Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
DOI: 10.29252/jafm.13.02.30534

Cold Flow Simulation of a 30 kWth CFB Riser with CPFD

D. Gündüz Raheem1,2†, B. Yılmaz1 and S. Özdoğan1

1 Marmara University, Institute of Pure and Applied Sciences, Kadikoy, Istanbul, 34722, Turkey
2 Istanbul Medeniyet University, Faculty of Engineering and Natural Sciences, Dept. of Mechanical Engineering, North Campus, 34700, Uskudar, Istanbul, Turkey

†Corresponding Author Email: duygu.gunduz@medeniyet.edu.tr

(Received May 17, 2019; accepted August 11, 2019)

ABSTRACT

A 30 kWth Circulating fluidized bed (CFB) combustor is experimentally and numerically investigated under cold flow conditions. Barracuda software based on Computational Particle Fluid Dynamics (CPFD) method is utilized for simulations. The influences of bed inventory and drag model on flow hydrodynamics were investigated considering pressure and velocity profiles and particle concentration. Two advanced drag models, namely Energy minimization multi-scale (EMMS) and Wen-Yu/Ergun were selected for this study. The simulations were performed with initial bed material masses of 3.79, 4.55 and 5.20 kg corresponding to 2.5, 3 and 3.5 diameters height of riser, respectively. With increasing bed inventory pressure drops and solid concentration increase. The axial particle velocities slightly change with bed inventory. The comparison of simulation results with experimental measurements was resulted in good agreement (<5%) with both models. The simulation with EMMS drag model predicted the pressure profiles more accurately than Wen-Yu/Ergun drag model. The profiles of particle volume fraction and axial velocity demonstrate that core-annulus flow pattern was captured by both models. But EMMS drag model was better in revealing the meso-scale structures at instantaneous particle concentration distribution. Moreover, the influence of particle size distribution on particle volume fraction and particle velocity profiles is also investigated with two drag models.

Keywords: CFB; Experimental; Cold flow; CPFD; Drag model.

NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_d$</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>$d$</td>
<td>riser inner diameter</td>
</tr>
<tr>
<td>$D_p$</td>
<td>drag function</td>
</tr>
<tr>
<td>$f_e$</td>
<td>coefficient for EMMS drag law</td>
</tr>
<tr>
<td>$F$</td>
<td>drag force</td>
</tr>
<tr>
<td>$g$</td>
<td>gravity</td>
</tr>
<tr>
<td>$m$</td>
<td>mass</td>
</tr>
<tr>
<td>$m_0$</td>
<td>initial bed mass</td>
</tr>
<tr>
<td>$n_p$</td>
<td>number of particles per average unit volume</td>
</tr>
<tr>
<td>$P$</td>
<td>pressure</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$z$</td>
<td>axial height</td>
</tr>
<tr>
<td>$\beta$</td>
<td>constant in Eq. (6)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
</tr>
<tr>
<td>$\tau_p$</td>
<td>interparticle stress</td>
</tr>
<tr>
<td>$\omega$</td>
<td>coefficient for EMMS drag law</td>
</tr>
</tbody>
</table>

1. INTRODUCTION

The CFB technology is extensively used in gas-solid interacting thermolysis processes including combustion, oxy-combustion, gasification and catalytic cracking. Among the advantages of CFB technology are fuel adaptability, low pollutant emissions and high heat and mass transfer rates (Kunii, 1980). High particle concentration and intense gas-solid interactions are the main key features of CFB. Fluid Dynamics parameters affecting the system performance are important for the scaling, laying out and operation improvement of the CFB system (Thapa et al., 2016). Several experimental studies are conducted to understand the complex hydrodynamics in CFB (Issangya et al., 2000; Malcus et al., 2000; Hu et al., 2009; Lim et al., 2012).

Experimental studies can present limited macro and local flow data, since it is not easy to study all parameters. Moreover, particularly for large capacity systems experimental methods are time consuming and costly. Computational Fluid Dynamics (CFD) modeling is an efficient and reliable approach to investigate the flow development in CFB systems with the great progress in computer science. Gas-solid hydrodynamics CFD models consist of two main techniques; Eulerian-
Eulerian (E-E) and Eulerian-Lagrangian (E-L) approaches. In the E-E approach, called Two Fluid Model (TFM) as well, the two phases are described as inter-penetrating fluids (Gidaspow, 1994). The equations of motion and continuity are solved for the fluid and particle phase (Hernández, 2008). The E-E approach has the advantage of lower computational cost, however, it predicts the flow characteristics of particles less accurately (Yeoh and Tu, 2009) due to the usage of single mean particle diameter rather than the particle size distribution (PSD). PSD has significant influences on the modeling results of CFB flow field. Therefore, continuity and momentum equations should consider a realistic particle size distribution rather than a specific particle size (Gidaspow, 1994).

On the contrary, the E-L approach using Discrete Element Method (DEM) considers particles individually. In this approach, for each individual particle the momentum equations are solved and interparticle collisions are also considered. By increasing the number of particles, simulations performed with Lagrangian methods become computationally more expensive. Thus, Lagrangian approach is limited to a relatively small particle numbers and two dimensional simulations (Taghipour et al., 2005; Deen et al., 2007).

The multi-phase particle-in-cell, namely MP-PIC, approach which uses E-L scheme is recently proposed due to the mentioned limitations of TFM and DEM (Andrews and O'Rourke, 1996; Snider, 2001). The fluid part is described as continuous phase however the particles are defined as Lagrangian computational particles having a particle size distribution (Snider, 2001) in the MP-PIC method. The continuous phase is modeled by Navier-Stokes equations, and the dynamics of the particle phase is modeled using the particle distribution function. The Lagrangian computational particles (called parcel) are not treated individually and they are assumed to have the same physical properties. Therefore, the large scale multiphase systems can be modelled by a comparatively small parcel numbers, that can reduce computational cost (Snider, 2007; Chen et al., 2013).

The MP-PIC method was implemented to the simulations of multiphase and thermal reacting flows in fluidized bed (Li et al., 2013; Berrouk et al., 2017; Xie et al., 2017). This approach has been utilized in several cold flow modeling studies including bubbling fluidized beds (Weber et al., 2013; Liang et al. 2014; Fotovat et al., 2015), risers of CFB’s (Chen et al., 2013; Shi et al., 2014; Shi et al., 2015a; Shi et al., 2015b; Wang et al., 2015) and full loop CFB systems (Wang et al., 2014; Tu and Wang, 2018).

There is still limited experience to represent details of multiphase flows in CFB. Effects of some important modeling criteria such as computational particles number requirement, the chosen drag models and the bed inventory have not been fully understood yet. In the two phase flows, the drag force controls the particle movement in the riser of a CFB system. Thus, the precision of the simulation depends highly on correct drag force prediction (Nikolopoulos et al., 2013). The drag force is solved using drag models in the utilized software. One of the conventional drag models is the Wen-Yu/Ergun model which is widely used in dense flows. EMMS drag model which takes meso-scale forms in the CFB riser into account and considers heterogeneous characteristics of the flow (Chen et al., 2016; Qi et al., 2007) is recently preferred. EMMS drag model is often utilized with two-fluid model (TFM) in literature (Zhang et al., 2008; Zhang et al., 2010; Song et al., 2014; Zeneli et al., 2015; Qiu et al., 2017). The CPFD approach which is based on MP-PIC method appears more cost-effective and efficient than TFM. In this study, EMMS drag model is utilized with CPFD approach for simulation of hydrodynamics of a CFB system.

Modelling hydrodynamics of fluidized bed systems under cold flow conditions is crucial to understand flow field characteristics. The developed flow field inside the CFB system influences the characteristics of the reactive systems significantly. Therefore, accurate prediction of solid-solid and solid-gas interactions for different operating conditions should be obtained using advanced models.

In the present work, a 30 kWth CFB combustor is investigated using the CPFD method under cold flow conditions. The cold flow experiments were conducted at TUBITAK Marmara Research Center facilities. In the numerical part, the same geometry is utilized for simulations. The three dimensional flow field is solved for transient conditions. Three different bed inventories are investigated with two drag models (Wen-Yu/Ergun and EMMS). The results are compared to available measurements. The hydrodynamics of the CFB system is examined in terms of distributions of pressure, velocity and solid concentration along the riser.

2. MODEL DESCRIPTION

2.1 Governing Equations

The MP-PIC approach solves the gas phase using Eulerian grids whereas the solid phase is accounted as Lagrangian particles. In this method, the motion of gas flow is determined by solving the Navier-Stokes equations which include a coupling between two phases by interphase drag force (Andrews and O'Rourke, 1996; Snider, 2001). Newton's second law of motion is utilized to define the particles motion.

The gas phase continuity equation is

$$\frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g u_g) = 0$$

where $u_g$, $\rho_g$ and $\theta_g$ represent the velocity, density and volume fraction of the gas, respectively.

The gas phase momentum equation is written as

$$\frac{\partial (\rho_g u_g)}{\partial t} + \nabla \cdot (\rho_g u_g u_g) = -\nabla \cdot \tau_{gg} - \rho_g \nabla \theta_g$$

The gas phase mass transfer equation is

$$\frac{\partial (\rho_g \theta_g)}{\partial t} + \nabla \cdot (\rho_g u_g \theta_g) = \nabla \cdot \alpha_{gg}$$

where $\alpha_{gg}$ represents the gas mass transfer coefficient.

The solid phase continuity equation is

$$\frac{\partial (\rho_s \theta_s)}{\partial t} + \nabla \cdot (\rho_s u_s \theta_s) = \nabla \cdot \alpha_{ss}$$

The solid phase momentum equation is

$$\frac{\partial (\rho_s u_s \theta_s)}{\partial t} + \nabla \cdot (\rho_s u_s u_s \theta_s) = -\nabla \cdot \tau_{ss} - \rho_s \nabla \theta_s$$

The solid phase mass transfer equation is

$$\frac{\partial (\rho_s \theta_s)}{\partial t} + \nabla \cdot (\rho_s u_s \theta_s) = \nabla \cdot \alpha_{ss}$$

where $\alpha_{ss}$ represents the solid mass transfer coefficient.
\[ \frac{\partial \theta u_x}{\partial t} + \nabla \cdot (\theta u_x u_x) = -\frac{1}{\rho_e} \nabla P - \frac{1}{\rho_e} F + \theta g \]
\[ + \frac{1}{\rho_e} \nabla \tau \]
(2)

where \( P \), \( g \) and \( F \) represent gas pressure, the gravitational acceleration and the momentum exchange rate between two phases, respectively. The fluid is assumed to be incompressible and both phases are isothermal.

\[ F = \int f \left[ D_p (u_x - u_p) \right] - \frac{1}{\rho_p} \nabla P \] dmdv
(3)

where \( u_p \) and \( \rho_p \) are the particle velocity and density. \( D_p \) is the drag function and \( f \) stands for probability distribution function obtained from Liouville equation.

The Lagrangian method is implemented to model particle motion with particle acceleration defined as

\[ \frac{du_p}{dt} = D_p (u_x - u_p) - \frac{1}{\rho_p} \nabla P + g - \frac{1}{\theta_p \rho_p} \nabla \tau_p \]
(4)

The first term in the right hand side of Eq. (4) is acceleration and the second term is pressure gradient. The remaining terms are gravity and the inter-particle stress gradient, \( \tau_p \), respectively. The particle velocity is presented by

\[ \frac{dx_p}{dt} = u_p \]
(5)

where \( x_p \) is the location of particles.

In the present model, the spatial gradients are used to solve the particle-particle collisions then they are interpolated to each particle. The volume fraction of particles is computed from particle volume mapped to the grid. Then, the particle stress is derived from particle volume fraction. The normal stress of particle is expressed as (Snider, 2001)

\[ \tau_p = \frac{P_x \theta_p^\beta}{\max ([\theta cp - \theta_p], \epsilon (1 - \theta_p))} \]
(6)

where \( P_x \) represents a positive constant, \( \theta cp \) is the volume fraction of particle at the close packing limit. \( \beta \) is a constant \((2 < \beta < 5)\). The constant \( \epsilon \) is a number having the order of \(10^{-7}\) to avoid the singularity at close packing (Snider, 2001).

### 2.2 Drag Models

In CPFD modeling of gas–solids flow, the selected drag model influences the simulation results significantly. Therefore, different drag models should be tested.

In this study, two drag models were used.

- Wen-Yu/Ergun (Gidaspow, 1994)
- EMMS (Yang et al., 2003)

The drag force exerted on a particle is given as

\[ F_p = m_p D (u_x - u_p) \]
(7)

where \( m_p \) and \( D \) represent the particle mass and the drag function, respectively. In most of the models, the drag function depends on flow conditions, geometry and the drag coefficient \( c_d \).

The Wen-Yu model is suggested for the particle volume concentrations within the range of 0.01 to 0.61 (Wen and Yu, 1966) whereas the Ergun drag model is suitable for the range from 0.47 to 0.7 (Ergun, 1952). By combining of two mentioned drag models, Wen-Yu/Ergun model is obtained and recommended for dense particulate flows (Gidaspow, 1994).

The Wen-Yu/Ergun model is defined as in Table 1. \( D_1 \) and \( D_2 \) are the Wen-Yu and Ergun drag functions, respectively. EMMS model is utilized to reveal the heterogeneous structure and capture the multiphase flow interactions in the dense region of CFB. The EMMS model is recommended for resolving the formation and breaking up of clusters (Chen et al., 2013). The EMMS drag law is implemented in CPFD as in Table 1.

### 3. EXPERIMENTAL SETUP

Experimental measurements were conducted in a 30 kW\(_h\) CFB system (Fig. 1) at TUBITAK-MRC premises.

The laboratory scale system contains a riser, a fuel feeding system, a return leg, cyclones, ash hoppers, primary air fan, ID fan and bag filter. The height of the CFB riser is 6 m with 0.108 m of the inner diameter.

The silica sand bed material was used having PSD given in Fig. 2. Its particle and bulk densities are determined to be 2650 kg/m\(^3\) and 1530 kg/m\(^3\) respectively. Inlet air velocity was set constant at 3 m/s. Pressures along the riser were measured with pressure transmitters which are placed at 0.32, 1.25, 2.248, 3.148, 4.036, 4.936, 5.84 m above the distributor plate. Three different particle masses were studied to observe the effect of solid bed inventory on the pressure distribution along the riser. Operating conditions and gas and particle properties used in measurements are listed in Table 2 in detail.

### 4. NUMERICAL SETUP

Barracuda\(^{®}\) software using CPFD scheme is used in the simulations. In the model, the riser of the experimental system is analyzed. The particle phase is silica sand and air is defined as fluid phase in the model. For the initial conditions, three initial particle masses (\(m_p=3.79, 4.55 \text{ and } 5.20 \text{ kg}\)) are set with PSD of Fig. 2 and air feeding is identical with experiments. Inlet and outlet boundary conditions are introduced as the same with experiments and their locations are shown in Fig. 3. The fluidization air is fed into system at the riser bottom with inlet velocity of 3 m/s; particles are recycled to the riser.
Table 1 Drag Coefficients for Wen-Yu Ergun and EMMS Model

| Wen-Yu/Ergun model | \( C_d = \begin{cases} 
D_1 & \theta_p < 0.75\theta_p \\
(D_2 - D_1) & 0.75\theta_p \geq \theta_p \geq 0.85\theta_p \\
D_2 & \theta_p > 0.85\theta_p 
\end{cases} \) |
|-------------------|--------------------------------------------------|
| EMMS model        | \( f_e = \begin{cases} 
\frac{1}{180\theta_p} \left( \frac{150\theta_p}{\theta_g} \cdot 1.75Re \right) & \theta_g < 0.74 \\
(1 + 0.15Re)^{0.67} & \theta_g \geq 0.74 \text{ and } Re < 1000 \\
0.44 Re 24 & \theta_g \geq 0.74 \text{ and } Re \geq 1000 
\end{cases} \) |
|                   | \( \omega = \begin{cases} 
-0.0214 + 0.00214 \left( \frac{4(\theta_g - 0.7463)^2}{4(\theta_g - 0.7463)^2 + 0.0044} \right) & 0.74 \leq \theta_g \leq 0.82 \\
-0.0101 + 0.0038 \left( \frac{4(\theta_g - 0.7789)^2}{4(\theta_g - 0.7789)^2 + 0.0040} \right) & 0.82 < \theta_g \leq 0.97 \\
-31.8295 + 32.895\theta_g & 0.97 < \theta_g \leq 1 
\end{cases} \) |

Fig. 1. 30kW CFB Combustor (schematic diagram on the left, a photograph of the system on the right) (Kayahan and Ozdogan, 2016).

located 0.143 m above the distributor. Pressure boundary condition is implemented at the outlet. The other simulation parameters are given in Table 2.

4.1 Grid Test

A grid was generated for the geometry given in Fig. 3. A grid independence analysis was implemented to make sure that the grid size does not affect the simulation results. Four grid sizes were used with approximately 20k, 40k, 50k and 80k cells. In Fig. 4, the distribution of the time averaged axial pressure was used to examine the influences of the grid size.

The simulation results for the 40k, 50k and 80k cells grid sizes are in good agreement but the results of 20k cells grid size deviate and predict higher pressure profile along the riser. The 40k cells grid size was chosen in simulations due to the computational cost without losing precision.

In Barracuda software, computational particles are defined as particle groups having the same physical properties. It is necessary to determine the computational particles number for CPFD calculations. Selection of this number affects the accuracy and computational efficiency of the simulations. For our study sensitivity analysis for computational particles number was done like grid sensitivity analysis. Five computational particles (parcel number) with 12084, 46464, 90288, 118772, 196040 which correspond to the particle numbers per average unit volume (np) value of 5, 20, 40, 60, 80, respectively, were tested (Fig. 5). The axial...
pressure profile simulations with 12084 and 46464 parcel numbers are omitted due to their higher deviation rates compared to those with the 90288, 118772 and 196040 parcels. Therefore, 90288 parcel number is selected for the simulations to save the computational cost.

Fig. 2. Particle size distribution (PSD) of the silica sand.

Table 2 Experimental and simulation parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Experimental</th>
<th>Simulation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Operating temperature (K)</td>
<td>298</td>
<td>298</td>
</tr>
<tr>
<td>Operating pressure (kPa)</td>
<td>101.325</td>
<td>101.325</td>
</tr>
<tr>
<td>Gas density (kg/m³)</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>Gas viscosity (Pa.s)</td>
<td>1.9×10⁻⁵</td>
<td>1.9×10⁻⁵</td>
</tr>
<tr>
<td>Superficial gas velocity (m/s)</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Particle density (kg/m³)</td>
<td>2650</td>
<td>2650</td>
</tr>
<tr>
<td>Total bed inventory, Mₚ (kg)</td>
<td>3.79; 4.55;</td>
<td>3.79; 4.55;</td>
</tr>
<tr>
<td>Initial bed material height (m)</td>
<td>2.5d; 3d;</td>
<td>2.5d; 3d;</td>
</tr>
<tr>
<td>Particle-particle interaction</td>
<td>-</td>
<td>40</td>
</tr>
<tr>
<td>Maximum momentum redirection from collision(%)</td>
<td>-</td>
<td>0.58</td>
</tr>
<tr>
<td>Close-pack particle volume fraction</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>Particle to wall interaction</td>
<td>-</td>
<td>0.99</td>
</tr>
<tr>
<td>Normal-to-wall retention coefficient</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>Tangent-to-wall retention coefficient</td>
<td>-</td>
<td>0.001</td>
</tr>
<tr>
<td>Diffuse bounce</td>
<td>-</td>
<td>0</td>
</tr>
<tr>
<td>Time Control</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Initial time step (s)</td>
<td>-</td>
<td>0.001</td>
</tr>
<tr>
<td>Total time (s)</td>
<td>-</td>
<td>100</td>
</tr>
<tr>
<td>Start time for averaging (s)</td>
<td>-</td>
<td>20</td>
</tr>
</tbody>
</table>

Fig. 3. Schematic diagram of the computational geometry.

Fig. 4. Pressure distribution along the CFB riser for 4 grid sizes.

Fig. 5. Pressure distribution along the CFB riser for 5 parcel numbers.
5. RESULTS AND DISCUSSION

5.1 The Effect of Solid Bed Inventory on Pressure Distribution

The solid bed inventory has an important influence on the pressure drops along the riser. The higher the solid bed mass, the greater the pressure drop (Li and Kwauk, 1980). Figures 6 and 7 present the experimental and simulation results with 2.5d, 3d and 3.5d static bed heights corresponding to 3.79, 4.55, and 5.20 kg bed material mass, respectively. The comparison of measured and computed time averaged pressure distributions along the riser for different bed inventories are illustrated in Fig. 6 with the Wen-Yu/Ergun Drag model and Fig. 7 with the EMMS model.

Both models predict higher pressure in the bottom section of the riser due to higher particle volume fractions parallel to experimental results. The variation of static bed height affects the pressure distribution in the riser as seen in Figs. 6 and 7. It is revealed by both models that pressure drops significantly at the lower section of the riser due to higher concentration of particles for all bed inventories. Moreover, EMMS model results in higher decrease in pressure compared to Wen-Yu/Ergun model in that region. Then, the decrease in pressure is observed less at the upper section of the riser. Wen-Yu/Ergun model predicts pressure drop more however EMMS model computes a slight change in that region.

![Fig. 6. Time averaged pressure distribution along the riser under different bed inventories (Wen-Yu/Ergun Drag model).](image)

The expected increase in pressure drops with the increase of the total bed mass is observed with both the simulated and experimental results. It is seen that by increasing the bed inventory, pressure drop along the lower section of the riser increases both in experiments and simulations. For both drag models, comparisons of the measured and simulated pressure profiles along the riser show good agreement (Figs. 6 and 7). Calculation errors are reasonable and less than 5%. Both models slightly overestimate the pressure data. Wen Yu/Ergun model has better predicted qualitatively the correct decreasing trend. However, the EMMS based drag model predicts better the pressure distribution quantitatively. Thus, EMMS model appears to predict the pressure profile along the riser more accurately.

![Fig. 7. Time averaged pressure distribution along the riser under different bed inventories (EMMS drag model).](image)

5.2 The Effect of Solid Bed Inventory on Particle Volume Fraction Average particle volume fraction

A conventional CFB riser may be divided into two regions which are called core and annulus. The gas velocity is quite low sometimes even negative in the annulus whereas it is higher than the superficial velocity in the core. The solids move upwards and downwards through the bed within the clusters. Solids moving up through the core mostly drift sideways by cause of hydrodynamic interactions as they meet with gas velocities negative or too low to bring the clusters upwards. Therefore, the clusters begin to move downward in the annulus region (Arena et al., 1988; Tung et al., 1988; Werther, 2005). Additional complex interactions between wall and particles may lead solid movements downwards near the wall region as well (Wang et al., 2015).

Figure 8 depicts the radial profiles of the time-averaged particle volume fraction at five locations in axial direction (z = 10d, 20d, 30d, 40d, 50d m) for different bed inventories by comparing the results of EMMS and the Wen-Yu/Ergun drag models. As it is seen from the figures; the axial particle volume fraction is higher at each axial position in the Wen-Yu/Ergun simulation than the corresponding data in the EMMS simulations. When the bed inventory increases, the concentration of particle also increases in the riser. Both models show the typical core-annulus flow especially at the dense region in the riser. The particle concentration changes from the axis to the wall and it is the highest near the wall and lowest along the axis of the riser. Wen-Yu/Ergun model shows a relatively more parabolic distribution while EMMS model predicts a slight change of particle concentration in the core region. The particle volume fraction decreases from bottom to upper regions and its profile is found much flatter in the upstream region of the riser due to more homogeneous flow in that region.

Figure 8. Average particle volume fraction under different bed inventories, a) Wen-Yu/Ergun based, left column b) EMMS based, right column Instantaneous particle volume fraction.

Figure 9 illustrates the distribution of instantaneous particle volume fraction for the bed inventory height of 3d using Wen-Yu/Ergun and EMMS models, and five axial heights (z=10d, 20d, 30d, 40d, 50d). This figure indicates that solid concentration is more close to the wall region in line with the averaged results of Fig. 8. In the simulation results with the EMMS model, the particle volume fraction shows non-uniform characteristics at each axial heights and particle clusters appear mostly near the wall area. However, in the Wen-Yu/Ergun simulation, particle distribution is nearly uniform axially without clusters presence and region separation. In the literature (Wang et al., 2008; Li et al., 2012; Chen et al., 2013), it is reported that EMMS drag model considers the heterogeneous configuration (i.e. cluster formation) in the CFB risers. Thus, it gives more precise results compared to homogeneous drag models (i.e. Wen-Yu/Ergun drag model).

5.3 The Effect of Solid Bed Inventory on Particle Velocity Distribution.

The axial particle velocity profiles in radial direction for different bed inventories at five different heights in the riser for both drag models are given in Fig. 10. Both models have predicted lower or even negative axial solid particles velocities near the wall, while the particle velocity is upward and reaches a maximum in the core region. It indicates that back mixing occurs and core-annulus flow structure is predicted (Yang et al., 2003). In the Wen-Yu/Ergun model particle velocities in the core region decreases regularly in the axial direction and becomes nearly flat in upper region of the riser. Moreover, it predicts particle velocities larger than that found by EMMS drag model. In the simulations, near-wall effects and core annulus flow pattern are correctly estimated at each height with both drag models. Differently from Wen-Yu/Ergun model, in the simulation with EMMS drag model, particle velocity distribution shows slightly the ‘M’ shaped profile as monitored by (Yang et al., 2004; Nikolopoulos et al., 2010; Wang and Liu, 2010). As shown in Figure 10, solid bed inventory appears to have a more profound effect on particle velocities in the simulation with Wen-Yu/Ergun model compared to the EMMS simulation. Wen-Yu/Ergun drag model gives slightly higher velocities at higher solid inventories however EMMS model finds almost no change.

5.4 The Effect of PSD on Flow Dynamics

CPFD approach has the capability of working with the particle size distribution for gas-solid flow modeling. In previous sections, the simulations were performed with a PSD shown in Fig. 2. In order to examine the effect of PSD on gas particle flow dynamics an average particle diameter case has been studied as in literature (Chen et al., 2013; Wang et al., 2014). The surface volume average particle diameter was used in simulations equal to 0.301mm. The computed radial profiles of time-averaged particle volume fraction with and without PSD at three locations in axial direction (z =10d, 20d, 30d), where variations are high, for the bed inventory.
height of 3d are shown in Figs. 11a and b. The results of Wen-Yu Ergun model and EMMS model are shown in Figs. 11a and 11b, respectively. The axial particle volume fraction is found higher at each axial position in the Wen-Yu/Ergun simulation than the EMMS simulations in line with the simulation results of Fig. 8.

The PSD affects the averaged particle volume fraction at the lower section of the riser (z=10d) significantly while it has limited influence at the upper sections of the riser predicted by the Wen-Yu Ergun model simulations. In addition, the particle volume fraction augmented highly near the wall while it decreased slightly in the core region at lower section of the riser since the number of large particles with PSD case is comparatively less than averaged particle diameter case. The particle volume fraction is found higher in the upper sections of the riser with PSD since the small-diameter particles can move freely and accumulate in those regions. In the EMMS model, small difference is observed between particle volume fractions simulated using the average particle size and that with the PSD as seen in Fig. 11b.

Figure 12 presents the axial particle velocity profiles in radial direction for average particle size at three different heights in the riser for both drag models. Particle velocities in the simulation with Wen-Yu/Ergun model are higher compared to the EMMS simulations and found large at the lower section with both models in line with the results in Fig. 10. Axial particle velocity is also negative near the wall and increases towards the center due to back mixing. It is seen that the velocity profiles become flatter with average particle diameter cases for all locations in axial direction in Wen-Yu Ergun model. Moreover, average particle velocities are obtained higher with PSD case. Slight differences are observed between the time-averaged axial solid velocity profiles with EMMS model simulated with the average particle diameter and the PSD case.

**CONCLUSION**

The cold flow model simulations of the 30 kW CFB combustor with CPFD showed good agreement with measured data.

The investigation of the effects of bed inventory and drag model on flow characteristics considering pressure profile, particle concentration and velocity profile with EMMS and Wen-Yu/Ergun drag models have led to the following conclusions in terms of simulations.

The simulated pressure profiles with both drag models resulted in accordance with available measurements along the riser. Both models slightly
Fig. 10. Average solid axial velocity distribution under different bed inventories, a) Wen-Yu/Ergun based, left column b) EMMS based, right column.

Fig. 11. Average particle volume fraction with and without PSD for h=3d, a) Wen-Yu/Ergun based, left column b) EMMS based, right column.

Fig. 12. Average solid axial velocity distribution with and without PSD for h=3d, a) Wen-Yu/Ergun based, left column b) EMMS based, right column.
overestimate the pressure data with errors less than 6%. Wen-Yu/Ergun model has better predicted qualitatively the correct decreasing pressure trend. However, the EMMS based drag model shows better prediction in the pressure distribution of the CFB riser quantitatively. Thus, EMMS model appears to predict the pressure profile more precisely.

In simulations with both drag models it is found that volume fraction of particles increases as the bed inventory increases. Whereas it decreases along the riser.

The core annulus structure has captured by both drag models, particularly at the dense particle region. Simulation with EMMS model considers heterogeneous characteristics (cluster formation) better than Wen-Yu/Ergun model.

The solid axial velocities slightly increase as the bed inventory increases in both drag models. Wen-Yu/Ergun model predicts one core annulus flow pattern. However, EMMS drag model shows M shaped profile as reported by (Yang et al., 2004; Nikolopoulos et al., 2010; Wang and Liu, 2010).

The influence of PSD is observed more in particle volume fraction distributions with Wen-Yu Ergun model. Utilizing a PSD has significantly decreased the particle volume fraction especially at the lower section of the riser. The effect of PSD cannot be captured well by EMMS model.

In the present study, it is found that the CPFD approach is a promising tool and predicts flow hydrodynamics accurately.

ACKNOWLEDGMENTS

The authors gratefully acknowledge Marmara University (BAP Project number: FEN-C-DRP-090217-0056), TUBITAK (Project ID:216M348) and Energy Institute of MRC-TUBITAK.

REFERENCES


Research 43(18), 5548-5561.


