A Study of Swirling Flows in a Cyclone Separator Using a Large Eddy Simulation *

Kazuyoshi MATSUZAKI**, Yasuhito KUDO***, Mizue MUNEKATA**** and Hideki OHBA*****

**Tokyo Electron LTD.,
650 Mitsuzawa Hosaka-cho Nirasaki, Yamanashi 407-0192 Japan
E-mail: kazuyoshi.matsuzaki@tel.com
***Ishikawajima-Harima Heavy Industries Co., Ltd.,
TOYOSU IHI BUILDING, 1-1, Toyosu 3-chome, Koto-ku, Tokyo 135-8710 Japan
****Dept. of Mechanical System Engineering, Kumamoto University,
2-39-1 Kurokami, Kumamoto, 860-8555 Japan
E-mail: munekata@kumamoto-u.ac.jp
*****Dept. of Mechanical System Engineering, Kumamoto University,
2-39-1 Kurokami, Kumamoto, 860-8555 Japan
E-mail: ohba@kumamoto-u.ac.jp

Abstract
A numerical simulation of swirling flows in a cyclone separator has been performed using a large eddy simulation (LES) based on a Smagorinsky model. The validity of the simulation and the nature of complex flow characteristics are discussed in the context of experimental results. In this work, particle motion is described using a one-way Lagragian method. Particle separation performance is assessed from particle tracking results. A result of this study investigation demonstrates that the LES has been sufficiently capable of predicting the complex swirling flows in a cyclone separator.

Key words: Swirling Flow, Cyclone Separator, Large Eddy Simulation, Particle Motion, Numerical Analysis

1. Introduction

A cyclone separator uses centrifugal force produced by a swirling flow to separate substances with the same specific weight but different size. It has been widely utilized in the refining process of powder materials. However, the state-of-the-art separation finesse is insufficient for fabrication of new design powder materials. The development of an advanced cyclone separator with high-accuracy particle separation is necessary because the design based on the traditional principle to separate large particles is not reasonably applicable. As a basis for development, it is essential to characterize the turbulent flow characteristics of complicated swirling flows in the cyclone separator because the motion of small particles is sensitive to fluctuations in the flows.

We developed an advanced tangential-type cyclone separator, and experimentally investigated the turbulent characteristics of the swirling flows using a model of cyclone with simplified structure in order to elucidate the relation between the flow characteristics and the particle separation (1)-(3). We therefore used numerical simulations to clarify the behavior of swirling flows. In previous numerical studies, Reynolds averaged numerical simulation (RANS) is used to investigate swirling flows in cyclone separators (4),(5). In such
simulations, it is difficult to describe unsteady phenomena particular to swirling flows in cyclone separators such as oscillating vortex core motion and vortex breakdown. Moreover, it is reported that a standard k-ε model which is widely used in RANS cannot accurately predict the flow characteristics of swirling flows (6).

In this study, a large eddy simulation (LES) is applied to describe the complex swirling flows in the cyclone separator. The validity of the method is confirmed through comparison with experimental results. Moreover, we try to predict the performance for particle separation of the cyclone separator with the method that individual particles are traced with a one-way Lagragian method.

In this paper, the one-way method used in this study is described in the next section, which includes the large eddy simulation for fluid and model formulation of particle motion. Computational results and discussions are presented in §3. We end with brief concluding remarks.

Nomenclature

\( A \) : Projected area of particle \((= \pi d_p^2 / 4)\)

\( C_1, C_2, C_3 \) : Coefficient for viscous

\( C_D \) : Drag coefficient

\( C_LR \) : Lift coefficient

\( C_T \) : Non-dimensional coefficient

\( C_s \) : Smagorinsky constant

\( d_p \) : Particle diameter (5, 10 µm)

\( D / Dt \) : Substantial derivative

\( g \) : Acceleration of gravity (9.81 m/s\(^2\))

\( H \) : Height of swirl chamber in the cyclone separator (340 mm)

\( I \) : Inertia moment of particle \((= \pi \rho d_p^5 / 60)\)

\( m \) : Mass

\( r, \theta, z \) : Cylindrical coordinate system

\( q_r \) : Radial flux \((= r \times v_r)\)

\( r_o \) : Radius of outlet pipe

\( R \) : Radius of swirl chamber in the cyclone separator (72 mm)

\( Re_p \) : Particle Reynolds number \((d_p \| u_p \| / \nu)\)

\( Re_R \) : Rotational Reynolds number of particle \((= d_p \| \omega_p \| / 4\nu)\)

\( T \) : Time normalized by \( R / V_{in} \)

\( u_i \) : Velocity component in \( x, y \) and \( z \) direction \((i = 1, 2, 3)\)

\( V_{in} \) : Inlet velocity (7.8 m/s)

\( v_r, v_{th}, v_z \) : Velocity component in \( r, \theta \) and \( z \) direction

\( x_i \) : \( x, y, z \) coordinate system \((i = 1, 2, 3)\)

\( \eta \) : Collection rate of particle

\( \Delta t \) : Non-dimensional time increment

\( \mu_f \) : Fluid viscosity (18.22×10\(^{-6}\) Pa s)

\( \nu \) : Kinematic viscosity (15.12×10\(^{-6}\) m\(^2\)/s)

\( \rho \) : Density

\( \omega \) : Rotational angular velocity

Subscript

\( f \) : Fluid

\( n \) : Time

\( p \) : Particle

\( R \) : Relative quantity between fluid and particle
2. Numerical Analysis Method

2.1 Cyclone Separator

Figure 1 shows the schematic view of the cyclone separator used in this simulation. This is the simplified model of cyclone corresponding to our previous study \(^{(3)}\). It has a swirl chamber with a radius \((R)\) of 72 mm and a height \((H)\) of 340 mm. The air flow enters through the top of the swirl chamber in the tangential direction and is exhausted through the outlet pipe with a radius \((r_0)\) of 43.2 mm \((0.6R)\), which is coaxially located below the swirl chamber. As shown in Fig.1, the outlet pipe is installed on the bottom of the swirl chamber \((\text{TYPE-A})\) or inside the swirl chamber by 85 mm \((0.25H)\) from the bottom of the swirl chamber \((\text{TYPE-B})\) in order to investigate the influence of the position of the outlet pipe for the swirl chamber on the swirling flows in the swirl chamber. The outlet pipe has a length of 680 mm \((2H)\) for \(\text{TYPE-A}\) and 760 mm \((2.25H)\) for \(\text{TYPE-B}\). The inlet mean velocity \((V_{in})\), which is the bulk velocity in the inlet duct, was about 7.8 m/s corresponding to our experimental study. The Reynolds number based on the radius of the swirl chamber and the inlet mean velocity was about 37,000. The cylindrical coordinate system as shown in Fig.1 is used and the origin is located at the center point on the bottom of the swirl chamber.

2.2 Large Eddy Simulation

The governing equations are the Navier-Stokes equation and the equation of continuity for incompressible viscous flow. They are transformed into cylindrical coordinates and the spatial filter operation is applied to them. The finite-difference method is used for the discretization. The variable \(q = r \times v_r\) is introduced in order to simplify the discretization at the singular point \(^{(7)}\). A fractional step method is used for the solution of the pressure. A Smagorinsky model \(^{(8)}\) with van Driest wall damping function is applied to subgrid scale (SGS) stresses. In this computation, the Smagorinsky constant \(C_s\) is set at 0.10. The computational code used in this study is based on DNS (Direct Numerical Simulation) \(^{(9)}\) added to the part of SGS stresses in the cylindrical coordinate. All spatial terms except for the convective terms are discretized with 2nd-order central difference scheme. The 3rd-order modified upstream-biased finite difference scheme is applied to the convective terms \(^{(10)}\). In the time integration, the 3rd-order Adams-Bashforth method is applied to the terms including the convection and the eddy viscosity. The 2nd-order Crank-Nicolson method is used for viscous terms.
A staggered grid system is used for variable array. A finer grid spacing in the $r$ and $z$ directions is generated as the walls are approached. The grid is a function of the hyperbolic tangent. The grid is uniform in the $\theta$ direction.

### 2.3 Particle Motion

The motion of a small rigid sphere is described by the following equations of translational motion and rotational motion \(^{11},^{12}\)

\[
\begin{align*}
\frac{m_p}{\rho_p} \frac{d u_{p}}{dt} &= \frac{1}{2} \rho_f \left| u_f \right| \left( C_D u_{Ri} + C_{LR} \frac{(u_g \times \omega_R)}{\|u_g\|} \right) \\
&+ 1.61 \left( \frac{u_f}{\rho_f} \right) \left( \frac{u_g \times \omega_f}{\|u_g\|} \right) + m_f \frac{D u_f}{dt} + \frac{1}{2} m_f \left( \frac{D u_f}{dt} - \frac{d u_{p}}{dt} \right) \\
&+ \left( m_p - m_f \right) g \delta_{ij} \\
I \frac{d \omega_p}{dt} &= - C_T \frac{1}{2} \rho_f \left( \frac{d_r}{2} \right)^{2} \left| u_f \right| \|\omega_f\| \\
&- C_T \frac{1}{2} \rho_f \left( \frac{d_r}{2} \right)^{2} \left| u_f \right| \|\omega_f\| \\
&+ \left( m_p - m_f \right) g \delta_{ij} \left( \frac{d_r}{2} \right)^{2} \left| u_f \right| \|\omega_f\|
\end{align*}
\]

The first term in the right-hand side of Eq. (1) denotes the drag force, the second term the lift force due to rotational motion, the third term Saffman lift force, the forth term the force due to fluid pressure gradient and viscous stresses, the fifth term the inertia force of added mass and the last term the buoyancy (or gravity) force. The empirical drag coefficient $C_D$ and lift coefficient $C_{LR}$ are given by the following equations, respectively.

\[
C_D = \frac{24}{Re_p} \left( 1 + 0.15 Re_p^{0.687} \right) \tag{3}
\]

\[
C_{LR} = \min \left[ 0.5, 0.25 \frac{d_r}{\|u_f\|} \right] \tag{4}
\]

where $Re_p$ is the particle Reynolds number. The right-hand side of Eq. (2) is the viscous torque against particle rotation \(^{13}\). $C_T$ is a non-dimensional coefficient which is a function of the rotational Reynolds number $Re_\theta$, given by
\[ C_f = \frac{C_1}{Re_R^{1/2}} + \frac{C_2}{Re_R} + C_3 \]  \hspace{1cm} (5)

where \( C_1, C_2 \) and \( C_3 \) are numerical factors as shown in Table 1.

The fluid velocity at a particle position is calculated by linear interpolation of the velocities at 8 grid points surrounding the particle and the velocity and rotational velocity of particles were integrated by the 3rd-order Adams-Bashforth method and the particle position by the 2nd-order Crank-Nicolson as follows.

\[ x_{i+1} = x_i + \frac{\Delta t}{2} (u_{pi} + u_{p,i}) \]  \hspace{1cm} (6)

<table>
<thead>
<tr>
<th>( Re_R )</th>
<th>( C_1 )</th>
<th>( C_2 )</th>
<th>( C_3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0 – 1</td>
<td>0</td>
<td>16 \pi</td>
<td>0</td>
</tr>
<tr>
<td>1 – 10</td>
<td>0</td>
<td>16 \pi</td>
<td>0.0418</td>
</tr>
<tr>
<td>10 – 20</td>
<td>5.32</td>
<td>37.2</td>
<td>0</td>
</tr>
<tr>
<td>20 – 50</td>
<td>6.44</td>
<td>37.2</td>
<td>0</td>
</tr>
<tr>
<td>50 – 100</td>
<td>6.45</td>
<td>32.1</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.4 Computational and boundary conditions

Figure 2 shows the computational grid for TYPE-B. The whole computational region consists of two blocks, Block-1 and 2, as shown in this figure. Block-1 is the computational region in the swirl chamber, and Block-2 one in the outlet pipe. Grid points in Block-1 for TYPE-A are 64x65x64 and those for TYPE-B are 64x65x84 in the \( r, \theta \) and \( z \) directions, respectively. 32x65x64 grid points for Block-2 are used. The grid resolution was determined by a grid screening and computational run time on . Non-dimensional time increment \( \Delta t \) for TYPE-A and B is set at 5\times10^{-4} and 4\times10^{-4}, respectively. After the fully developed flow fields are achieved, time history data of the 200,000 time steps is taken in order to calculate the time average flow fields. Neumann conditions are imposed on pressure at the solid walls. Non-slip conditions are used for velocities on the walls. Uniform flow is prescribed at the inlet part. Velocities on the outlet boundary are obtained by solving Orr-Sommerfeld equations. Pressure condition on the outlet boundary is given by the pressure equation obtained from substituting the Orr-Sommerfeld equation into Navier-Stokes equation in \( r \) direction \(^{(14)}\).

We would remark the attention of our simulation below. In case that the computation in the smaller radius of the outlet pipe, (\( r_0 \leq 40 \) mm), the velocity in the region of cyclone center increases and a complex flow field could be formed in that region. Since the anisotropy in grid system becomes larger in the center region, in which is inherent problem in cylindrical coordinate system, it was confirmed that the instability on computation occurs. In the simulation with the smaller radius of the outlet pipe, more stable time marching method (implicit scheme) and scheme of convective term could be required. Moreover, the fully length of the outlet pipe should be cordonned. These issues would be studied in the future.

Particles are lycopodium with a density of 700 kg/m³. The diameters \( d_p \) are set at 5 and 10 \( \mu \)m in this simulation. Initially, 100,000 particles are distributed randomly in the inlet part of the swirl chamber. The motions of fluid and particles are computed simultaneously with the one-way method. Inter-particle collisions are not considered because the solid volume fraction is very small (\( O(10^{-19}) \)). Impulsive equations are solved when particles collide with the walls \(^{(12)}\), and particle velocity and rotational velocity in the post collision
are computed. The particle properties used in this study are listed in Table 2. The coefficient of restitution and friction are used in the solution of impulsive equation at the particle-wall collision.

3. Computational Results and Discussion

3.1 Comparison of time-mean velocity field with experimental results

Figure 3 shows the time-mean velocity distribution in the $r$-$\theta$ plane at $z = 170$ mm for TYPE-A. The upper half of the figure includes contour map of the axial velocity, the bottom half are plots of velocity vectors. The positive value of axial velocity displays upward flow, and the negative one dose downward flow. The experimental results are from experiments with an outlet pipe radius of 36 mm. As mentioned in §2.4, though the computation could not be performed with the same diameter of outlet pipe as the experiment and it is difficult to be rigorously compared, the computational results are in qualitative agreement with the experimental ones. The axial velocity for the simulation shows the distinct regions with positive and negative value, which suggest that the swirling flow has a spiral vortex structure, just as the experimental results do. As seen in the velocity vector, the results for the simulation display the profile of a Rankine confound vortex corresponding to the experimental results although the region of a forced vortex for the simulation is slightly larger than that of the experiment due to the differences in size of outlet pipe. It is confirmed that the vortex core for the simulation and center line of the swirl chamber is not co-axially located and the extent is slightly larger than that for the experiment.

Figure 4 shows the time-mean velocity distribution in the $r$-$\theta$ plane at $z = 105$ mm for TYPE-B. The experiment results with the radius of the outlet pipe of 27 mm are used for comparison. The $r$-$\theta$ plane in Fig.3 is located at 20 mm above the top of the outlet pipe. As seen in these figures, it is found that the computational results are in good qualitative agreement with the experimental ones. Though the diameter of the outlet pipe for the simulation has a different one for the experiment and their differences in the location of vortex core and in the values of the axial and tangential velocity are evident, the results for the simulation capture the dominant characteristics of the swirling flows as shown in Fig.3.

The time-mean velocity distributions in the $r$-$\theta$ plane at $z = 20$ mm for TYPE-B are presented in Fig.5. In contour map, the axial velocity is positive near the outer wall of the outlet pipe and negative near the inner wall of the swirl chamber. This tendency can be seen in the experimental results. Examining the velocity vectors for the simulation, we can confirm that a rotating flow which has almost uniform tangential velocity in the $r$ direction is formed. Such a tendency probably suggests that the flows between the outer wall of the outlet pipe and the inner wall of the swirl chamber roll up along the outer wall of the outlet pipe with a uniform rotating. This can be figured out by the time-mean velocity distribution in the $r$-$z$ plane shown in Fig.6. From this figure, the rolling-up is clearly confirmed in both
the simulation and the experiment. It is also found that the flow pattern obtained from the computation is similar to that from the experiment though the axial velocity for the experiment is larger near the center of the swirl chamber than that for the computation due to their difference in the diameter of the outlet pipe.

From above discussion, it is considered that the LES predicts the fundamental characteristics of the swirling flow in the cyclone separator shown in the experimental results although rigorous comparison between the LES and the experiment cannot be made.
due to their difference in the diameter of the outlet pipe.

Figure 7 shows the 3-D time-mean flow field in the swirl chamber obtained from the LES. In this figure, red bold lines display positions of the vortex core and color of the streamlines corresponds to the magnitude of the velocity shown in the color bar. The inlet direction shown in the upper side of the swirl chamber is normal to the paper. From these figures, it is found that the flows in the swirl chamber form a three-dimensionally spiral vortex. The vortex core near the inlet part largely deviates from the center of the swirl chamber. The extent for TPYE-B is larger than that for TYPE-A.

Fig.5 Time-mean velocity distribution in the $r$-$\theta$ plane at $z = 20$ mm for TYPE-B, (a) experimental results ($r_o = 27$ mm), (b) computational results

Fig.6 Time-mean velocity distribution in the $r$-$z$ plane at $\theta = 150^\circ$ for TYPE-B, (a) experimental results ($r_o = 27$ mm), (b) computational results
3.2 Prediction of performance for particle separation

In this section, the prediction of the performance for the particle separation of the cyclone separator is performed from the results of the particle tracing. Figure 8 shows the time history of the collection rate of particles for TYPE-A and B. The collection rate of particles $\eta$ is calculated by the following equation.

$$\eta = \frac{N_{\text{out}}}{N_{\text{total}}} \times 100 \, \% \quad (7)$$

where $N_{\text{out}}$ is the number of particles which flow out from the outlet pipe and $N_{\text{total}}$ is the total number of particles mixed into the swirl chamber. By taking time history of $\eta$, it could be estimated how speed particles are collected. Moreover, the saturating value displays a collection rate which is frequently used as a performance parameter of cyclone separator. The longitudinal axis shows the collection rate and the lateral axis shows the non-dimensional time. As seen in this figure, $\eta$ increases exponentially with increasing time. Finally, the collection rates of about 90% and 80% are achieved respectively for particles with a diameter of 5 and 10 µm in case of TYPE-A. In case of TYPE-B, the collection rates of about 90% and 70% are achieved respectively. The increasing rate in $\eta$ for TYPE-A is larger than that for TYPE-B. Therefore, it is oblivious that the performance of particle separation for TYPE-A is higher than that for TYPE-B. This is probably because many...
particles for TYPE-B are not collected into the outlet pipe directly; particles fall down to the bottom of the swirl chamber, then rise by the rolling-up flow along the outer wall of the outlet pipe and finally enter into the outlet pipe. These behaviors can be observed in the comparison of the instantaneous particle distributions between TYPE-A and B shown in Fig.9 (a) and (b) respectively. These figures are snapshot at non-dimensional time $T = 20$ ($T = 0$ means the time in which particle are mixed into the chamber.). From Figure 9(b) and (c), it can be seen that particles with a diameter of 5 µm are distributed uniformly in the swirl chamber and particles with a diameter of 10 µm accumulate in the upper side of the swirl chamber. This is probably because the axial velocity of larger particles in the upstream decreases by the movement to the wall side due to the centrifugal force; the decrease in the particle axial velocity leads to the increase in particle arrival time to the outlet pipe located in the downstream. Therefore, it is considered that the collection rate of smaller particles is higher than that of larger particles.

The prediction method of performance for particle separation proposed in this study could accurately simulate the particle trajectories with the differences in the inserted height of the outlet pipe and particle diameter although the number of counting particles is much smaller than practical one. This would be very useful for establishment for the simple model for performance of particle separation and helpful for the design of cyclone separator with high-accurate particle separation.

4. Conclusion

The purpose of this study is to grasp the turbulent flow characteristics of complicated swirling flows in a cyclone separator and the relationships between the turbulent characteristics and particle motions. In this study, numerical simulations of the swirling flows in two types of the cyclone separator, which have different positions of the outlet pipe (TYPE-A and B), were performed using a large eddy simulation (LES). In addition, the performance for the particle separation was estimated by particles tracing in which individual particles are traced by a Lagragian method with a one-way method. It was shown that the LES accurately predicts the primary characteristics of the swirling flows in the cyclone separator shown in the experimental results, and the validity of our LES to the complicated swirling flow was confirmed. As results of the prediction of the performance of the particle separation, it was shown that the collection rate for TYPE-A is higher than that for TYPE-B. This is probably because many particles for TYPE-B fall down to the bottom
of the swirl chamber, then rise by the rolling-up flow along the outer wall of the outlet pipe and finally are collected into the outlet pipe. Moreover, the prediction method of performance for particle separation proposed in this study, which is based on the one-way method, could simulate the particle trajectories with the differences in the inserted height of the outlet pipe and particle diameter although the number of counting particles is much smaller than practical one. It is considered that this would be very useful for establishment for the simple model for performance of particle separation.

References


