Modeling a Biomass Fluidizing Bed With Side Port Air Injection

Mirka Deza
Virginia Polytechnic Institute and State University, mdeza@iastate.edu

Francine Battaglia
Virginia Polytechnic Institute and State University

Theodore J. Heindel
Iowa State University, theindel@iastate.edu

Follow this and additional works at: http://lib.dr.iastate.edu/me_conf

Part of the Acoustics, Dynamics, and Controls Commons, Biochemical and Biomolecular Engineering Commons, Computer-Aided Engineering and Design Commons, and the Fluid Dynamics Commons

Recommended Citation
http://lib.dr.iastate.edu/me_conf/135

This Conference Proceeding is brought to you for free and open access by the Mechanical Engineering at Iowa State University Digital Repository. It has been accepted for inclusion in Mechanical Engineering Conference Presentations, Papers, and Proceedings by an authorized administrator of Iowa State University Digital Repository. For more information, please contact digirep@iastate.edu.
Modeling a Biomass Fluidizing Bed With Side Port Air Injection

Abstract
Fluidized beds are used to gasify materials such as coal or biomass in the production of producer gas. Modeling these reactors using computational fluid dynamics is advantageous when performing parametric studies for design and scale-up. While two-dimensional simulations are easier to perform than three-dimensional simulations, they may not capture the proper physics. This paper compares two- and three-dimensional simulations with experiments for a reactor geometry with side port air injection. The side port is located within the bed region so that the injected air can help promote mixing. Of interest in this study is validating the hydrodynamics of fluidizing biomass. Two operating conditions of the fluidized bed are studied for superficial gas velocities of 1.5Umf and 3.0Umf, where Umf is the minimum fluidization velocity. The material used to represent biomass is ground walnut shell because it tends to fluidize uniformly and falls within the Geldart type B classification. The simulations are compared to experimental data of time-averaged local gas holdup values using X-ray computed tomography. Results indicate that for the conditions of this study, two-dimensional simulations overpredict the gas holdup trends when compared to the experiments. However, the three-dimensional simulations compare exceptionally well with the experiments, thus predicting the fluidization hydrodynamics, irrespective of flowrate or complexity due to the side air port. Furthermore, the study demonstrates the importance of using a three-dimensional model for bubbling fluidized beds with complex physics.

Keywords
biomass, computational fluid dynamics, fluidized bed, hydrodynamics, injection port

Disciplines
Acoustics, Dynamics, and Controls | Biochemical and Biomolecular Engineering | Computer-Aided Engineering and Design | Fluid Dynamics

Comments
This is a conference proceeding from ASME 2009 Fluids Engineering Division Summer Meeting 1 (2009): 1151, doi:10.1115/FEDSM2009-78372. Posted with permission.
Serial Title: Proceedings of the ASME Fluids Engineering Division Summer Conference--2009; presented at 2009 ASME Fluids Engineering Division Summer Conference, Au

Article Author: Deza, M., Heindel, T.J., and Battaglia, F

Article Title: Modeling a Biomass Fluidizing Bed with Side Port Air Injection


Volume: Issue: Month/Year: 2009 Pages: 1151-1159

OCLC/Docline: 505417241

Fax: (515) 294-1885 Ariel: 129.186.176.86

Lender String: *UIU,GZM,COD,AZU,TXH,RCE,ORE,LHL

Download Date: 20141023

Shelf __________________ Sort __________________
CI __________________ Unshlv Stx __________________
Cardex __________________ Staff __________________
Other Loc/Notes __________________

Initials/date 1st _________ Initials/date 2nd _________
MODELING A BIOMASS FLUIDIZING BED WITH SIDE PORT AIR INJECTION

Mirka Deza and Francine Battaglia
Department of Mechanical Engineering
Virginia Polytechnic Institute and State University
Blacksburg, Virginia 24061
*fbattaglia@vt.edu

Theodore J. Heindel
Department of Mechanical Engineering
Iowa State University
Ames, Iowa 50011

ABSTRACT

Fluidized beds are used to gasify materials such as coal or biomass in the production of producer gas. Modeling these reactors using computational fluid dynamics is advantageous when performing parametric studies for design and scale-up. While two-dimensional simulations are easier to perform than three-dimensional simulations, they may not capture the proper physics. This paper compares two-and three-dimensional simulations with experiments for a reactor geometry with side port air injection. The side port is located within the bed region so that the injected air can help promote mixing. Of interest in this study is validating the hydrodynamics of fluidizing biomass. Two operating conditions of the fluidized bed are studied for superficial gas velocities of $1.5U_{mf}$ and $3.0U_{mf}$, where $U_{mf}$ is the minimum fluidization velocity. The material used to represent biomass is ground walnut shell because it tends to fluidize uniformly and falls within the Geldart type B classification. The simulations are compared to experimental data of time-averaged local gas holdup values using X-ray computed tomography. Results indicate that for the conditions of this study, two-dimensional simulations overpredict the gas holdup trends when compared to the experiments. However, the three-dimensional simulations compare exceptionally well with the experiments, thus predicting the fluidization hydrodynamics, irrespective of flowrate or complexity due to the side air port. Furthermore, the study demonstrates the importance of using a three-dimensional model for bubbling fluidized beds with complex physics.

Keywords: Biomass, Computational fluid dynamics, Fluidized bed, Hydrodynamics, Injection port

INTRODUCTION

Fluidized bed technology is receiving great attention with the desired aim to improve the gasification of biomass, thus improving reactor design and efficiency. The challenge in designing gasifiers, particularly those using biomass, is two-fold. First, fluidization is a dynamic process so that traditional methods to monitor and measure the fluid behavior are difficult [1]. The bed material consists of a feedstock as well as an inert material, creating an opaque media which is difficult to visualize. Second, biomass media range in density, particle size and shape, which can result in poor mixing during the gasification process. A review by Cui and Grace [2] has directly identified that biomass hydrodynamics is unique and requires further characterization and modeling in order to improve fluidized bed processes.

The hydrodynamics associated with gas-solid interactions are particularly important when computationally modeling biomass. Several drag models have been reported in the literature to account for the gas-solid hydrodynamics of fluidized beds. Taghipour et al. [3] compared the Syamlal–O’Brien, Gidaspow, and Wen–Yu models with experimental data and found that for relatively large Geldart B particles, the models predicted the hydrodynamics of the bed reasonably well. Another extensive model comparison in fluidized beds was made by Mahinpey et al. [4], comparing bed expansion and pressure drop with different superficial velocities in a fluidized bed using the Di Felice, Gibilaro, Koch, Syamlal–O’Brien, Arastoopur, Syamlal–O’Brien (adjusted), Di Felice (adjusted), Gidaspow, Zhang–Reese, and Wen–Yu drag models. All of the models gave acceptable qualitative agreement with experimental data; however results for the adjusted models of Syamlal–O’Brien and Di Fe-
lice showed an improvement in quantitative predictions of the bed hydrodynamics. The aforementioned drag model studies used glass beads as the solid particle in the fluidized beds; however none of the drag models were tested using biomass particles. The authors [5] validated two drag models, namely, the Syamlal-O'Brien model [6] and the Gidaspow model [7] for biomass fluidization using ground walnut shell as the medium. They also performed parametric studies to characterize the walnut shell properties, such as sphericity and coefficient of restitution, which is information not easily obtained experimentally. They found that the Gidaspow model better predicted the fluidized biomass bed for all cases.

The reactor design is also crucial in order to achieve improvements in reactor performance, especially when using biomass. As part of the reactor design, side air ports strategically placed along the reactor column can help promote and improve mixing and reactions. The features of the ports, such as location, diameter, and flowrate, increase the complexity of the reactor design even though the goal is to enhance reactor performance. Most of the research on side gas injection in fluidized beds has focused on predicting the morphology of the gas-solid distribution around the jet [8-10]. Several correlations have been developed to predict penetration lengths of horizontal gas jets and a summary can be found in Li et al. [11]. These studies have concluded that penetration varies with velocity of gas injection and diameter of the injection port and bed material properties such as density, viscosity, and particle diameter.

Some computational models have been conducted in fluidized beds with one injection port. Tyler and Mees [12] studied three discretization schemes and qualitatively compared their 2D simulations with experiments. They found that a higher order Superbee total variational diminishing (TVD) scheme predicts horizontal jet and bubble formation better than minmod TVD and hybrid schemes. A recent study by Li [11] performed simulations of a circulating fluidized bed with one horizontal jet using a 3D model and compared jet penetration lengths with experiments and empirical correlations. They proposed a scaled Gilbaro drag model to account for agglomeration of fine particles, and found the quantitative comparison of bed height and jet penetration length were in good agreement with the experiments.

Investigations are now pursued to determine the biomass fluidization hydrodynamics for a reactor with a single side port. Franka et al. [13] experimentally studied a 10.2 cm diameter reactor using glass beads and various materials to represent biomass. In a companion paper, Deza et al. [14] performed a preliminary CFD modeling study to validate the hydrodynamics of glass bead fluidization in a reactor with a side air port. The study compared predictions of two-dimensional (2D) and three-dimensional (3D) simulations with experiments [13] and found good correspondence. In this work, simulations of a fluidized bed of ground walnut shell will be compared and validated with experiments for the same conditions.

NUMERICAL MODEL

A multiphase Eulerian-Eulerian model is employed in Multiphase Flow with Interphase eXchanges (MFIX) [6] and assumes that each phase behaves as interpenetrating continua with its own physical properties. The instantaneous variables are averaged over a region that is larger than the particle spacing but smaller than the flow domain. Volume fractions are introduced to track the fraction each phase occupies in the averaging volume, where \( e_g \) is the gas phase volume fraction (also referred to as the void fraction) and \( e_s \) is the solid phase volume fraction. Assuming a single gas phase and solid phase, the volume fractions must satisfy the relation that \( e_g + e_s = 1 \). The solid phase is described with an effective particle diameter \( d_p \), and characteristic material properties, and solved using a conservation equation for the solid phase.

The continuity equations for the gas phase and the solid phase, respectively, are:

\[
\frac{\partial}{\partial t} (e_g \rho_g) + \nabla \cdot (e_g \rho_g \mathbf{u}_g) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t} (e_s \rho_s) + \nabla \cdot (e_s \rho_s \mathbf{u}_s) = 0 \tag{2}
\]

where \( (\rho) \) is the density and \( (\mathbf{u}) \) the velocity vector.

The momentum equations for the gas and solid phases have the form:

\[
\frac{\partial}{\partial t}(e_g \rho_g \mathbf{u}_g) + \nabla \cdot (e_g \rho_g \mathbf{u}_g \mathbf{u}_g) = -e_g \nabla P_g + \nabla \cdot \tilde{\mathbf{s}}_g + \mathbf{I}_g + e_g \rho_g \mathbf{g} \tag{3}
\]

\[
\frac{\partial}{\partial t}(e_s \rho_s \mathbf{u}_s) + \nabla \cdot (e_s \rho_s \mathbf{u}_s \mathbf{u}_s) = -e_s \nabla P_s + \nabla \cdot \tilde{\mathbf{s}}_s - \mathbf{I}_s + e_s \rho_s \mathbf{g} \tag{4}
\]

The expressions on the left side are the net rate of momentum increase and the net rate of momentum transfer by convection. The right side includes contributions for buoyancy caused by the fluid pressure gradient, the stress tensors (\( \tilde{\mathbf{s}} \)), gravity (\( \mathbf{g} \)), and the interaction forces (\( \mathbf{I} \)) accounting for the momentum transfer between the gas and solid phases; this will be discussed in detail later in this section. The constitutive equations for the gas phase tensor can be found in [6].

The granular temperature \( \theta \) for the solid phase can be related to the granular energy, defined as the specific kinetic energy of the random fluctuating component of the particle velocity. The resulting transport equation for the granular temperature [15] is:

\[
\frac{3}{2} \left[ \frac{\partial}{\partial t} (e_s \rho_s \theta) + \nabla \cdot (e_s \rho_s \mathbf{u}_s \theta) \right] = \mathbf{\tilde{s}}_s : \nabla \mathbf{u}_s - \nabla \cdot \mathbf{q}_\theta + \gamma_{sign} - \gamma \theta + \phi_g \tag{5}
\]
where \( q_0 \) is the diffusive flux of granular energy, \( \gamma_{slip} \) is the production of translational fluctuation kinetic energy due to gas-solid slip, \( \gamma_9 \) is the rate of granular energy dissipation due to inelastic collisions [16], and \( \phi_y \) is the transfer of granular energy between the gas phase and solid phase. Since the numerical simulations will model a cold-flow fluidized bed, the energy equation will not be employed in MFIX and therefore is not presented here.

Kinetic theory for granular flow is used to calculate the solid stress tensor \( \sigma_x \) and solid-solid interaction force \( F_{gs} \) in the rapid granular flow regime [6]. There are two distinct flow regimes in granular flow: a viscous or rapidly shearing regime in which stresses arise due to collisional or translational momentum transfer, and a plastic or slowly shearing regime in which stresses arise due to Coulomb friction between solids in close contact. A blending function to provide a smooth transition between each regime is employed [17]. Further details related to the constitutive relations in Eqs. (3-5) can be found in the MFIX theory guide [6].

Drag Modeling

The interaction force \( F_g \) in the momentum Eqsns. (3) and (4) accounts for the gas-solid momentum transfer, where:

\[
F_g = F_g(u_s - u_g) \tag{6}
\]

The drag force \( F_g \) is expressed as the product of the coefficient for the interphase force between the gas and solid phases and the slip velocity between the two phases \( u_s - u_g \). The coefficient for the interphase force is different for each drag model.

It should be noted that for cases where the particle diameter is not perfectly spherical, the particle diameter used in the correlations is modified. The sphericity is the particle property that indicates how spherical a particle is, where a sphericity of unity signifies that the particle is a perfect sphere. Therefore, the modified particle diameter is \( d_{p} = \psi \bar{d}_p \), where \( \bar{d}_p \) is the mean diameter and \( \psi \) is the estimated sphericity of the actual particles.

The Gidaspow model [18] calculates the interphase drag force coefficient using two correlations depending on the local void fraction value and a blending function. For void fractions less than 0.8 the Ergun equation is used to calculate the interphase force coefficient and for void fractions greater than or equal to 0.8 the Wen-Yu equation is used. To avoid a discontinuity between the models, the blending function \( \varphi_{gs} \) introduced by [18] is:

\[
\varphi_{gs} = \frac{\arctan[150 \times 1.75(0.2 - (1 - \epsilon_g))] + 0.5}{\pi} \tag{7}
\]

The interphase drag force for the Gidaspow model is expressed as:

\[
F_g = (1 - \varphi_{gs}) F_{g(\text{Ergun})} + \varphi_{gs} F_{g(\text{Wen-Yu})} \tag{8}
\]

where \( F_g \) for the Ergun equation valid for \( \epsilon_g < 0.8 \) is:

\[
F_{g(\text{Ergun})} = 150 \frac{(1 - \epsilon_g)^2 \mu_g}{\epsilon_g d_p^2} + 1.75 \frac{(1 - \epsilon_g) \rho_g}{d_p} |u_s - u_g| \tag{9}
\]

and \( F_g \) for the Wen-Yu equation valid for \( \epsilon_g \geq 0.8 \) is:

\[
F_{g(\text{Wen-Yu})} = \frac{f(Re)}{d_p} (1 - \epsilon_g) \mu_g |u_s - u_g| \epsilon_g^{-2.65} \tag{10}
\]

where

\[
f(Re) = \frac{3}{4} \left[ \frac{24}{\epsilon_g Re} (1 + 0.15(\epsilon_g Re)^{0.687}) \right] \tag{11}
\]

Solution Methodology

To discretize the governing equations in MFIX, a finite volume approach for a staggered grid is used to reduce numerical instabilities [19]. Velocities are stored at the cell surfaces, and scalars, such as void fraction and pressure, are stored at the center of the cell. Discretization of time derivatives are first-order and discretization of spatial derivatives are second-order. An important feature is the use of a higher-order discretization scheme for the convective terms, known as the Superbee method [20], which improves convergence and accuracy of the solution. A modification of the SIMPLE algorithm is used to solve the governing equations [19]. The first modification uses an equation for the solid volume fraction that includes the effect of the solids pressure to help facilitate convergence for both loosely and densely packed regions. The second modification uses a variable time-stepping scheme to improve convergence and execution speeds.

Problem Description

The fluidized bed reactor used in the experiments consisted of a 9.5 cm ID, 40 cm tall acrylic tube. Air entered a plenum chamber with two 1.0 cm pressure taps used for measuring pressure drop across the bed. Air left the plenum through a distributor plate drilled with 100, 0.1 cm diameter holes; each hole was spaced 0.4 cm apart on a square grid. X-ray computer tomography (CT) imaging was used to provide time-averaged local and global gas holdup. Complete details of the experimental methods are found in [21].

For all simulations, air is uniformly provided at the bottom of the domain (see Fig. 1) to equal the superficial gas velocity and atmospheric pressure is specified at the exit. The side port injection is also modeled with a uniform air velocity at the inlet. The no-slip condition is used to model the gas-wall interactions and a partial-slip condition is used for the particle-wall interactions [22]. Table 1 summarizes the ground walnut shell particle properties and flow conditions. The sphericity and coefficient of restitution were numerically estimated based on previous work [5], whereas the other properties were provided from
the experiments. Two inlet gas velocities are used; the lower velocity of \( U_g = 1.5U_{mf} \) represents a mild bubbling bed and the higher velocity of \( U_g = 3.0U_{mf} \) represents an industrial reactor flowrate [23]. Three side port injection flowrates are tested: a base case where \( Q_s = 0 \) (cylinder with no injection port), and two additional cases where \( Q_s = 5\%Q_{mf} \) and \( 10\%Q_{mf} \), where \( Q_{mf} \) is the minimum fluidization volumetric flowrate.

The first domain tested in the CFD modeling is a two-dimensional (2D) plane in Cartesian coordinates, which represents the centerplane of the cylinder with a 10.2-cm diameter, as shown in Fig. 1. Previous work of Xie et al. [17] and Deza et al. [5,24] have shown very good agreement using a 2D approach for a cylinder with no side air injection when the flow is limited to the bubbling regime for Geldart B particles. The second domain tested is fully three-dimensional (3D) in cylindrical coordinates.

Based on the grid resolution study of Deza et al. [24], the work herein uses a uniform grid with \( 40 \times 60 \) rectangular cells for the 2D domain, which were found to sufficiently produce grid-independent results with an estimated numerical error less than 1%. The grid resolution for the 3D domain in the radial and axial directions is based on the same discretization as the 2D domain, with the addition of a uniform discretization in the azimuthal direction. For the 3D simulations, the grid is discretized using \( 40 \times 60 \times 16 \) cells that form parallelepiped cells due to the circular cross-section of the domain. The time step used by MFIX automatically adjusts to help the simulation converge. It was found that the time step decreased by one-half when performing 3D simulations versus 2D simulations, and it decreased when the flowrate increased. The mean time step for the 3D simulation with \( 3.0U_{mf}, 10\%Q_{mf} \) was on the order of 0.0003 s.

### RESULTS AND DISCUSSION

As a starting point, the base case (\( Q_{mf} = 0 \) and \( U_g = 27.6 \) cm/s) is used to confirm that the CFD modeling predicts the same pressure drop through the bed for both two- and three-dimensions. The pressure drop across the walnut shell fluidized bed versus the superficial gas inlet velocity is shown in Fig. 2, comparing the experimental measurements to that predicted using MFIX. Once the bed is fluidized at \( U_{mf} = 18.4 \) cm/s [13], the measured pressure drop is approximately constant at 470 Pa whereas the predicted pressure drop is approximately 560 Pa. It should be noted that the CFD predictions are in very close agreement with the theoretical pressure drop of 572 Pa. The slight discrepancy with experiments is most likely due to error associated with the irregular particle sizes for ground walnut shell. Furthermore, the simulations utilized a single particle diameter of 550 \( \mu \)m, whereas the experiments have a particle diameter range of

---

**Table 1: Properties and Flow Characteristics for Walnut Shell**

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle diameter, ( d_p ) (cm)</td>
<td>0.055</td>
</tr>
<tr>
<td>Particle density, ( \rho_p ) (g/cm(^3))</td>
<td>1.3</td>
</tr>
<tr>
<td>Bulk density, ( \rho_b ) (g/cm(^3))</td>
<td>1.48</td>
</tr>
<tr>
<td>Sphericity, ( \psi ) (-)</td>
<td>0.6</td>
</tr>
<tr>
<td>Coefficient of restitution, ( \epsilon ) (cm/s(^2))</td>
<td>0.85</td>
</tr>
<tr>
<td>Minimum fluidization velocity, ( U_{mf} ) (cm/s)</td>
<td>18.4</td>
</tr>
<tr>
<td>Initial void fraction, ( \epsilon^i ) (-)</td>
<td>0.56</td>
</tr>
<tr>
<td>Low superficial velocity, ( U_g ) = 1.5(U_{mf}) (cm/s)</td>
<td>27.6</td>
</tr>
<tr>
<td>High superficial velocity, ( U_g ) = 3.0(U_{mf}) (cm/s)</td>
<td>55.2</td>
</tr>
</tbody>
</table>

---

Figure 1: Schematic of the bed chamber and freeboard in a cylindrical reactor, including the side port injector.

Figure 2: Superficial gas velocity versus pressure drop comparing 2D and 3D simulations with experiments.
500-600 μm. It is particularly encouraging that for the base case, the 2D and 3D simulations are almost identical.

Figure 3(a-c) presents contours of the void fraction for the walnut shell fluidized bed at $U_g = 1.5U_{mf}$ and $Q_s = 5\%Q_{mf}$. The 2D and 3D simulations, Fig. 3a and c, respectively, are time-averaged from 5 to 40 s (which represents 3500 time realizations) and are compared to the experiments (Fig. 3b). Also shown is the void fraction averaged across the bed width versus axial direction (Fig. 3d), which identifies that the bed height expands to approximately 11 cm after fluidization. These side-by-side comparisons help elucidate the hydrodynamic features between both the 2D and 3D simulations and their agreement with the experiments. The side port air injection tends to cause a slight nonuniformity of the fluidized media near the port, which is accompanied by higher void fraction (i.e., more gas). Overall the bed uniformly fluidizes; this feature is observed in the experiment and the 3D simulation also predicts the same fluidization hydrodynamics. The 2D simulation predicts relatively uniform
fluidization, however, there is more gas present within the center of the bed, displacing the solid particles. Figure 3d confirms that the 2D simulation predicts higher void fraction, which increases from 0.6 to 0.7 within the fluidizing bed. The 3D simulation predicts slightly lower void fraction but it is very uniform, as indicated by the constant value of 0.62. Increasing the side air injection flowrate to \( Q_s = 10\% Q_{mf} \), Fig. 4(a-d) provides evidence that the results are still in very good agreement. However, the 2D simulation predicts a more nonuniform solid distribution within the bed region.

In an effort to further quantify and contrast the simulations with the experiments, time-averaged void fraction profiles at two axial locations, \( y = 4 \) and 8 cm, are shown in Fig. 5 for \( 1.5U_{mf} \) and \( 10\% Q_{mf} \). The experimental data are shown (symbols) for two perpendicular slices (X-slice and Y-slice). The variations in the experimental data are attributed to the nonuniform inlet conditions that result from the 100 discrete air inlet holes in the distributor plate. It should be noted that the orientation of the experimental X- and Y-slice locations, which are mutually perpendicular, is arbitrary. The 3D volume could be rotated about the central axis for slightly different X- and Y-slice experimental data. However, in this case the Y-slice corresponds a vertical slice through the center of the side port air injector. The 2D and 3D predictions for local void fraction profiles shown in Fig. 5 compare well with the experiments at \( y = 4 \) cm but at \( y = 8 \) cm the 2D data deviates from the other data. Similar discrepancies have been shown by others [3,25-27].

The remaining two cases are for \( U_g = 3.0U_{mf} \) for \( Q_s = 5\%Q_{mf} \) and \( 10\%Q_{mf} \), as shown in Figs. 6 and 7, respectively. Examining the 2D void fraction contours (Fig. 6a and Fig. 7a), the nonuniformity of the fluidization is very apparent near the bed expansion height of 15 cm. The 3D simulations (Fig. 6c and Fig. 7c) compare remarkably well with the experiments (Fig. 6b and Fig. 7b), which is very exciting because the inlet gas velocity \( U_g \) is large. The larger inlet gas velocity in combination with the side air injection suggests improved mixing throughout the bed, with a mean void fraction of 0.7. The 3D simulation slightly underpredicts the hydrodynamics, however, these discrepancies are most likely attributed to the single particle size used in the computational modeling or the estimate for the particle sphericity. The time-averaged void fraction profiles at \( y = 4 \) and 8 cm are shown in Fig. 8 for \( 1.5U_{mf} \) and \( 10\% Q_{mf} \). The 3D prediction compares very well with the experiment, whereas the 2D simulation significantly overpredicts the presence of gas near the lower region of the bed (Fig. 8(a)). The discrepancies between the 2D data and the experiment are less pronounced with bed height.

Figures 6-8 clearly elucidate the importance of modeling a 3D domain to capture the hydrodynamics for fluidizing biomass. Although the authors found differences between the 2D simulations and experiments for a fluidized bed of glass beads [14], overall there was good agreement. It would seem that the computational modeling is more sensitive when predicting the hydrodynamics for biomass. The most likely reasons are the irregular shape of biomass particles and the lack of freedom for the particles to move azimuthally, thus limiting the validity for using a two-dimensional domain.

CONCLUSIONS

Computational fluid dynamics was used to compare predictions of fluidization hydrodynamics with experiments of a reactor with side air injection. Two- and three-dimensional simulations were performed to determine if both modeling approaches would
Figure 6: Average void fraction for the $3.0U_{mf}$ and $5\%Q_{mf}$ fluidized bed comparing the (a) 2D simulation, (b) experiment, (c) 3D simulation, and (d) averaged across the bed width versus axial direction.

Figure 7: Average void fraction for the $3.0U_{mf}$ and $10\%Q_{mf}$ fluidized bed comparing the (a) 2D simulation, (b) experiment, (c) 3D simulation, and (d) averaged across the bed width versus axial direction.

capture the salient bed features. Ground walnut shells were used for the biomass bed material, thus extending the previous work using a well-characterized bed medium of glass beads. Two superficial inlet gas velocities (low and high) were tested with three side air injection flowrates. The findings showed that 2D simulations overpredicted the fluidized bed expansion and the results did not demonstrate a uniformly fluidizing bed. The 3D simulations compared well for all cases; however, the remarkably close correspondence for the higher inlet gas velocity was not expected and the 3D simulations virtually collapse with the experiments. This study demonstrates the importance of using a 3D model for a truly 3D flow in order to capture the hydrodynamics of the fluidized bed for a complicated flow and geometry. Work is underway to continue these studies for other biomass materials (see for example [13]) to further validate the computational modeling and expand the range of flow parameters.
Figure 8: Average void fraction profiles for the $3.0U_{mf}$ and $10\%Q_{mf}$ fluidized bed comparing 2D and 3D simulations with the experiment at (a) $y = 4$ cm and (b) $y = 8$ cm.

**NOMENCLATURE**

- $d$: Particle diameter
- $e$: Coefficient of restitution
- $g$: Gravitational acceleration
- $q$: Diffusive flux of granular energy
- $u$: Velocity vector
- $t$: Time
- $C_D$: Drag coefficient
- $F$: Drag force
- $I$: Interphase momentum transfer
- $P$: Pressure
- $Q$: Volumetric flowrate
- $Re$: Reynolds number
- $U$: Fluidization velocity

**Greek Letters**

- $\varepsilon$: Void fraction
- $\varphi$: Blending function
- $\gamma$: Granular energy dissipation due to inelastic collisions
- $\mu$: Dynamic viscosity
- $\rho$: Density
- $\sigma$: Stress tensor
- $\psi$: Particle sphericity
- $\Phi$: Transfer of granular energy between phases
- $\theta$: Granular temperature

**Superscripts/Subscripts**

- $b$: Bulk
- $g$: Gas phase
- $mf$: Minimum fluidization
- $s$: Solid phase

**ACKNOWLEDGMENT**

The authors would like to thank the generous financial support from the ISU Institute for Physical Research and Technology and the Grow Iowa Values Fund. Additional appreciation is extended to the High Performance Computing Center at Iowa State University for their computer and technical support.

**REFERENCES**


Proceedings of the
ASME FLUIDS ENGINEERING DIVISION
SUMMER CONFERENCE
– 2009 –

VOLUME 1
PART B

GAS-LIQUID TWO-PHASE FLOWS
NUMERICAL METHODS FOR MULTIPHASE FLOWS
TRANSPORT PHENOMENA IN MANUFACTURING
PROCESSES AND MATERIALS PROCESSING
TRANSPORT PHENOMENA IN MIXING
TRANSPORT PHENOMENA IN ENERGY CONVERSION
FROM CLEAN AND SUSTAINABLE RESOURCES
FUNDAMENTAL ISSUES AND PERSPECTIVES IN FLUID MECHANICS

presented at
2009 ASME FLUIDS ENGINEERING DIVISION SUMMER CONFERENCE
AUGUST 2–6, 2009
VAIL, COLORADO USA

sponsored by
THE FLUIDS ENGINEERING DIVISION, ASME

ASME
Three Park Avenue • New York, N.Y. 10016
Statement from By-Laws: The Society shall not be responsible for statements or opinions advanced in papers... or printed in its publications (7.1.3)

INFORMATION CONTAINED IN THIS WORK HAS BEEN OBTAINED BY ASME FROM SOURCES BELIEVED TO BE RELIABLE. HOWEVER, NEITHER ASME NOR ITS AUTHORS OR EDITORS GUARANTEE THE ACCURACY OR COMPLETENESS OF ANY INFORMATION PUBLISHED IN THIS WORK. NEITHER ASME NOR ITS AUTHORS AND EDITORS SHALL BE RESPONSIBLE FOR ANY ERRORS, OMISSIONS, OR DAMAGES ARISING OUT OF THE USE OF THIS INFORMATION. THE WORK IS PUBLISHED WITH THE UNDERSTANDING THAT ASME AND ITS AUTHORS AND EDITORS ARE SUPPLYING INFORMATION BUT ARE NOT ATTEMPTING TO RENDER ENGINEERING OR OTHER PROFESSIONAL SERVICES. IF SUCH ENGINEERING OR PROFESSIONAL SERVICES ARE REQUIRED, THE ASSISTANCE OF AN APPROPRIATE PROFESSIONAL SHOULD BE SOUGHT.

For authorization to photocopy material for internal or personal use under circumstances not falling within the fair use provisions of the Copyright Act, contact the Copyright Clearance Center (CCC), 222 Rosewood Drive, Danvers, MA 01923, Tel: 978-750-8400, www.copyright.com.

Requests for special permission or bulk reproduction should be addressed to the ASME Technical Publishing Department.

ISBN No. 978-0-7918-4372-7

Copyright © 2009 by ASME
All Rights Reserved
Printed in U.S.A.