A Brief Review on Improving the Collection Efficiency of a Square Cyclone Separator Using Variable Frequency Drive

[1]Vivek R, [2]Venkatesh S, [3]Kathiravan A, [4]Jeevan Prasath and [5]Sam Daniel M
[1,2,3,4,5]Department of Mechanical Engineering, Sri Eshwar College of Engineering, Coimbatore, Tamil Nadu, India.
[1]vivek.r@sece.ac.in

Article Info
S. Venkatesh et al. (eds.), *1st International Conference on Emerging Trends in Mechanical Sciences for Sustainable Technologies*, Advances in Computational Intelligence in Materials Science.
Doi: https://doi.org/10.53759/acims/978-9914-9946-6-7_11

Abstract - The study examines the way that a square cyclone separator separates solid particles. In terms of space economy and simplicity of integration into industrial processes, the square cyclone separator is a relatively novel design that provides benefits over conventional circular cyclone separators. The study evaluates the separator's particle removal effectiveness, pressure drop, and gas flow rate using both numerical models and experimental measurements. The construction of a prototype separator and testing of its performance under various operating circumstances comprise the experimental tests, while computational fluid dynamics (CFD) software is used for the numerical calculations. In order to ultimately increase process efficiency and product quality, the study's findings will offer insights into the design and optimisation of square cyclone separators for industrial applications and reduce maintenance costs. In order to separate solid particles from gas streams in industrial settings, this project intends to develop and build a square cyclone separator. The system has a square chamber that separates the gas flow from the solid particles using centrifugal force. With a maximum operating pressure of 10 psi, the separator will be made of a combination of steel and plastic components. Measurements of the particle removal efficiency, pressure drop across the separator, and gas flow rate will be used to assess the separator's efficacy. The ultimate purpose of this project is to create an affordable, effective square cyclone separator that can be quickly incorporated into current industrial processes to enhance overall performance and lower maintenance costs.

Keywords: Cyclone Separator, Flow Pattern, CFD Analysis, VFD Drive, Collection Efficiency.

I. INTRODUCTION

Since the late 1800s, cyclone separators have been employed in several industrial processes to remove dust or other particles from gases. They are straightforward to maintain and simple with no moving components, but the physics driving the flow process inside of them is complicated. Scaling rules are used to adapt various standard geometries to a particular set of operating circumstances. The performance of these standard geometries is well established. To improve the cooling stability of electrical equipment, high-quality air preparation is necessary, but filters have a short filtering life, need to be cleaned often, and increase air resistance. A low-cost and low-maintenance way of eliminating particulate matter from air or other gas streams is to use cyclone separators [1,2,3]. They transform the inertia force of gas particles into a centrifugal force, resulting in an outer vortex. Chemical processes involve separation of streams based on physical principles, such as Gas-Liquid (vapor-liquid) separation, Gas-Solid separation, Liquid-Liquid separation (immiscible), Liquid-Solid, and Solid-Solid separation. Gravitational force or centrifugal force can be used to enhance separation. In several sectors, cyclone separators are frequently used to remove particles from the gas flow using centrifugal force. They are suitable for a variety of industrial applications and feature a straightforward design [4,3,5]. The main components of gas cyclones are pressure drop, fractional efficiency, and collection efficiency. Advanced experimental instruments have been utilized to measure airflow parameters.

II. DESIGN CONSIDERATION

Lapple created the classical cyclone design (CCD) approach in the early 1950s, and it is regarded as a standard procedure. The right number of rotations for various types of cyclones cannot be predicted since the cyclone entrance velocity is not taken into account. The design engineer must be knowledgeable of the flow conditions, PM concentrations, PSD, and the type of cyclone to be developed in order to employ the CCD procedure (high efficiency, conventional, or high throughput). All principle dimensions must be provided by the cyclone type as a function of the cyclone barrel diameter, and the PSD must take the form of mass-fraction versus aerodynamic equivalent diameter of the

PM (D). The performance of the cyclone separator depends heavily on its shape. The input length, cylinder diameter, and cone length are all significant variables that have an impact on the separation efficiency [6,7]. The cyclone separator's internal flow pattern is impacted by the design of the inlet. The fluid will enter the cyclone separator smoothly, eliminating turbulence and facilitating effective particle separation thanks to an effective intake design. Design of the outflow: The flow of the separated particles out of the cyclone separator is influenced by the design of the outlet. The particles will be effectively collected without being re-entrained into the fluid if the exit is well-designed. The cyclone separator's internal flow pattern may be impacted by the presence of a wall close by. The separator should be constructed with enough room between the wall and the intake to provide a smooth flow of fluid in order to lessen the impacts of walls. Particle size describes while building a cyclone separator, particle size is a crucial factor to take into account. The efficiency of the separation and the pressure drop across the separator are both impacted by the particle size. For effective particle separation with the least amount of pressure loss, the design should be tuned. Material selection: The resistance of the material used in the cyclone separator's construction to abrasion, corrosion, and high temperatures should be taken into consideration [8,9,10]. To guarantee that the cyclone separator can survive the forces encountered during operation, the material should also be chosen based on its strength and longevity. Operational circumstances for designing the cyclone separator, the operating conditions, including the flow rate and pressure, must be taken into account. To guarantee that the cyclone separator can function well under the anticipated operating circumstances, the design should be optimised [11,12].

This work presents a study on a gas–solid cyclone separator used in a complex cement production plant. The main objective of the study is based on the performance evaluation and optimization of the cyclone separator in terms of particle separation and heat transfer efficiencies, while keeping pressure losses under control. The thermal interaction is between two gas–solid mixtures, one at 850°C and the other at 600°C, respectively. The solid phase consists mostly of calcium carbonate subsequently intended to the so-called baking process for the production of clinker and ultimately cement. A first model has been setup using experimental data as boundary conditions to assess the physical model behavior and the CFD solver parameters. After that, five additional models with different geometries have been analyzed to evaluate the influence of the vortex finder length on the separation efficiency and on the heat exchange performance [13,14,15]. By increasing the length of the results show a global improvement in the separation efficiency of up to 5% if compared to the geometry without the. Further, the increase in the length determines a monotonic decrease of temperature at the exit but a monotonic increase of pressure losses. In the second part of this work, using one of the previous models with a study of the influence of the particle diameter on the separation efficiency has been performed. The increase of particle diameter causes an increase of the separation and thermal exchange performance, decreasing at the same time the pressure drop. The numerical approach for all the cases is based on implicit unsteady simulations using the Eulerian Multiphase model [18, 19]. Modeling cyclone separators continues to be a significant challenge despite being a widespread and crucial component in many different sectors. Unsteady flow dynamics (CFD) simulations have been proven to be a successful method for predicting the circular flow pattern in such a separator device. Although CFD simulations are very realistic, the amount of computer power needed to run them prevents wider usage. This is how this work introduces a data-driven reduced-order model that considers the flow, known as Physics-Informed Neural Networks (PINN). The field behaviour and the conservation of mass and momentum in the Navier-Stokes equations. The results demonstrate that PINN can faithfully represent complex flow behaviour by combining pressure and velocity fields. PINN Conservation's momentum is valued and used as an output [20]. Modeling cyclone separators continues to be a significant challenge despite being a widespread and crucial component in many different sectors. Unsteady flow dynamics (CFD) simulations have been proven to be a successful method for predicting the circular flow pattern in such a separator device. Although CFD simulations are very realistic, the amount of computer power needed to run them prevents wider usage. This is how this work introduces a data-driven reduced-order model that considers the flow, known as Physics-Informed Neural Networks (PINN). The field behaviour and the conservation of mass and momentum in the Navier-Stokes equations. The results demonstrate that PINN can faithfully represent complex flow behaviour by combining pressure and velocity fields. PINN Conservation's momentum is valued and used as an output. [17,21,25]. The cyclones, one of the dust collection devices, are used to separate particles or liquid from the carrier gas by centrifugal forces. Their simplicity, lack of moving parts, and adaptation to high pressure and temperature conditions make them durable and efficient. In the operation of cyclones, the effects of particles, carrier gas, cyclone dimensions, and other factors are important operational requirements. The geometric structure of a high efficiency standard cyclone is studied and modelled in this study using the Solid works program. Computational Fluid Dynamics (CFD) was used to determine cyclone separation performance. Flow analysis was performed, and cut-off diameters were determined using statistical learning methods.

III. COMPUTATIONAL FLUID DYNAMICS (CFD)

A square cyclone separator's behaviour in terms of gas and particle flow may be modelled using computational fluid dynamics (CFD). The design and operating parameters of the separator may be optimised with the use of CFD simulations, which can offer insights into the fluid dynamics and particle behaviour inside the separator. A CAD programme must be used to simulate the separator's shape in order to do a CFD simulation of a square cyclone separator. The fluid and particle parameters, inlet conditions, and boundary conditions are then specified in a CFD programme once the geometry has been imported. In order to forecast flow behaviour and particle separation efficiency, the CFD programme solves the governing equations for fluid and particle motion. Particle paths, particle concentration distribution, and pressure drop through the separator may all be predicted using CFD models. The separator's shape and operating parameters may be optimised using the simulation findings to obtain the highest particle separation efficiency and the lowest pressure drop. CFD simulations may be used to analyse the impact of various factors on the separator's performance in addition to improving the separator's design [22, 23]. CFD simulations may be used, for instance, to examine how inlet velocity, particle size, and particle concentration affect separation efficiency and pressure drop. Overall, square cyclone separators may be designed and optimised using CFD models, which offer insights into the fluid dynamics and particle behaviour inside the separator and help increase its effectiveness and performance. The present study is mainly focused on proposing an effective way to improve the efficiency of a square cyclone separator. For this purpose, a dipleg is attached under the square cyclone to investigate its effect on the performance of the square cyclone. A three-dimensional Computational Fluid Dynamics (CFD) simulation is done by solving the Reynolds-Averaged Navier-Stokes equations with the Reynolds Stress Model (RSM) turbulence model and applying the EulerianLagrangian two-phase method. The turbulent dispersion of particles is predicted by the application of the Discrete Random Walk (DRW) model. The numerical results demonstrate that using dipleg produced an increase in pressure drop but it positively enhances the separation efficiency of the square cyclone. Using dipleg significantly increases the separation efficiency of the square cyclone, especially at higher inlet velocities. This can be more obvious when using dipleg which is minimized the 50% cut size of square cyclone by about 26.3%. The main objective of the present study was to analyze the effect of varying the inlet shape on the flow field and overall performance in a square cyclone. Three diferent inlet shapes, namely straight, inclined, and bend inlets, were specifically proposed to improve the low separation effeciency of the square cyclone. A three-dimensional numerical simulation was done by solving the Reynolds averaged Navier–Stokes equations with the Reynolds stress model turbulence model and applying the Eulerian–Lagrangian two-phase method. The turbulent dispersion of particles was predicted by the application of the discrete random walk model. Comprehensive comparisons of pressure drop, separation efficiency, 50% cut sizes, and flow feld were performed to demonstrate the mechanism of how the inlet shape infuenced the square cyclone performance. The results extracted from computational fluid dynamics simulation demonstrated that although using inclined and bend inlets produced an increase in pressure drop, they positively enhanced the separation efficiency of the square cyclone. Among all inlet shapes, inclined inlet significantly helped the square cyclone to collect finer particles, as it reduced the 50% cut sizes approximately 29.4% at an inlet velocity of 28 m/s. [16, 25, 27] New theoretical methods for computing travel distance, numbers of turns, and cyclone pressure drop have been developed to accurately estimate cyclone performance. Cyclone cut-points for different dusts were traced from measured cyclone overall collection efficiencies and the theoretical model for calculating cyclone overall efficiency. Experimental results indicate that optimal cyclone design velocities should be determined based on standard air density. The tangential inlet generates the swirling motion of the gas stream, which forces particles towards the outer wall where they spiral in the downward direction. The particles are collected in the dustbin located at the bottom of the conical section of the cyclone body, and the cleaned gas leaves through the exit pipe at the top. Cyclone Separator is a commonly used device to separate dust particles from gas and dust flow. The project presents the design and development of Cyclone Separators based on CFD along with simulations. The present work is based on the performance of flour mill Cyclone Separator for different inlet velocity. In the present investigation the characteristics of the standard cyclone are studied and its effect on performance parameters like pressure drop and efficiency [26]. Cyclone Separator is designed with single tangential inlet and two outlets at the barrel top and bottom area. The study was performed for air-dust flow, based on an experimental study available in the literature, where a conventional Cyclone Separator model was used. This study focuses on the effect of geometrical parameters on the flow field pattern and performance of tangential inlet cyclone separators using a computational fluid dynamics (CFD) approach. The objective of the study is threefold: to study the optimized staircase design by understanding the pressure and velocity variations; to study the performance of a cyclone separator by varying its geometrical parameters; and to compare it to that without a collector. Four geometrical factors have significant effects on the cyclone performance: inlet width, cone-tip diameter, cone height, and inlet geometry [28-30]. The flow field pattern has been simulated and analyzed with the aid of velocity components and static pressure contour plots. The CFD model was used to predict pressure and velocity variations in cyclone geometries based on Stair and high-efficiency design. During the gas-solid separation process, cyclone separators are useful for removing particles from harsh environments with high pressure and temperature. The separation effectiveness and speed of the cyclone separators define their significant performance. lower pressure. The performance is affected by the intake characteristics, including the number of inlets, inlet shape, and inlet gas velocity about cyclone separators. The purpose of this research is to investigate experimentally the impact of a dual inlet on the effectiveness of cyclone separators for separation.

IV. NUMERICAL ANALYSIS

The air flow and particle behaviour in square cyclone separators are studied using computer methods in numerical analysis. This investigation can shed light on the elements influencing the separator's performance, enabling the design and usage circumstances to be improved. Square cyclone separators may be studied using a variety of numerical analytical methods, such as finite element analysis (FEA), finite volume analysis (FVA), and computational fluid dynamics (CFD). Due to its capacity to capture complicated fluid dynamics, CFD is the approach that is most frequently employed for the numerical study of square cyclone separators. The flow behaviour and particle paths inside the separator may be predicted using CFD simulations, giving information on the pressure drop across the separator and separation effectiveness. The performance of the separator may also be examined using CFD simulations in relation to various design and operational factors [31-33]. CFD simulations, for instance, may be used to investigate the impact of intake geometry, inlet velocity, particle size, and particle concentration on separation efficiency and pressure drop. Square cyclone separators may also be numerically analysed using FEA and FVA in addition to CFD. FEA is helpful for examining the separator's structural integrity and forecasting stresses and deformations in the separator's component parts. On the other hand, the separator's fluid dynamics and heat transfer may be studied using FVA, which can reveal information about the temperature distribution and pressure drop. In general, numerical analysis of square cyclone separators is a helpful tool for optimising the separator's design and operating circumstances, giving insights into the fluid dynamics and particle behaviour inside the separator, and enhancing the separator's performance.

In the present study, through Computational Fluid Dynamics techniques, the performance characterization of a new Stairmand-type separator cyclone was carried out using the commercial software ANSYS Fluent. Four models for the geometrical cyclone separator were built, namely model A as per the dimensions reported in the literature and models B, C, and D by applying square and circular shape cavities as a passive flow control technique on the surface of its cylindrical section. The Navier-Stokes equations [34] with the RSM turbulence model were formulated to solve the continuous phase of the cyclone separator and, the Lagrangian approach was adopted to track the solid particles with one way-coupling. The proposed model's separation efficiency and pressure drop were compared against those recorded in the previous studies reported in the literature. Model D was the cyclone separator that stood out as the most valuable by demonstrating a separation efficiency and pressure drop decrement of 0.42% and 6.01%, respectively. The study a system for real-time analysis of some involving factors in the efficiency of gas-liquid separators is developed based on the weighing method. An ultrasonic atomizer generates water drops in a size range of 1-10 μm with the same frequency during the test. A cyclone separator is selected and effect of the developing flow rate and shape of the mini-riser as a part of connecting assembly to the cyclone separator is investigated. Further an efficient electrostatic precipitator (ESP) with outcome of single-phase airflow is employed in the downstream of the cyclone to separate remaining droplets and produce the same pressure loss during the test. Circular, triangular, rectangular and square cross section areas with the same hydraulic diameter of 14 mm were examined. The highest cyclone efficiency was recorded for the triangular and rectangular risers comparing to the circular riser at the same length and hydraulic diameter, so the connecting risers with corner could improve the separation efficiency. With the aid of Reynolds stress turbulence, the influence of modelling velocity fluctuations on the prediction of cyclone separator collection efficiency has been studied numerically. Simulating huge eddies using the RSTM model (LES) [35-38]. Modeling using Eulerian-Lagrangian theory method of the CFD code To mimic the three-dimensional, Flu 6.3.26 has been used. Turbulent gas-solid flows that are turbulence-free in a Stair and high efficiency cyclone. The hypothetical 0Results and experimental observations from the literature have been compared. The data analysis demonstrates that the RSTM and LES have successfully anticipated the mean flow region. The current study's findings show that the LES performs well in terms of predicting a fluctuating flow field and collection effectiveness for each and every particle. The cyclone study has developed new theoretical methods for computing travel distance, number of turns, and cyclone pressure drop. The flow pattern and cyclone dimensions determine the travel distance in a cyclone, and the number of turns is calculated based on this travel distance. The new theoretical analysis of cyclone pressure drop was tested against measured data at different inlet velocities and gave excellent agreement. Cyclone cut-points for different dusts were traced from measured cyclone overall collection efficiencies and the theoretical model for calculating cyclone overall efficiency. Regression fit from traced and theoretical cut-point diameters was used to develop cut-point correction models for 2D and 2D cyclones. Experimental results indicate that optimal cyclone design velocities should be determined based on standard air density, and it is important to consider the air density effect on cyclone performance in the design of cyclone abatement systems. The two operational metrics for evaluating the performance of cyclone separators are pressure drop and particle collection effectiveness. Although there are several useful models for forecasting the cyclone pressure drop during the design stage, there are very few methods for assessing particle collection efficiency. In this work, a better mathematical model based on Lapple formula was created to calculate the cut diameter in cyclone separators, a measure of particle collecting efficiency. The cut diameters are represented by the modified Lapple formula with R 2 = 0.9969 and a relative mean square error (RMSE) of 2.533*10-9. A brand-new empirical regression model (R = 0.9619) was also published. Both models' average errors were extremely near to zero. Performance tests indicated that both models can be used confidently to predict cut diameter in cyclone separators. This research investigates the impact of surface roughness on cyclone performance and flow field. The flow inside the cyclone separator is modeled as a three-dimensional turbulent gas flow with solid particles treated as a discrete phase. The Reynolds Stress turbulence model is used to predict the continuous gas flow, while the particle motions are modeled using a Lagrangian approach. Numerical simulation results are compared with experimental data and mathematical models. The analysis of computed results shows that an increase in relative roughness due to corrosion, wear, or particle accumulation on the inner walls has a significant impact on the tangential velocity, cyclone separation efficiency, and pressure drop, especially for high inlet velocities. The decrease in cyclone collection efficiency and pressure drop with an increase in surface roughness is more noticeable for high relative roughness values.

V. TURBULENCE FLOW PATTERN

The complexity of the turbulence flow pattern in a square cyclone separator relies on a number of variables, such as the gas inlet velocity, particle size, and separator geometry [39]. Typically, a swirling motion is created when the gas stream enters the square cyclone separator from the tangential direction, separating the solid particles from the gas stream. The gas stream creates a vortex that revolves around the centre of the separator chamber as it descends towards the separator's bottom. As a result of the rotation's centrifugal forces, the solid particles move towards the chamber's outer walls and eventually fall to the separator's bottom. Square cyclone separator operation must take into account turbulence flow because it has an impact on the pressure drop across the separator and the effectiveness of the separation. A vortex is created in the separator chamber of a square cyclone separator by tangentially introducing the gas stream, which separates the solid particles from the gas flow. It is possible for this vortex flow to become turbulent, which would diminish the effectiveness of separation and heighten the pressure drop. High gas velocities, erratic intake flow, and uneven separator geometry are a few of the causes that might create turbulence flow in a square cyclone separator. Several actions can be taken, such as modifying the input shape and size, tangential velocity, adding a separator to decrease turbulent flow and increase separation efficiency, and introducing a vortex finder. The pressure drop might rise and the separation efficiency can decrease if the vortex flow becomes turbulent. As the gas velocity goes over a critical point, turbulent flow results, making the flow unstable and unpredictable. As a result of the particles' random motion in a turbulent flow, the separation efficiency and pressure drop may be compromised. Several design changes, such as adjusting the tangential velocity and adding a baffle or vortex detector, can be implemented to minimise turbulent flow and increase separation efficiency. Moreover, the turbulence flow pattern in a square cyclone separator may be predicted and examined using numerical simulations with computational fluid dynamics (CFD) software. In general, an understanding of the turbulence flow pattern is crucial for the design and optimisation of square cyclone separators to produce effective particle separation and minimal pressure loss. presents a series of hexagonal cyclones with hexahedral outer tube and outer cone parts. By comparison with Hoffmann cyclone and square cyclone, we confirm the submicrometer particle separation performance of the hexagonal cyclone separator based on the computational fluid dynamics method. The Reynolds stress turbulence model combined with the discrete phase model is used to simulate the three-dimensional gas cyclone separator. The simulation results of the hexagonal cyclone demonstrate an excellent balance of transient separation and swirling flow separation. Two types of optimized hexagonal cyclones are designed, and three parameters are introduced. The results indicate that increasing the hexagonal twist angle and hexagonal outer body round corner diameter leads to higher overall pressure drop and lower collection efficiency in the hexagonal cyclone. The particle residence time in unit volume of the hexagonal cyclone is shorter, and the wall wear rate was lower as well. This represents a substantial potential savings in energy and costs.[12] MoizHussain et al, A cyclone separator is a commonly used device to separate dust particles from gas and dust flow. This project presents the design and development of cyclone separators based on CFD, along with simulations. The study was performed for air-dust flow, based on an experimental study available in the literature. The simulation of flow was done with the help of CFD software, and verification was done with theoretical analysis. Results showed that with an increase in inlet velocity from 8 to 24 m/s, efficiency increases by 5–7% and particle diameter increases. It is recommended that the device be operated within an inlet velocity range of 10-15 m/s to achieve proper operation because the pressure drop values are close to theoretical values. This study examines the effect of various cone and cylinder height variations on the functionality of cyclone separators. The three heights, H/D = 0.5, the major cyclone body diameter, D, and the values of 1.0 and 1.5 are examined. A straight (conventional) profile, a concave, and a triangle are all significant geometrical elements. One tangential intake and two exits are located at the top and bottom of the barrels of the cyclone separator. CFD software was used to simulate the flow, and theoretical analysis was used to verify the results. The efficiency increases significantly with an increase in inlet velocity from 8 to 24 m/s and particle diameter from 5 to 25 micrometers. To ensure proper device operation, it is advised to operate within the inlet velocity range of 10 to 15 m/s. This cyclone study developed new theoretical methods for computing travel distance, number of turns, and cyclone pressure drop. Particle motion in the cyclone outer vortex was analyzed to establish a force- balance differential equation. Cyclone cut-points for different dusts were traced from measured cyclone overall collection efficiencies and the theoretical model for calculating cyclone overall I efficiencies. The cut-point correction models (K) for 1D, 3D, and 2D, 2D cyclones were developed through regression fit from traced and theoretical cut-points. Two sets of inlet design velocities determined by the different air densities were used for the tests. Experimental results indicate that optimal cyclone design velocities should be 16 m/s (3200 ft/min) and 15 m/s (3000 ft/min). It is important to consider the effect of air density on cyclone performance in the design of cyclone performance.

VI. PRESSURE DROP CHARACTERIZATION

Cyclone pressure drop is another key characteristic to be addressed while developing a cyclone system. Calculating the pressure drop in the quantity of inflow velocity heads (Hv) and translating it to a static pressure drop (P) is the Lapple method of pressure drop estimate. Nevertheless, the Lapple pressure drop calculation does not take any vertical dimensions into account as contributing to pressure drop, which leads to the false assertion that an increase in vertical dimensions will boost cyclone efficiency. To predict the cyclone pressure drop associated with cyclone dimensions, a new scientific methodology is required. An essential factor in a square cyclone separator's design and functioning is the pressure drop across it. High pressure drops might lead to increased energy use and decreased separator effectiveness. To maximise the separator's efficiency, it is crucial to define the pressure drop across it. The geometry of the separator, the inflow velocity, and the particle size and concentration all have an impact on the pressure drop through a square cyclone separator. Experimentally or by numerical simulations utilising computational fluid dynamics, the pressure drop can be determined. Pressure sensors are used to measure the pressure differential between the cyclone separator's input and exit in order to experimentally characterise the pressure drop. After that, you may determine the pressure drop by deducting the exit pressure from the intake pressure. To ascertain the impact of the operating parameters on the pressure drop, the pressure drop can be monitored under various operating situations. A square cyclone separator's pressure drop may be described using CFD simulations as well. The separator's fluid dynamics and particle behaviour are modelled in the simulations, and the pressure drop across the separator is calculated. The simulations may be used to analyse how various operational factors affect the pressure drop and improve the separator's design to reduce the pressure drop. Many design changes, including enlarging the separator diameter, lowering the intake velocity, and lengthening the cyclone cone, can be made to lessen the pressure drop across a square cyclone separator. These changes can enhance the separator's internal fluid dynamics, lowering turbulence and increasing separation effectiveness while minimising pressure loss. In conclusion, a square cyclone separator's design and operation include pressure drop characterization as a key component. By quantifying the pressure drop, the performance of the separator may be adjusted, increasing its efficiency and lowering energy usage. The efficient removal of fine dust from exhaust gas reduces the harm to human health and minimizes material waste. A circumfluent cyclone (CFC) was designed to perform with increased gas-solid separation efficiency and decreased pressure drop compared to conventional cyclone separators, and its flow pattern and velocity distribution were investigated. Commercial computational fluid dynamics software was used to simulate the 3D-gas-flow field using the Reynolds stress model (RSM). The simulation results (e.g., for the velocity profile, pressure drop, and separation efficiency) were in good agreement with the experimental results and indicated that about 78% of the gas flows directly out of the CFC through the inner cylinder and then the vortex finder, which significantly shortens the cleaned gas flow path and thereby reduces friction loss. The CFC also decreases the shortcut flow rate near the vortex finder entrance, reducing particle escape. Finally, the discrete particle model (DPM) was used to predict the flow pattern of particles of different sizes. In the present study, through Computational Fluid Dynamics techniques, the performance characterization of a new Stairmand-type separator cyclone was carried out using the commercial software ANSYS Fluent. Four models for the geometrical cyclone separator were built, namely model A as per the dimensions reported in the literature and models B, C, and D by applying square and circular shape cavities as a passive flow control technique on the surface of its cylindrical section. The Navier-Stokes equations with the RSM turbulence model were formulated to solve the continuous phase of the cyclone separator and, the Lagrangian approach was adopted to track the solid particles with one way-coupling. The proposed model’s separation efficiency and pressure drop were compared against those recorded in the previous studies reported in the literature. Model D was the cyclone separator that stood out as the most valuable by demonstrating a separation efficiency and pressure drop decrement of 0.42% and 6.01%, respectively. Pressure drop and collection effectiveness affect cyclone performance. This study tries to improve cyclone performance by optimising the outlet pipe's size. An Eulerian-Lagrangian approach-based numerical technique was employed. The By resolving the three-dimensional, incompressible turbulence flow governing equations, the behaviour of the cyclone was investigated. The Reynolds Stress method was used to model the turbulent flow. Model. The particle equation of motion was solved to get the particle trajectories. The collection efficiency was determined by releasing a predetermined quantity of particles at the cyclone's entrance and counting the particles that were gathered. By contrasting the numerical findings with previously published experimental measurements, the model was confirmed. It was discovered via according to this study, pressure drops as exit pipe diameter increases. The pressure drop in a tangential inflow cyclone can be predicted using a new mathematical model that is presented in this paper. Using a wall friction coefficient based on surface roughness and frictional losses in the cyclone body, the model determines the pressure drop. Reynolds figure. Losses from the entrance and exit are also included in by adding new geometrical parameters to the model. the strain The relationship between cyclone size and drop coefficient is obtained and operational circumstances. 12 studies are used to validate the model. The literature presents various cyclones. the comparison of reported model outcomes with forecasts and measurements Literature demonstrates that the new model accurately forecasts the experimental outcomes under a variety of operational situations, including0.3-220 litres per second.[24] The main objective of the current study is to present a workable technique for increasing the efficiency of a square cyclone separator. In order to investigate its effect on the cyclone performance, a dip leg is inserted beneath the square cyclone. Three-dimensional computational fluid dynamics (CFD) simulations are carried out by solving the Reynolds-averaged Navier-Stokes equation. Equations based on the Reynolds Stress Model (RSM) and the Eulerian-Lagrangian two- phase method. The software programme forecasts turbulent particle dispersion. The discrete random walk (DRW) paradigm The numerical results demonstrate that while using dip leg increased pressure drop, it also improved how well materials were segregated by the square-cyclone.

VII. CONCLUSION

The article concludes that the optimisation of the vortex finder diameter, cone angle, and other geometrical aspects of the square cyclone separator. Using guiding vanes or other sophisticated flow control systems to regulate flow behaviour and lessen turbulence inside the separator. Improvement of separation effectiveness by modification of the cyclone intake design to lessen the influence of the feed stream. Using cutting-edge materials for the separator's construction allows it to endure the abrasiveness of the Al2O3 particles. Including an electrostatic precipitator or a fabric filter system to boost the Al2O3 particle collecting efficiency. Overall, these techniques can greatly increase the effectiveness of collecting Al2O3 particles in a square cyclone separator. While choosing the best strategy to boost the system's efficiency, it's crucial to take the individual operating circumstances and the properties of the Al2O3 particles into account.In conclusion, a square cyclone separator works well for removing particles from liquid or gas streams. Because to its distinctive square design, it offers a number of benefits over conventional cylindrical cyclones, including a lower size and increased collecting efficiency. The particle size distribution, particle shape, and operating conditions are only a few of the variables that affect how well the square cyclone separator performs in particle separation. It is crucial to take these elements into account and choose the proper geometrical parameters and flow control systems in order to maximise the separator's performance. The system's separation efficiency can also be increased by using cutting-edge components and collecting methods like fabric filters or electrostatic precipitators. Overall, a well-constructed and maintained square cyclone separator is a dependable and effective way to separate particles, and it may be used in a variety of sectors, such as the chemical, petrochemical, and pharmaceutical ones.

References

[1]. K. W. Chu, B. Wang, D. L. Xu, Y. X. Chen, and A. B. Yu, "CFD–DEM simulation of the gas–solid flow in a cyclone separator," Chemical Engineering Science, vol. 66, no. 5, pp. 834–847, Mar. 2011, doi: 10.1016/j.ces.2010.11.026.

[2]. Moiz Hussain, Mayur K M, Pradeep D, Prithvi B Srinivas S, 2020. "Design and Analysis of Cyclone Separator," International Journal of Scientific & Engineering Research, Vol. 11, no. 6, ISSN 2229-5518.

[3]. J. Chen, Z. Jiang, and J. Chen, "Effect of Inlet Air Volumetric Flow Rate on the Performance of a Two-Stage Cyclone Separator," ACS Omega, vol. 3, no. 10, pp. 13219–13226, Oct. 2018, doi: 10.1021/acsomega.8b02043.

[4]. R. Vivek, S. Venkatesh, V. Manoj, M. Kumar, and Mohanasundaram, "A brief review on improving materials particulates using cyclone separator by geometrical and turbulence factors," Materials Today: Proceedings, vol. 69, pp. 1076–1079, 2022, doi: 10.1016/j.matpr.2022.08.170.

[5]. Barth W, "Design and layout of the cyclone separator on the basis of new investigations," Brennstoff-Warme-Kraft, 1956.

[6]. S. Pandey and L. S. Brar, "On the performance of cyclone separators with different shapes of the conical section using CFD," Powder Technology, vol. 407, p. 117629, Jul. 2022, doi: 10.1016/j.powtec.2022.117629.

[7]. L. X. Zhou and S. L. Soo, "Gas—solid flow and collection of solids in a cyclone separator," Powder Technology, vol. 63, no. 1, pp. 45–53, Oct. 1990, doi: 10.1016/0032-5910(90)80006-k.

[8]. S. Venkatesh, 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, vol. 383, pp. 93–103, May 2021, doi: 10.1016/j.powtec.2021.01.031.

[9]. B. Wang, D. L. Xu, K. W. Chu, and A. B. Yu, "Numerical study of gas–solid flow in a cyclone separator," Applied Mathematical Modelling, vol. 30, no. 11, pp. 1326–1342, Nov. 2006, doi: 10.1016/j.apm.2006.03.011.

[10]. D. Fal, "Introduction to Industrial Gas Cleaning," Academic Press: San Diego, CA, 1989, pp 55−90.

[11]. F. Zhou, G. Sun, Y. Zhang, H. Ci, and Q. Wei, "Experimental and CFD study on the effects of surface roughness on cyclone performance," Separation and Purification Technology, vol. 193, pp. 175–183, Mar. 2018, doi: 10.1016/j.seppur.2017.11.017.

[12]. S. Venkatesh, S. P. Sivapirakasam, and M. Sakthivel, "Importance of Dust Collectors in Waste Management and Recycling in Industries," Green Materials and Advanced Manufacturing Technology, pp. 23–34, Dec. 2020, doi: 10.1201/9781003056546-2.

[13]. S. Ganegama Bogodage and A. Y. T. Leung, "CFD simulation of cyclone separators to reduce air pollution," Powder Technology, vol. 286, pp. 488–506, Dec. 2015, doi: 10.1016/j.powtec.2015.08.023.

[14]. E. Balestrin, R. K. Decker, D. Noriler, J. C. S. C. Bastos, and H. F. Meier, "An alternative for the collection of small particles in cyclones: Experimental analysis and CFD modeling," Separation and Purification Technology, vol. 184, pp. 54–65, Aug. 2017, doi: 10.1016/j.seppur.2017.04.023.

[15]. L. S. Brar, R. P. Sharma, and K. Elsayed, "The effect of the cyclone length on the performance of Stairmand high-efficiency cyclone," Powder Technology, vol. 286, pp. 668–677, Dec. 2015, doi: 10.1016/j.powtec.2015.09.003.

[16]. C. CORTES and A. GIL, "Modeling the gas and particle flow inside cyclone separators," Progress in Energy and Combustion Science, vol. 33, no. 5, pp. 409–452, Oct. 2007, doi: 10.1016/j.pecs.2007.02.001.

[17]. C. W. Haig, A. Hursthouse, S. Mcilwain, and D. Sykes, "An empirical investigation into the influence of pressure drop on particle behaviour in small scale reverse-flow cyclones," Powder Technology, vol. 275, pp. 172–181, May 2015, doi: 10.1016/j.powtec.2015.02.011.

[18]. A.-N. Huang, K. Ito, T. Fukasawa, K. Fukui, and H.-P. Kuo, "Effects of particle mass loading on the hydrodynamics and separation efficiency of a cyclone separator," Journal of the Taiwan Institute of Chemical Engineers, vol. 90, pp. 61–67, Sep. 2018, doi: 10.1016/j.jtice.2017.12.016.

[19]. E. Kashani, A. Mohebbi, and M. G. Heidari, "CFD simulation of the preheater cyclone of a cement plant and the optimization of its performance using a combination of the design of experiment and multi-gene genetic programming," Powder Technology, vol. 327, pp. 430–441, Mar. 2018, doi: 10.1016/j.powtec.2017.12.091.

[20]. R. D. Luciano, B. L. Silva, L. M. Rosa, and H. F. Meier, "Multi-objective optimization of cyclone separators in series based on computational fluid dynamics," Powder Technology, vol. 325, pp. 452–466, Feb. 2018, doi: 10.1016/j.powtec.2017.11.043.

[21]. A. A. Vegini, H. F. Meier, J. J. Iess, and M. Mori, "Computational Fluid Dynamics (CFD) Analysis of Cyclone Separators Connected in Series," Industrial & Engineering Chemistry Research, vol. 47, no. 1, pp. 192–200, Nov. 2007, doi: 10.1021/ie061501h.

[22]. W. I. Mazyan, A. Ahmadi, H. Ahmed, and M. Hoorfar, "Increasing efficiency of natural gas cyclones through addition of tangential chambers," Journal of Aerosol Science, vol. 110, pp. 36–42, Aug. 2017, doi: 10.1016/j.jaerosci.2017.05.007.

[23]. S. Venkatesh, M. Sakthivel, M. Avinasilingam, S. Gopalsamy, E. Arulkumar, and H. P. Devarajan, "Optimization and experimental investigation in bottom inlet cyclone separator for performance analysis," Korean Journal of Chemical Engineering, vol. 36, no. 6, pp. 929–941, Jun. 2019, doi: 10.1007/s11814-019-0266-2.
[24]. B. E. Launder, "Second-moment closure: present… and future?," International Journal of Heat and Fluid Flow, vol. 10, no. 4, pp. 282–300, Dec. 1989, doi: 10.1016/0142-727x(89)90017-9.
[25]. Ansys, Inc. ANSYS Workbench Help. Release 17.0; ANSYS, Inc:Canonsburg, Pennsylvania, 2017.
[26]. H. Wang, W. Nie, W. Cheng, Q. Liu, and H. Jin, "Effects of air volume ratio parameters on air curtain dust suppression in a rock tunnel's fully-mechanized working face," Advanced Powder Technology, vol. 29, no. 2, pp. 230–244, Feb. 2018, doi: 10.1016/j.apt.2017.11.007.
[27]. D. Misiulia, A. G. Andersson, and T. S. Lundström, "Effects of the inlet angle on the flow pattern and pressure drop of a cyclone with helical-roof inlet," Chemical Engineering Research and Design, vol. 102, pp. 307–321, Oct. 2015, doi: 10.1016/j.cherd.2015.06.036.
[28]. X. Gao, J. Chen, J. Feng, and X. Peng, "Numerical investigation of the effects of the central channel on the flow field in an oil–gas cyclone separator," Computers & Fluids, vol. 92, pp. 45–55, Mar. 2014, doi: 10.1016/j.compfluid.2013.11.001.
[29]. Y. Shi, Z. Zhu, and Y. Fan, "New form of geodetic coordinate system taking two length quantity as coordinate parameters," Frontiers of Architecture and Civil Engineering in China, vol. 3, no. 1, pp. 105–110, Mar. 2009, doi: 10.1007/s11709-009-0014-5.
[30]. S. Demir, "A practical model for estimating pressure drop in cyclone separators: An experimental study," Powder Technology, vol. 268, pp. 329–338, Dec. 2014, doi: 10.1016/j.powtec.2014.08.024.
[31]. Y. Huang, M. Zhang, J. Lyu, Z. Liu, and H. Yang, "Effects of gas leakage on the separation performance of a cyclone. Part 1: Experimental investigation," Chemical Engineering Research and Design, vol. 136, pp. 900–905, Aug. 2018, doi: 10.1016/j.cherd.2018.03.047.
[32]. J. Yang, G. Sun, and M. Zhan, "Prediction of the maximum-efficiency inlet velocity in cyclones," Powder Technology, vol. 286, pp. 124–131, Dec. 2015, doi: 10.1016/j.powtec.2015.07.024.
[33]. S. Yuu, T. Jotaki, Y. Tomita, and K. Yoshida, "The reduction of pressure drop due to dust loading in a conventional cyclone," Chemical Engineering Science, vol. 33, no. 12, pp. 1573–1580, 1978, doi: 10.1016/0009-2509(78)85132-x.
[34]. Z. Madadi, F. Hassanibesheli, S. Esmaeili, L. Hedayatifar, and A. A. Masoudi, "Surface growth by cluster particles: Effects of diffusion and cluster's shape," Journal of Crystal Growth, vol. 480, pp. 56–61, Dec. 2017, doi: 10.1016/j.jcrysgro.2017.10.010.
[35]. J. Valente de Oliveira, A. Szabo, and L. N. de Castro, "Particle Swarm Clustering in clustering ensembles: Exploiting pruning and alignment free consensus," Applied Soft Computing, vol. 55, pp. 141–153, Jun. 2017, doi: 10.1016/j.asoc.2017.01.035.
[36]. L. Wang, C. Wu, and W. Ge, "Effect of particle clusters on mass transfer between gas and particles in gas-solid flows," Powder Technology, vol. 319, pp. 221–227, Sep. 2017, doi: 10.1016/j.powtec.2017.06.046.
[37]. K. Wu et al., "Study on the distribution characteristics of flexible filamentous particle clusters in a fluidized bed dryer," Powder Technology, vol. 331, pp. 7–19, May 2018, doi: 10.1016/j.powtec.2018.02.040.
[38]. S. Venkatesh, M. Sakthivel, S. Sudhagar, and S. A. A. Daniel, "Modification of the cyclone separator geometry for improving the performance using Taguchi and CFD approach," Particulate Science and Technology, vol. 37, no. 7, pp. 799–808, May 2018, doi: 10.1080/02726351.2018.1458354.
[39]. F. Ş. Kılavuz and Ö. Y. Gülsoy, "The effect of cone ratio on the separation efficiency of small diameter hydrocyclones," International Journal of Mineral Processing, vol. 98, no. 3–4, pp. 163–167, Mar. 2011, doi: 10.1016/j.minpro.2010.11.006.