Particuology xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Particuology

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

Numerical simulation of a dense solid particle flow inside a cyclone separator using the hybrid Euler–Lagrange approach

Paweł Kozołub[b], Adam Klimanek[a], Ryszard A. Białecki[a], Wojciech P. Adamczyk[a,*]

[a] *Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, Gliwice, Poland*
[b] *AMEC Foster Wheeler Energia Polska, Staszica 31, Sosnowiec, Poland*

ARTICLE INFO

Article history:
Received 20 April 2015
Received in revised form
12 September 2016
Accepted 22 September 2016
Available online xxx

Keywords:
Cyclone separator
Two-phase flow
Solid particle transport
Computational fluid dynamics
Hybrid Euler–Lagrange

ABSTRACT

This paper presents a numerical simulation of the flow inside a cyclone separator at high particle loads. The gas and gas–particle flows were analyzed using a commercial computational fluid dynamics code. The turbulence effects inside the separator were modeled using the Reynolds stress model. The two phase gas–solid particles flow was modeled using a hybrid Euler–Lagrange approach, which accounts for the four-way coupling between phases. The simulations were performed for three inlet velocities of the gaseous phase and several cyclone mass particle loadings. Moreover, the influences of several submodel parameters on the calculated results were investigated. The obtained results were compared against experimental data collected at the in-house experimental rig. The cyclone pressure drop evaluated numerically underpredicts the measured values. The possible reason of this discrepancies was disused.

Introduction

Cyclone separators are widely used in various industries to separate solid particles from air or process gases (Hoffmann & Stein, 2008). These devices utilize the centrifugal force created by the swirling motion of fluid as a separation mechanism. The popularity of the cyclone separator results from its simple design, low manufacturing and maintenance costs, relative high efficiency, and the ability to function in a wide range of temperatures and pressures. Cyclone separators found an important application in circulating fluidized bed (CFB) installations, where they are used for the separation of recirculating material from fluegases. Moreover, proper operating conditions for a cyclone separator have a great impact on the gas–solid flow conditions inside the CFB unit (Basu, 2006).

The pressure drop created by the cyclone and the particle collection efficiency are key parameters that characterize the cyclone separator performance. These parameters depend on the separator geometry and operating conditions (e.g., gas inlet velocity, particle mass loading). There are many semiempirical correlations used for the estimation of cyclone flow characteristics, which result from experimental work carried out on various cyclone installations (Altmeyer et al., 2004; Bohnet, Gottschalk, & Morweiser, 1997; Dirgo & Leith, 1985; Ray, Hoffmann, & Postma, 2000). These relations could be very useful for rapid cyclone design and optimization. Nevertheless, the flow pattern inside these devices is very complex, and many physical aspects are not considered in this methodology. Turbulence effects and detailed mutual gas–particle and particle interaction mechanisms are not considered (Cortes & Gil, 2007). Similarly, the influence of geometry on the operation of the separator can be factored in to such equations only to a limited extent.

Computational methods make it possible to simulate the flow behavior inside the cyclone separator in a more detailed manner. Computational fluid dynamics (CFD) allows investigators to calculate the flow parameters inside the cyclone separator using mathematical models that describe the fluid flow and provide a more thorough insight into the flow field characteristics. Detailed experimental velocity profiles obtained using laser Doppler anemometry (LDA) (Obermair, Woisetschläger, & Staudinger, 2003; Peng et al., 2002; Solero & Coghe, 2002) and particle image velocimetry (Liu, Zheng, Jia, Jiao, & Zhang, 2006; Liu, Zheng, Jia, & Zhang, 2007) indicate a strong anisotropy of flow structures inside the cyclone. Thus, an accurate CFD model describing the flow should be equipped with a proper turbulence model (Gronald & Derksen, 2011; Shalaby, Pachler, Wozniak, & Wozniak, 2005; Slack, Prasad, Bakker, & Boysan, 2000). Moreover, the presence of the particles adds to the complexity of the model. The different interactions between the solid and gas phases should be

---

* Corresponding author.
E-mail address: wojciech.adamczyk@polsl.pl (W.P. Adamczyk).

http://dx.doi.org/10.1016/j.partic.2016.09.003

Please cite this article in press as: Kozołub, P., et al. Numerical simulation of a dense solid particle flow inside a cyclone separator using the hybrid Euler–Lagrange approach. *Particuology* (2016), http://dx.doi.org/10.1016/j.partic.2016.09.003

Nomenclature

C_D	Drag coefficient
d_D	Particle diameter, m
$\bar{d}_D$	Average particle diameter, m
$\mathbf{g}$	Gravitational acceleration, m/s^2
$g_{0,ss}$	Radial distribution function
$\mathbf{I}$	Unit tensor
K_{fs}	Drag coefficient calculated for the dispersed phase, kg/(m^3 s)
$\dot{m}$	Mass flow rate, kg/s
p_s	Granular pressure, N/m^2
$p_{s,fric}$	Friction of granular pressure, N/m^2
p	Fluid pressure, N/m^2
S	Source term, N/m^3
t	Time, s
$\mathbf{u}_f$	Fluid phase velocity vector, m/s
$\mathbf{u}_p$	Particle velocity vector, m/s
$\mathbf{u}_s$	Average velocity of the solid phase, m/s
V_{cell}	Computational cell volume, m^3
V_p	Particle volume, m^3
$\mathbf{x}_p$	Particle coordinate vector, m
x_i	Coordinate component, m
α_f	Fluid phase volume fraction
α_s	Solid phase volume fraction
α_s^{fr}	Friction packing limit
γ_s	Dissipation of granular energy, kg/(m^{3}s)
Θ	Granular temperature, m^2/s^2
λ_s	Solid phase bulk viscosity, kg/(ms)
μ_s	Solid phase dynamic viscosity, kg/(ms)
ρ_f	Fluid phase density, kg/m^3
ρ_p	Particle density, kg/m^3
$\boldsymbol{\sigma}_s$	Granular stress tensor, N/m^2
$\boldsymbol{\tau}_f$	Stress tensor for viscous forces in fluid phase, N/m^2
ϕ	Angle of internal friction
φ_{fs}	Interphase exchange term

considered depending on the amount of solid particulate matter (Cortes & Gil, 2007; Elghobashi, 1994).

Some simulations consider the one-way coupling between the gas and particle phase in which the gas phase influences the particle movement (Derksen, 2003; Elsayed & Lacor, 2011b; Shalaby, Wozniak, & Wozniak, 2008; Shukla, Shukla, & Ghosh, 2011; Wang, Xu, Chu, & Yu, 2006). The particle presence is modeled mainly using the Euler–Lagrange (EL) approach, in which particles are tracked in a straightforward manner within the Lagrangian frame of reference. Nevertheless, the effects of the bidirectional coupling between the fluid and particle matter and the particle collisions are neglected in this approach, making the numerical model appropriate only for dilute solid gas mixtures.

More elaborate numerical studies of the two-phase flow are presented (Derksen, Sundaresan, & Van Den Akker, 2006; Derksen, Van den Akker, & Sundaresan, 2008; Wan, Sun, Xue, & Shi, 2008), where two-way interphase coupling is applied. This model introduces the influence of the moving particles on the gaseous phase flow field. The presence of the particles reduces the swirl inside the cyclone and attenuates the turbulence effects. However, this approach still does not account for the particle–particle collisions.

The application of the four-way coupling is presented in Chu, Wang, Xu, Chen, and Yu (2011) and Chu et al. (2009), where simulations using the discrete element method (DEM) are reported. The introduced model accounts for mutual gas–particle interactions and interparticle collisions, which are important for flows with high solid particle loading. Although the DEM model accounts for all the physical aspects present in the dense particle flow, the computing cost associated with the modeling of particle–particle collisions is very high. However, some modification of the DEM model implementation may reduce the computational cost of running simulations. The four-way coupling is also considered in the simulations using the Euler–Euler approach (Kharoua, Khezzar, & Nemouchi, 2011; Qian, Huang, Chen, & Zhang, 2007), in which the solid particles are treated as a continuum. However, each particle size needs to be represented as an individual phase in this methodology, making the numerical simulations extremely expensive and causing convergence problems.

A majority of the numerical models of the cyclone presented in literature are valid only for low and moderate solid mass loadings, and only few are applicable for very dense particulate flows, e.g. cyclones in CFB installations (Basu, 2006; Dewil, Baeyens, & Caerts, 2008). Therefore, an appropriate mathematical model should be applied for simulations of such cases. Under these conditions, mutual gas and particle and interparticle interactions should be considered in order to represent the real physics of the flow. This paper presents the application of a hybrid EL approach in the CFD modeling of a dense solid particle flow inside the cyclone separator. This technique incorporates the four-way coupling to take into account all the interactions between the continuum and dispersed phases, which is essential for predicting high solid particle content. The numerical model used in the simulations was validated using measured data, such as the pressure drop within cyclone.

Experimental

In this work, an experimental rig was built in order to validate the numerical model. The pressure drop in the cyclone separator was measured for different inlet gas velocities and particle loadings. The experimental results thus obtained were used to assess the accuracy of the currently used numerical model and disclose its shortcomings.

The experimental rig consisted of three main devices: cyclone, fan, and particle feeder (Fig. 1). The measuring equipment installed in the rig consisted of a flow meter, thermocouple, temperature transducer, set of measuring ports, valve terminal, pressure transducer, pressure converter, and personal computer for data acquisitions.

The cyclone used in the experiments was built from polymethyl methacrylate. The geometry of the separator was replicated from the Stairmand high efficiency cyclone design (Cortes & Gil, 2007; Hoffmann & Stein, 2008). The cyclone walls were transparent in order to observe the particle flow patterns for which the future vision system would be applied. The cyclone dimensions and its geometric features are presented in Fig. 2.

Air at ambient pressure and temperature was supplied to the cyclone by the fan. The flow load of the fan was maintained by an inverter, which controlled the revolutions of the electric motor of the fan. The airflow rate was measured using a flow meter equipped with pressure and temperature sensors during the experiment in order to calculate the air parameters in future. Eight ports were installed on the walls of the inlet duct and vortex finder (four on each element) of the rig for static pressure measurements, as shown in Fig. 2. The ports were connected to the pressure transducer through the valve terminal. The pressure was measured at specified time intervals for a given port. The results were calculated as the average value for a given time range set to 20 s. The pressure transducer converted the pressure signal into direct current, which was recorded by an in-house application created in the LabVIEW environment (National Instruments, 2012).

The cyclone pressure-drop was measured for two operating modes: pure gas and gas laden by solid particles. The particles were

Please cite this article in press as: Kozołub, P., et al. Numerical simulation of a dense solid particle flow inside a cyclone separator using the hybrid Euler–Lagrange approach. *Particuology* (2016), http://dx.doi.org/10.1016/j.partic.2016.09.003