They investigated the reliance of the efficiency and pressure drop to the different geometrical parameters. Safikhani [32] studied the multi-objective optimization of the new cyclones using the combination of CFD, GDMH models and NSGA II algorithm. Zhiao et al. [33] investigated a dynamic cyclone with rotational impeller using experimental and numerical methods. Their focus was over the efficiency of separation. Jiao et al. [34] experimentally studied the efficiency of dynamic cyclones. Results revealed that the position of the rotational separator and the way floor opens to the input elevation are very influential on the efficiency of the cyclone. In this study the numerical simulation of flow field in new design dynamic cyclone separator is investigated. This novel type emerges from the

combination of dynamic cyclone and new design cyclones.

2. NUMERICAL MODELING

2.1. Geometry The design of new cyclone is based on the idea that the vortex length and separation performance can be increased by decreasing friction losses in a cyclone. The new design differs from a classical cyclone with the separation space. It has inner and outer cylinders without a conical part, and a vortex limiter [30]. The idea of new design dynamic cyclones is rooted in the studies when in order to increase efficiency and decrease pressure drop a rotational separator is designed for the new design cyclone. The geometry used in dynamic new design cyclones as can be seen in Figure 1 is similar to that of new design static cyclone and the only difference is the rotational impeller.

2.2. Governing Equation For a non-compressible fluid continuum and momentum equations can be written as the following:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} R_{ij} \quad (2)$$

Where \bar{u}_i is average speed, x_i is the location, \bar{P} is the average pressure, ρ is the constant density of gas, ν is kinetic viscosity and $R_{ij} = \bar{u}_i \bar{u}_j$ is Reynolds stress tensor. Here, $\acute{u}_i = u_i - \bar{u}_i$ is the i th fluctuating velocity component.

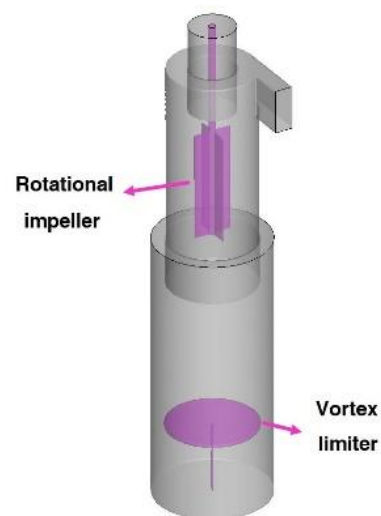


Figure 1. Deposition efficiency on a single square in channel

The RSTM provides differential transport equations for evaluation of the turbulence stress components where the turbulence production terms are defined as follows:

$$\frac{\partial}{\partial t} R_{ij} + \bar{u}_k \frac{\partial}{\partial x_k} R_{ij} = \frac{\partial}{\partial x_k} \left(\frac{\nu_t}{\sigma^k} \frac{\partial}{\partial x_k} R_{ij} \right) - \left[R_{ik} \frac{\partial \bar{u}_j}{\partial x_k} + R_{jk} \frac{\partial \bar{u}_i}{\partial x_k} \right] - C_1 \frac{\varepsilon}{K} \left[R_{ij} - \frac{2}{3} \delta_{ij} K \right] - C_2 \left[P_{ij} - \frac{2}{3} \delta_{ij} P \right] \quad (3)$$

$$P_{ij} = - \left[R_{ik} \frac{\partial \bar{u}_j}{\partial x_k} + R_{jk} \frac{\partial \bar{u}_i}{\partial x_k} \right] P = \frac{1}{2} P_{ij} \quad (4)$$

With P being the fluctuating kinetic energy production. ν_t is the turbulent (eddy) viscosity; and $\sigma^k = 1$, $C_1 = 1.8$, $C_2 = 0.6$ are empirical constants [35].

The transport equation for the turbulence dissipation rate, ε , is given as follows:

$$\frac{\partial \varepsilon}{\partial t} + \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial \varepsilon}{\partial x_j} \left[\left(\nu + \frac{\nu^t}{\sigma^\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] - C^{\varepsilon 1} \frac{\varepsilon}{K} R_{ij} \frac{\partial \varepsilon}{\partial x_j} - C^{\varepsilon 2} \frac{\varepsilon^2}{K} \quad (5)$$

In Equation (5), $K = \frac{1}{2} \overline{u'_i u'_i}$ is the fluctuating kinetic energy, and ε is the turbulence dissipation rate. The values of constants are $\sigma^\varepsilon = 1.3$, $C^{\varepsilon 1} = 1.44$ and $C^{\varepsilon 2} = 1.92$.

To solve the couple of velocity and pressure, SIMPLE algorithm is used. For this study in two-phase equations a one way coupling is used and Eulerian-Lagrangian model is used to model two-phase state. In this model continuous phase is assumed to be the gas and discrete phase is particles. Discrete equations are numerically solved for the particles using Rang-Kutta method. To calculate the path of particles in the fluid, discrete phase model (DPM) is used to track particles individually. Equation of motion for small particles includes non-linear effect of drag force and gravitational forces which is presented in following equation:

$$\frac{du_i^p}{dt} = \frac{3\nu C_D Re_p}{4d^2 S} (u_i - u_i^p) + g_i \quad (6)$$

$$C_D = \frac{24}{Re_p} \quad \text{for} \quad Re_p < 1 \quad (7)$$

$$C_D = \frac{24}{Re_p} \left(1 + 1 \frac{1}{6} Re_p^{\frac{2}{3}} \right) \quad \text{for} \quad 1 < Re_p < 400 \quad (8)$$

$$Re_p = \frac{|u_j - u_j^p| d}{\nu} \quad (9)$$

where u_i^p is particle's velocity, x_i is the location of the particle, d is the diameter of the particle, S is the ratio of particle density to flow density and g_i is gravity acceleration. The right side of Equation (6) is the drag force due to relative slippage of particles and flow. Drag force, generally, is the impelling force and drag coefficient can be computed using Equations (7) and (8) and Re_p is Reynolds number of the particles which is calculated using Equation (9). Impact of the particles to the walls of the cyclone is treated with elastic coefficient of 0.8 and it is assumed that impact between particles is negligible.

2. 2. Grid Generation Grid generation of the new design dynamic cyclone is depicted in Figure 2. This type of cyclone has 2 to 2.5 million number of grids. In order to be sure that results are independent from grids, eight different gridding systems are used. Figure 3 shows the independence of pressure drop from gridding system.

2. 3. Validation To attain confidence about the simulations, it is necessary to compare the simulation results with the available data. As can be seen from

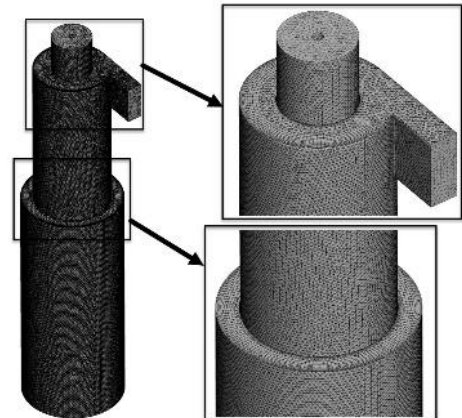


Figure 2. Deposition efficiency on a single square in channel

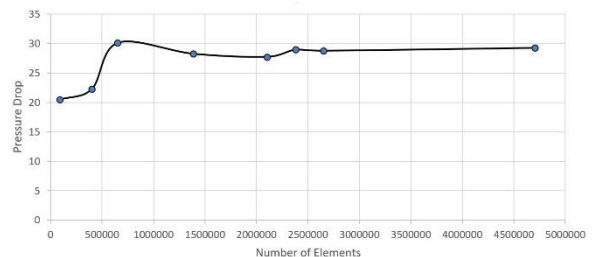


Figure 3. Deposition efficiency on a single square in channel

Figure 4, one can approve that numerical values are well correlated with experimental and analytical values. For validating the collection efficiency results, Table 1 compares the present numerical results and the related experimental data of Wang [36] for different PSDs in conventional cyclones. In Table 1, D_{50} is the diameter of a particle which has 50% probability of separation and 50% probability of escaping and is a criterion for comparing efficiency of cyclones. Physical characteristics of A–E particles are shown in Table 1. MMD and GSD are mass median diameter and geometric standard deviation, respectively. As shown in this table, a good agreement is observed between numerical predictions and experimental results.

3. RESULTS AND DISCUSSION

In this section, first the effect of variation in geometrical parameters on inlet velocity, pressure drop in the cyclone and turbulence inside new design dynamic cyclone is investigated. Then, comparing collection efficiency and pressure drop between static and dynamic new design cyclones with similar boundary conditions is followed.

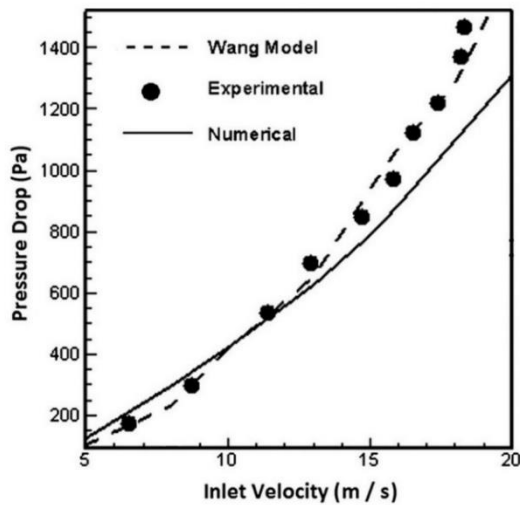


Figure 4. Deposition efficiency on a single square in channel

TABLE 1. Strouhal number for different geometric cases

Dust	PSD		D_{50}	
	ρ_p	MMD/GSD	Experimental	Numerical
A	1.77	20/2	2.74	2.41
B	1.82	21/1.9	3.75	3.32
C	1.87	23/1.8	3.60	3.151
D	1.52	19/1.4	-	7.1
E	2.37	13/1.7	4.40	4.07

Considered parameters include impeller diameter and number of blades, rotational speed of impeller, location of impeller, length of impeller and inlet speed.

3. 1. Effect of the Impeller Diameter

Three different diameters of impeller are investigated. As can be seen from Figures 5 and 6, turbulence in new design dynamic cyclone with minimum impeller diameter is the lowest. With increase in impeller diameter, first turbulence is increased. More increase in impeller diameter leads to a decrease in kinetic energy. As can be observed, increasing impeller diameter results in a reduction in pressure drop. However, collection efficiency shows a maximum for the middle impeller diameter.

3. 2. Effect of the Number of Blades

As shown in Figures 7 and 8, with increase in number of blades,

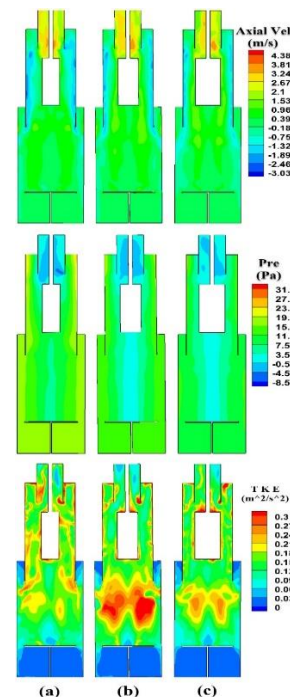


Figure 5. Effects of the impeller diameter on axial velocity, pressure distribution and turbulent kinetic energy contours

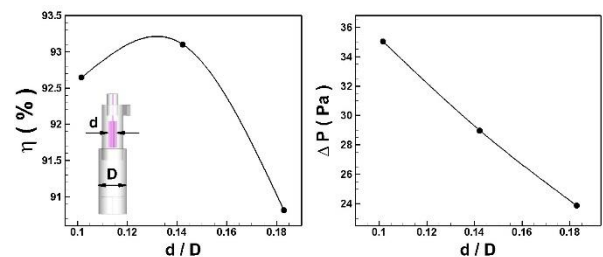


Figure 6. Effects of the impeller diameter on pressure drop and collection efficiency

axial velocity increases in the outlet cylinder. With lower number of blades, kinetic energy of turbulence slightly decreases. However, with increase in number of blades in a new design dynamic cyclone, turbulence kinetic energy dramatically decreases. As shown decreasing number of blades from an optimum value will lead to a decrease in both collection efficiency and pressure drop. However, increasing number of blades results in more severe decrease in collection efficiency and pressure drop.

3. 3. Effect of the Impeller Rotational Speed This is a prominent parameter which was studied in three different levels. As can be observed from Figures 9 and 10, higher turbulence kinetic energy can be achieved with higher impeller rotational speed. The area of which low turbulence existed with low rotational speed,

decrease dramatically with higher speeds. Moreover, turbulence increases in the lower cylinder. As depicted, pressure drop and collection efficiency have minimum and maximum points respectively. With increase in rotational speed, first pressure drop increases and then is constant. Furthermore, increasing rotational speed first leads to an increase in collection efficiency and then it efficiency reduces.

3. 4. Effect of the Location of Impeller To study the effect of the location of impeller it is located in three different locations. As illustrated in Figures 11 and 12, when the impeller installed at the maximum height, the

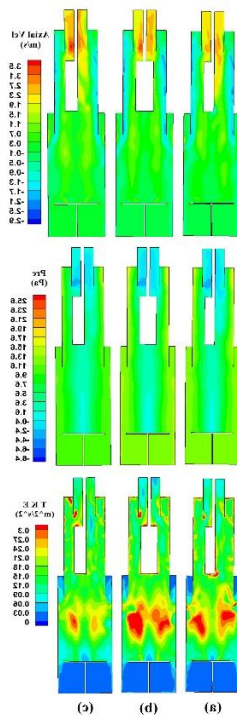


Figure 7. Effects of the impeller rotational speed on axial velocity, pressure distribution and turbulent kinetic energy contours

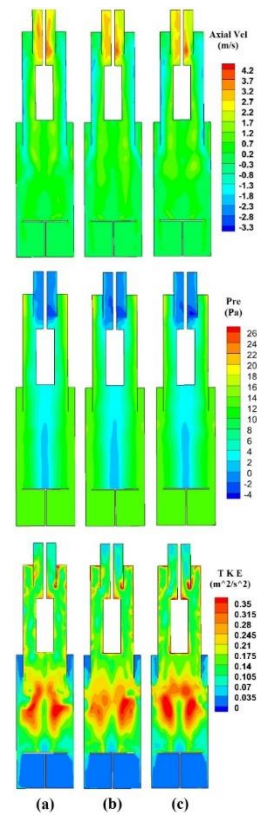


Figure 9. Effects of the impeller rotational speed on axial velocity, pressure distribution and turbulent kinetic energy contours

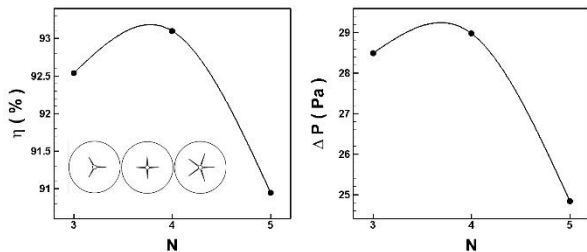


Figure 8. Effects of the number of blades on pressure drop and collection efficiency

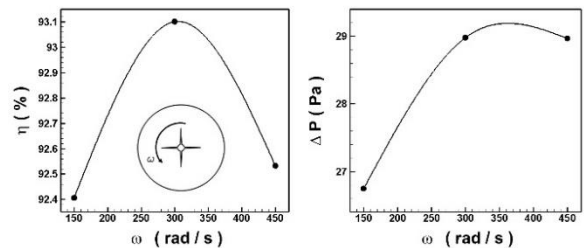


Figure 10. Effects of the impeller rotational speed on pressure drop and collection efficiency

axial speed in output cylinder decreases, a low pressure zone occurs in the central line of the cyclone and turbulence energy increases in the lower cylinder. On the other hand, installing impeller at the lowest height increases axial velocity in output cylinder and low pressure zone emerges in the output cylinder and turbulence kinetic energy in the lower cylinder stands at its lowest state. As depicted, the highest collection efficiency coincides with highest level of impeller installation, and with lowering impeller it gradually decreases. This decrease is due to the fact that particles exit before entering the rotational field of the impeller. The highest pressure drop occurs when impeller is installed at the highest position, but with its descend, pressure drop decreases. When impeller gets down from the middle position, pressure drop increases.

3. 4. Effect of the Impeller Length To investigate the effect of impeller length, three different sizes of impeller were investigated. As can be observed from Figures 13 and 14, with increase in length of the impeller, axial velocity increases within the cyclone and turbulence energy is maximized. Moreover, as illustrated, increasing the length of the impeller leads to an increase in collection efficiency. For pressure drop, first it dramatically decreases with increasing length of impeller and then its decrease becomes slight and

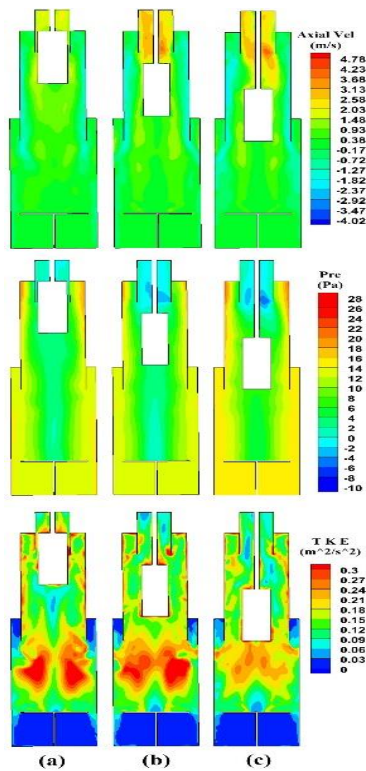


Figure 11. Effects of the location of impeller on axial velocity, pressure distribution and turbulent kinetic energy contours

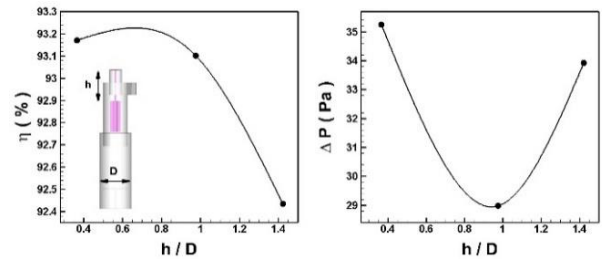


Figure 12. Effects of the location of impeller on pressure drop and collection efficiency

negligible. It can be implied that, for the new design dynamic impeller, increasing the length of the impeller is suitable for both efficiency and pressure drop.

3. 5. Effect of the Inlet Velocity Different inlet velocities were studied and as can be observed from Figures 15 and 16 with increase in inlet velocity, axial velocity increases through entire cyclone and pressure variation increase within the cyclone. Furthermore, with increasing inlet velocity, first, efficiency increases in a high rate and then its rate of increase decreases. Nevertheless, pressure drop increases with inlet velocity.

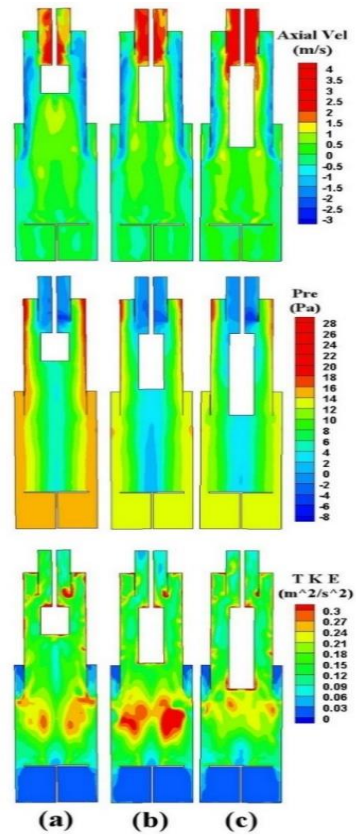


Figure 13. Effects of the impeller length on axial velocity, pressure distribution and turbulent kinetic energy contours

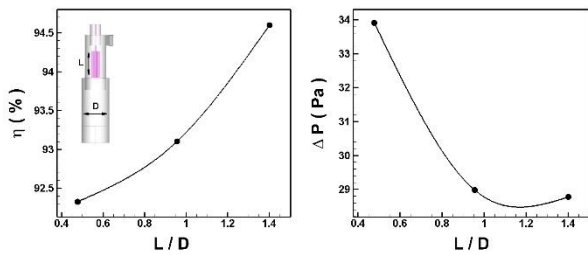


Figure 14. Effects of the impeller length on pressure drop and collection efficiency

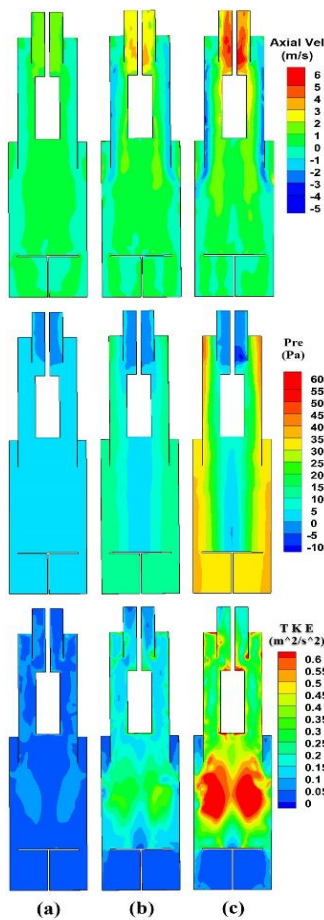


Figure 15. Effects of the inlet velocity on axial velocity, pressure distribution and turbulent kinetic energy contours

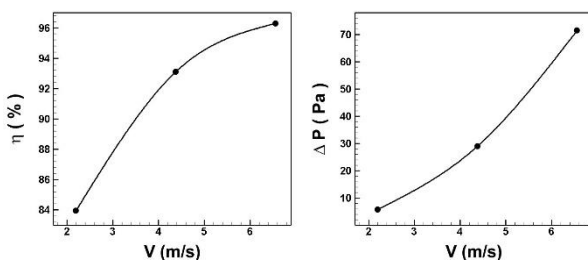


Figure 16. Effects of the inlet velocity on pressure drop and collection efficiency

3. 6. Comparison of Pressure Drop and Efficiency in Static New Design, Dynamic New Design and Conventional Static Cyclones

As can be seen from Figure 17, the minimum pressure drop occurs in new design dynamic cyclones and maximum pressure drop occurs for static new design cyclones. High pressure drops are even severe for higher speeds in new design cyclones. Low values for pressure drop is an advantage of a good cyclone design and dynamic new design cyclone has achieved this. Figure 17 proves that in very low speeds the minimum efficiency for static new design cyclones and dynamic new design cyclone stands very close to the static cyclones at the top. With increasing speed, maximum efficiency belongs to the dynamic new design cyclones and static ones possess the lowest efficiency.

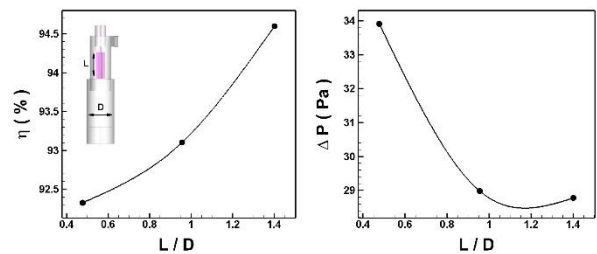


Figure 17. Comparison of collection efficiency and pressure drop for dynamic and static cyclones and static conventional cyclones

4. CONCLUSION

In this paper, the numerical simulation of flow field in the new design dynamic cyclone separator was investigated. This novel type emerges from the combination of dynamic cyclones and new design cyclones. The comparison of collection efficiency and pressure drop in new design dynamic cyclones, new design static cyclones and the conventional static cyclones were investigated. The effects of different geometrical parameters such as impeller diameter, number of vanes, length and the location of impeller was investigated. The Reynolds averaged Navier–Stokes equations with Reynolds stress turbulence model (RSM) were solved by the use of finite volume method based on the SIMPLE pressure correction algorithm in the computational domain. The Eulerian-Lagrangian computational procedure was used to predict particles tracking in the cyclones. The velocity fluctuations were simulated using the Discrete Random Walk (DRW). The following results were observed:

1. Collection efficiency of the dynamic new design cyclones is higher than other two types with similar geometrical parameter and boundary conditions.
2. Pressure drop in dynamic new design is lower than that of other two types.

3. Small particles yield higher efficiency due to intensive rotation with higher speed with the impeller inside the cyclone.
4. Installing impeller at the lower positions leads to a decrease in efficiency.
5. With increase in impeller diameter, pressure drop in dynamic new design decreases.
6. With increase in impeller length, efficiency increases and pressure drop decreases.
7. Increasing inlet velocity results in increasing collection efficiency and increasing pressure drop.
8. Design of the dynamic new design cyclones can be manipulated for different flow regimes with the aim of maximum collection efficiency with change in rotational speed of impeller and re-location of the vortex limiter.

5. REFERENCES

1. McK, R., "Fundamentals of cyclone design and operation", *Proceedings of . Australasian Institute of Mining and Metallurgy*, Vol. 152, (1949), 203.
2. Stairmand, C.J., "The design and performance of cyclone separators", *Transactions of the Institution of Chemical Engineers*, Vol. 29, No., (1951), 356-383.
3. Elsayed, K. and Lacor, C., "The effect of cyclone inlet dimensions on the flow pattern and performance", *Applied Mathematical Modelling*, Vol. 35, No. 4, (2011), 1952-1968.
4. Cortes, C. and Gil, A., "Modeling the gas and particle flow inside cyclone separators", *Progress in Energy and Combustion Science*, Vol. 33, No. 5, (2007), 409-452.
5. Avci, A., Karagoz, I. and Surmen, A., "Development of a new method for evaluating vortex length in reversed flow cyclone separators", *Powder Technology*, Vol. 235, (2013), 460-466.
6. Zhao, B., Su, Y. and Zhang, J., "Simulation of gas flow pattern and separation efficiency in cyclone with conventional single and spiral double inlet configuration", *Chemical Engineering Research and Design*, Vol. 84, No. 12, (2006), 1158-1165.
7. Qian, F. and Zhang, M., "Effects of the inlet section angle on the flow field of a cyclone", *Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment Engineering-Biotechnology*, Vol. 30, No. 11, (2007), 1564-1570.
8. Qian, F. and Wu, Y., "Effects of the inlet section angle on the separation performance of a cyclone", *Chemical Engineering Research and Design*, Vol. 87, No. 12, (2009), 1567-1572.
9. Erdal, F.M. and Shirazi, S.A., "Effect of the inlet geometry on the flow in a cylindrical cyclone separator", Vol. 128, No. 1, (2006), 62-69.
10. Zhao, B., Shen, H. and Kang, Y., "Development of a symmetrical spiral inlet to improve cyclone separator performance", *Powder Technology*, Vol. 145, No. 1, (2004), 47-50.
11. Xiang, R. and Lee, K., "Exploratory study on cyclones of modified designs", *Particulate Science and Technology*, Vol. 19, No. 4, (2001), 327-338.
12. Chuah, T., Gimbin, J. and Choong, T.S., "A cfd study of the effect of cone dimensions on sampling aerocyclones performance and hydrodynamics", *Powder Technology*, Vol. 162, No. 2, (2006), 126-132.
13. Xiang, R., Park, S. and Lee, K., "Effects of cone dimension on cyclone performance", *Journal of Aerosol Science*, Vol. 32, No. 4, (2001), 549-561.
14. Yoshida, H., Fukui, K., Yoshida, K. and Shinoda, E., "Particle separation by inoya's type gas cyclone", *Powder technology*, Vol. 118, No. 1-2, (2001), 16-23.
15. Qian, F., Zhang, J. and Zhang, M., "Effects of the prolonged vertical tube on the separation performance of a cyclone", *Journal of Hazardous Materials*, Vol. 136, No. 3, (2006), 822-829.
16. Kaya, F. and Karagoz, I., "Numerical investigation of performance characteristics of a cyclone prolonged with a dipleg", *Chemical Engineering Journal*, Vol. 151, No. 1-3, (2009), 39-45.
17. Yoshida, H., Kwan-Sik, Y., Fukui, K., Akiyama, S. and Taniguchi, S., "Effect of apex cone height on particle classification performance of a cyclone separator", *Advanced Powder Technology*, Vol. 14, No. 3, (2003), 263-278.
18. Keza, A., "The effect of a counter-cone position on cyclone performance", *Separation Science and Technology*, Vol. 47, No. 16, (2012), 2250-2255.
19. Xiang, R. and Lee, K., "Effects of exit tube diameter on the flow field in cyclones", *Particulate Science and Technology*, Vol. 26, No. 5, (2008), 467-481.
20. Chen, J. and Liu, X., "Simulation of a modified cyclone separator with a novel exhaust", *Separation and Purification Technology*, Vol. 73, No. 2, (2010), 100-105.
21. Keza, A., "Division of outlet flow in a cyclone vortex finder— the cfd calculations", *Separation and Purification Technology*, Vol. 75, No. 2, (2010), 127-131.
22. Hoffmann, A., De Groot, M., Peng, W., Dries, H. and Kater, J., "Advantages and risks in increasing cyclone separator length", *AIChE Journal*, Vol. 47, No. 11, (2001), 2452-2460.
23. Safikhani, H., Akhavan-Behabadi, M., Shams, M. and Rahimyan, M., "Numerical simulation of flow field in three types of standard cyclone separators", *Advanced Powder Technology*, Vol. 21, No. 4, (2010), 435-442.
24. Wang, J.-J., Wang, L.-Z. and Liu, C.-W., "Effect of a stick on the gas turbulence structure in a cyclone separator", *Aerosol Science and Technology*, Vol. 39, No. 8, (2005), 713-721.
25. Farahani, S.N., Tahmasbi, V., Safikhani, H. and Abbassi, A., "Effects of using ribs on flow pattern and performance of cyclone separators", *Engineering Applications of Computational Fluid Mechanics*, Vol. 5, No. 2, (2011), 180-187.
26. Su, Y. and Mao, Y., "Experimental study on the gas-solid suspension flow in a square cyclone separator", *Chemical Engineering Journal*, Vol. 121, No. 1, (2006), 51-58.
27. Safikhani, H., Shams, M. and Dashti, S., "Numerical simulation of square cyclones in small sizes", *Advanced Powder Technology*, Vol. 22, No. 3, (2011), 359-365.
28. Su, Y., Zheng, A. and Zhao, B., "Numerical simulation of effect of inlet configuration on square cyclone separator performance", *Powder Technology*, Vol. 210, No. 3, (2011), 293-303.
29. Kim, H.-T., Lee, K. and Kuhlman, M., "Exploratory design modifications for enhancing cyclone performance", *Journal of Aerosol Science*, Vol. 32, No. 10, (2001), 1135-1146.
30. Karagoz, I., Avci, A., Surmen, A. and Sendogan, O., "Design and performance evaluation of a new cyclone separator", *Journal of Aerosol Science*, Vol. 59, (2013), 57-64.
31. Safikhani, H. and Mehrabian, P., "Numerical study of flow field in new cyclone separators", *Advanced Powder Technology*, Vol. 27, No. 2, (2016), 379-387.
32. Safikhani, H., "Modeling and multi-objective pareto optimization of new cyclone separators using cfd, anns and nsga ii algorithm", *Advanced Powder Technology*, Vol. 27, No. 5, (2016), 2277-2284.

33. Jiao, J., Zheng, Y., Wang, J. and Sun, G., "Experimental and numerical investigations of a dynamic cyclone with a rotary impeller", *Chemical Engineering and Processing: Process Intensification*, Vol. 47, No. 9-10, (2008), 1861-1866.
34. Jiao, J., Zheng, Y., Sun, G. and Wang, J., "Study of the separation efficiency and the flow field of a dynamic cyclone", *Separation and Purification Technology*, Vol. 49, No. 2, (2006), 157-166.
35. Launder, B.E., Reece, G.J. and Rodi, W., "Progress in the development of a reynolds-stress turbulence closure", *Journal of Fluid Mechanics*, Vol. 68, No. 3, (1975), 537-566.
36. Wang, L., "Theoretical study of cyclone design", Texas A&M University, (2005),

Numerical Study of Flow Field in New Design Dynamic Cyclone Separators

H. Safikhani, F. Esmaili, S. Salehfard

Department of Mechanical Engineering, Faculty of Engineering, Arak University, Arak, Iran

PAPER INFO

چکیده

Paper history:

Received 04 September 2019

Received in revised form 16 October 2019

Accepted 08 November 2019

Keywords:

Dynamic Cyclone

New Design Cyclone

Collection Efficiency

CFD

Discrete Random Walk

در این مقاله، مدل‌سازی عددی جریان سیال در سیکلون‌های دینامیکی نسل جدید انجام شده است. تأثیر پارامترهای هندسی و غیرهندسی مختلف مانند قطر، طول، محل و سرعت دورانی پروانه بر پارامترهای عملکردی سیکلون بررسی شده است. معادلات میانگین‌گیری شده ناویر-استوکس در جریان حل شده‌اند و مدل دوفازی اویلر-لاگرانژ یک‌راهه نیز تحلیل شده است. نتایج نشان می‌دهد که در سیکلون‌های دینامیکی نسل جدید افت فشار کمتر و بازده جداسازی بیشتر از سیکلون‌های سنتی می‌باشد. همچنین، مشخص گردید که نصب پروانه در جایگاه پایین‌تر باعث افزایش بیشتر بازده می‌گردد.

doi: 10.5829/ije.2020.32.02b.22