Powder Technology 274 (2015) 135–145

Contents lists available at ScienceDirect

Powder Technology

journal homepage: www.elsevier.com/locate/powtec

Numerical simulation of an internal flow field in a uniflow cyclone separator

Jeongseog Oh *, Sangil Choi, Jeonggeun Kim

Advanced Combustion Laboratory, Korea Institute of Energy Research, Daejeon, Republic of Korea

ARTICLE INFO

Article history:
Received 20 June 2014
Received in revised form 21 November 2014
Accepted 5 January 2015
Available online 14 January 2015

Keywords:
Uniflow cyclone separator
Particle separation
RSM
LES
DPM

ABSTRACT

The characteristics of a particle separator were numerically investigated using the concept of a uniflow cyclone. The objective of the current study was to predict the internal flow field and to study the effect of flow streams on particle movement in a uniflow cyclone separator. The motion of solid particles in a flow field was simulated using the Eulerian–Lagrangian approach. Inlet temperature ($T_{in} = 300$–1100 K) and pressure ($P_{in} = 1$–9 bar) were varied for the initial conditions. Dust–gas mixtures (i.e., particle-laden flow) were injected into the separator inlet at $u_{in} = 3$–15 m/s. Calculation results showed that the Eulerian–Lagrangian approach was useful for modifying the two-phased viscous turbulence flow. The recirculation zone was predicted under a vortex finder, while a helical flow developed in the carrier gas outlet. Separation efficiency decreased with an increase in dust–gas temperature and pressure and increased with an increase in particle loading, inlet velocity, and particle diameter.

1. Introduction

Particle removal is an essential step in the coal gasification process because recirculated syngas in a gasifier includes unburned coal and ash particles. Particle concentration in a dust–gas mixture (i.e., particle-laden flow) is known to be related to syngas production yield and wall erosion. The process of particle separation in the dust–gas mixture is important for increasing overall gasifier performance and for cost-effective syngas production.

Several types of separators have been suggested for particle removal from a dust–gas mixture, including an aero-type cyclone separator, bag filter or membrane type separator, and electrostatic type separator. The aero-type separator uses aerodynamic and centrifugal forces to remove particles. It has the advantage of easier installation and cheaper maintenance cost compared to the bag filter or membrane type separator and the electrostatic type separator, even though it has the disadvantage of limited separation efficiency and range (i.e., a rapid decrease in separation efficiency for fine particles).

The aero-type separator is categorized as either a reverse-flow cyclone separator or a uniflow separator against the streamline direction of the dust–gas mixture inside a separator [1–3]. The reverse-flow cyclone separator (in general, Lapple cyclone) has the advantage of being a well-known and common device, while the uniflow separator has the advantage of higher performance. The uniflow cyclone separator (UCS) is suggested to maximize collection efficiency and minimize pressure loss [4]. Oh et al. reported an optimized design geometry of the UCS (i.e., the length of the gas outlet tube and a diverging channel) on the basis of a previous study [5] and noted the importance of the internal flow pattern to cyclone performance characteristics.

Numerical simulation can reduce design time and resource cost and so has been effectively used for studying and developing cyclone separators. Flow patterns inside the cyclone separator and separator performance (i.e., particle collection efficiency and pressure loss) are summarized in Table 1 [6–17].

High performance computing (HPC) technology was introduced to study separator performance and to design optimized configuration. Three-dimensional structured [8] and unstructured [7] meshes were used for separator modeling with the FLUENT program. Overall computational cost is known to be exponentially proportional to the turbulent Reynolds number [7].

Visualizing and analyzing the flow pattern inside a cyclone separator is useful because the flow pattern is related to the pressure drop and the gas–solid interaction affects particle collection efficiency [13]. The Reynolds average Navier–Stokes (RANS) model and a large eddy simulation (LES) model have been used to predict the time-averaged or time-dependent flow field in a cyclone separator [10,11,14,17]. In general, a Reynolds stress model is adequate to simulate an anisotropic viscous turbulent flow field inside a separator [8,9], while a renormalization group (RNG) $k-\varepsilon$ model shows under-predicted axial and over-predicted tangential velocity [18].

Computational fluid dynamics (CFD) can be used to investigate the geometric effects of a reverse-flow cyclone on separator performance.

---

* Corresponding author at: Advanced Combustion Laboratory, Korea Institute of Energy Research, Daejeon, Republic of Korea, 305-343. Tel.: +82 42 860 3479; fax: +82 42 860 3133.

E-mail address: jeongs5@kier.re.kr (J. Oh).

http://dx.doi.org/10.1016/j.powtec.2015.01.015
0032-5910/

Nomenclature

ASMM	algebraic slip mixture model
CFD	computational fluid dynamics
C_{DPM}	DPM concentration (kg/m^3)
DEM	discrete element model
DPM	discrete phase model
D_P	diameter of a coal particle (μm)
D_m	mean diameter of a coal particle (μm)
d_{in}	diameter of an inlet tube (mm)
d_{out}	diameter of an outlet tube (mm)
Eu	overall Euler number ($=\Delta P/[½\rho_A u_{in}^2]$)
g	acceleration of gravity (m/s^2)
HPC	high performance computing
LES	large eddy simulation
m_A	mass flow rate of air (g/s)
m_P	mass flow rate of coal particles (g/s)
PREST	pressure-staggered option
P_{stat}	static pressure (Pa)
P_{tot}	total pressure (Pa)
P_∞	pressure of ambient air (bar)
ΔP	pressure drop in a uniflow cyclone separator (Pa)
QUICK	quadratic upstream interpolation for convective kinetics
Re_i	Reynolds number of species i ($=u_i \times d_{in}/\nu_i$)
RANS	Reynolds average Navier–Stokes
RKE	realizable $k-\varepsilon$ model
RNG	renormalization $k-\varepsilon$ model
RSM	Reynolds stress model
r	radial distance (mm)
SKE	standard $k-\varepsilon$ model
SIMPLE	semi-implicit method pressure-linked equations
T_∞	temperature of surrounding air (°C)
UCS	uniflow cyclone separator
u_A	air velocity in a separator (m/s)
u_{in}	dust–gas velocity in separator inlet (m/s)
u'_i	fluctuating velocity to direction i (m/s)
u'_i/u_m	turbulent intensity to direction i (%)
u_m	mean velocity (m/s)
x	axial distance (mm)
μ_A	viscosity of air (kg/m/s)
ν_A	kinematic viscosity of air (m^2/s)
ρ_A	density of air (kg/m^3)
ρ_P	bulk density of a coal particle (kg/m^3)
τ_R	flow residence time from an inlet to a gas outlet (s)

Previous results have shown that geometric configurations, such as the cyclone cut-off diameter [19], cone-tip diameter [20], vortex finder shape [21], and cyclone inlet or outlet shape [22,23], influence the separation efficiency and pressure drop.

Particle collection efficiency increases with a decrease in cyclone height due to a reduction in tangential velocity [24], and the pressure drop decreases with an increase in the inlet section angle from 0° to 45° due to a reduction of the shortcut flow rate at a fixed inlet velocity [9,12].

Most previous work focuses on how to increase separator performance in the reverse-flow cyclone separator. To our knowledge, there is no numerical investigation of an unsteady flow field in the UCS. In the current study, the characteristics of a particle separator were investigated using the concept of a uniflow cyclone. Numerical simulation was carried out with the general-purpose CFD program ANSYS Fluent ver. 12.01 [25]. The objective of the current study is to predict the internal flow field and to study the effect of flow streams on particle movement in a UCS.

2. Numerical methods

2.1. Turbulent flow modeling

The three-dimensional flow field in a UCS was simulated using CFD. The conservation equations for mass and momentum in an incompressible Newtonian flow are as follows [25]:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{v}\right) = S \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho \vec{v}\right) + \nabla \cdot \left(\rho \vec{v}\,\vec{v}\right) = -\nabla p + \nabla \cdot \left(\overline{\overline{\tau}}\right) + \rho \vec{g} + \vec{F} \tag{2}$$

where ρ is the fluid density, v is the fluid velocity, p is the static pressure, τ is the stress tensor ($=\mu\left[\left(\nabla\vec{v}+\nabla\vec{v}^{T}\right)-\frac{2}{3}\nabla\cdot\vec{v}I\right]$), μ is the molecular viscosity, I is the unit tensor, ρg is the gravitational body force, and F is the external body force.

The Reynolds stress model (RSM) was used to modify the viscous turbulent flow in the UCS. The turbulence transport equation for RSM is as follows [25]:

$$\begin{aligned}\frac{\partial}{\partial t}\left(\rho\overline{u'_i u'_j}\right)+\frac{\partial}{\partial x_k}\left(\rho u_k\overline{u'_i u'_j}\right) = {} & -\frac{\partial}{\partial x_k}\left[\rho\overline{u'_i u'_j u'_k}+\overline{p\left(\delta_{kj}u'_i+\delta_{ik}u'_j\right)}\right] \\ & +\frac{\partial}{\partial x_k}\left[\mu\frac{\partial}{\partial x_k}\left(\overline{u'_i u'_j}\right)\right]-\rho\left(\overline{u'_i u'_j}\frac{\partial u_j}{\partial x_k}+\overline{u'_j u'_k}\frac{\partial u_i}{\partial x_k}\right) \\ & -\rho\beta\left(g_i\overline{u'_j\theta}+g_j\overline{u'_i\theta}\right)+\overline{p\left(\frac{\partial u'_i}{\partial x_j}+\frac{\partial u'_j}{\partial x_i}\right)u'_j} \\ & -2\mu\overline{\frac{\partial u'_i}{\partial x_k}\frac{\partial u'_j}{\partial x_k}}+S\end{aligned} \tag{3}$$

where t is time, ρ 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 eddy viscosity, and S is the source term.

The two terms on the left hand side of Eq. (3) indicate local time derivative and convection, respectively from left to right. The terms on the right hand side of Eq. (3) indicate turbulent diffusion, molecular diffusion, stress production, buoyancy production, pressure strain, and dissipation, respectively from left to right. The terms for turbulent diffusion, buoyancy production, pressure strain, and dissipation are needed for modeling [26].

2.2. Discrete phase modeling

The motion of solid particles in a flow field was simulated using the Eulerian–Lagrangian approach with a discrete phase method (DPM), i.e., the gas phase was treated as a continuum by solving Navier–Stokes equations and the solid phase was calculated by tracking particles in the flow field because the solid phase flow and gas-phase flow cannot be calculated simultaneously [18]. The volume fraction of the dispersed second phase (i.e., particle loading) did not exceed 10%, indicating that the volumetric flow rate of solid particles was sufficiently lower than that in the gas phase. It was assumed particle–particle interaction to be negligible and that particles did not affect the flow field (i.e., one-way coupling). A stochastic tracking method was used for modeling the turbulent dispersion of particles. The force balance equation for particle movement in a Lagrangian reference frame is as follows [25]:

$$\frac{du_p}{dt}=F_D\left(u_A-u_p\right)_x \tag{4}$$