Chapter 3

CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed
3.1. Introduction

Liquid–solid fluidised beds continue to attract increasing attention due to their inherent versatility for several industrial applications in hydrometallurgical, biochemical, environmental and chemical process industries (Epstein, 2003). Due to advantages such as the absence of high shear zones and uniform distribution of solids, liquid–solid fluidised beds provide a viable option to replace mechanically agitated reactors for achieving cost reduction and improvements in product quality. However, due to the lack of information on various design and operating aspects of liquid–solid fluidised beds, it is likely that their introduction to large scale applications may not be realised as soon as desirable. Significant contributions have been made by several authors (Kiared et al., 1997; Limtrakul et al., 2005) to improve the understanding of the hydrodynamics of liquid–solid fluidised beds through experimental and theoretical investigations.

In comparison to reactors such as bubble column reactors, the flow patterns of solids in liquid fluidised beds is not yet fully understood in terms of circulation patterns and energy dissipation. Circulation phenomena of solids have been observed to be dominant in liquid fluidised beds due to non uniform solid holdup profiles and solid velocity profiles. For this reason, computational fluid dynamics (CFD) has been promoted as a useful tool for understanding multiphase reactors (Dudukovic et al., 1999) for reliable design and scale up.

Hydrodynamics and solids expansion in liquid fluidised beds have been extensively studied by several authors (Richardson and Zaki, 1954; Latif and Richardson, 1972; Gibilaro et al., 1986) and reviewed by Di Felice (1995). Kiared et al. (1997) investigated the motion of solids in liquid fluidised beds using non-invasive
radioactive particle tracking technique. According to their investigation, in the fully
developed region of the bed, the flow structure consisted of a core and an annulus in
which the solids underwent distinct upward and downward movements. Yang and
Renken (2003) developed a more accurate relationship linking the apparent drag
force, the effective gravitational force and the voidage to propose a generalised
correlation for liquid particle interaction which is applicable for intermediate regime.
This correlation along with Richardson and Zaki equation is applicable for laminar,
intermediate and turbulent regimes. Recently, Limtrakul et al. (2005) have reported
comprehensive experimental results for solid holdup and solids velocity profiles in
liquid fluidised beds using non-invasive gamma ray based techniques. The non-
invasive measurement techniques such as computer tomography (CT), computer-
aided radioactive particle tracking (CARPT) are used for the prediction of phase
holdup and solid velocity profiles respectively of liquid–solid fluids beds. This study
provides the data needed for CFD validation. Based on the experimental observations,
they have reported that the time-averaged solid holdup distribution is axisymmetric
with high value at the wall and low value at the center and the average solid holdup
can be predicted reasonably well with the modified Richardson-Zaki equation
(Garside and Al-Dibouni, 1977).

Roy and Dudukovic (2001) have carried out experimental investigations on
the fluid dynamics of liquid–solid risers using non-invasive flow methods and created
a database for solids holdup distribution, the solids instantaneous and ensemble-
averaged velocity patterns, as well as the solids residence time distribution in the
riser. They used this database for validating their two fluid Euler–Euler CFD model.
Cheng and Zhu (2005) developed a CFD model for simulating the hydrodynamics of
liquid–solid circulating fluidised bed reactor. They included turbulence and kinetic theory of granular flow in the governing equations to model the high Reynolds number two phase flows with strong particle–particle interactions and used FLUENT 4.5.6 for their CFD simulations. They reported strong non-uniformities in flow structure for the larger particle system. Doroodchi et al. (2005) used CFD approach to investigate the influence of inclined plates on the expansion behavior of solids in a liquid fluidised bed containing two different sized particles. Their model is based on the solution of Eulerian muliphase equations with two different particle sizes with continuous phase of water. The hindered settling behavior was included in their model via the inclusion of a volume fraction dependent drag law. The authors validated their computational model with their own experiments performed with ballotini particles demonstrating a significant increase in particle sedimentation rate due to introduction of inclined plates into the conventional fluidised bed. However, comparatively less information is available regarding CFD modeling of the solids flow pattern in a liquid–solid fluidised beds in contrast to the extensive knowledge of gas–solid fluidised beds and bubble column reactors.

In this chapter, the flow pattern of solids and liquid motion in liquid fluidised beds are simulated using CFD for various design and operating conditions. The data of Limtrakul et al. (2005) is chosen for the purpose of validating the numerical results obtained through CFD. The liquid fluidised beds used in the experimental study of Limtrakul et al. (2005) are two plexiglas columns: 0.1 m i.d. with 2 m height and 0.14 m i.d. with 1.5 m height. The liquid phase is chosen as water. The solid phase is chosen as glass beads of size 1 and 3 mm with a density of 2900 kg/m$^3$ and 2500 kg/m$^3$ respectively. They also used acetate beads of 3 mm size with a density of
The present work also aims to evaluate the influence of various interphase drag force models, inlet boundary condition, grid resolution, time step sensitivity as well as a comparison between 2D and 3D simulation on the predictive capabilities of the numerical investigation. Based on the flow pattern of solids motion predicted by CFD, a solid mass balance in the center and wall regions of the fluidised bed and various energy flows are computed.

3.2. CFD Model

The simulation of liquid fluidised bed was performed by solving the governing equations of mass and momentum conservation using ANSYS CFX software. A multi-fluid Eulerian model, which considers the conservation of mass and momentum of fluid and solid phases, was applied.

Continuity equations:

Liquid phase

\[ \frac{\partial}{\partial t} (\varepsilon_l \rho_l) + \nabla \cdot (\rho_l \varepsilon_l \vec{u}_l) = 0 \]  \hspace{1cm} \text{(3.1)}

Solid phase

\[ \frac{\partial}{\partial t} (\varepsilon_s \rho_s) + \nabla \cdot (\rho_s \varepsilon_s \vec{u}_s) = 0 \]  \hspace{1cm} \text{(3.2)}

where \( \varepsilon_l, \varepsilon_s \) are the volume fractions of liquid and solid phase respectively which satisfy the relation

\[ \varepsilon_l + \varepsilon_s = 1 \]  \hspace{1cm} \text{(3.3)}

\( \vec{u}_l, \vec{u}_s \) are the liquid and solid phase velocities respectively and \( \rho_l, \rho_s \) are the liquid
and solid phase densities respectively.

Momentum equations:

Liquid phase

$$\frac{\partial}{\partial t}(\rho_l, e_l, \bar{u}_l) + \nabla \cdot (\rho_l, e_l, \bar{u}_l \bar{u}_l) = -e_l \cdot \nabla P + \nabla \cdot \left( e_l \mu_{\text{eff},l} \left[ \nabla \bar{u}_l + \left( \nabla \bar{u}_l \right)^T \right] \right) + \rho_l e_l \tilde{g} + F_{D,ls}$$

.................(3.4)

Solid phase

$$\frac{\partial}{\partial t}(\rho_s, e_s, \bar{u}_s) + \nabla \cdot (\rho_s, e_s, \bar{u}_s \bar{u}_s) = -e_s \cdot \nabla P - \nabla P_s + \nabla \cdot \left( e_s \mu_{\text{eff},s} \left[ \nabla \bar{u}_s + \left( \nabla \bar{u}_s \right)^T \right] \right) + \rho_s e_s \tilde{g} - F_{D,ls}$$

.................(3.5)

where P is the pressure, which is shared by all the phases, $\mu_{\text{eff}}$ is the effective viscosity, $\nabla P_s$ is the collisional solids stress tensor that represent the additional stresses in solid phase due to particle collisions, $\tilde{g}$ is the gravity vector, and the last term ($F_{D,ls}$) represents interphase drag force between the liquid and solid phases.

The most popular constitutive equation for solids pressure is due to Gidaspow (1994) viz.,

$$\nabla P_s = G(e_s) \nabla e_s$$

...............(3.6)

where $G(e_s)$ is the elasticity modulus and it is given as

$$G(e_s) = G_0 \exp(c(e_s - e_{\text{sm}}))$$

...............(3.7)

as proposed Bouillard et al. (1989), where $G_0$ is the reference elasticity modulus and is set to 1 Pa, $c$ is the compaction modulus which is set to 100 for the present simulation and $e_{\text{sm}}$ is the maximum packing parameter.

For the continuous phase (liquid phase) the effective viscosity is calculated as

$$\mu_{\text{eff},l} = \mu_{l} + \mu_{T,l} + \mu_{\text{vp}}$$

...............(3.8)

where $\mu_{l}$ is the liquid viscosity, $\mu_{T,l}$ is the liquid phase turbulence viscosity or shear
induced eddy viscosity, which is calculated based on the k–ε model of turbulence as
\[ \mu_{T,l} = c_p \rho_l \frac{k^2}{\varepsilon} \] ........................(3.9)
where the values of \( k \) and \( \varepsilon \) come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate, \( \mu_{ip} \) represents the particle induced turbulence and is given by the equation proposed by Sato et al. (1981) as
\[ \mu_{ip} = c_p \rho_l \varepsilon_s d_l |\bar{u}_s - \bar{u}_l| \] ........................(3.10)
The values used for constants in the turbulence equations are summarised in Table 3.1.

**Table 3.1.** Standard values of the parameters used in the Turbulence model

<table>
<thead>
<tr>
<th>( C_\mu )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
<th>( C_{\varepsilon 1} )</th>
<th>( C_{\varepsilon 2} )</th>
<th>( C_{\mu b} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.3</td>
<td>1.44</td>
<td>1.92</td>
<td>0.6</td>
</tr>
</tbody>
</table>

The interphase drag force, which is generally, computed from the knowledge of the drag coefficient \( C_D \), particle Reynolds number and solids volume fraction is given by
\[ \bar{F}_{D,ls} = C_{D,ls} \frac{3}{4} \rho_l \frac{\varepsilon_s}{d_p} |\bar{u}_s - \bar{u}_l| (|\bar{u}_s - \bar{u}_l|) \] ........................(3.11)
where \( C_{D,ls} \) is the interphase drag coefficient.

The following drag models are used for representing the drag coefficient between solid and liquid phases.

**Drag model 1: Gidaspow (1994)**
\[ C_{D,ls} = \frac{150 \varepsilon_s^2 \mu_l}{\varepsilon_l d_p^2} + 1.75 \varepsilon_s \rho_i (u_s - u_i) \] \((\varepsilon_l < 0.8)\) ........................(3.12)
\[ C_{D,ls} = \frac{3}{4} C_d \rho_i \frac{\varepsilon_s}{d_p} (\bar{u}_s - \bar{u}_l) f(\varepsilon_l) \] \((\varepsilon_l > 0.8)\) ........................(3.13)
where
\[ C_D = \frac{24}{Re_p} \left(1 + 0.15Re_p^{0.687}\right), \quad Re_p \leq 1000 \] .................(3.14)

\[ C_D = 0.44, \quad Re_p \geq 1000 \] .................(3.15)

and \[ f(\varepsilon_i) = \varepsilon_i^{-2.65} \] .................(3.16)

**Drag model 2: Di Felice (1994)**

\[ C_D = \frac{3}{4} C_d \rho_l \frac{\varepsilon_i}{d_p} (\bar{u}_s - \bar{u}_i)f(\varepsilon_i) \] .................(3.17)

where
\[ f(\varepsilon_i) = \varepsilon_i^{-x} \] .................(3.18)

where \( x \) is given
\[ x = 3.7 - 0.65\exp\left[-\frac{1}{2}\left(1.5 - \log_{10}Re_p\right)^2\right] \] .................(3.19)


\[ C_D = \frac{3}{4} C_d \rho_l |u_s - u_i| \varepsilon_i \varepsilon_s \] .................(3.20)

and \[ C_d = \sqrt{0.63 + 4.8\sqrt{f/Re_i}} \] .................(3.21)

where \( f \) is the ratio of the falling velocity of a superficial to the terminal velocity of a single particle and is given by Kmiec (1982) as

\[ f = 0.5 \left(A - 0.06Re_i + \sqrt{(0.06Re_i)^2 + 0.12Re_i(2B - A) + A^2}\right) \] .................(3.22)

where
\[ A = \varepsilon_i^{4.14} \] .................(3.23)

\[ B = \begin{cases} \varepsilon_i^{2.65}, & \varepsilon_s < 0.15, \\ 0.8 \varepsilon_i^{1.28}, & \varepsilon_s \geq 0.15. \end{cases} \] .................(3.24)
3.3. Numerical Simulation

ANSYS CFX software code is used for simulating the hydrodynamics of liquid–solid fluidised bed. Tables 3.2 and 3.3 summarise the model parameters/conditions used for the simulation of solid motion in liquid fluidised beds.

**Table 3.2. Simulation process conditions**

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D and 3-D simulation</td>
<td>column Diameter 0.14 m, height 1.5 m</td>
</tr>
<tr>
<td>Grid size</td>
<td>coarse mesh with 25000 nodes, finer mesh with 40000 nodes</td>
</tr>
<tr>
<td>Time step</td>
<td>0.001–0.01 s</td>
</tr>
<tr>
<td>Inlet boundary</td>
<td>fully developed velocity profile, uniform inlet velocity</td>
</tr>
<tr>
<td>Column diameter</td>
<td>diameter: 0.1 m, 0.14 m</td>
</tr>
<tr>
<td>Particle size</td>
<td>1 mm, 3 mm</td>
</tr>
<tr>
<td>Particle density</td>
<td>1300–2500 kg/m³</td>
</tr>
<tr>
<td>Superficial liquid velocity</td>
<td>0.07–0.13 m/s</td>
</tr>
</tbody>
</table>

**Table 3.3. Simulation model parameters**

<table>
<thead>
<tr>
<th>Solid</th>
<th>Glass beads</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (kg/m³)</td>
<td>2500</td>
</tr>
<tr>
<td>Size (mm)</td>
<td>3, 1</td>
</tr>
<tr>
<td>(U_{mf}) (m/s)</td>
<td>0.0412, 0.014</td>
</tr>
<tr>
<td>Solid holdup (-)</td>
<td>0.683, 0.593</td>
</tr>
<tr>
<td>Bed voidage (-)</td>
<td>0.317, 0.417</td>
</tr>
<tr>
<td>Initial bed height (m)</td>
<td>0.369, 0.366</td>
</tr>
</tbody>
</table>

**3.3.1. Flow Geometry and Boundary conditions**

Figure 3.1 depicts typical numerical mesh used for simulation. The upper section of the simulated geometry, or freeboard, was considered to be occupied by liquid only. Inlet boundary conditions were employed at the bottom of the bed to specify a uniform liquid inlet velocity. The liquid is introduced at all the computational cells of the bottom of the column. Pressure boundary condition was employed at the top of the freeboard. This implies outlet boundary conditions on
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

pressure, which was set at a reference value of $1.013 \times 10^5$ Pa. The lateral walls were modeled using the no-slip velocity boundary conditions for the liquid phase and the free slip assumption for the solid phase.

![Figure 3.1](image.png)

**Figure 3.1.** (a) 2D (b) 3D mesh of liquid fluidised bed

The numerical simulations of the discrete governing equations were achieved by finite volume method. Pressure-velocity coupling was achieved by the SIMPLE algorithm. The governing equations were solved using the advanced coupled multi-grid solver technology of ANSYS CFX. The second order equivalent to high-resolution discretisation scheme of momentum, volume fraction of phases, turbulent
kinetic theory and turbulence dissipation rate was chosen. During the simulations, the standard values of under relaxation factors were used. For time dependent solution the second order implicit time discretisation was used. The simulations were carried out till the system reached the pseudo steady state. Once the fully developed quasi-steady state is reached, the time averaged quantities are calculated. For all the simulations, the time averaged quantities are performed in the time interval 50–150s. The axial and azimuthal average is then performed along the axial direction within the middle section of the column. The convergence criteria for all the numerical simulation is based on monitoring the mass flow residual and the value of 1.0e-04 is set as converged value. This convergence is monitored as a function of number of iterations at each time.

3.4. Results and Discussion

3.4.1. Comparison between 2D and 3D simulation

Figure 3.2 provides a comparison of time averaged solid holdup and solid velocity obtained through 2D and 3D CFD simulation. From Figures 3.2(b) & 3.2(d) it is evident that 3D CFD simulation provides a more accurate prediction of solid motion involving the core–annulus pattern and hence only 3D simulation was chosen for further studies in this work.
3.4.2. Grid resolution study

Two type of meshes were used in this study i.e., mesh 1 contains a medium mesh of around 25000 nodes and mesh 2 contains 40000 nodes. The simulation was performed using a liquid superficial velocity of 0.07 m/s. Figure 3.3 illustrates the effect of different meshes on time averaged axial solid velocity. It shows that both meshes are giving the same pattern of axial solid velocity and there is not much difference in prediction of solid velocity profiles. So, in order to reduce the computational time, medium mesh was used for further simulation.

3.4.3. Effect of time step

Time dependent simulations were performed with time step in the range of 0.01–0.001 sec. The various time steps viz., 0.01, 0.005 and 0.001 sec were used for testing the accuracy of solution. Figure 3.4 shows the predicted solid volume fraction
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

at 5 sec for different time step of 0.01, 0.005 and 0.001 s. It can be shown that there is not much variation of solid holdup prediction for the time step values 0.005s and 0.001s. A computational time a value of 0.005 s was set as the time step for the simulation studies in this work.

Figure 3.3. Influence of mesh sensitivity on the time averaged axial solid velocity at superficial liquid velocity of 0.07 m/s.

Figure 3.4. Influence of time sensitivity studies on the solid holdup (a) 0.01 s (b) 0.005s (c) 0.001s
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

3.4.4. Effect of drag force models

Figure 3.5 shows the effect of drag force models proposed by Gidaspow (1994), Di Felice (1994) and Syamlal and O'Brien (1988) by comparing the variation of axial solid velocity against dimensionless radius position. Table 3.4 depicts the influence of drag force models by comparing the bed expansion and solid holdup with experimental data reported by Limtrakul et al. (2005).

![Figure 3.5. Influence of different drag force models on the time averaged axial solid velocity of fluidised at a superficial liquid velocity of 0.07 m/s.](image)

<table>
<thead>
<tr>
<th>Drag force Model</th>
<th>Bed Expansion</th>
<th>Solid holdup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gidaspow (1994)</td>
<td>0.59</td>
<td>0.43</td>
</tr>
<tr>
<td>Di Felice (1994)</td>
<td>0.68</td>
<td>0.36</td>
</tr>
<tr>
<td>Syamlal and O'Brien (1988)</td>
<td>0.586</td>
<td>0.43</td>
</tr>
</tbody>
</table>

Table 3.4. Comparison of bed expansion and solid holdup prediction from different drag force models and experimental data
**CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed**

Even though the models proposed by Syamlal and O'Brien and Gidaspow match closely with the experimental data of Limtrakul et al. (2005) (average error of 0.2–0.7% for solid holdup), the drag model proposed by Syamlal and O'Brien overpredicts the axial solid velocity profiles. Based on these observations the Gidaspow drag model was used in the present study.

### 3.4.5. Effect of inlet feed condition

The effect of two types of inlet velocity profiles \( V_{in} = V_{max} \left(1 - r/R\right)^{1/7} \), uniform velocity profile) of liquid feed was evaluated with the experimental results in the present study. Table 3.5 presents the effect of different inlet conditions on bed expansion and solid holdup. The fully developed inlet profile gives lower bed expansion and higher solid holdup than the velocity profiles assuming uniform velocity as shown in Table 3.5.

<table>
<thead>
<tr>
<th>Type of feed inlet conditions</th>
<th>Bed Expansion</th>
<th>Solid holdup</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experimental</td>
<td>CFD</td>
</tr>
<tr>
<td>Fully developed velocity profile</td>
<td>0.59</td>
<td>0.586</td>
</tr>
<tr>
<td>Uniform velocity profile</td>
<td>0.59</td>
<td>0.586</td>
</tr>
</tbody>
</table>

Table 3.5. Comparison of bed expansion and solid holdup on the type of velocity profiles at the inlet

Table 3.6 gives the CFD model parameters used in the numerical investigation.
Table 3.6. Parameters employed in the CFD simulation

<table>
<thead>
<tr>
<th>Description</th>
<th>Method used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mode of simulation</td>
<td>3D</td>
</tr>
<tr>
<td>Grid size</td>
<td>25000 nodes</td>
</tr>
<tr>
<td>Time step</td>
<td>0.005 s</td>
</tr>
<tr>
<td>Drag model</td>
<td>Gidaspow Model</td>
</tr>
<tr>
<td>Inlet boundary</td>
<td>Uniform inlet velocity</td>
</tr>
</tbody>
</table>

3.4.6. Comparison of solid holdup between experimental and CFD results

Figure 3.6 shows the time averaged solid holdup as a function of dimensionless radial position along with the experimental results reported by Limtrakul et al. (2005). The solid holdup is defined as the volume fraction of the solid phase in the liquid–solid mixture. The solid holdup profile predicted by the CFD simulation matches closely with experimental data at the center of the column and varies at the wall region of the column with an average error of 2.6 %. The enhanced deviation at the wall may be due to wall effects which have not been explicitly considered in the present study. Table 3.7 shows the averaged solid holdup obtained by the experimental and the CFD simulation at various operating conditions. It is observed that the solid holdup obtained from the CFD simulation is able to predict the experimental results reported by Limtrakul et al. (2005) with an average error of 2–14 %.
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

Figure 3.6. Azimuthally averaged solid holdup profile obtained by CT scan and CFD simulation, 0.14 m diameter column, 0.003 m glass beads $U_l=0.07$ m/s

Table 3.7. Experimental validation of average solid holdup predicted by the CFD

<table>
<thead>
<tr>
<th>Column size (m)</th>
<th>Superficial Liquid velocity (m/s)</th>
<th>Solid particle</th>
<th>Holdup from Experimental Data (Limtrakul et al., 2005)</th>
<th>Holdup from the present CFD simulation</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.14</td>
<td>0.07</td>
<td>Glass beads (3mm)</td>
<td>0.44</td>
<td>0.42</td>
<td>+4.5</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Glass beads (1mm)</td>
<td>0.51</td>
<td>0.48</td>
<td>+5.9</td>
</tr>
<tr>
<td>0.13</td>
<td>0.1</td>
<td>Glass beads (3mm)</td>
<td>0.35</td>
<td>0.3</td>
<td>+14.3</td>
</tr>
<tr>
<td>0.1</td>
<td>0.065</td>
<td>Glass beads (3mm)</td>
<td>0.25</td>
<td>0.255</td>
<td>-2.0</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Glass beads (3mm)</td>
<td>0.48</td>
<td>0.43</td>
<td>+10.4</td>
</tr>
</tbody>
</table>

3.4.7. Solid motion in liquid fluidised bed

Experimental studies of solid motion reported by Limtrakul et al. (2005) show that multiple solids cell circulations patterns exist for all conditions of liquid fluidised
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

bed operations. However CFD simulation exhibits only a single solid circulation cell which is also in agreement with the observations of Roy et al. (2005) in a liquid–solid riser. Figure 3.7 shows the vector plot of time averaged solid velocity on the different planes at typical operating conditions ($U_i=0.07$ m/s) for glass beads. The existence of a single recirculation cell with solids ascending along the column at the center and descending along the wall is evident from the simulation results. CFD simulation of axial solid velocity at various dimensionless radial positions is depicted in Figure 3.8. The agreement between the experimental and simulation results is quite satisfactory.

![Figure 3.7. Typical time averaged azimuthally averaged axial solid velocity profile](image)

**Figure 3.7.** Typical time averaged azimuthally averaged axial solid velocity profile

### 3.4.8. Effect of Column Diameter

In this work, two columns of 0.1 m and 0.14 m in diameter are used to study the effect of column diameter. The simulation results of the effect of column diameter
on axial solid velocities are compared with the experimental results in Figure 3.9 and it shows that the axial solid velocities increase with increase in column diameter, at superficial liquid velocity of 0.07 m/s.

**Figure 3.8.** Axial solid velocity profiles as a function of radial position at a superficial velocity of 0.07 m/s

**Figure 3.9.** Effect of column size for 0.003 m glass beads at $U_l = 0.07$ m/s
3.4.9. Effect of Particle size and density

Acetate beads ($\rho_s = 1300 \text{ kg/m}^3$) and glass beads ($\rho_s = 2500 \text{ kg/m}^3$) with particle sizes, 0.001m and 0.003 m were used to study the effect of particle size and density. Figure 3.10 (a, b) shows that the axial solid velocities increase with increase in particle diameter and density leading to larger inversion point (the point at which axial solid velocity is zero) for both CFD simulation and the experimental results reported by Limtrakul et al. (2005). Table 3.8 depicts the comparison of the inversion points for different operating conditions. The smaller size particle of 1 mm glass beads has a smaller value of inversion point compared to that of glass beads of 3 mm size. Song and Fan (1986) mentioned that due to higher value of apparent viscosity of slurry, the inversion point is reduced for systems with particles having smaller sizes.
Figure 3.10. (a) Effect of particle type ($U_l$ for glass beads =0.007 m/s, $U_l$ for acetate=0.024 m/s) and (b) Effect of particle size ($U_l$ for 3 mm =0.007 m/s, $U_l$ for 1mm =0.024 m/s) on axial solid velocity

Table 3.8. Comparison of inversion points for different operating conditions

<table>
<thead>
<tr>
<th>Column Diameter</th>
<th>Solid properties</th>
<th>Inversion Points</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Experimental</td>
</tr>
<tr>
<td>0.14 m Glass beads (2500 kg/m³,3mm)</td>
<td>0.72</td>
<td>0.77</td>
</tr>
<tr>
<td>0.14 m Glass beads (2900 kg/m³,1mm)</td>
<td>0.62</td>
<td>0.69</td>
</tr>
<tr>
<td>0.14 m Acetate beads (1300 kg/m³,3mm)</td>
<td>-</td>
<td>0.64</td>
</tr>
<tr>
<td>0.1m Glass beads (2500 kg/m³,3mm)</td>
<td>-</td>
<td>0.72</td>
</tr>
</tbody>
</table>

3.4.10. Effect of liquid superficial velocity

The increase in superficial liquid velocity increases the energy input to the system, leading to enhanced bed expansion and solid motion. Figure 3.11 shows the
effect of liquid superficial velocity on the time averaged axial solid velocity. The CFD predictions of axial solid velocity give the same pattern as that obtained from the experimental data.

![Figure 3.11](image)

**Figure 3.11.** Effect of superficial liquid velocity on time averaged axial solid velocity

### 3.4.11. Turbulence parameters

To further validate the CFD simulation results, a comparison of the turbulence parameters viz., turbulence intensities, and shear stress profiles with the experimental data provided by Limtrakul et al. (2005) was made. Figure 3.12 shows the root-mean-square (rms) axial ($u_r'$) and radial ($u_r'$) velocities of solids along the radial position. Figures 3.12a and b show that the axial RMS velocities are roughly twice those of the corresponding radial components. Similar to the observations made by Devanathan et al. (1999) in gas–liquid bubble columns systems and Roy et al. (2005) in liquid–solid
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

A typical comparison of the experimental and the simulation results is depicted in Figures 3.12 and 3.13.

Figure 3.12. (a) Variation of radial rms velocities along the radial position
(b) Variation of axial rms velocities along the radial position
3.4.12. Computation of solids mass balance

Based on the validation of CFD model predictions discussed earlier, a mass balance of solids in the center and wall region was computed to verify the conservation of solid mass in the liquid–solid fluidised bed i.e. the net solid volume flow rate in center region should equal the net solid volume flow rate in the wall region represented mathematically as

Solid upflow rate in the core region = \[ \int_{0}^{R_i} 2\pi r \varepsilon_i(r) V_z(r) \, dr \]  \hspace{1cm} \text{(3.25)}

Solid downflow rate in the annular region = \[ 2\pi \int_{R_i}^{R} r \varepsilon_i(r) V_z(r) \, dr \]  \hspace{1cm} \text{(3.26)}

where \( \varepsilon_i(r) \) is the time averaged radial solid holdup profile and \( V_z(r) \) is the time averaged axial solid velocity and \( R_i \) is the radius of inversion, defined as the point at which the axial solids velocity is zero. The radial solid holdup profile at each of the operating conditions proposed by Roy et al. (2005) is given by

\[ \varepsilon_i(r) = \frac{m+2}{m+2+2C} \left[ 1 + C \left( \frac{r}{R} \right)^m \right] \]  \hspace{1cm} \text{(3.27)}

Similarly an expression that has been observed to describe the radial profile of the
axial solids velocity (Roy et al., 2005) is

\[ V_z(r) = V_z(0) + \alpha_1 \left( \frac{r}{R} \right)^n - \alpha_2 \left( \frac{r}{R} \right)^{n\alpha_1/\alpha_2} \]  

\[ \text{...............(3.28)} \]

In equation 3.28, \( V_z(0) \) is the centerline axial solids velocity, and \( \alpha_1 \) and \( \alpha_2 \) are empirical constants determined through curve fitting. The exponent \( n \) defines the curvature of the velocity profile.

The net volumetric solid flow rates computed from equations (3.26) and (3.27) are shown in Table 3.9. The relative deviation of volumetric solid flow between core and wall region is observed in the range of 10–15%. This finding may be compared with observation of Kiared et al. (1997) who investigated the net solid flow rate in the center and wall region and obtained the relative deviation for volumetric mass rate in the range of 23–27%.

<table>
<thead>
<tr>
<th>Column Size (m)</th>
<th>Liquid superficial velocity (m/s)</th>
<th>Solid Particle</th>
<th>Volumetric flow rate of solid in center (m³/s)</th>
<th>Volumetric flow rate of solid in wall (m³/s)</th>
<th>Relative deviations (in %)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.14</td>
<td>0.07 Glass beads (3mm)</td>
<td>1.614E-05</td>
<td>1.86E-05</td>
<td>13.1</td>
<td></td>
</tr>
<tr>
<td>0.07</td>
<td>Glass beads (1mm)</td>
<td>1.236E-05</td>
<td>1.506E-05</td>
<td>17.9</td>
<td></td>
</tr>
<tr>
<td>0.1</td>
<td>Glass beads (3mm)</td>
<td>8.303E-06</td>
<td>8.563E-06</td>
<td>3.0</td>
<td></td>
</tr>
<tr>
<td>0.13</td>
<td>Glass beads (3mm)</td>
<td>6.3507E-06</td>
<td>5.629E-06</td>
<td>12.8</td>
<td></td>
</tr>
<tr>
<td>0.024</td>
<td>Acetate beads (3mm)</td>
<td>5.3572E-06</td>
<td>5.339E-06</td>
<td>0.03</td>
<td></td>
</tr>
</tbody>
</table>

**3.4.13. Computation of various energy flows**

It is informative to investigate the various energy flows into the two-phase fluidised bed and make an order-of-magnitude estimate of the various terms in the
energy flows. Extensive work has been carried out by Joshi (2001) to understand the energy transfer mechanism in gas–liquid flows in bubble column reactors. A similar attempt is made in this work. In liquid–solid fluidised beds, the input energy from liquid is distributed to the mean flow of the liquid and the solid phases. Also, a part of the input energy is used for liquid phase turbulence and some part of the energy gets dissipated due to the friction between the liquid and solid phases. Apart from these energy dissipation factors, some of the other energy losses due to solid fluctuations, collisions between particles, between particles and column wall are also involved in two-phase reactors. Since the present CFD simulation is based on Eulerian–Eulerian approach, these modes of energy dissipation could not be quantified. Hence, these terms are neglected in the energy calculation.

In general, the difference between the input and output energy should account for the energy dissipated in the system. Thus, the energy difference in this work is calculated as

\[ \text{Energy difference} = \text{Energy entering the fluidised bed (E}_i\text{) } - \text{Energy leaving the fluidised bed by the liquid (E}_{\text{out}}\text{) } - \text{Energy gained by the solid phase (E}_T\text{) } - \text{Energy dissipated by the liquid phase turbulence (E}_e\text{) } - \text{Energy dissipated due to friction at the liquid–solid interface (E}_{\text{Bis}}\text{)} \]

\[ \text{Energy difference} = \text{Energy entering the fluidised bed (E}_i\text{) } - \text{Energy leaving the fluidised bed by the liquid (E}_{\text{out}}\text{) } - \text{Energy gained by the solid phase (E}_T\text{) } - \text{Energy dissipated by the liquid phase turbulence (E}_e\text{) } - \text{Energy dissipated due to friction at the liquid–solid interface (E}_{\text{Bis}}\text{)} \]

\[ \text{Energy entering the fluidised bed (E}_i\text{) by the incoming liquid and gas} \]

The energy entering the fluidised bed due to the incoming liquid and gas flow is given by

\[ E_i = \frac{\pi}{4} D^2 h g V_1 (\varepsilon_s \rho_s + \varepsilon_l \rho_l) \]

where \( D \) is the diameter of the column, \( H \) is the expanded bed height, \( V_1 \) is superficial
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

Liquid velocity, $\varepsilon_l, \varepsilon_s$ are the liquid and solid volume fractions respectively and $\rho_l, \rho_s$ are the liquid and solid densities respectively.

Energy leaving the fluidised bed ($E_{out}$) by the outflowing liquid

The liquid leaving the bed possess both potential energy and kinetic energy by virtue of its expanded bed height and are given as

$$E_{pl} = \frac{\pi}{4}D^2HV_l\rho_l g$$ ................................(3.31)

$$E_{ul} = \frac{1}{2}\rho_l \frac{\pi}{4}D^2V_i^3$$ ................................(3.32)

$$E_{out} = E_{pl} + E_{kl}$$ ..................................(3.33)

Energy gained by the solid phase ($E_T$)

The solid flow pattern in the present study shows a single circulation pattern, as depicted in Figure 3.7. The energy gained by the solids for its upward motion in the center region is the sum of the potential energy and kinetic energy of the solids in the center region and are given by

$$E_{ps} = \rho_s gH \frac{\pi}{4}D_c^2v_s$$ ..................................(3.34)

$$E_{ks} = \frac{1}{2}\rho_s \frac{\pi}{4}D_c^2v_s^3$$ ..................................(3.35)

$$E_T = E_{ps} + E_{ks}$$ ..................................(3.36)

where $v_s$ is the time averaged solid velocity in the center region, and $D_c$ is the diameter of the center region.

Energy dissipation due to liquid phase turbulence ($E_g$)

Since $k-\varepsilon$ model for turbulence is used in this work, the energy dissipation rate per unit mass is given by the radial and axial variation of $\varepsilon$. Hence, the energy dissipated due to liquid phase turbulence is calculated as
Energy dissipation at the liquid–solid interface ($E_{Bls}$)

The net rate of energy dissipated between liquid–solid phases is calculated based on the drag force and slip velocity between liquid and solid and is summed over all the particles.

For a single particle at an infinite expanded state ($\varepsilon = 1$), the interaction can be represented as the sum of drag and buoyancy forces. Hence, the force balance for a single particle is

$$mg = \text{drag} + \text{buoyancy}$$

$$\frac{\pi}{6} d_p^3 (\rho_s - \rho_l) = C_d \frac{\pi}{4} d_r^2 (U_l - U_s) |U_l - U_s| \frac{\rho_l}{2} \quad \ldots \ldots (3.38)$$

For multiple particles, the above equation can be written as

$$\frac{\pi}{6} d_p^3 (\rho_s - \rho_l) f(\varepsilon) = C_d \frac{\pi}{4} d_p^2 (U_l - U_s) |U_l - U_s| \frac{\rho_l}{2} \quad \ldots \ldots (3.39)$$

Lewis et al. (1952), Wen and Yu (1966); and Kmiec (1981) presented the above equation in the form of

$$\frac{\pi}{6} d_p^3 (\rho_s - \rho_l) \varepsilon^n = C_d \frac{\pi}{4} d_p^2 (U_l - U_s) |U_l - U_s| \frac{\rho_l}{2} \quad \ldots \ldots (3.40)$$

where $n = 4.65$ (Lewis et al.), $n = 4.7$ (Wen and Yu) and $n = 4.78$ (Kmiec).

Yang and Renken (2003) developed an equilibrium force model for liquid–solid fluidised bed and derived an empirical correlation for equilibrium between forces to account for laminar, turbulent and intermediate region as given by

$$C_d \frac{\pi}{4} d_p^2 (U_l - U_s) |U_l - U_s| \frac{\rho_l}{2} = \frac{\pi}{6} d_p^3 (\rho_s - \rho_l) \left(a \varepsilon^{4.78} + (1 - a) \varepsilon^{2.78}\right) \quad \ldots \ldots (3.41)$$

$$a = 0.7418 + 0.9674 Ar^{-0.5} \quad 1 < Re_t < 50, \quad 24 < Ar < 3000$$
CFD Simulation of Hydrodynamics of Liquid-Solid Fluidised Bed

\[ a = 0.7880 - 0.00009 \text{Ar}^{0.625} \quad \text{50} < \text{Re}_t < 500, \quad 3000 < \text{Ar} < 10^5 \]

The total drag force is thus equal to the product of drag force for single particle and multiplied by the total number of particles namely,

\[
F_T = \frac{\pi}{4} D^2 H \varepsilon_s g(\rho_s - \rho_l) \left(a \varepsilon^{4.78} + (1 - a) \varepsilon^{2.78} \right) \quad \text{…………………..(3.42)}
\]

The rate of energy transferred to the solid from liquid motion is computed from equations (3.41) and (3.42) as

\[
E_B = \frac{\pi}{4} D^2 H \varepsilon_s g(\rho_s - \rho_l) \left(a \varepsilon^{4.78} + (1 - a) \varepsilon^{2.78} \right) V_s \quad \text{…………………..(3.43)}
\]

where \( V_s \) is the slip velocity.

The values calculated for these terms along with the energy difference (in terms of %) are presented in Table 3.10. It can be observed that energy difference is in the range of 2–9% for various operating conditions. This can be attributed to the fact that the energy losses due to particle–particle collisions and particle–wall collisions are not included in this present energy calculation. It can also be seen from Table 3.10 that the energy required for solid motion is more around 70–80% of total energy dissipation of fluidised bed.
3.5. Conclusions

CFD simulation of hydrodynamics and solid motion in liquid fluidised bed were carried out by employing the multi-fluid Eulerian approach. Adequate agreement was demonstrated between the CFD simulation results and the experimental findings reported by Limtrakul et al. (2005) using non invasive techniques to measure solid holdup, solid motion and turbulence parameters. The predicted flow pattern demonstrates that the time averaged solid velocity profile exhibits axisymmetric with downward velocity at the wall and maximum upward velocity at the center of the column and higher value of solid holdup at the wall and lower value of that at the center. The CFD simulation exhibits a single solid circulation cell for all the operating conditions, which is consistent with the observations reported by various authors. Based on the predicted flow field by CFD model, the focus has been on the computation of the solid mass balance and computation of various energy flows in fluidised bed reactors. The result obtained shows a deviation in the range of 10–15% between center and wall region for solid flow balance calculations. In the computation of energy flows, the energy difference observed is in the range of 2–9%.

In the present study, the influence of various interphase drag models on solid motion in liquid fluidised bed was studied. The drag models proposed by Gidaspow (1994), Syamlal and O’Brien (1994), and Di Felice (1988) can qualitatively predict the flow pattern of solid motion inside the fluidised bed, in which the model proposed by Gidaspow gives the best agreement with experimental data.
Table 3.10. Various energy flows in the liquid fluidised bed

<table>
<thead>
<tr>
<th>Column Size (m)</th>
<th>U₁ (m/s)</th>
<th>Solid type</th>
<th>Eᵢ  (Eqn.3.30)</th>
<th>Eᵢout (Eqn.3.33)</th>
<th>EᵢT  (Eqn.3.36)</th>
<th>Eᵦ  (Eqn.3.37)</th>
<th>EᵦB  (Eqn.3.43)</th>
<th>Difference (in %)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.14</td>
<td>0.07</td>
<td>Glass beads (3mm)</td>
<td>10.05</td>
<td>6.11</td>
<td>2.66</td>
<td>0.13</td>
<td>0.44</td>
<td>7.06</td>
</tr>
<tr>
<td></td>
<td>0.07</td>
<td>Glass beads (1mm)</td>
<td>8.58</td>
<td>4.32</td>
<td>3.34</td>
<td>0.05</td>
<td>0.07</td>
<td>9.3</td>
</tr>
<tr>
<td></td>
<td>0.1</td>
<td>Glass beads (3mm)</td>
<td>18.35</td>
<td>12.65</td>
<td>3.76</td>
<td>0.2</td>
<td>1.18</td>
<td>3.05</td>
</tr>
<tr>
<td></td>
<td>0.13</td>
<td>Glass beads (3mm)</td>
<td>27.09</td>
<td>19.59</td>
<td>3.73</td>
<td>0.36</td>
<td>1.95</td>
<td>5.4</td>
</tr>
<tr>
<td></td>
<td>0.024</td>
<td>Acetate beads (3mm)</td>
<td>1.97</td>
<td>1.72</td>
<td>0.18</td>
<td>4e-04</td>
<td>0.02</td>
<td>2.3</td>
</tr>
<tr>
<td>0.1</td>
<td>0.07</td>
<td>Glass beads (3mm)</td>
<td>10.16</td>
<td>6.18</td>
<td>2.52</td>
<td>0.14</td>
<td>0.48</td>
<td>8.26</td>
</tr>
</tbody>
</table>
To identify the CFD methodology to enhance the accuracy of numerical simulation comparison between 2D and 3D simulation, the effect of grid sensitivity, time step sensitivity and effect of inlet feed conditions were investigated and a comprehensive CFD methodology was established to model the liquid–solid fluidised bed.