A Novel Design to Overcome Detrimental Effect of High-Temperature Operating Conditions on Separation Efficiency of Square Cyclone

Hosseini, Ebrahim; Atarzadeh, Mohammad Amin,*+; Shekarzadeh, Mehdi.
Department of Mechanical Engineering, Ahvaz Branch, Islamic Azad University, Ahvaz, I.R. IRAN

ABSTRACT: *In this research, a numerical study was conducted to analyze a novel design for overcoming the detrimental effect of high-temperature operating conditions on the separation efficiency of a square-based cyclone. This kind of cyclone can be utilized in the Circulating Fluidized Bed (CFB) boiler and it was shown to become one of the most appropriate cleaning tools for high-temperature gases. Previous studies found that the separation efficiency of a cyclone reduced remarkably with the increment of inlet temperature resulting in weaker swirling flow over cyclones at extremely high temperatures. Hence, it is vital to develop an effective approach to prevent this detrimental impact. The novel cyclone design is based on the idea of altering the inlet shape on the flow field and enhancing the cyclone collecting efficiency in high-temperature operating conditions. Three separate inlet configurations, namely flat, oblique, and curved inlets were particularly developed and investigated numerically through the Computational Fluid Dynamics (CFD) method to maximize the low separation efficiency of a square cyclone influenced by the high-temperature operating condition. For simulating the flow of particles, the Eulerian-Lagrangian methodology is implemented for solving Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations. The Discrete Random Walk (DRW) is utilized for modeling fluctuations of velocity. Numerical results indicated that using oblique and curved inlets generated a raising in pressure drop but they significantly improved the separation efficiency of the cyclone separator. Among all inlet shapes, oblique inlet dramatically enhanced the separation efficiency by up to 15% at an inlet velocity of 12 m/s.*

KEYWORDS: *Gas cyclone; CFD, Eulerian-Lagrangian, Inlet shape; Temperature.*

INTRODUCTION

Cyclones are used for a wide range of applications, including air pollution control and industrial-related technologies. Gas cyclones have been proven as a crucial particle separation tool because of their

* To whom correspondence should be addressed.
+ E-mail: mohamadamin.atarzadeh@gmail.com
1021-9986/2023/6/1908-1916 9/$/5.09

advantageous properties such as simplicity in design, reasonable operating costs, and excellent compatibility with harsh operating conditions [1]. Cyclones are often categorized as square or conventional [2-5]. The conventional cyclone with a round cross-section was the most extensively utilized in the industry for CFB boilers. CFB boilers are widely used in the power generating industry providing clean coal combustion. The large body of typical cyclones became a significant disadvantage with the growth of CFB boilers owing to the thick refractory wall needing a lengthy start-up time. A square cyclone can be used as an option to effectively deal with these limitations. This gas cyclone offers several benefits over the conventional circular cyclone, including design simplicity, and faster start-stop time [6-8]. The square cyclone is smaller than the cylindrical cyclone, the enhanced membrane wall arrangement minimizes the volume of the multi-phase components as well as the start and stop time, and it is effectively integrated with the boiler [9]. However, this sort of gas cyclone had a low separation efficiency. Few studies have attempted to investigate the particle separation efficiency of square cyclones and their turbulent characteristics. For example, *Wasilewski et al*. [10] focused on the flow field of a gas cyclone with different vortex finder sizes. Their study analyzed several vortex finder arrangements and determined that the geometric form of the vortex finder has a substantial impact on particle removal efficiency. *Venkatesh et al*. [11] experimentally and numerically considered three square cyclones that were connected in the series arrangement. Their experimental work was done to evaluate collection efficiency and pressure drop. Their series arrangement cyclone's collecting efficiency is predicted to be 61%. The series arrangement layout reduces the pressure drop to 14.3%. Moreover, the flow pattern outcome is more consistent with the experimental data. *Safikhani et al*. [12] numerically studied the particle separation efficiency of small square and round (traditional) cyclones. They found that the pressure drop in cyclones is less than that in round one. In a study, *Fatahian et al*. [13] demonstrated that incorporating a linearizer into a conventional or square cyclone improves particle separation efficiency. *Fatahian et al*. [14] computationally considered the effect of the dual inverse cone on the performance of the square gas cyclone. Their research confirmed that the dual inverse cone raises pressure drop and collection efficiency marginally.

It is important to remove solid particles from a high-temperature gas in various industrial operations, such as chemical material manufacturing, cleaning the outlet gas of industrial units' chimneys, and several operations in thermal power plants [15]. By utilizing proper materials and production techniques, a cyclone separator may be utilized in extreme working conditions. Thermo-physical properties of fluids change with temperature. As a result, the performance of a cyclone is affected by the temperature of its working fluid [15]. According to the existing literature, few studies have been done to consider the performance of cyclone separators under extreme operating conditions. *Safikhani et al*. [1] conducted a numerical analysis of the flow field in the new-style cyclones with five distinct wall temperature profiles. This new design was developed on the concept of expanding the vortex length to boost cyclone collection efficiency and pressure drop. They revealed that in all temperature profiles, the novel design cyclones exhibited better separation efficiency and reduced pressure drop, proving their advantages over their conventional ones. *Gimbun et al*. [16] explored the role of temperature and velocity on cyclone pressure drop. They realized that when the inlet temperature goes up, the pressure drop reduces. *Karagoz* and *Kaya* [17] analyzed the heat transfer characteristics in the cyclone separator. They proved that heat transfer from gas cyclone walls started to rise as inflow velocity increased. *Siadaty et al*. [15] carried out a detailed examination to evaluate the influence of flow and particle temperatures on cyclone efficiency. Their results indicated that raising the inlet temperature reduces the maximum tangential velocity at a specific segment.

Based on the above-mentioned literature review, it was found that the role of inlet temperature on separation efficiency and gas flow patterns for cyclone separators, particularly square cyclones, is still poorly known. In our previous study [18], the square cyclone performance under high-temperature operating conditions was numerically analyzed using 3D CFD modeling. Computational results revealed that when inlet gas temperature increased, particle separation efficiency reduced considerably due to weaker swirling flow within gas cyclones. The majority of the past works lacked a sufficient method for overcoming the reduction in separation efficiency under high-temperature

Table 1: Dimensions (D=200 mm)

Dimensions	D_e/D	a/D	b/D	s/D	h/D	H/D	B/D
Square cyclone	0.5	0.75	0.2	1.2	2	4	0.25

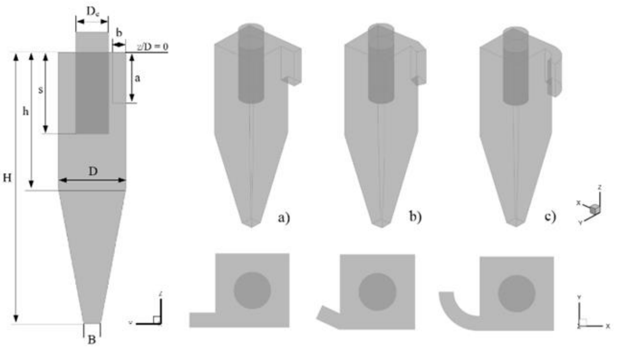

Fig. 1: Gas cyclone with a) Flat inlet b) Oblique inlet c) Curved inlet.

operating circumstances.

Therefore, the novelty of the present research is to present an appropriate design to overcome the negative impact of the high-temperature operating condition on the efficiency of a square cyclone. To enhance the low separation efficiency of a square cyclone impacted by inlet temperature, three distinct inlet designs, namely flat, oblique, and curved inlets, were selected and numerically examined using the CFD approach.

NUMERICAL ANALYSIS

Cyclone geometry

Fig. 1 depicts the gas cyclone with various inlet sections, such as flat, oblique, and curved inlets. Based on Su and Mao's [19] experimental results, the square cyclone (with a flat inlet) was created. The base cyclone's dimensions are also presented in Table 1. As it can be observed, only the inlet section's form was altered; all other components remained the same.

Mathematical formulations

A 3D CFD simulation was done to consider the flow within the cyclone. The URANS equations for the incompressible flow can be defined as [20]:

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

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

Where $\bar{u}_i$ represents the mean velocity, x_i indicates the spatial position, ρ is the gas density, and ν corresponds to the gas kinematic viscosity. Here $\bar{P}$ shows the mean pressure and $R_{ij} = \overline{u_i' u_j'}$ is the Reynolds stress tensor, where $u_i' = u_i - \bar{u}_i$ denote the ith fluctuating velocity component.

For evaluating the flow within cyclones, the LES approach provides more accurate results than the Reynolds Stress Turbulence Model (RSTM), although it has a larger computing cost. In the present study, the RSTM [21] was utilized to analyze turbulent airflow in a square cyclone in the present CFD simulation. The RSTM computes differential transport equations for evaluating turbulence stress components, with the following turbulence production terms [21]:

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

where, P_{ij} can be defined as [31]:

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

with P_f is the fluctuating energy production, ν_t represents the turbulent (eddy) viscosity, and $\sigma^k = 1$, $C_1 = 1.8$, $C_2 = 0.6$ are empirical constants. The transport equation for turbulence dissipation rate, ε, is expressed as follows [22]:

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

$K = \frac{1}{2}\overline{u_i' u_i'}$ represents the fluctuating kinetic energy, and ε corresponds to the turbulence dissipation rate. The values of constants are $\sigma^{\varepsilon} = 1.3$, $C^{\varepsilon 1} = 1.44$, $C^{\varepsilon 2} = 1.92$.

The CFD code Ansys Fluent 2021 R1 was used to model the two-phase flow by using the Eulerian-Lagrangian method. By solving momentum equations, the gas phase was assumed continuous in this methodology, whereas the solid phase (discrete phase) was solved by tracking particles through the flow field. Particle-particle interactions can be neglected if the dispersed second phase

Table 2: Thermo-physical properties [18,26,27]

T (K)	ρ (kg/m³)	μ×10⁵ (Pa.s)
293	1.188	1.7894
700	0.4975	3.388

has a low volume fraction. Since the particle volume fraction was lower than 10^{-3} in most zones, the interaction between the particles was negligible and the two-way coupling approach investigating the interaction between the particles and the gas stream was employed in the present work. The Discrete Phase Model (DPM) was adopted for evaluating particle trajectories in a Lagrangian reference frame by characterizing the density and size of individual particles.

The stochastic tracking model may predict particle dispersion because of turbulence in the fluid phase. Through the adoption of stochastic methods, the stochastic tracking model incorporates the influence of fluctuations of instantaneous turbulent velocity on particle trajectories. In the present CFD simulation, to evaluate turbulent particle dispersion, the DRW model was utilized. The Rosin–Rammler equation was employed to account for the particle size distribution. Morsi and Alexander [23] correlation is applied to calculate the drag coefficient of spherical particles in terms of relative Reynolds number (Re_p).

The particle equation of motion is expressed as [24]:

$$\frac{du_{pi}}{dt} = \frac{18\mu}{\rho_p d_p^2}\frac{C_D Re_p}{24}\left(u_i - u_{pi}\right) + \frac{g_i(\rho_p - \rho)}{\rho_p} \quad (6)$$

$$\frac{dx_{pi}}{dt} = u_{pi} \quad (7)$$

where the term $\frac{18\mu}{\rho_p d_p^2}\frac{C_D Re_p}{24}\left(u_i - u_{pi}\right)$ denote the drag force per unit particle mass.

In Equation (6), ρ_p and d_p indicate the particle density and diameter, respectively, C_D is the drag coefficient, u_i and u_{pi} represent the instantaneous gas and particle velocities, respectively. g_i indicates the gravitational acceleration and Re_p denote the relative Reynolds number, which is described as:

$$Re_p = \frac{\rho_p d_p |u - u_p|}{\mu} \quad (8)$$

Where μ denotes the dynamic viscosity.

Numerical setting

The finite volumes method (CFD code Ansys Fluent 2021 R1) was employed in this research to numerically solve the governing equations of gas flow. The present CFD simulation started with a steady solver (gas alone) for 10000 iterations before switching to an unsteady solver (particle tracking) with a time step of 10^{-4} seconds. The residence time was influenced by cyclone volume and the volumetric flow rate of the gas. The influence of turbulence was investigated using the RSTM and a standard wall function based on the strong swirl flow in a gas cyclone [2,18,25]. The case was modeled in which 10,000 Fly ash particles with a density of 1989.7 kg/m³ were injected into the square cyclone from the inlet surface with a size distribution of 1 to 32 µm from a coal-fired boiler in a power plant.

At the gas inlet segment, the inlet velocity boundary condition was specified, and it was supposed that the particle and inlet velocity of the gas was equal. The inflow velocity varies from 12 to 28 m/s. The hydraulic diameter was set to 0.0857 m and the turbulence intensity was set to 4% [2,18]. The outflow was assumed for the gas exit. Furthermore, the cyclone walls had a no-slip condition, and the dust outlet segment was adjusted to trap the DPM condition. Table 2 presents details of thermo-physical parameters of the gas phase (air) including viscosity and density at low and high temperatures. The temperature data were selected to match the ranges of temperature which is reported in previous studies [18,26,27]. Heat transfer was not solved in the present study, and the gas flow was assumed to be isothermal but at diverse temperatures. On the other hand, the effect of the temperature dependence of gas thermo-physical properties on the performance of gas cyclones was analyzed.

Grid independence study and validation

The proper grid resolution is crucial to reveal the reliability of the numerical simulation. To verify the grid independence study, three different grid levels were adopted. In the first case, a coarse-type mesh was produced which contains 480,000 elements. In the second case, a medium-type with 720,000 elements was developed. In the third case, a fine-type mesh with 1,080,000 elements was generated. Following that, Fig. 2 illustrates the pressure profiles for three different meshes at z/D=-0.25. The results confirmed mesh independence because the pressure difference between two subsequent meshes was less than 1%. Therefore, for all subsequent numerical simulations, a grid with 720,000 cells (medium mesh) was utilized. This mesh achieved a fair computational cost while maintaining a sufficient amount of grid independence.

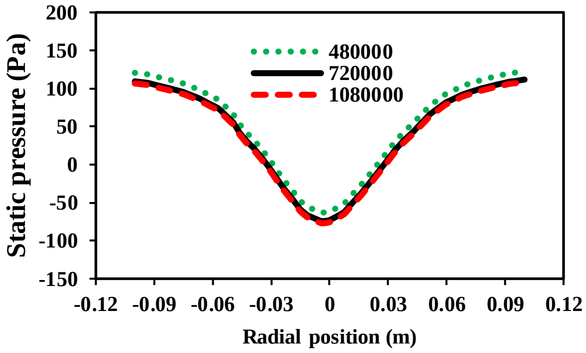

Fig. 2: Radial static pressure profiles at v=12 m/s

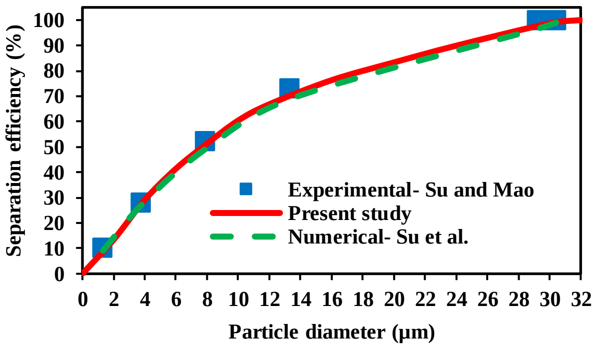

Fig. 4: Comparing the separation efficiency with refs. [3,19] at v=20 m/s

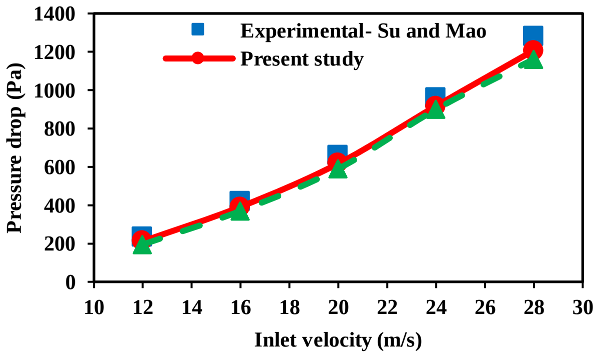

Fig. 3: Comparing the pressure drop with refs. [3,19]

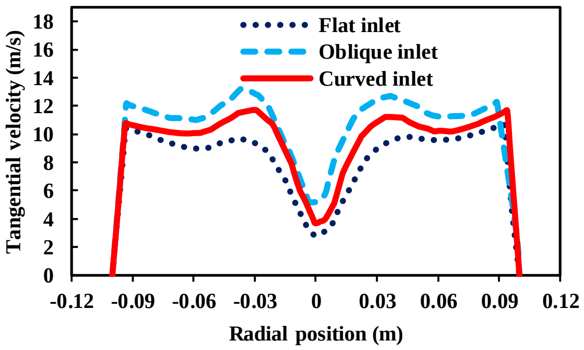

Fig. 5: Comparing the tangential velocity distribution at z/D= -0.25 and v=12 m/s

To gain confidence in the simulation, the results must be compared to existing numerical or experimental studies. The mesh validation refinement ratio is 0.9963, which is quite near to unity. Since the findings are in the asymptotic range and independent of the number of cells, the grid has been sufficiently refined. For validating the current computational model, the results were compared with the experimental study done by *Su* and *Mao* [19] and the numerical study of *Su et al.* [3] for the square cyclone depicted in Figs 3 and 4. The condition of CFD modeling was chosen to be similar to those utilized in previous studies [3,19]. The present finding agrees well with the experimental and numerical results published in previous works. The difference is less than 8%, which is within an acceptable range. This demonstrates that the present methodology is quite well to predict the performance of a square cyclone in terms of separation efficiency and other parameters.

RESULTS AND DISCUSSION

Tangential velocity

Fig. 5 shows the influence of the inlet part on the tangential velocity distribution at z/D = -0.25 for a square cyclone with v=12 m/s.

The tangential velocity distribution in all gas cyclones was composed of a free vortex in the outer zone and a forced vortex in the interior region (V-shaped), which revealed the predicted Rankine-type vortex [18]. Tangential velocity is dispersed somewhat equally along the center axis of the square cyclone. 67Moreover, the zone between the inner and outer vortexes had the highest tangential velocity. A comparison of the findings revealed that changing the geometry of the inlet zone has a direct effect on the distribution of the tangential velocity. Tangential velocity profiles in the free and forced vortex zones differed dramatically when the inlet geometry changed from flat to oblique and curved. When the tangential velocity profiles at the wall remained relatively constant, both oblique and curved inlets substantially enhanced the tangential velocity in the other areas. The maximum tangential velocity was around 1.1 times the inlet velocity for the square cyclone with an oblique inlet, while it was expected to be 0.98 times and 0.89 times the inlet velocity for the square cyclone with a curved inlet and

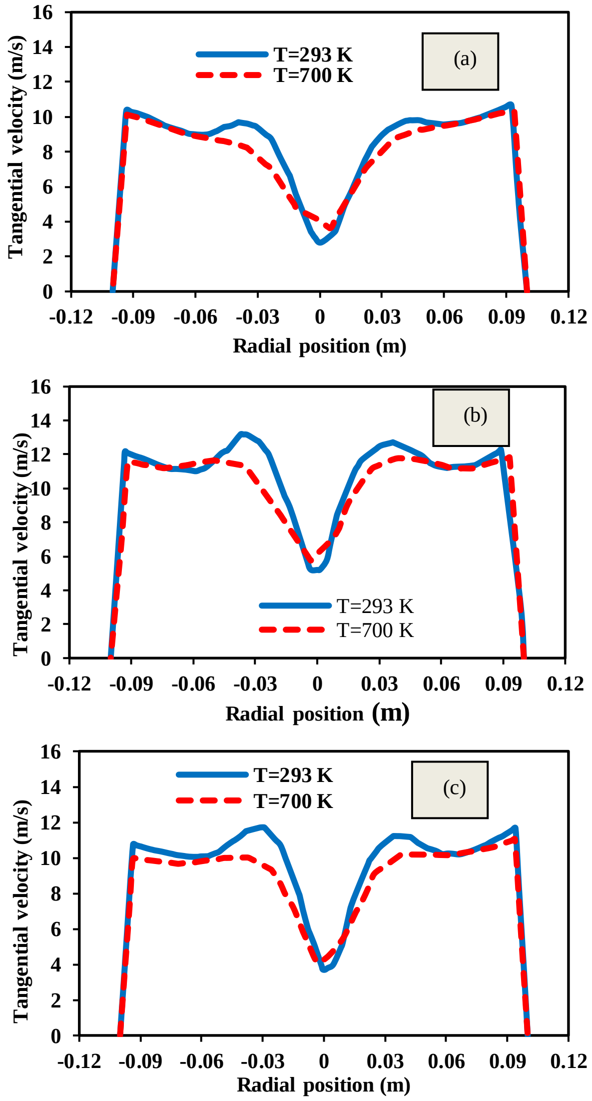

Fig. 6 Comparing the tangential velocity at different gas temperatures a) Flat b) Oblique c) Curved inlets

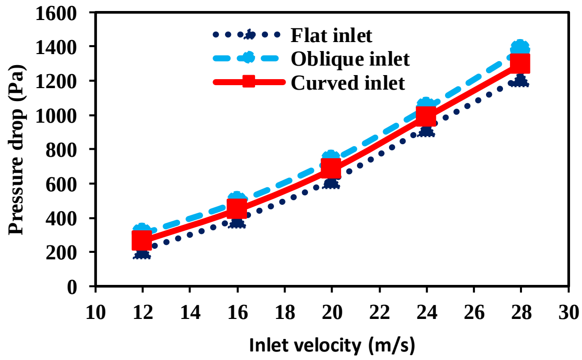

Fig. 7: Influence of inlet velocity on pressure drop at T=293 K

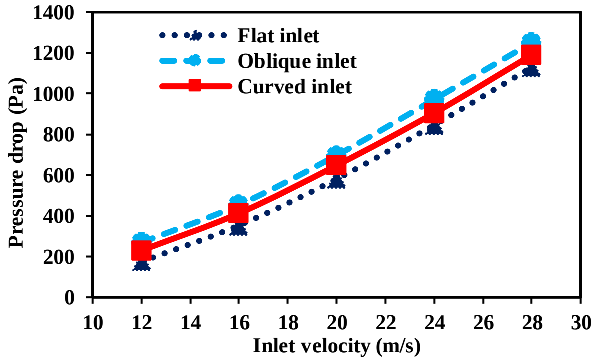

Fig. 8: Influence of inlet velocity on pressure drop at T=700 K.

a flat inlet, respectively. It is possible to derive that the cyclone with oblique and curved inlets generated greater tangential velocity values than the base cyclone. This demonstrated that the oblique and curved inlets might improve tangential velocity, resulting in a considerable augment in separation efficiency.

Fig. 6 demonstrates the impact of inlet temperature on the tangential velocity profile for three distinct inlet geometries. For investigating the influence of gas temperature on tangential velocity, the minimum and maximum gas temperatures (T=293 K and T=700 K) were studied. Tangential velocity profiles along the wall remain essentially constant as the temperature of the gas rises, but tangential velocity profiles in free and forced vortex zones change significantly. The maximum tangential velocity of all cyclones fell considerably as the inlet temperature increased. Enhanced separation efficiency comes from higher tangential velocity. Therefore, the separation efficiency of the square cyclone was decreased at high temperatures.

Pressure drop

Fig. 7 demonstrates the pressure drop (Pa) for gas cyclones with various inlet geometries versus inlet velocity. In general, when the inflow velocity increased, the pressure drop rose slightly. The inlet angle has a direct impact on the direction of gas entry, which affects pressure drop and separation efficiency. Consequently, the inlet direction and shape are critical elements for cyclone flow pattern and performance. It is apparent that altering the design of the inlet section had a substantial impact on cyclone performance and flow pattern since it directly altered the entering direction of gas flow. When compared to the base square cyclone with a flat entrance, both square

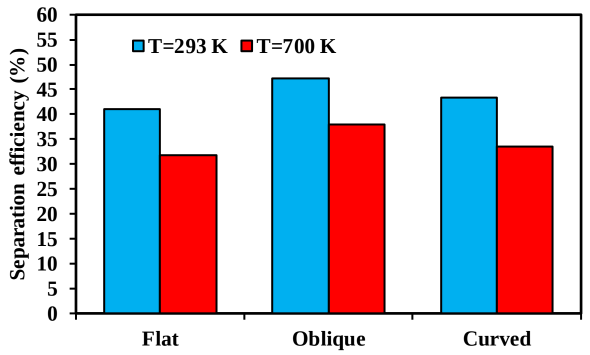

Fig. 9: Comparison between separation efficiencies of all cyclones at v=12 m/s.

cyclones with oblique and curved inlets had a larger pressure drop. This was more noticeable at lower inlet velocities, particularly at v=12 m/s. Using an oblique inlet enhanced the pressure drop by roughly 43% compared to the cyclone with a flat cyclone.

Fig. 8 clearly shows the changes in pressure drop versus inlet temperature. It is evident that the pressure drop is augmented by raising inlet velocity, a trend that has been seen in previous studies [2,5]. Increasing the inlet temperature substantially reduced the pressure drop in all circumstances. This was more visible at higher inflow velocities when the cyclone was dominated by a swirl flow. This is defined as an augmentation in dynamic viscosity produced by the increment of inlet flow temperature, which causes greater shear stress and a reduction in fluid tangential velocity in the cyclone body. The greater the fluid's tangential velocity, the larger the dissipation. This is why pressure drops significantly at low temperatures. So when the density of the gas raises and the viscosity reduces at low temperatures, the tangential velocity raises, increasing pressure drop.

Particle Separation Efficiency

Fig. 9 shows the effect of inlet temperature on the separation efficiency of a gas cyclone with flat, oblique, and curved inlet configurations at v=12 m/s. The separation efficiency was found using the DPM model based on the number of particles exiting from the bottom outlet. For comparison, a particle size of d_P=16 m is chosen. It is evident that augmenting the inlet temperature caused decreasing the separation efficiency. It is mainly because when temperature increases, tangential velocity is reduced. At the lowest temperature (T=293 K), the maximum separation efficiency was estimated, while the lowest separation efficiency was attained at the highest temperature (T=700 K). By increasing the inlet temperature from 293 K to 700 K, the separation efficiency of the cyclone with a flat inlet was reduced 23%. This was attributed to higher tangential velocity being generated as the inlet velocity raised. By comparing the separation efficiency of all square cyclones, it was concluded that the inlet configurations had a substantial effect on the cyclone's performance. Only the square cyclone with oblique and curved inlets yielded higher separation efficiencies than the flap inlet cyclone at both temperatures. Consequently, the suggested oblique inlet shape enhanced separation efficiency, particularly at higher inlet temperatures. Using oblique and curved inlet shapes instead of the inlet shape considerably helped the cyclone in collecting particles, indicating the possibility to operate at higher inlet temperatures.

CONCLUSIONS

The inlet temperature influence on the flow field and separation efficiency of the square cyclone at low and high temperatures were examined in the present research using CFD modeling. In order to augment the separation efficiency of the cyclone in high-temperature operating conditions, only the inlet shape is modified and other parts of the cyclone have remained unchanged. The Eulerian-Lagrangian technique was utilized to analyze the airflow and particle dynamics in the square cyclone. The precision of CFD modeling was demonstrated by comparing predicted pressure drop and separation efficiency to experimental data of ref. [19] and numerical results of ref. [3], which showed good agreement.

- Generally, the idea of altering the direction of incoming gas flow through the inlet segment was promising for performance improvement of the square cyclone.
- Changes in inlet flow temperature harmed cyclone performance and flow pattern.
- The maximum tangential velocity was derived about 1.1 times the inlet velocity with an oblique inlet, while it was expected 0.98 times and 0.89 times the inlet velocity with a curved inlet and square cyclone with the flat inlet.
- When the inlet temperature was raised, the maximum tangential velocity of all cyclones decreased dramatically.

- By increasing the inlet temperature from 293 K to 700 K, the separation efficiency with a flat inlet was reduced 23%. While using an oblique inlet instead of a flat one improved the separation efficiency by up to 15%.

Received :Jul.03, 2022 ; Accepted : Oct.24, 2022

REFERENCES

[1] Safikhani H., Rafiee M., Ashtiani D., Numerical Study of Flow Field in New Design Cyclones with Different Wall Temperature Profiles: Comparison with Conventional Ones, *Adv. Powder Technol.*, **32(9)**: 3268-3277 (2021).

[2] Fatahian H., Fatahian E., Nimvari M. E., Ahmadi G., Novel Designs for Square Cyclone Using Rounded Corner and double-inverted cones shapes, *Powder Technol.*, **380**: 67-79 (2021).

[3] Su Y., Zheng A., Zhao B., Numerical Simulation of Effect of Inlet Configuration on Square Cyclone Separator Performance, *Powder Technol.*, **210(3)**: 293-303 (2011).

[4] Fatahian H., Fatahian E., Improving the Efficiency of a Square Cyclone Separator Using the Dipleg–A CFD-Based Analysis, *Iran. J. Chem. Chem. Eng. (IJCCE)*, **41(2)**: 670-681 (2022).

[5] Fatahian E., Fatahian H., Hosseini E., Ahmadi G., A Low-Cost Solution for the Collection of fine Particles in Square Cyclone: A Numerical Analysis, *Powder Technol.*, **387**: 454-465 (2021).

[6] Nejad J. V. N., Kheradmand, S., The Effect of Arrangement in Multi-Cyclone Filters on Performance and the Uniformity of Fluid and Particle Flow Distribution, *Powder Technol.*, **399**: 117191 (2022).

[7] Pandey S., Brar L. S., On the performance of Cyclone Separators with Different Shapes of the Conical Section Using CFD, *Powder Technol.*, **407**: 117629 (2022).

[8] Venkatesh S., Kumar R. S., Sivapirakasam S. P., Sakthivel M., Venkatesh D., Arafath S. Y., Multi-Objective Optimization, Experimental and CFD Approach for Performance Analysis in Square Cyclone Separator, *Powder Technol.*, **371**: 115-129 (2020).

[9] Wang S., Fang M., Luo Z., Li X., Ni M., Cen K., Instantaneous Separation Model of a Square Cyclone, *Powder Technol.*, **102(1)**: 65-70 (1999).

[10] Wasilewski M., Brar L. S., Ligus G., Experimental and Numerical Investigation on the Performance of Square Cyclones with Different Vortex Finder Configurations, *Sep. Purif. Technol.*, **239**: 116588 (2020).

[11] Venkatesh S., Sivapirakasam S. P., Sakthivel M., Ganeshkumar S., Prabhu M. M., Naveenkumar M., Experimental and Numerical Investigation in the Series Arrangement Square Cyclone Separator, *Powder Technol.*, **383**: 93-103 (2021).

[12] Safikhani H., Shams M., Dashti S., Numerical Simulation of Square Cyclones in Small Sizes, *Adv. Powder Technol.*, **22(3)**: 359-365 (2011).

[13] Fatahian H., Fatahian E., Nimvari M. E., Improving Efficiency of Conventional and Square Cyclones Using Different Configurations of the Laminarizer, *Powder Technol.*, **339**: 232-243 (2018).

[14] Fatahian H., Hosseini E., & Fatahian E., CFD Simulation of a Novel Design of Square Cyclone with Dual-Inverse Cone, *Adv. Powder Technol.*, **31(4)**: 1748-1758 (2020).

[15] Siadaty M., Kheradmand S., Ghadiri F., Study of Inlet Temperature Effect on Single and Double Inlets Cyclone Performance, *Adv. Powder Technol.*, **28(6)**: 1459-1473 (2017).

[16] Gimbun J., Chuah T. G., Fakhru'l-Razi A., Choong T. S., The Influence of Temperature and Inlet Velocity on Cyclone Pressure Drop: a CFD Study, *Chem. Eng. Process.*, **44(1)**: 7-12 (2005).

[17] Karagoz I., Kaya F., CFD Investigation of the Flow and Heat Transfer Characteristics in a Tangential Inlet Cyclone, *Int. Commun Heat Mass.*, **34(9-10)**: 1119-1126 (2007).

[18] Hosseini E., Fatahian H., Ahmadi G., Eshagh Nimvari M., Fatahian E., CFD Study on the Effect of Gas Temperature on the Separation Efficiency of Square Cyclones, *J. Braz. Soc. Mech. Sci.*, **43(9)**: 1-13 (2021).

[19] Su Y., Mao Y., Experimental Study on the Gas–Solid Suspension Flow in a Square Cyclone Separator, *Chem. Eng. Technol.*, **121(1)**: 51-58 (2006).

[20] Elsayed K., Lacor C., Modeling and Pareto Optimization of Gas Cyclone Separator Performance Using RBF Type Artificial Neural Networks and Genetic Algorithms, *Powder Technol.*, **217**: 84-99 (2012).

[21] Launder B. E., Reece G. J., Rodi W., Progress in the Development of a Reynolds-Stress Turbulence Closure, *J. Fluid Mech.*, **68(3)**: 537-566 (1975).

[22] Hoekstra A. J., Derksen J. J., Van Den Akker H. E. A., An Experimental and Numerical Study of Turbulent Swirling Flow in Gas Cyclones, *Chem. Eng. Sci.*, **54(13-14)**: 2055-2065 (1999).

[23] Morsi S. A. J., Alexander A. J., An investigation of Particle Trajectories in Two-Phase Flow Systems, *J. Fluid Mech.*, **55(2)**: 193-208 (1972).

[24] Elsayed, K., & Lacor, C., Numerical modeling of the flow field and performance in cyclones of different cone-tip diameters, *Comput. Fluids*, **51**(1): 48-59 (2011).

[25] Song C., Pei B., Jiang M., Wang B., Xu D., Chen Y., Numerical Analysis of Forces Exerted on Particles in Cyclone Separators, *Powder Technol.*, **294**: 437-448 (2016).

[26] Shin M. S., Kim H. S., Jang D. S., Chung J. D., Bohnet M., A Numerical and Experimental Study on a High Efficiency Cyclone Dust Separator for High temperature and Pressurized Environments, *Appl. Therm. Eng.*, **25(11-12)**: 1821-1835 (2005).

[27] Siadaty M., Kheradmand S., Ghadiri, F., Research on the Effects of Operating Conditions and Inlet Channel Configuration on Exergy Loss, Heat Transfer and Irreversibility of the Fluid Flow in Single and Double Inlet Cyclones, *Appl. Therm. Eng.*, **137**: 329-340 (2018).