

The Effect of Vertex Finder and Dust Outlet on Collection Performance in Square Cyclones Separators with Downward Gas Exit

Ryom Junho, Jin Songchol & Jon Choljin

Faculty of Thermal Engineering, Kim Chaek University of Technology, Pyongyang, Democratic People's Republic of Korea

Received: 20.09.2025 | Accepted: 01.10.2025 | Published: 21.10.2025

*Corresponding Author: Jon Choljin

DOI: [10.5281/zenodo.17403762](https://doi.org/10.5281/zenodo.17403762)

Abstract

In this study, computational fluid dynamic method is used to predict and evaluate the flow field and collection performance inside square cyclone separators. Two cyclones with different dust outlet were studied using the Reynolds stress model (RSM). The cyclone flow field pattern has been simulated and analyzed with the aid of static pressure, and velocity components. In addition the cyclone collection efficiency based on one-way discrete phase modeling has been investigated. The obtained results demonstrate that collection performance of the square cyclone separator with vertical dust-outlet is very well. The simulation results agree well with the published experimental results.

Keywords: CFD, Square cyclone, downward exit, Flow field, collection performance, CFB boiler.

Original Research Article

Copyright © 2025 The Author(s). This is an open-access article distributed under the terms of the Creative Commons Attribution-NonCommercial 4.0 International License (CC BY-NC 4.0).

1. INTRODUCTION

The circular cross-section cyclone with upward gas exit has been proved excellent performance for the small-sized CFB boilers [1-3]. With the development of large CFB boilers, the cyclone separator with downward gas exit has been explored by Chen et al. [4] and Liu et al. [5], which shows the convenience in construction and easy integration with the boiler. Later, the square cyclone separator is suggested for the Π -arrangement of the CFB boiler, which is convenient to install heat transfer surfaces on the inner separator walls.

The previous research for the flow field in the cyclones separator with downward gas exit [6-12] only were concentrated at the space between vertex and gas exit, won't obtained the flow characteristics at each sections in the cyclones. Raoufi et al. [12] predicted the flow field inside a square cyclone with upward and downward gas exit. In this study, the flow characteristics within square cyclones with

downward gas exit is simulated in details.

Computational fluid dynamics (CFD) has a great potential to predict the flow field characteristics and the pressure drop of the cyclone [10]. Up to now, there are many CFD simulations have been conducted on the conventional circular cyclone. Khairy [2] numerically optimized the cone-tip diameter of the circular cross-section cyclones with upward gas exit. Only a little investigation on the square cyclone can be found in open publications.

In study, two square cyclone separators with different downward dust outlet (incline and vertical dust outlet) are used, and characteristics of flow field and collection performance in two square cyclone separators are investigated and compared.

2. Numerical simulation

2.1. Model description

The conservation equations of mass and

momentum for incompressible Newtonian flows are as follows:

$$\frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g \vec{v}_g) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} (\rho_g \vec{v}_g) + \nabla \cdot (\rho_g \vec{v}_g \vec{v}_g) = -\nabla P + \nabla \cdot (\bar{\tau}) - \rho_g \vec{g} + \vec{F} \quad (2)$$

Where ρ_g is the fluid density, v_g is the fluid velocity, p is the static pressure, τ is the stress tensor, μ is the molecular viscosity, g is the acceleration of gravity, and F is the external body force.

Reynolds stress model (RSM) has been regarded as an efficient method to predict the highly swirling turbulent flow within cyclone separators vortex by compared with experimental data[8, 12], which is written as follows:

$$\begin{aligned} \frac{\partial}{\partial t} (\rho_g \overline{u'_i u'_j}) + \frac{\partial}{\partial x_k} (\rho_g u_k \overline{u'_i u'_j}) &= \frac{\partial}{\partial x_k} [\rho_g \overline{u'_i u'_j u'_k} + P(\delta_{kj} u'_i + \delta_{ik} u'_j)] \\ + \frac{\partial}{\partial x_k} [\mu \frac{\partial}{\partial x_k} (\overline{u'_i u'_j})] - \rho_g (\overline{u'_i u'_j} \frac{\partial u_j}{\partial x_k} + \overline{u'_j u'_k} \frac{\partial u_i}{\partial x_k}) &- \rho_g \beta (g \overline{u'_j \theta} + g \overline{u'_i \theta}) \\ + P(\frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i}) u'_j - 2\mu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} + S \end{aligned} \quad (3)$$

where t is time, ρ_g is the density of fluid, u'_i is the fluctuating velocity to direction i ($=u_i - u_m$), u_i is the velocity to direction i , u_m is the mean velocity to direction i , $\overline{u'_i u'_j}$ is the Reynolds stress tensor, β is the coefficient of thermal expansion, p is the pressure,

μ is the molecular viscosity, and S is the source term.

The motion equation of small particles, considering the effects of nonlinear drag and gravitational forces, is given by [13]:

$$\frac{du_p}{dt} = \frac{3\nu C_D \text{Re}_p}{4d^2 s} (u_g - u_p)_x + g \quad (4)$$

Where u_p is the velocity of the particle and ν is the kinetic viscosity, d is the particle diameter, s is the ratio of particle density to fluid density, and g is the acceleration of gravity.

2.2. Simulation procedure

As shown in Fig.1, two cyclone separators have the same geometry and dimension excepting that the dust outlet is different. The detailed geometries of the square cyclone separators are listed

in Table 1.

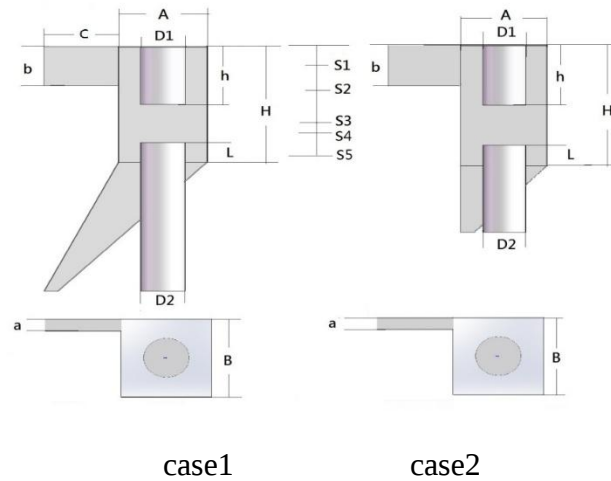


Fig.1. the detailed geometry of two cyclones

Table1. Detailed geometry of the square cyclone separators (mm)

a	B	A	B	c	D1	D2	H	h	L	S1	S2	S3	S4	S5
20	60	120	120	100	60	60	180	90	30	30	70	120	130	170

It is very important for accuracy and convergence of calculation to select suitable numerical schemes for solving the governing and transport equations with RSM and DPM models. In this study, semi-implicit method pressure-linked equation (SIMPLE) is used for dealing with pressure velocity coupling, pressure staggered option (PRESTO) for the finite-difference pressure interpolation, and quadratic upstream interpolation for convective kinetics (QUICK) for the momentum equations. Second-order upwind is employed for calculating turbulent kinetic energy, turbulent dissipation rate, and Reynolds stress.

3. Results and discussions

3.1. The flow pattern of two cyclones

In this study, the flow field is explored at different axial sections of the cyclone, i.e., the inlet annular space, the space between vertex finder and gas-exit, and the space below gas-exit. As shown in Fig.1, three sections are Section S1 (the inlet annular space), S3 (the space between vertex finder and gas-exit) and S5 (the space below gas exit) are at 30, 120

and 170 mm from top of two cyclones.

The velocity field in the cyclone includes tangential, axial and radial components. Since the flow in a cyclone is strongly swirling, the tangential velocity is more important than the axial and radial components. Fig.2 presents the radial profiles of tangential velocity at three different sections for cases 1 and 2, respectively. It is clear that the pattern of the tangential velocity profile is the similar, but the magnitude increases with an increase of the gas inlet velocity from 20 m/s to 25 m/s. Furthermore, the tangential velocity profiles at S1 and S3 sections in two cases are consisted of two regions, i.e., inner region and outer region (see Figs. 2a, 2b, 2d and 2e). The tangential velocity increases from the center to the wall in the inner region, whilst the tangential velocity decreases with radius in the outer region. This means that the tangential velocity profile exhibits the Rankine type vortex feature. In the S5 section of the cyclones, however, the tangential velocity profiles exhibit “W” type feature (see Figs. 2c and 2f).

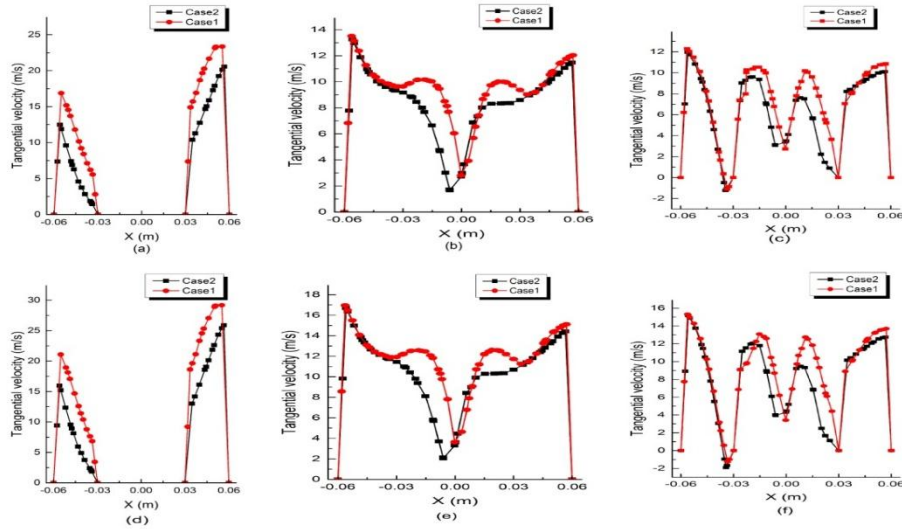


Fig.2. Comparison between the radial profiles for the tangential velocities of different inlet velocities at different sections. (a), (b), (c): at S1, S3, S5 at the inlet velocity $u=20\text{m/s}$. (d), (e), (f): at S1, S3, S5 at the inlet velocity $u=25\text{m/s}$

Fig.3 shows the radial profiles of axial velocity is the similar at three sections when the gas inlet velocity between 20 m/s and 25 m/s. At S1 section in Figs. 3a and 3d, the axial velocity profiles exhibit an obvious asymmetrical pattern. In the left region the gas moves downward with a maximum value at its center

area. In the right region, however, the gas flows downward near the wall of vertex finder, then flows upward near the wall of the cyclone. This is largely contribute to the effect of the corner vortex. At S3 section in Figs. 3b and 3e, the axial velocity is downward in the near wall region and has a greater amount in the right side, which shown a good agreement with the experimental data([10]). At S5 section in Figs. 3c and 3f, the trend of axial velocity is similar to that at S2 section, which implies that the axial velocity profiles display a very asymmetrical shape.

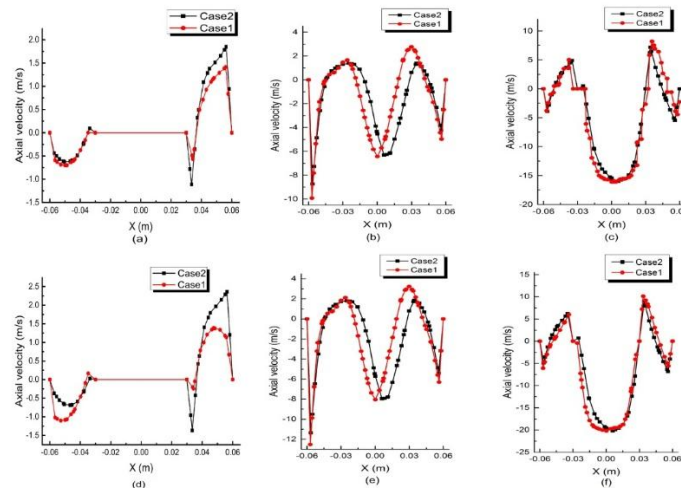


Fig.3. Comparison between the radial profiles for the axial velocities of different inlet velocities at different sections. (a), (b), (c): at S1, S3, S5 at the inlet velocity $u=20\text{m/s}$. (d), (e), (f): at S1, S3, S5 at the inlet velocity $u=25\text{m/s}$

3.2. The performance of two cyclone separators

3.2.1. The pressure field

The static and dynamic pressure drop between inlet and outlet is shown as a function of

inlet velocity for two cases in Fig.4. In two cyclones static and dynamic pressure drop are all proportional to inlet velocity, one of case2 is lower than one of case1 at inlet velocities $u=20\text{m/s}$ and $u=25\text{m/s}$.

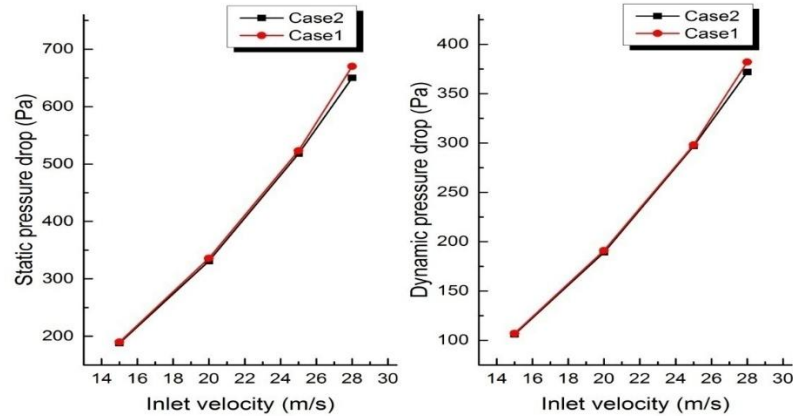


Fig.4. Comparison of static and dynamics pressure drop at two cyclones

3.2.2. The collection efficiency

Fig.5 shows the collection efficiency for particles of diameter $d=30\mu\text{m}$ at two square cyclones. As shown at the figure, the collection efficiency for the square cyclone of case2 is greater than one of case1, in particular, if the inlet velocity is small, the collection efficiency is significantly increased, for

example, if the inlet velocity $u=10\text{m/s}$, the collection efficiency is more than 10% higher, as the inlet velocity increases, the difference is gradually reduced. The collection efficiency of case2 is higher than one of case2 because there isn't the inclined dust outlet, is decreased the disadvantageous effect on the re-scattering of collected particles.

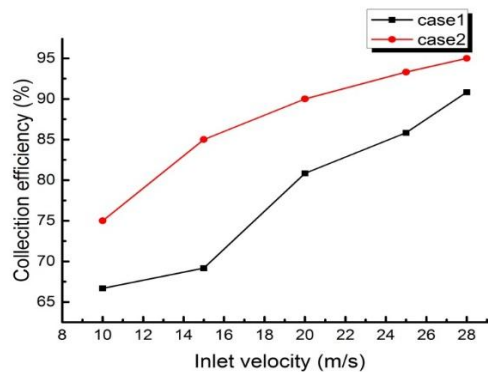


Fig.5. Comparison of the collection efficiency for particles of diameter $d=30\mu\text{m}$ at two cyclones

4. Conclusions

- 1) Except exhaust exit, the tangential velocity

profiles at all the sections in two cyclones exhibit Rankine type vortex feature where tangential velocity in inner region increases with radius,

tangential velocity in outer region decreases with radius. In the exhaust exit of the cyclones, the tangential velocity profiles exhibit “W” type feature where the tangential velocity from center to wall respectively increases with radius and decreases with radius.

2) The axial velocity profiles exhibit a severe asymmetrical feature, the magnitude of the tangential and axial velocity for case1 is greater than one for case2.

3) Static and dynamic pressure drop in two cyclones are all proportional to inlet velocity, in the cyclone separator of case2, pressure loss is also lower than one of case1.

4) The collection efficiency for particles of diameter $d=30\mu\text{m}$ for the square cyclone of case2 is greater than one of case1, in particular, if the inlet velocity is small, the collection efficiency is significantly increased, for example, if the inlet velocity $u=10\text{m/s}$, the collection efficiency is more than 10% higher.

5) The type of the inclined dust outlet of square cyclones is disadvantageous to increase the collection performance.

REFERENCES

- [1] W.D. Griffiths, F. Boysan, Computational fluid dynamics (CFD) and empirical modeling of the performance of a number of cyclone samplers, *Journal of Aerosol Science* 27 (1996) 281–304.
- [2] Khairy Elsayed, Chris Lacor, The effect of cyclone vortex finder dimensions on the flow pattern and performance using LES, *Computers & Fluids* 71 (2013) 224–239.
- [3] Jolius Gimbuna, T.G. Chuahb, Thomas S.Y. Choongb, A. Fakhru’l-Razi, Prediction of the effects of cone tip diameter on the cyclone performance, *Aerosol Science* 36 (2005) 1056–1065.
- [4] H. Chen, Z. Lin, D. Liu, New type of separator for CFBs with low pressure drop and high efficiency, in: *Proceedings of the 10th Int. Conf. on Fluidized Bed Combustion*, 1 (1989) 413–417.
- [5] D. Liu, H. Chen, Z. Lin, Design of PICFB boilers, in: *Proceedings of the 11th Int. Conf. on Fluidized Bed Combustion ASME*, 1 (1991) 585–588.
- [6] Shurong Wang, Mengxiang Fang, Zhongyang Luo, Xuantian Li, Mingjiang Ni, Kefa Cen, Instantaneous separation model of a square cyclone, *Powder Technology* 102(1999) 65–70.
- [7] H. Safikhani, M. Shams, S. Dashti, Numerical simulation of square cyclones in small sizes, *Advanced Powder Technology* 22 (2011) 359–365.
- [8] Yaxin Su, Anqiao Zheng, Bingtao Zhao, Numerical simulation of effect of inlet configuration on square cyclone separator performance, *Powder Technology* 210 (2011) 293–303.
- [9] Yaxin Su, Bingtao Zhao, Anqiao Zheng, Simulation of Turbulent Flow in Square Cyclone Separator with Different Gas Exhaust, *Industrial & Engineering Chemistry Research*, 50 (2011) 12162–12169.
- [10] Y. Su, Y. Mao, Experimental study on the gas–solid suspension flow in a square cyclone separator, *Chemical Engineering Journal* 121 (2006) 51–58.
- [11] Y. Su, The turbulent characteristics of the gas–solid suspension in a square cyclone separator, *Chemical Engineering Science* 61 (2006) 1395 – 1400.
- [12] Arman Raoufi, Mehrzad Shams, Homayoon Kanani, CFD analysis of flow field in square cyclones, *Powder Technology* 191 (2009) 349–357.
- [13] B. E. Launder, G. J. Reece, W. Rodi, Progress in the development of a Reynolds stress turbulent closure[J]. *J Fluid Mechanics*. 68 (1975) 537–538.

- [14] ANSYS Inc., ANSYS FLUENT 12.0 theory guide, Canonsburg, 2009.
- [15] F. Kaya, I. Karagoz, Performance analysis of numerical schemes in highly swirling turbulent flows in cyclones, Current Science 94 (10) (2008) 1273–1278.