
Ramin Moradi  
Ferdowsi University of Mashhad  
ramin.moradi2020@gmail.com

Amin Deyranlou  
Ferdowsi University of Mashhad  
deeyranlou.amin@gmail.com

Mohammad Moghiman  
Ferdowsi University of Mashhad  
moghiman@um.ac.ir

Abstract

This paper presents a numerical study of the gas–powder flow in a typical Lapple cyclone with division of gas and particle flow in a vortex finder. The Navier-Stokes equations along with the RNG k-ε turbulent model are solved numerically. The separation efficiency and the trajectory of particles are simulated and the effects of the particle size on the separation efficiency and the particle residence time are investigated. The effect of the particle density on the particle size in the range which results 100% cyclone separation efficiency and particle residence time is investigated. Large particles generally have a higher concentration in the wall region and small particles have a higher concentration in the inner vortex region. Particles enter from different sides give different separation efficiency and trajectory. A particle with a size exceeding a critical diameter or a critical density would stagnate on the wall of the cyclone's cone. This phenomenon is regarded as a main reason for the deposition on the inner conical surface in such cyclones used in cement industry.

Key words: Cyclone, Gas-solid flow, Particle separator, Computational fluid dynamics, RNG model

1. Introduction

Gas cyclone separators are widely used in industries to separate dust from gas or for the product recovery because of its geometrical simplicity, economical proficiencies in power supply and flexibility [1]. They are widely used in petrochemical and process industries for the removal of the particles from their carrying fluids [2]. Using suitable materials and proper methods of construction, cyclones can operate at high temperature and pressure conditions [2]. They are simple in construction, do not require extensive cost and maintenance, and show a relatively high level of efficiency (especially for particles bigger than 10 microns). There are various methods for constructing cyclones. A classical centrifugal collector with a reversed-flow cyclone and a tangential inlet is the most frequently used [3].

The gas–solid suspension flow behaviors in cyclones have been subject of many experimental, theoretical and numerical studies [4, 5, 6]. In the literatures, there are two typical ways for improving cyclone performance. The first modification is about cyclone's
shape. Optimization of cyclone dimensions and adding double inlet spiral-shape entrance are examples of such studies. Double inlet spiral-shape entrance can improve the symmetry of the gas flow pattern and enhances the particle separation efficiency [7]. The second modification is about flow characteristics like velocity inlet condition or temperature. For example inlet gas velocity is an effective parameter on the cyclone performance and increases pressure drop [1].

The dispersed phase modeling is the simplest model for the numerical simulation. The method assumes that the influence of the dispersed phase of the particles on the flow field of the continuous phase and particle–particle interactions may be neglected. Gujun Wan et al. [2] studied the gas flow in a cyclone by using LES model and Wang et al. [1] solved the gas flow equations in a typical Lapple cyclone by using the Reynolds stress model. In this study, the RNG k-ε model under gravitational condition is used by the commercial software package "FLUENT 6.3" to study the gas–solid flow in a typical Lapple cyclone separator.

The simulation of the particle flow in the cyclone is performed based on the gas flow. The particle tracking method is used to calculate the separation efficiency. It is assumed that once the particle touches the bottom of the cyclone dust-bin, it is collected and the computation can be terminated for this particle (the particle is trapped). The fractional collection efficiency can be obtained by calculating the ratio of the trapped and the released particles (from gas inlet) or on the basis of the escaped particles (escape from the vortex finder). The first method is used in this analysis.

\[
\text{efficiency} = \frac{\text{number of trapped particles}}{\text{number of released particles}} \quad (1)
\]

2. Mathematical Modeling
2.1. Computational Model

Fluid flows are described by a set of nonlinear, partial differential equations, namely the Navier-Stokes equations [8]:

\[
\rho \frac{\partial \tilde{u}^i}{\partial x^j} = \frac{\partial p}{\partial x^i} + \frac{\partial}{\partial x^j} \left[ \mu \left( \frac{\partial \tilde{u}^i}{\partial x^j} + \frac{\partial \tilde{u}^j}{\partial x^i} \right) \right] + \frac{\partial \tau^{ij}}{\partial x^j} \quad (2)
\]

and the continuity equation:

\[
\frac{\partial u'}{\partial x'} = 0 \quad (3)
\]

where the superscripts i,j = 1,2,3 indicate the vector components in the Cartesian coordinate system, \( u, p, \rho \) and \( \mu \) are the fluid velocity, pressure, density and molecular viscosity, respectively, and

\[
\tau^{ij} = -\rho \frac{\partial u' u'}{\partial x'} \quad (4)
\]

is the Reynolds stress tensor which represents the effects of the turbulent fluctuations on the fluid flow.

A practical way of simulating fluid flows in the engineering applications is to use turbulence models which are solved for the mean fluid velocity and the pressure. The RNG k-
The ε model employs the following expression to correlate the Reynolds stress tensor $\tau_{ij}$ to the mean fluid velocity:

$$
\tau_{ij} = -\frac{2}{3} \rho k \delta_{ij} + \mu_i (\frac{\partial u^i}{\partial x^j} + \frac{\partial u^j}{\partial x^i})
$$

(5)

where $k$ is the turbulent kinetic energy, $\delta_{ij}$ is the Kronecker delta and the eddy viscosity $\mu_i$ is determined by the RNG theory formula:

$$
\mu_i = \mu [1 + \left( \frac{c_p \rho}{\mu} \frac{k}{\varepsilon} \right)^2]
$$

(6)

which is valid across the full range of turbulent fluid flow conditions from the low to the high Reynolds number flows. The turbulent kinetic energy $k$, and the dissipation rate $\varepsilon$ in the RNG k-ε model are formulated by the following two transport equations. The kinetic energy equation is described as

$$
\rho u^i \frac{\partial k}{\partial x^j} = \alpha_i \frac{\partial}{\partial x^j} \left( \mu_{eff} \frac{\partial k}{\partial x^i} \right) + \varphi - \rho \varepsilon
$$

(7)

and the dissipation equation is

$$
\rho u^j \frac{\partial \varepsilon}{\partial x^j} = \alpha_i \frac{\partial}{\partial x^j} \left( \mu_{eff} \frac{\partial \varepsilon}{\partial x^i} \right) + c_{\varepsilon} \frac{\varepsilon}{k} \phi - c_{\varepsilon} \rho \frac{\varepsilon^2}{k} - R
$$

(8)

where

$$
\varphi = \mu_{eff} \left( \frac{\partial u^i}{\partial x^j} + \frac{\partial u^j}{\partial x^i} \right) \frac{\partial u^j}{\partial x^i}
$$

(9)

$$
R = \frac{2 \mu}{\rho} S_{ij} \frac{\partial u^i}{\partial x^j} \frac{\partial u^j}{\partial x^i}
$$

(10)

$$
S_{ij} = \frac{1}{2} \left( \frac{\partial u^i}{\partial x^j} + \frac{\partial u^j}{\partial x^i} \right)
$$

(11)

$$
\mu_{eff} = \mu + \mu_i
$$

(12)

in the RNG k-ε model, the value of $R$ is given by:

$$
R = \rho \frac{c_p \eta^3 (1 - \eta / \eta_0)}{1 + \beta \eta^3} \frac{\varepsilon^2}{k}
$$

(13)

where

$$
\eta = \frac{k}{\varepsilon} \left( \frac{2 S_{ij} S_{ij}}{\varepsilon} \right)^{1/2}
$$

(14)

2.2. Computational Domain

Figure 1 shows the cyclone dimensions and figure 2 shows the computational domain, containing 120753 tetrahedral meshes. Using from the cyclone boundary conditions presented by Wang et al. [1], the inlet velocity of the gas and the particles are both assumed 20 m/s. The zero gradient boundary condition is applied for the tube’s outlet and the gas pressure at the
top of the vortex finder is assumed 1 atm and the cement density is taken 3320 kg/m$^3$. Three grid sizes are tested in the computation domain, containing 47750, 95350, 120753 cells, respectively. The results show that the grid size with 95350 cells, keeps either the accuracy and reduces time expense as well. The results are reported for the gas pressure, flow filed pattern, particle residence time and the collector efficiency. The effects of the particle size and the particle diameter are investigated too.

2.3. Results and Discussion

2.3.1. Pressure field

Figure 3 shows the static pressure contour. The pressure decreases in the radial direction toward the center and results a negative pressure zone. The largest pressure gradient occurs along the radial direction since there is a highly intensified forced vortex.
The gas enters to the cyclone with a high pressure. Since the gas stream collides with the upstream and forms a chaotic flow close to the vortex finder, Point A becomes a low pressure area in this region. This phenomenon can separate the inlet flow from the stream which is entered to the cyclone and produces the low pressure area. It would increase the energy loss and the pressure drop in the cyclone. This is the main cause of the short-circuiting flow and often results in a high pressure drop. To overcome this problem, it is suggested that the inlet shape should be modified [9].

2.3.2 Effect of the particle size on the separation efficiency

In the Figure 4, the efficiency of the cyclone with respect of the particle diameter is reported. Particle diameter varies from the 0.2 to 100 microns. The diameter of the particles is applied directly from the injection tab of the FLUENT package. The air and the particles velocity inlet are 20 m/s, and the constant particles density 3320 kg/m3 is applied. The cyclone efficiency would be 100 % whenever the particles size varies in the range of 3-35 microns. For the particle diameter larger than 35 microns, the efficiency becomes zero and for the values less than 3 microns it would be less than 100%. It should be noted that in this study if any particles remain in the cyclone, the suitable results are not concluded. So particles with diameter larger than 35 microns have efficiency equals to zero whenever they are not escaped to the atmosphere.

![Figure 4: effect of the particle diameter on the efficiency](image)

Each particle released from the inlet may have three possible final trajectories:

- Escapes from the dust outlet and enters to the cyclone dust bin (this is the aim of the cyclone design).
- Escapes from the vortex finder to the atmosphere or another cyclone.
- Sticks near the wall usually at the beginning of the conical part of the cyclone and circulates. It may be caused deposition on the cyclone wall.

In the figure 5, different trajectories presented. According to the particle diameter, the distance that it moves in the cyclone would be varied; consequently different residence times
will result. Trajectory of 35-microns particle - that is the critical size - is longest and the particle touches the bottom of the cyclone hardly.

![Particle Trajectory Image]

Figure 5: particle trajectory

2.3.3. Effect of the particle density on the cyclone performance

The range of the particle diameter in which the cyclone efficiency being 100%, affected by the particle density. In the figure 6 the critical range which results maximum efficiency is illustrated for the four particles (wood, gypsum, cement, steel). As the density increases, the diameter range in which the maximum cyclone efficiency would be obtained decreases. Therefore it must be noted that the particle material has an important effect on the cyclone performance.

![Particle Density vs Efficiency Image]

Figure 6: effect of the particle density on the performance

2.3.4. Effect of the particle diameter on the residence time

Particle trajectory depends on the node at the inlet area of the cyclone in which particles released. In figures 7 and 8, particles released from the same node at the middle of the inlet area. Figure 7 presents the particle residence time on the particle diameter. As the particle size increases, it has to travel longer distance in the cyclone till escaping to the dust bin (trapped),
so the residence time would be increased. Figure 5 obviously illustrates the above results. Residence time would be more crucial whenever there are some reactions between fluid and the particles that causes combustion or between two discrete particles in which one particle acts as an absorbent for the filtering purposes.

2.3.5. Effects of the particle density on the residence time

Residence time is defined as the time span in which a particle moves along the cyclone from its inlet to the outlet of the cyclone. Figure 8 illustrates the effect of the particle density on the residence time. It is predictable from the figure 7 that as particles being heavier, they stand more time in the cyclone till they escape to the dust bin if they are being in the critical diameter range. Results are shown in the figure 8. The residence time would be increased as the particle density increases. It can conclude that particle diameter has a more intense effect on the residence time. Thus for increasing the particle residence time, particle diameter should be increased.

3. Conclusions

In this paper effects of the particle size and the particle density are studied numerically on the flow field, solid phase pattern and the separation efficiency of a simple cyclone with one inlet which are using in the cement industry.

The cyclone efficiency depends on the particle size and the particle density. If the particles which are entered to the cyclone are not being in the critical range that result 100% cyclone efficiency, the cyclone can't separate them from the flow effectively. Therefore the particles will escape to the atmosphere or may remain in the cyclone and making some troubles for the cyclone performance like deposition. An important factor for increasing the Particle residence time in a cyclone is the particle weight. As the particle size and the particle density increases, the residence time increases too. However the particle diameter is more effective on the particle residence time.

Researchers try to reduce all energy dissipating factors. The friction occurs due to the fluid viscosity is one of the main energy destruction resources. This friction, in turns causes the
pressure drop in the mechanical facilities, thus it causes energy loss. Therefore it is important to have a best design in the different parts of the cyclone. The shape and the gas inlet location are important factors in pressure drop.

References