Fluid Dynamics of Gas – Solid Fluidized Beds

Germán González Silva¹, Natalia Prieto Jiménez¹ and Oscar Fabio Salazar
State University of Campinas
Brazil

1. Introduction

Fluidization refers to the contact between a bed of solids and a flow of fluid. As a result, the solid particles are transformed into a fluid-like behavior that can be used for different purposes. The fluidized bed reactor is one of the most important technologies for gas-solid heterogeneous operations chemical or petrochemical, considering catalytic or non-catalytic processes (Kunii and Levenspiel 1991). The most important industrial applications include catalytic cracking, coal combustion and biomass combustion. One of the most relevant type of fluidized bed reactor is the ascendant flow reactor, which is also known as riser. The riser reactors consist of a tubular column in which both solid and gas flow upwards. The first fluidized bed gas generator was developed in Germany by Fritz Winkler in the 1920s. Later in the 1930s, the american petroleum industry started developing the fluidized bed technology for oil feedstock catalytic cracking, becoming the primary technology for such applications (Tavoulareas 1991).

Inside the riser reactor, solid particles have a wide range of residence time, which is a disadvantage that reduces the overall conversion and the selectivity of the chemical reactions. For that reason it has recently grown the interest in a new type of gas-solid circulating reactor known as downer. In this reactor the gas and the solid flow cocurrently downward, creating hydrodynamic features comparable to a plug flow reactor and allowing a better control over the conversion, the selectivity and the catalyst deactivation. The concept of downer reactor gas-solid appeared in the 1980s, with the first studies on the fluid dynamics of gas-solid suspensions (Kim and Seader 1983) and with the first downer reactors for patents developed by Texaco for the FCC process (Gross Benjamin and Ramage Michael P 1981; Niccum Phillip K and Bunn Jr Dorrance P 1983). In these studies it is observed that in the downer reactor has a uniform distribution of two-phase flow along the reactor, also observed that the contact time is very low, achieving a 20% decrease in the amounts of coke produced during the FCC process. Applications, differences, advantages and disadvantages to these types of fluidized bed reactors can be found in various publications (Ancheyta 2010; Gonzalez, 2008; Yi Cheng et al. 2008; Crowe 2005; Wen-ching Yang 2003; Grace 1997; Gidaspow 1994; Geldart 1986)

2. Fluidization regimes and particle classification

Fluidization occurs when a gas or liquid is forced to flow vertically through a bed of particles at such a rate that the buoyed weight of the particles is completely supported by the drag force imposed by the fluid.
2.1 Flow regimes in fluidized beds
As the superficial gas velocity, $U$, is increased stepwise beyond the minimum fluidization velocity, it is observed different types of flow regimes. The principal ones are schematically shown in Figure 1. The flow regimes are listed by increasing value of $U$ as follows:
- Bubble-free bed expansion
- Bubbling fluidization
- Slug flow
- Turbulent fluidization
- Fast fluidization and dense suspension upflow

Fig. 1. Flow regimes of gas–solid fluidization.

The bubbling regime is one of the most studied flow regimes in gas-solid fluidization. Bubbles coalesce and break-up as fluid flow is increased. Finally, the bubbles become large enough to occupy a substantial fraction of the cross-section of the small diameter columns (Vejahati 2006). These large bubbles are called slug, as shown in the third column of Figure 1.

2.2 Particle classification
The behavior of solids fluidized by gases fall into four clearly recognizable groups, characterized by density difference $(\rho_s - \rho_f)$ and mean particle size. The features of the groups are: powders in group A exhibit dense phase expansion after minimum fluidization and prior to the commencement of bubbling; those in group B bubble at the minimum fluidization velocity; those in group C are difficult to fluidize at all and those in group D can
form stable spouted beds (Geldart 1973). Desirable properties of particles and gas for fluidized bed are delineated in Table 1.

<table>
<thead>
<tr>
<th>Property</th>
<th>Desirable Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean diameter</td>
<td>50 μm to 1.6 mm</td>
</tr>
<tr>
<td>Size distribution</td>
<td>Neither too narrow or too broad, e.g., 90&lt;sup&gt;th&lt;/sup&gt; to 10&lt;sup&gt;th&lt;/sup&gt; decile ratio 5 to 25</td>
</tr>
<tr>
<td>Density</td>
<td>Wide range of values possible, but uniform from particle to particle</td>
</tr>
<tr>
<td>Shape</td>
<td>Rounded and with length to thickness ration no larger than ~3</td>
</tr>
<tr>
<td>Surface roughness</td>
<td>Smooth</td>
</tr>
<tr>
<td>Surface stickiness</td>
<td>Avoid sticky surfaces</td>
</tr>
<tr>
<td>Attrition resistance</td>
<td>Usually strong as possible</td>
</tr>
<tr>
<td>Hardness</td>
<td>Avoid resilience, but also excessive hardness</td>
</tr>
<tr>
<td>Density</td>
<td>No restriction, but higher value improves properties</td>
</tr>
<tr>
<td>Viscosity</td>
<td>No restriction</td>
</tr>
<tr>
<td>Relative humidity</td>
<td>Typically 10 to 90%</td>
</tr>
</tbody>
</table>

Table 1. Desirable properties of particles and gases for Gas-Solid fluidization (Jesse Zhu et al. 2005)

3. Experimental measurement techniques

For better understanding of these phenomena and to facilitate the solution of mathematical models is necessary to make an analysis of experimental data. This experimental analysis requires specialized measurement techniques are able to explain the flow field must also be automated to minimize human involvement in the process of collecting data. The measurement techniques, to capture the important fluids dynamic behavior of the two-phase flow, can be classified as non-intrusive (NMT) and intrusive (IMT) techniques. The intrusive techniques are generally probes used to study local basic flow phenomena. Some of these are intended only as research instruments. The most common parameters that are measured with such probes are solids mass flows, radial and axial solids concentration, solids velocities, and distribution.

The particles can be deposited in the measuring device reducing its performance or causing malfunction. Besides this, the flow area reduction makes of the intrusive devices not the best solution. Non-intrusive techniques to characterize the flow within a fluidized bed are more desirable because it does not disturb the flow behavior. In the Table 2 and Table 3 classification techniques are included and recent successes have been achieved.
<table>
<thead>
<tr>
<th>NMT</th>
<th>Ref for more details</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laser Doppler Anemometry (LDA)</td>
<td>LDA is a technology used to measure velocities of small particles in flows. The technique is based on the measurement of laser light scattered by particles that pass through a series of interference fringes (a pattern of light and dark surfaces). The scattered laser light oscillates with a specific frequency that is related to the velocity of the particles.</td>
</tr>
<tr>
<td>X-ray</td>
<td>Radiographic techniques based either based on electromagnetic radiation such as X and y rays. The transmission of X-rays or γ-rays through a heterogeneous medium is accompanied by attenuation of the incident radiation, and the measurement of this attenuation provides a measure of the line integral of the local mass density distribution along the path traversed by the beam.</td>
</tr>
<tr>
<td>γ-ray</td>
<td></td>
</tr>
<tr>
<td>Radioactive Particle Tracking (RPT)</td>
<td>Technique to measure velocity field and turbulent parameters of multiphase flow. This is based on the principle of tracking the motion of a single tracer particle as a marker of the solids phase. The tracer particle contains a radioactive element emitting γ-rays. This radiation is received by an ensemble of specific detector.</td>
</tr>
<tr>
<td>Particle Image Velocimetry (PIV)</td>
<td>PIV measures whole velocity fields by taking two images shortly after each other and calculating the distance individual particles travelled within this time. The displacement of the particle images is measured in the plane of the image and used to determine the displacement of the particles.</td>
</tr>
</tbody>
</table>

Table 2. Non-intrusive measurement techniques.
4. **Computational fluid dynamics (CFD)**

Computational Fluid Dynamics (CFD) is a technique which uses conservation principles and rigorous equations of fluid flow (Navier-Stokes) along with specialized turbulence models (k-ε, k-ω, SST among others). These models are more accurate and fundamentally more acceptable than empirical ones. The empirical models are approximations that assemble different phenomena to remove a number of unknown parameters. For this reason, these models are not reliable and therefore should not be generalized.

The CFD models can be divided into two groups: the *Eulerian-Eulerian* model in which the gas and solid phases are considered as two interpenetrating continuum flows; and the *Eulerian-Lagrangian* model that consider the gas as a fluid phase and the solids as discrete phase. The *Eulerian-Lagrangian* model calculates the trajectory of each individual particle using Newton’s second law. The interaction between particles can be described by the potential energy or the dynamic of collisions. This method has the advantage of knowing exactly the particle trajectory and the system variables. However, this requires high computational effort, higher yet when gas and solid velocity fields are coupled.

### 4.1 Governing equations

Governing equations for *Eulerian-Eulerian* model are here presented in tensor notation.

#### 4.1.1 Continuity equations

The gas and solid continuity equations are represented by:

\[
\frac{\partial}{\partial t}\left(\alpha_g \rho_g \right) + \nabla \cdot \left(\alpha_g \rho_g \vec{v}_g \right) = 0
\]  

(1)
\[ \frac{\partial}{\partial t} \left( \alpha_s \rho_s \right) + \nabla \cdot \left( \alpha_s \rho_s \vec{v}_s \right) = 0 \]  

(2)

Where \( \alpha, \rho \) and \( \vec{v} \) are volume fraction, density and the vector velocity, respectively. No mass transfer is allowed between phases.

### 4.1.2 Momentum equations

The gas phase momentum equation may be expressed as:

\[ \frac{\partial}{\partial t} \left( \alpha_s \rho_s \vec{v}_s \right) + \nabla \cdot \left( \alpha_s \rho_s \vec{v}_s \vec{v}_s \right) = -\alpha_s \nabla p + \nabla \cdot \left[ \tau_s \right] + \alpha_s \rho_s \vec{g} + \beta \left( \vec{v}_s - \vec{v}_g \right) \]  

(3)

\( p \) and \( \vec{g} \) are fluid pressure and gravity acceleration. \( \beta \) is the drag coefficient between the phases \( g \) and \( s \). The stress tensor is given by:

\[ \tau_s = \alpha_s \mu_s \left[ \nabla \vec{v}_s + \left( \nabla \vec{v}_s \right)^T \right] - \frac{2}{3} \alpha_s \mu_s \nabla \vec{v}_s \]  

(4)

The solid phase momentum equation may be written as:

\[ \frac{\partial}{\partial t} \left( \alpha_s \rho_s \vec{v}_s \right) + \nabla \cdot \left( \alpha_s \rho_s \vec{v}_s \vec{v}_s \right) = -\alpha_g \nabla \alpha_s + \nabla \cdot \left[ \tau_s \right] + \alpha_s \rho_s \vec{g} + \beta \left( \vec{v}_s - \vec{v}_s \right) \]  

(5)

\[ \tau_s = \alpha_s \mu_s \left[ \nabla \vec{v}_s + \left( \nabla \vec{v}_s \right)^T \right] - \frac{2}{3} \alpha_s \mu_s \nabla \vec{v}_s \]  

(6)

\( G \) is the modulus of elasticity given by:

\[ G = \exp \left[ C_G \left( \alpha_s - \alpha_{s,\text{max}} \right) \right] \]  

(7)

Where \( \alpha_{s,\text{max}} \) is the maximum solid volume fraction and \( \beta \) is the interface momentum transfer proposed by Gidaspow, (1994):

\[
\left\{ \begin{array}{ll}
\beta = 150 \frac{\alpha_s \left( 1 - \alpha_s \right) \mu_s}{\alpha_s d_p^2} + 1.75 \frac{\alpha_s \rho_s \left| \vec{v}_s - \vec{v}_s \right|}{d_p} & |\alpha_s| \leq 0.8 \\
\beta = \frac{3}{4} \frac{C_D \alpha_s \rho_s \left| \vec{v}_s - \vec{v}_s \right|}{d_p} \alpha_s^{-2.65} & |\alpha_s| > 0.8
\end{array} \right.
\]  

(8)

Where \( d_p \) and \( C_D \) are the particle diameter and the drag coefficient, based in the relative Reynolds number \( (Re_s) \):

\[ C_D = \left\{ \begin{array}{ll}
24 \left( 1 + 0.15 Re_s^{0.687} \right) & |Re_s| \leq 1000 \\
0.44 & |Re_s| > 1000
\end{array} \right. \]  

(9)

\[ Re_s = \frac{\rho_s \left| \vec{v}_s - \vec{v}_s \right|}{\mu_s} \]  

(10)
4.1.3 Energy equation
The gas and solid energy equations can be written as:

\[
\frac{\partial}{\partial t} \left( \alpha_g \rho_g H_g \right) + \nabla \cdot \left( \alpha_g \rho_g \vec{v}_g H_g \right) = \nabla \cdot \left( \alpha_g \lambda_g \nabla T_g \right) + \gamma \left( T_g - T_s \right) + \alpha_s \rho_s \sum_r \Delta H_r \frac{\partial C_r}{\partial t} \quad (11)
\]

\[
\frac{\partial}{\partial t} \left( \alpha_s \rho_s H_s \right) + \nabla \cdot \left( \alpha_s \rho_s \vec{v}_s H_s \right) = \nabla \cdot \left( \alpha_s \lambda_s \nabla T_s \right) + \gamma \left( T_s - T_g \right) \quad (12)
\]

Where
\[H = \text{Specific enthalpy}\]
\[T = \text{Temperature}\]
\[\gamma = \text{Interface heat transfer coefficient: } \gamma = Nu \lambda / d_p\]
\[\lambda = \text{Thermal conductivity}\]

4.2 Turbulence models
Turbulence is that state of fluid motion which is characterized by random and chaotic three-dimensional vorticity. When turbulence is present, it usually dominates all other flow phenomena and results in increased energy dissipation, mixing, heat transfer, and drag. The physical turbulence models provide the solution the closure problem in solving Navier–Stokes equations. While there are ten unknown variables (mean pressure, three velocity components, and six Reynolds stress components), there are only four equations (mass balance equation and three velocity component momentum balance equations). This disparity in number between unknowns and equations make a direct solution of any turbulent flow problem impossible in this formulation. The fundamental problem of turbulence modeling is to relate the six Reynolds stress components to the mean flow quantities and their gradients in some physically plausible manner.

The turbulence models are summarized in Table 4

<table>
<thead>
<tr>
<th>Family group</th>
<th>Models</th>
<th>Description and advantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds–Averaged Navier–Stokes (RANS)</td>
<td>Zero equation models</td>
<td>Provides good predictions for all types of flows, including swirl, and separation. Longer calculation times than the RANS models.</td>
</tr>
<tr>
<td></td>
<td>One equation models</td>
<td>Provides excellent results for all flow systems. LES solves the Navier-Stokes equations for large scale motions of the flow models only the small scale motions.</td>
</tr>
<tr>
<td></td>
<td>Two equation models</td>
<td>Provides good predictions for all types of flows, including swirl, and separation. Longer calculation times than the RANS models.</td>
</tr>
<tr>
<td></td>
<td>( \kappa - \varepsilon )</td>
<td></td>
</tr>
<tr>
<td></td>
<td>( \kappa - \omega )</td>
<td></td>
</tr>
<tr>
<td>Reynolds Stress Model (RSM)</td>
<td>Smagorinsky-Lilly model</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Dynamic subgrid-scale model</td>
<td></td>
</tr>
<tr>
<td></td>
<td>RNG – LES model</td>
<td></td>
</tr>
<tr>
<td></td>
<td>WALLE model</td>
<td></td>
</tr>
</tbody>
</table>
Family group | Models | Description and advantages
--- | --- | ---
| Detached Eddy Simulation (DES) | The difficulties associated with the use of the standard LES models, has lead to the development of hybrid models (like that DES) that attempt to combine the best aspects of RANS and LES methodologies in a single solution strategy. |
| Direct Numerical Simulation (DNS) | The most exact approach to turbulence simulation without requiring any additional modeling beyond accepting the Navier–Stokes equations to describe the turbulent flow processes. |

Table 4. Summary of turbulence models.

### 4.3 System discretization

The most important numerical methods used to approximate the partial differential equations by a system of algebraic equations in terms of the variables at some discrete locations in space and time (called “discretization method”) are the Finite Volume (FV), the Finite Difference (FD) and the Finite Element (FE) methods. In this book, the finite volume method and the commercial software CFX® 12.0 were chosen; the solution domain is discretized in a computational mesh that can be structured or unstructured.

**Finite volume (FV) method**

The FV discretization method is obtained by integrating the transport equation around a finite volume. The general form of transport equations is given by:

\[
\frac{\partial \left( \rho \phi \right)}{\partial t} + \nabla \cdot \left( \rho \vec{v} \phi \right) = \nabla \cdot \left( \Gamma_\phi \nabla \phi \right) + S_\phi
\]

\[(13)\]

i. Transient term

ii. Convective term

iii. Diffusive term

iv. Source term

The transport equations are integrated in each computational cell using the divergence theorem over a given time interval \(\Delta t\):

\[
\int_{t}^{t+\Delta t} \left\{ \frac{\partial \left( \rho \phi \right)}{\partial t} + \nabla \cdot \left( \rho \vec{v} \phi \right) \right\} dV + \oint_{\partial V} \rho \vec{v} \phi \cdot dA = \oint_{\partial V} \Gamma_\phi \nabla \phi \cdot dA + \int_{V} S_\phi dV \]

\[(14)\]

Linearization and interpolation techniques can be clarified considering the finite volume \(P\) shown in Figure 3.

In agreement with Figure 3 notation, diffusive term can be represented as

\[
\oint_{\partial V} \Gamma_\phi \nabla \phi \cdot dA = \frac{\Gamma_\phi A_w}{h_w} \left( \phi - \phi_W \right) = D_w \left( \phi - \phi_W \right)
\]

\[(15)\]
4.4 Source term linearization
A generic source term may be written as

$$S_{\phi P}V_P = S_{C}^\phi + S_{P}^\phi \phi_P$$  \quad (16)$$

Where $S_{\phi P}$ is the value of source term in the center of the cell $P$ and $V_P$ is the volume of computational cell centered on node $P$. The method to represent $S_{\phi P}$ was suggested by Patankar, 1980.
This type of linearization is recommended since the source term decreases with increasing \( \Phi \). The source term coefficients are represented by:

\[
S^*_P = S^*_{\Phi P} + \left( \frac{dS_{\Phi P}}{d\Phi} \right) \left( \phi_P - \phi_P^* \right) \tag{17}
\]

\[S^*_C = \left[ S^*_{\Phi P} - \left( \frac{dS_{\Phi P}}{d\Phi} \right) \phi_P^* \right] V_P \tag{18}\]

\[S^*_P = \left( \frac{dS_{\Phi P}}{d\Phi} \right) \phi_P^* V_P \tag{19}\]

4.4.1 Spatial discretization

The most widely used in CFD is first and second order Upwind methods. In the first order one, quantities at cell faces are determined by assuming that the cell-center values of any field variable represent a cell-average value and hold throughout the entire cell. The face value \((\Phi_w)\) are equal to the cell-center value of \( \Phi \) in the upstream cell.

\[\int \rho \phi \phi \cdot dA = \rho v_w A_w \phi_W = C_w \phi_W \tag{20}\]

Where, \(C_w\) is the west face convective coefficient. \(A_w\) can be represented by:

\[A_w = \text{MAX}(C_w, 0) + D_w \tag{21}\]

In the second order one, quantities at cell faces are computed using a multidimensional linear reconstruction approach (Jespersen and Barth 1989). In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. Thus, the face value \(\Phi_w\) is computed using the following expression:

\[\phi_w = \frac{3}{2} \phi_W - \frac{1}{2} \phi_{WW} = \phi_W + \frac{1}{2} (\phi_W - \phi_{WW}) \tag{22}\]

The east face coefficient and matrix coefficient are shown below

\[\phi_e = \frac{3}{2} \phi_P - \frac{1}{2} \phi_W \tag{23}\]

\[A_w = \text{MAX}(C_w, 0) + \frac{1}{2} \text{MAX}(C_e, 0) + D_w \tag{24}\]

4.4.2 Temporal discretization

Temporal discretization involves the integration of every term in the differential equations over a time step \(\Delta t\). A generic expression for the time evolution of a variable \(\Phi\) is given by

\[\frac{\partial \Phi}{\partial t} = F(\Phi) \tag{25}\]
Where the function $F$ incorporates any spatial discretization. The first-order accurate temporal discretization is given by

$$\frac{\phi^{n+1} - \phi^n}{\Delta t} = F(\phi)$$  \hspace{1cm} (26)

And the second-order discretization is given by

$$\frac{3\phi^{n+1} - 4\phi^n + \phi^{n-1}}{2\Delta t} = F(\phi)$$  \hspace{1cm} (27)

5. Case studies
In order to give a better introduction with regards to the simulation of fluidized beds, in this chapter there are presented three case studies that were carried out by using a CFD software package.

The case studies were carried out using simulations in dynamic state. These simulations were set up taking into account the average value of the Courant number, which is recommended to be near 1. Besides this, it was used a constant step time, in this way was possible to have numerical stability during the execution of each of the simulations.

5.1 Cases 1 and 2
Lab scale riser reactor (Samuelsberg and B. H. Hjertager 1996; V Mathiesen 2000). Riser height, 1 m; riser diameter, 0.032 m. Experimental data and LES - Smagorinsky simulations were compared for three velocities with initial particle bed, 5cm.

5.1.1 Mesh parameters and boundary conditions
- Control volumes number: 100.000
- $\Delta x = 2$ mm
- Matrix determinant $> 0.5$ and minimum angle $> 50^\circ$

The boundary conditions for both cases are shown in Table 5 and Table 6.

In addition, tests were made with a 500.000 control volume mesh with same block distribution (the description of volume distribution in the meshes, are presented in Table 7). Obtaining similar results with the 100.000 control volume mesh. Both meshes are shown in Figure 4.

<table>
<thead>
<tr>
<th>In</th>
<th>Gas velocity = 0.36; 1.42 m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Particle mass flow equal to the output</td>
</tr>
<tr>
<td>Out</td>
<td>$Opening =$ atmospheric pressure</td>
</tr>
<tr>
<td>Wall</td>
<td>Particles = <em>free slip</em> and <em>No slip</em></td>
</tr>
<tr>
<td></td>
<td>Gas = no slip</td>
</tr>
<tr>
<td>Initial height</td>
<td>Bed height = 0,05 m</td>
</tr>
<tr>
<td>Particles</td>
<td>60 $\mu$m; 1600 kg/m3</td>
</tr>
</tbody>
</table>

Table 5. Boundary conditions for the Case 1.
In Gas velocity = 1 m/s
Particle mass flow equal to the output

Out Opening = atmospheric pressure

Wall Particles = No slip
Gas = No slip

Initial height Bed height = 0.05 m

Particles 120 μm, 2400 kg.m⁻³

Table 6. Boundary conditions for the Case 2.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>dx/dp</th>
<th>Volumes Number</th>
<th>Δdx</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>15</td>
<td>99900</td>
<td>0.05</td>
</tr>
<tr>
<td>II</td>
<td>10</td>
<td>467313</td>
<td>0.08</td>
</tr>
</tbody>
</table>

Table 7. Volume discretization of the meshes.

Fig. 4. Schematic diagram of the Table 7 meshes. Up: Mesh I. Down: Mesh II
Numeric calculations performed (Vreman, Geurts, and Kuerten 1997; Chow and Moin 2003) showed that the required values to obtain an accurate numerical solution, it is necessary to use a ratio $\Delta dx \leq 0.25$ for the second order spatial scheme, and a ratio $\Delta dx < 0.5$ for the sixth order scheme.

The values of $\Delta dx$ presented in Table 7 are within the range recommended in the literature (Chow and Moin 2003; Agrawal et al. 2001; van Wachem 2000; Ahmed and Elghobashi 2000; Vreman, Geurts, and Kuerten 1997).

Figure 5 presents the solid volume fraction time evolution for the mesh II with superficial velocity 1 m/s. At the beginning, the solids present in the riser are forced to flow in the upward direction, similar to a plug flow.

When the bed of solids starts to expand, it is observed high solid particle concentration at the center of the tube and near the walls (Figure 5). This reordering of solid particles is a counteraction in order to offer a lower resistance to the gas flow. This type of flow regime is known as pre-fluidized bed. It is important to mention that one of most relevant characteristics of the fluidization is the high contact area between the solid particles and the fluid. In this way, a cubic meter of particles of 100 micron contains a superficial area of around 30000 m$^2$. The advantage of this high surface area is reflected in a high mass and heat transfer rates between the solid and the fluid.

Fig. 5. Evolution of the volume fraction field in a fluidized bed at 0, 11, 35, 70, 90, 132, 165, 185, 198, 220, 242, 264, 275, 290, and 317 ms.

Figure 6 shows the similarity between results presented by Miller and Gidaspow (1992). Here it is represented the regions of high and low solid concentration. Near the walls velocity is negative and near the center velocity is positive.

The annular-core behavior is something that detrimental in the units of Fluid Catalytic Cracking (FCC), since big fraction of the oil is converted in a region where the catalyst works less efficient. In addition to this, the particles that flow at center core are expose to bigger concentrations of oil compounds, which is something that produces faster deactivation of the catalyst. One the strategies to solve this issue is to inject pressurized gas in perpendicular direction to the flow in the reaction zone. Another solution is to include rings connected to walls, with the purpose of redirecting the solids from the wall towards the center.
Fig. 6. Comparison of solid phase velocity profile presented by Miller and Gidaspow (1992) with the CFD simulations (- ▲ -) and experimental data performed by Samuelsberg and B. H. Hjertager (1996) (●).

To get an impression regarding the flow behavior inside the column, the time averaged solid volume fraction is plotted at different column heights, 0.16 m, 0.32 m and 0.48 m (Figure 7). Here it can be observed the strong tendency of the solid particles to be near the wall.

Fig. 7. Axial profile of the solid phase volume fraction fields in the center (left) and radial profiles at 0.48 m, 0.32 m, 0.16 m (right up to down). Superficial velocity 0.36 m s⁻¹
5.2 Case 3
Pilot plant scale riser reactor (Bader, R., Findlay, J. and Knowlton, TM 1988). Riser height: 13 m, riser diameter 0.3 m. Entrance with angle 60°, gas superficial velocity 3.7 m and solids flux 98 kg/(s.m^2) as shown in Figure 8.

Fig. 8. Solids volumetric fraction in the center of the riser. Simulation time 15 sec. Left to right: LES Smagorinsky, LES WALE, LES Dynamic model, Detached Eddy Simulation (DES).

In the Figure 8 can be observed that the solid particles enter to the reactor uniformly distributed, after a short distance these particles start falling due to the gravity and they start flowing over the wall of the inclined pipe. After this, the solids fall into a turbulent zone where they get mixed. Some of the particles will continue falling over the vertical wall opposite to the entrance. The core-annular zone is formed at some height in the middle of the column.
6. Conclusions

Computational fluids dynamics is a very powerful tool understanding the behavior of multiphase in engineering applications. Large eddy simulation (LES) turbulence method provides a very detailed description of two-phase flow, which makes it suitable for simulation models that are validated with experimental data. By applying the LES method, it is possible to characterize different regions of a fluidized bed (core-annulus). LES can be considered as a valuable method for development and validation of closure models that include additional phenomena like heat exchange, mass transfer and chemical reactions.

It is important to constantly monitor the simulation, using parameters such as the Courant number, creating a function that calculates the maximum and average number of the control volume courant. The average value is recommended that is near or less than unity.

Finally, it is important to comment that success in the validation of experimental data depends on the appropriate choice of the experimental technique used to measure variables.

7. Acknowledgements

The author G. Gonzalez is grateful to PETROBRAS and the National Council for Scientific and Technological Development (CNPq) for the financial support to this research.

8. References


van Buijtenen, Maureen S., Willem-Jan van Dijk, Niels G. Deen, J.A.M. Kuipers, T. Leadbeater, and D.J. Parker. 2011. “Numerical and experimental study on multiple-

www.intechopen.com


This book provides a broad range of topics on fluid dynamics for advanced scientists and professional researchers. The text helps readers develop their own skills to analyze fluid dynamics phenomena encountered in professional engineering by reviewing diverse informative chapters herein.

How to reference
In order to correctly reference this scholarly work, feel free to copy and paste the following: