Powder Technology 249 (2013) 339–352

Contents lists available at ScienceDirect

Powder Technology

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

CFD simulation of hydrodynamics on the dense zone on a 65 t/h oil shale-fired high–low bed CFB boiler

Qing Wang *, Jianbo Xiao, Hongpeng Liu

Engineering Research Centre of Oil Shale Comprehensive Utilization, Ministry of Education, Northeast Dianli University, Jilin 132012, Jilin Province, China

ARTICLE INFO

Article history:
Received 4 October 2012
Received in revised form 25 July 2013
Accepted 29 July 2013
Available online 14 August 2013

Keywords:
Oil shale
High–low bed CFB
Eulerian–Eulerian model
Kinetic theory of granular flow

ABSTRACT

Gas–solid flow behavior in a 65 t/h oil shale-fired high–low bed CFB boiler obtained by the revamping of a 75 t/h pulverized coal-fired boiler has been simulated using a Eulerian–Eulerian model (EEM) with kinetic theory of granular flow by the commercial CFD software package, Fluent. Two-dimensional (2D) transient and three-dimensional (3D) steady flows were simulated for the gas and the solid phase, respectively. The comparative study with regard to turbulence and drag model was performed by 2D simulation. The simulated results agreed reasonably with the experimental data and showed that Swirl-modified RNG k-ε-Per phase model and Gidaspow drag model could predict preferably the internal circulation process. Gas–solid flow profiles were obtained by 3D steady simulation for solid velocity, pressure, solid volume fraction, and granular temperature and the internal circulation characteristics of the boiler were further understood in detail. The results showed that the pressure difference between the main and side bed and the distributions of solid velocity and volume fraction illustrated the mechanism of internal circulation process. The fluidized velocity in the side beds is lower and wear of immersed tubes is also lower. The granular temperature is higher near the immersed tube bundle. This research established the foundation for the design and large-scale of high–low bed CFB.

1. Introduction

Oil shale is considered as a kind of vital supplementary energy for a conventional source of energy in the world [1]. Oil shale is an organic-rich fine-grained sedimentary rock containing significant amounts of kerogen from which shale oil and combustible gas can be extracted. It can also be burnt directly in furnaces as a low-grade fuel for power generation and district heating [2]. In the oil shale combustion, Estonia, China and Israel have accumulated a large number of experiences to generate electricity and heat.

In 2005, a 75 t/h pulverized coal-fired boiler was successfully revamped to a 65 t/h oil shale-fired CFB boiler employed by high–low bed CFB combustion technology [3]. It was installed in Suixi Cement Factory, Guangdong province, China, and put into operation in 2006. The retrofit project was provided by Northeast Dianli University and the high–low bed CFB boiler technology introduced from Germany by Jiangxi Jianglian Energy & Environment Co., Ltd. was used. High–low bed CFB belongs to the category of internally circulating fluidized bed and has been widely used in energy and chemical industry because of several advantages over conventional CFBs, such as low immersed tube erosion rate and long retention time of fuel, recently [4].

In recent years, the researches on hydrodynamics of internally circulating fluidized bed were carried primarily through experiment and CFD modeling. The effects of distributor type, draft tube height, gap height [5,6], annulus height [7], gas velocity [8] and annulus temperature [9] on the circulation rate of solids [10], gas bypassing, volume fraction and velocity distribution have been investigated by many researchers through the experimental method. Moreover, Abellon et al. [11] determined the retention time and the circulation rate of particles in the internally circulating fluidized bed through the experimental method. However, detailed consideration of gas–solid flow is extremely difficult by means of tests for the industrial scale high–low bed CFB. Nowadays, with the enhancement of computer performance, CFD simulation have enjoyed great development and played increasing key role in the design, optimized operation and large-scale for the complex gas–solid flow as well as high–low bed CFB [12]. Accordingly, Yu et al. [13] studied the gas–solid flow in an internally circulating fluidized bed via numerical simulation with EEM model. The simulations were conducted to assess the effect of changes to four designs or operating parameters: gas distributor plate angles, presence of a heat exchange tube bundle, superficial fluidizing velocities and initial solid packing heights. Tian et al. [14] researched the hydrodynamics of the gas–solid flow in a fluidized bed with uneven gas supply using the discrete element method (DEM). Xiong et al. [15] solved the two-fluid model based on a smoothed particle hydrodynamics (SPH) method. Bokkers et al. [16] researched the large-scale dense gas–solid bubbling fluidized beds using discrete bubble model (DBM). The direct numerical simulation (DNS) was also used in CFB simulations [17]. Besides, according to the special structure of the differential-velocity bed, a simulation calculation of the coal combustion and the physical and chemical process was carried out with the cell model by Zhao et al. [18].

* Corresponding author. Tel.: +86 13843225761; fax: +86 432 64806281.
E-mail address: rlx888@126.com (Q. Wang).

0032-5910/$ – see front matter
http://dx.doi.org/10.1016/j.powtec.2013.07.037

This paper presents two-dimensional transient and three-dimensional steady CFD simulation of gas–solid flow behavior in the dense zone of the 65 t/h oil shale-fired high–low bed CFB boiler mentioned above with Eulerian–Eulerian model (EEM) incorporating the kinetic theory of granular flow. The comparative study with regard to turbulence and drag model was performed by 2D simulation and the detailed gas–solid flow simulation is given by 3D simulation.

2. Modeling

The simulation is performed using the commercial CFD software package, Fluent. An overview on the governing equations and closure models are given.

2.1. Governing equations

The conservation equation of mass is as follows:

$$\frac{\partial}{\partial t}\left(\alpha_g\rho_g\right)+\nabla\cdot\left(\alpha_g\rho_g v_g\right)=0 \quad (1)$$

$$\frac{\partial}{\partial t}(\alpha_s\rho_s)+\nabla\cdot(\alpha_s\rho_s v_s)=0. \quad (2)$$

In these equations parameters α_i, ρ_i and ν_i represent the volume fraction, density and velocity of phase i (i = gas, solid), respectively.

The conservation equation of momentum considers as follows:

$$\frac{\partial}{\partial t}\left(\alpha_g\rho_g v_g\right)+\nabla\cdot\left(\alpha_g\rho_g v_g v_g\right)=-\alpha_g\nabla p+\nabla\cdot\tau_g+\alpha_g\rho_g g+\beta\left(v_s-v_g\right) \quad (3)$$

$$\frac{\partial}{\partial t}(\alpha_s\rho_s v_s)+\nabla\cdot(\alpha_s\rho_s v_s v_s)=-\alpha_s\nabla p-\nabla p_s\nabla\cdot\tau_s+\alpha_s\rho_s g+\beta\left(v_g-v_s\right). \quad (4)$$

In these equations, β represents the interphase momentum exchange coefficient. Furthermore, the term τ_i stands for stress tensors of phase i. These terms are defined as follows:

$$\tau_g=\alpha_g\mu_g\left(\nabla v_g+\nabla v_g^T\right)+\alpha_g\left(\lambda_g-\frac{2}{2}\mu_g\right)\nabla\cdot v_g^I \quad (5)$$

$$\tau_s=\alpha_s\mu_s\left(\nabla v_s+\nabla v_s^T\right)+\alpha_s\left(\lambda_s-\frac{2}{2}\mu_s\right)\nabla\cdot v_s I. \quad (6)$$

2.2. Turbulence modeling

Turbulence model theory is established in the scheme of turbulence research due to the requirement of flow equations further closures, and also the two equation models based on the eddy viscosity model were adopted in this paper. At present, the two equation models, especially the Standard k-ε model proposed by Launder and Spalding [19], are used extensively in the engineering fields. However, the accuracy of computational results is unconvincing under the condition of bending streamline flow with this model. Accordingly, the developed RNG k-ε [20] and Realizable k-ε model are able to solve preferably the problem with regard to high strain rate and bending degree, and yet the turbulence viscosity calculation formula can be changed and the ε equation can be improved in the Realizable k-ε model. Furthermore, the mixture, dispersion and per phase approach have been applied for the description of solid phase turbulence.

2.3. Kinetic theory of granular flow

The kinetic theory of granular flow considering the dynamics of interparticle collisions and determining the turbulent kinetic energy of the particles is developed by Savage and Jeffrey [21], Jenkins and Savage [22], Lun et al. [23], P.C. Johnson and R. Jackson [24], which allows the determination of the pressure and viscosity of the solid in place of empirical relations, and used in the numerical analysis of fluidized bed by Gidaspow [25].

The equation for granular temperature may be written as:

$$\frac{3}{2}\left[\frac{\partial}{\partial t}(\alpha_s\rho_s\Theta_s)+\boldsymbol{\nabla}(\alpha_s\rho_s\nu_s\Theta_s)\right]=(-p_s I+\tau_s):\boldsymbol{\nabla}\nu_s+\boldsymbol{\nabla}\cdot(k_{\Theta s}\boldsymbol{\nabla}\Theta_s)-\gamma_{\Theta s}-\mathbf{3}\beta\Theta_s \quad (7)$$

where the Θ_s represents the granular temperature, and the term is defined as follows:

$$\Theta_s=\frac{1}{3}(\nu_s')^2 \quad (8)$$

ν'_s represents the ensemble averaged magnitude of the randomly fluctuating velocity of the solid particles.

The solid pressure p_s is calculated according to Syamlal and Lun et al. [23,26], which is composed of two parts: suspension mechanism that dominates in the dilute flow regions and a collision mechanism that is key in the dense flow regions.

$$p_s=\alpha_s\rho_s\Theta_s+2\rho_s(1+e_{ss})\alpha_s^2 g_0\Theta_s \quad (9)$$

where e_{ss} is the particle–particle restitution coefficient and 0.9–0.99 are used in studies found in the literatures [27–29]. In this work, 0.95 are tried. g_0 is the radial distribution function that the following expression was used by Lun et al. [23]:

$$g_0=\left[1-\left(\frac{\alpha_s}{\alpha_{s,\max}}\right)^{1/3}\right]^{-1}. \quad (10)$$

The solid shear viscosity is composed of three parts:

$$\mu_s=\mu_{s,col}+\mu_s,kin+\mu_s,fr \quad (11)$$

Collisional viscosity:

$$\mu_{s,col}=\frac{4}{5}\alpha_s\rho_s d_s g_0(1+e_{ss})\left(\frac{\Theta_s}{\pi}\right)^{1/2} \quad (12)$$

Kinetic viscosity [26]:

$$\mu_{s,col}=\frac{\alpha_s\rho_s d_s\sqrt{\Theta_s\pi}}{6(3-e_{ss})}\left[1+\frac{2}{5}g_0\alpha_s(1+e_{ss})(3e_{ss}-1)\right] \quad (13)$$

Friction viscosity [30]:

$$\mu_s,fr=\frac{p_s\sin\phi}{2\sqrt{I_{2D}}} \quad (14)$$

where ϕ is the angle of internal friction and I_{2D} is the second invariant of the deviatoric stress tensor.

The solid bulk viscosity accounts for the resistance of the granular particles to compression and expansion, and is given by Lun et al. [23]:

$$\lambda_s=\frac{4}{3}\alpha_s\rho_s d_s g_0(1+e_{ss})\left(\frac{\Theta_s}{\pi}\right)^{1/2}. \quad (15)$$

The collisional dissipation of energy fluctuation that attributes to the inelastic collision between particles is [23]:

$$\gamma_{\Theta s}=\frac{12\left(1-e_{ss}^2\right)g_0}{d_s\sqrt{\pi}}\rho_s\alpha_s^2\Theta_s^{3/2}. \quad (16)$$