Numerical Simulation Research on Gas-Solid Two Phase Flow in Oil Shale Circulating Fluidized Bed

Qing Wang¹, Jun-tao Feng¹, Bai-zhong Sun¹, Yong-qing Qi², Deng-feng Chen², Jin-long Luo¹

¹. Northeast Dianli University, Jilin, Jilin Province, China
². Daqing Oilfield Limited Company, Daqing City, China

Abstract

Oil shale circulating fluidized bed combustion technology is a new technology and it is the most economical and efficient combustion way for using Oil shale resources. Numerical simulation of CFB is very important in the prediction of its flow behavior. In this paper, a gas-particle computational fluid dynamics (CFD) flow model was developed based on the EULERIAN two-fluid model. In the modeling, a similar constitutive equation was applied to both phases. The turbulence was simulated with $k – \varepsilon$ turbulence model in the gas phase and the Gidaspow drag coefficient model was applied to coupling force between the gas-solid two phases. The computational model was used to simulate the flow behavior in an oil shale circulating fluidized bed under different operating conditions. In addition, several numerical experiments were carried out to understand the influence of particle size, gas superficial velocity and solids volume fraction on the flow characteristics.

© 2012 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Hainan University.

Open access under CC BY-NC-ND license.

Keywords: oil shale; CFB; gas-solid two phase flow; numerical simulation

1. Introduction

Oil shale is a kind of potential and tremendous energy. Among the fossil fuels, its reserve in terms of the calorific value is only next to coal and holds the second place[1]. The combustion in a circulating fluidized bed is the best choice for the industrial application of oil shale. Although the use of CFB equipment has opened wide possibilities for improving industrial technologies for oil shale combustion, detailed understanding of its complex gas-particle hydrodynamic behavior is still lacking. While simulation research can provide certain understanding not easily obtained through experiment in the lab. Investigations about the CFB modeling that consider the hydrodynamic behavior have been carried out by
many researchers. K. Smolders and J. Baeyens[2] introduced a hydrodynamic model of a low-density CFB. The simulation was performed mainly to predict the axial variation of solids holdup, but not the radial one. M.R. Hyre and L.R. Glicksman[3] simulated the upper region of a CFB and predicted the axial and the radial variation of solids holdup by assuming two or more phases. Recently, Van Wachem[4] have given a comparative analysis of CFD models based on gas and solid phase continuity equations, momentum balances and appropriate constitutive equations. These CFD models are the most general and can deal with different geometries.

Although published models have a similar structure, some significant differences are found in the sub-models. Generally the axial and radial distribution of voidage, velocity, pressure and particle size are not considered together in these models. By using the similar CFD models, a dynamic two dimensional model was developed considering the hydrodynamic behavior of CFB in this paper. A numerical study was performed using this model to understand the hydrodynamics of gas–solid flow in the riser of oil shale CFB. The simulations were done using the geometrical configuration of a CFB test rig at the Northeast Dianli University[5]. The furnace is higher than the conventional boiler to ensure sufficient residence time for oil shale, which would increase the burnout level of oil shale and improve boiler efficiency. The simulation experiments were done by using FLUENT and the models took into account the axial and radial distribution of velocity for solid phase, solids volume fraction and pressure drop along the axial direction.

2. CFD model

Both gas and solid phase are applied as the continuous phase in the EULERIAN two-fluid model, and two phases can be described by similar continuity equations and momentum equations. The EULERIAN model also considers two phases co-occupying the same space and fully interpenetrating. Therefore, the flow in bed can be understood as the motion and interaction of two kinds of fluid.

Continuity equations:

\[ \varepsilon_g + \varepsilon_s = 1 \]  
(1)

\[ \frac{\partial}{\partial t} \left( \varepsilon_g \rho_g \right) + \nabla \cdot \left( \varepsilon_g \rho_g v_g \right) = 0 \]  
(2)

\[ \frac{\partial}{\partial t} \left( \varepsilon_s \rho_s \right) + \nabla \cdot \left( \varepsilon_s \rho_s v_s \right) = 0 \]  
(3)

Momentum equations:

\[ \frac{\partial}{\partial t} \left( \varepsilon_g \rho_g v_g \right) + \nabla \cdot \left( \varepsilon_g \rho_g v_g v_g \right) = -\varepsilon_g \nabla p + \nabla \cdot \tau_g + \varepsilon_g \rho_g g + K_{gs} \left( v_g - v_g \right) \]  
(4)

\[ \frac{\partial}{\partial t} \left( \varepsilon_s \rho_s v_s \right) + \nabla \cdot \left( \varepsilon_s \rho_s v_s v_s \right) = -\varepsilon_s \nabla p - \nabla \rho_s + \nabla \cdot \tau_s + \varepsilon_s \rho_s g + K_{gs} \left( v_g - v_s \right) \]  
(5)

Stress tensors:
Syamlal[6], Wen-Hu[7] and Gidaspow[8] have done a lot of research on $g_s$, and got several empirical correlations called the drag force model. Gidaspow drag coefficient model combines the results of previous studies and makes some partial correction. The Gidaspow drag model has been the most appropriate model for the simulation of fluidized bed.

Momentum exchange coefficient:

$$K_{gs} = \frac{3}{4} C_d \frac{\varepsilon_g \rho_s \mu_s}{d_s} \left( \frac{\varepsilon_g - \varepsilon_0}{\varepsilon_g} \right)^{2.65} \left( \varepsilon_g \geq 0.8 \right)$$

$$K_{gs} = 150 \left( \frac{1 - \varepsilon_g}{\varepsilon_0} \right) \mu_g + 1.75 \left( \frac{\varepsilon_g \rho_s \mu_s}{d_s} \right)^{2.65} \left( \varepsilon_g \leq 0.8 \right)$$

where:

$$C_d = \frac{24}{\text{Re}} \left( 1 + 0.15 \text{Re}^{0.687} \right) \quad \left( \text{Re < 1000} \right)$$

$$C_d = 0.44 \quad \left( \text{Re > 1000} \right)$$

The conservation equations employed contain some unknown variables. Therefore, the EULERIAN model requires some additional closure laws to describe the hydrodynamic of particles.

Radial distribution function:

$$g_0 = \left[ 1 - \left( \frac{\varepsilon_s}{\varepsilon_{s,\text{max}}} \right)^{1/3} \right]^{-1}$$

Particle phase pressure:

$$p_s = \varepsilon_s \rho_s \left[ 1 + 2(1+e)\varepsilon_s g_0 \right] \Theta$$

Particle phase viscosity:

$$\bar{\eta}_s = \frac{4}{3} \varepsilon_s^2 \rho_s d_p g_0 (1+e) \sqrt{\Theta \pi}$$

Particle phase shear viscosity:

$$\mu_s = \frac{2 \mu_{s,\text{dil}}}{(1+e)g_0} \left[ 1 + \frac{4}{5} (1+e) g_0 \varepsilon_s \right]^2$$

$$+ \frac{4}{5} \varepsilon_s^2 \rho_p g_0 (1+e) \sqrt{\Theta / \pi}$$
Where: \( \mu_{s,dil} = \frac{5}{96} \rho_s d_p \sqrt{\pi} \)

Gas and solid phase bulk viscosity:

\[
\lambda_s = 0, \lambda_s = \frac{4}{3} \varepsilon_s \rho_s d_s g_{0,ss} (1 + e) \sqrt{\frac{\Theta}{\pi}}
\]

(14)

Granular temperature:

\[
\frac{3}{2} \left[ \frac{\partial}{\partial t} (\varepsilon_s \Theta) + \frac{\partial}{\partial x_i} (\varepsilon_s u_i \Theta) \right] = \Pi \frac{\partial u_{j,s}}{\partial x_i} + \frac{\partial}{\partial x_i} \left( \Gamma \frac{\partial \Theta}{\partial x_i} \right) - \gamma_s - 3 \beta \Theta
\]

(15)

Where:

\[
\Pi = -p \delta_y + \varepsilon_s \frac{\partial u_{k,s}}{\partial x_k} + \mu_s \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right)_s
\]

(16)

\[
\Gamma = \frac{2 \Gamma_{\Theta, dil}}{(1 + e) g_0} \left[ 1 + \frac{6}{5} (1 + e) g_0 \varepsilon_s \right]^2
\]

\[
+ 2 \varepsilon_s^2 \rho_s d_p g_0 (1 + e) \sqrt{\frac{\Theta}{\pi}}
\]

(17)

with:

\[
\Gamma_{\Theta, dil} = \frac{75}{384} \rho_s d_s \sqrt{\Theta \pi}
\]

(18)

3. Simulation results and discussion

The CFB test rig for oil shale combustion being set up in Northeast Dianli University mainly consists of two zones: dense and dilute phase region[9]. The entity structure of the CFB test rig is shown in Figure 1. The dense region diameter is 80mm and the dilute region diameter is 99mm. The secondary air inlet is located at 1.1m and the furnace height is 8.9m from distributor. The software GAMBIT is used to mesh the working region and the map type mesh is used for most of the system except for the dense region, for which submap type mesh is used in order to describe the gas-solid flow structure with sufficient accuracy. The simulation work is done by using Fluent 6.3. For the gas, a no-slip boundary is used at the wall. For the solid, the partial slip model is selected with a coefficient of 0.6. The phase coupled SIMPLE method is chosen for pressure-velocity coupling. The other operating parameters used for the simulation are being shown in Table1.
Figure 1. The entity structure of CFB

Table 1. Parameters used in simulation work

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle diameter (mm)</td>
<td>0.3, 0.5, 0.8, 1.2</td>
</tr>
<tr>
<td>Particle density (kg/m³)</td>
<td>2100</td>
</tr>
<tr>
<td>Shape factor</td>
<td>0.6</td>
</tr>
<tr>
<td>Gas velocity (m/s)</td>
<td>4, 5, 6, 7</td>
</tr>
<tr>
<td>Solid void fraction ($\varepsilon_{max}$)</td>
<td>0.65</td>
</tr>
<tr>
<td>Initial bed height (mm)</td>
<td>400</td>
</tr>
<tr>
<td>Circulation rate (kg/h)</td>
<td>40</td>
</tr>
<tr>
<td>Time step (s)</td>
<td>0.001</td>
</tr>
</tbody>
</table>

3.1 Fluidization process

Figure 2. The solid fluidization process
The solid fluidization process is shown in Fig. 2. As shown in the Figure, at the beginning the gas phase move upward along the side wall and after a certain height the gas phase distributes evenly throughout the bed. The solids phase move upward axially as the plug flow regime at 0.2 seconds. The particles start to form cluster phase at 0.4 seconds. The particles move from core region to the annulus region radially at 0.6 seconds. The phases like bubble phase start to appear and they mainly occur in the central region at 0.8 seconds. The amount of the bubble phase increases and the bubble phase drive solid phase upward rapidly at 1.0 seconds.

3.2 Radial profiles of particle velocity

The radial profiles of particle velocity at different riser heights are shown in Fig. 3. The simulated results show that particles move upward at a higher velocity in the central region and move downward at a slower velocity in the side wall region. This flow character causes internal circulation of particles and extends the residence time of solid particles in the furnace which is extremely beneficial for oil shale combustion. The flow at four different bed heights is consistent with the core-annulus flow style described in the literature. In addition, the figure shows that the particle velocity increases axially, and decreases after a certain height.

Fig. 4 presents the effect of different mean particle diameters on radial profiles of particle velocity at 3m. As shown in figure, the particle velocity profiles become flatter and the range of annulus area relatively reduces as the mean particle diameter increases. This is due to the larger mean particle diameter and it causes an increase in the distribution of voidage that decrease the flow velocity and particle distribution turns out to be more uniform. This feature further confirms the superiority of the wide range of particle diameters in CFB.

Fig. 5 shows the effect of superficial gas velocity on radial profiles of solid velocity. The simulated results show that at the same axial height, with the gas velocity increases, the particle velocity also increases and the range of the annulus region becomes narrow. Because the solid particles are entrained up by the upflowing gas flow and a high gas velocity causes an increase in the turbulent

![Figure 3. The radial profiles of particle velocity at different riser heights.](image-url)
dissipation effect that would prevent the formation of particle clusters to a certain extent.

3.3 Radial profiles of solids volume fraction

Fig. 6 shows the radial profiles of the solid volume fraction at different riser heights. Overall, the particle volume fraction is smaller in the core region and increases gradually along the radial direction. This feature can indirectly prove the existence of core-annulus flow style. In addition, the solid volume fraction increases axially, and decreases at a certain height. By the discussion of Fig. 3 and Fig. 6, the dilute phase region should be analyzed in two regions: In the bottom zone, particle velocity and solids volume fraction are gradually increasing; In the upper zone, particle velocity and solids volume fraction relatively decrease.
Fig. 6. The radial profiles of solid volume fraction at different riser heights.

Fig. 7. The effect of superficial gas velocity on radial profiles of solid volume fraction.

Fig. 7 shows the effect of superficial gas velocity on the radial profiles of the solids volume fraction. The simulated results show that solids volume fraction decreases as gas velocity increases and the range of the annulus region is narrow. Because more solids move out of the furnace with the gas velocity increasing that causes the incidence of solid cluster formations becomes smaller. This law shows that a higher gas velocity can effectively prevent the formation of particle clusters.

3.4 Axial profiles of pressure

Fig. 8 shows the effect of gas velocity on the axial profiles of pressure. It is clearly seen in Fig. 8 that the predicted values fit very closely the experimental data. Both simulated results and experimental data show that the total pressure decreases as the gas velocity increasing and the absolute values of pressure gradient decrease monotonically with increasing distance from the riser entrance, because more solids move from bottom to the top while the amount of solids in the CFB remains unchanged and the interface of dense and the dilute phase region will move upward.
4. Conclusions

1) EULERIAN two-fluid model can simulate the particle movement rule in an oil shale circulating fluidized bed. The simulation results have a good agreement with experiment. The simulated results show that gas-solid flow is still consistent with core-annulus flow style.

2) According to the axial solid volume concentration profile, the dilute region can be analyzed in two regions: In the bottom zone, particle velocity and solids volume fraction gradually increase and fluctuations appear evident; In the upper zone, particle velocity and solids volume fraction relatively decrease and distribute more evenly. This law shows that the particle cluster formation has a significant effect on the axial in homogeneity in a furnace.

3) High gas velocity can effectively reduce the incidence of the particle cluster formation, which means the performance of the bed is better at higher gas velocity. Therefore the air distributor or other measures should be taken to increase gas velocity to ensure better heat and mass transfer characteristics in the furnace.

Acknowledgment

The authors are grateful for financial support from the Major Special Projects of National Science Ministry. (2008ZX05018-004) and supported by Doctoral Program Foundation of Northeast Dianli University (BSJJXM-200901).

References


