Hydrodynamic Modeling of Fluidized Beds of Single Component and Binary Particle System in Compartmented Gasifier

S.K. Wee
School of Engineering and Science,
Curtin University of Technology Sarawak Campus,
CDT 250, 98009 Miri, Sarawak, Malaysia
wkhur@yahoo.com

A. Gorin, H.B. Chua, H.M. Yan
School of Engineering and Science,
Curtin University of Technology Sarawak Campus,
CDT 250, 98009 Miri, Sarawak, Malaysia
alexander.g@curtin.edu.my

Abstract— **In the present study, three-dimensional simulation using Computational Fluid Dynamics (CFD, FLUENT 6.3.26) and experiments were conducted on a transparent cold-flow gas-solid bubbling Compartmented Fluidized Bed Gasifier (CFBG) model with identical diameter as the pilot-scaled CFBG. Using Eulerian-Eulerian granular model with closure laws according to the kinetic theory of granular flow that based on modified drag model simulations were conducted to model the hydrodynamics of CFBG in terms of gas-solid flow pattern, bed expansion ratio, bed pressure drop and bubble diameter. The model was resolved by Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) algorithm in CFD. Different inerts like river sand, quartz sand and alumina were used to examine the hydrodynamics behaviors both in single component and binary system where biomass, palm shell, was mixed with river sand. The modeling predictions compared reasonably well with experimental bed expansion ratio measurements and qualitative gas-solid flow patterns. Pressure drops predicted by the simulations were in relatively close agreement with experimental measurements. Furthermore, the simulated bubble diameters showed similarities with the Darton et al bubble size equation.**

Keywords- CFD; fluidized bed; single component; binary system; hydrodynamics

I. Introduction

Fluidized bed reactors are widely utilized in industry applications including pharmaceutical, food, chemical, petrochemical, mining and etc due to its effectiveness of heat and mass transfer between gas and solid phases [1]. Biomass, such as palm shell, is abundantly available in Malaysia since Malaysia oil palm is ranked number one fruit crops in year 2007 [2]. As such, it captured the attention of researchers to study the potential of these indigenous resources for renewable energy. Therefore, palm shell is selected as feedstock in current study. Understanding the hydrodynamics of a single component and binary mixture fluidized bed is essential for selecting the correct operating parameters for the appropriate fluidization regime due to its highly complex behavior. Previous works have been done in compartmented reactors of laboratory scale at different geometry and configuration of internals either in cold or hot conditions [3-6]. The study is then extended to pilot scale reactor at cold condition [7-10].

Computational fluid dynamics (CFD) with the advent of increased computational capabilities offers a new approach to understanding and modeling the complex phenomena in gas-solid environs. Due to reasonable computing time the Eulerian -Eulerian model is the preferred choice for simulating macroscopic hydrodynamics. The Eulerian-Eulerian model treats each phase as continuous and interpenetrating continuum. The solid particles are generally considered having identical diameter and density. The integral balances of continuity, momentum and energy for both phases are solved in this model.

Intensive studies [11-20] have been carried out to first validate the CFD multiphase code before its practical application. The bubbles motion and properties has been examined in 2D and 3D simulations [12, 13]. Paola et al's work [14] reported that the 2D and 3D simulation shows similar voidage profile and consistent formation, shape and size of bubbles for uniform shape of reactor. Several authors [11-16] carried out the modeling using Geldart A and Geldart B particles incorporated with different drag laws. The computational analyses of binary mixture fluidized bed were also the subject of study in recent years. Gidaspow et al. [21] extended the kinetic theory of dense gases to binary granular mixture with unequal granular temperature between the particle phases. The hydrodynamics of binary mixture with different sizes in a riser were studied by Mathiesen et al [22] using a CFD model. Huilin et al. [23] gave an extension to binary mixtures of particles using kinetic theory of dense gases, and simulated flow behavior of particles of binary mixture in the bubbling fluidized bed [24].

In this study, the Eulerian-eulerian CFD model has been applied to study the hydrodynamics of CFBG in terms of gas-solid flow pattern, bed expansion ratio, bed pressure drop and bubble diameter. Different inerts like river sand, quartz sand and alumina were used to examine the hydrodynamics behaviors both in single component and binary system where biomass, palm shell, was mixed with river sand. Model predictions were validated with experimentally determined data except the predicted bubble diameters were compared with Darton et al equation [25].

II. Methodolody

A. Experimental Setup

The schematic experimental set up is shown in Fig. 1. The CFBG as illustrated in Fig. 2 is made of the mild steel as the body structure and Perspex material as the cover to ease the visual observation during the experiment. The CFBG of the overall diameter of 0.66m and a height of 1.8m, partitioned into two compartments, the combustor and the gasifier, in a ratio of 65:35 respectively. Perforated plate distributor with orifice diameter of 3mm is used to uniformly distribute the fluidizing agent, ambient air at free area of 0.27% and 0.32% in triangular pitch arrangement for gasifier and combustor accordingly. The air is regulated in the range of 1 – 2 U_{mf} to maintain the bubbling mode of fluidization. Pressure drop are measured using water manometers at three locations to indicate the total pressure drop, across distributor and bed respectively. The inert particle used is the Geldart B particles, river sand, of 272 microns in mean size diameter and 2620 kg/m^3 in density. Biomass material used in the present work is palm shell of 1200 kg/m^3 in density. Different weight percent of palm shell (2%, 10% and 15%) with various palm shell sizes (2-2.36mm, 2.36-4.27mm and 4.75-9.5mm) were studied. The details of the experiment procedures have been reported elsewhere [7, 9, 10].

Authors would like to thank the Ministry of Science, Technology and Innovation and Malaysia Palm Oil Board for the financial support.

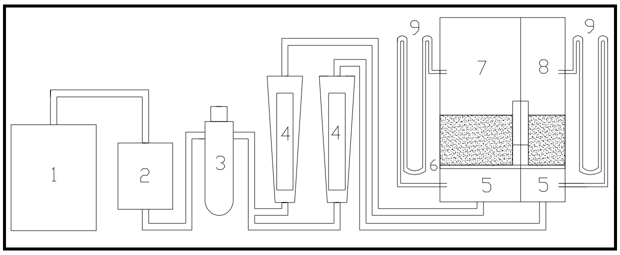

Figure 1. Experimental setup - 1: compressor; 2: dryer; 3: pressure regulator; 4: rotameter; 5: plenum; 6: perforated distributor; 7: combustor; 8: gasifier; 9: manometer

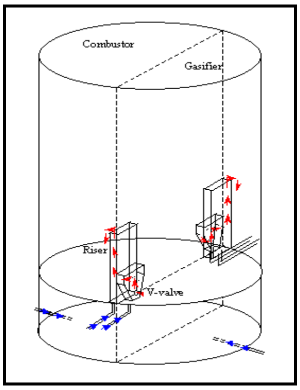

Figure 2. Isometric view of CFBG

B. *Computational Model*

The simulation of fluidized bed is performed by using the CFD commercial package, FLUENT 6.3.26 in 3D due to the complex geometry of CFBG. A multi-fluid Eulerian-Eulerian model, which considers the conservation of mass and momentum for the gas and fluid phases, is applied. The kinetic theory of granular flow is used to describe the solids phase stress. The governing equations can be summarized as below:

Mass conservation equations of gas (g) and solid (s) phases:

$$\frac{\partial}{\partial t}.(\alpha_g.\rho_g) + \nabla.(\alpha_g.\rho_g.\vec{v}_g) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}.(\alpha_s.\rho_s) + \nabla.(\alpha_s.\rho_s.\vec{v}_s) = 0 \tag{2}$$

Momentum conservation equations of gas and solid phases:

$$\begin{aligned}&\frac{\partial}{\partial t}.(\alpha_g.\rho_g.\vec{v}_g) + \nabla.(\alpha_g.\rho_g.\vec{v}_g^{\,2})\\&= -\alpha_g.\nabla p + \nabla.\overline{\overline{\tau}}_g + \alpha_g.\rho_g.\vec{g} + K_{gs}.(\vec{v}_g - \vec{v}_s)\end{aligned} \tag{3}$$

$$\begin{aligned}&\frac{\partial}{\partial t}.(\alpha_s.\rho_s.\vec{v}_s) + \nabla.(\alpha_s.\rho_s.\vec{v}_s^{\,2})\\&= -\alpha_s.\nabla p - \nabla p_s + \nabla.\overline{\overline{\tau}}_s + \alpha_s.\rho_s.\vec{g} + K_{gs}.(\vec{v}_g - \vec{v}_s)\end{aligned} \tag{4}$$

Fluctuation energy conservation of solid particles:

$$\begin{aligned}&\frac{3}{2}\left[\frac{\partial}{\partial t}.(\rho_s.\alpha_s.\Theta_s) + \nabla.(\rho_s.\alpha_s.\vec{v}_s.\Theta_s)\right]\\&= (-p_s\overline{\overline{I}} + \overline{\overline{\tau}}_s) : \nabla.\vec{v}_s + \nabla.(k_{\Theta s}.\nabla.\Theta_s) - \gamma_{\Theta s}\end{aligned} \tag{5}$$

Where α is the volume fraction, ρ is the density, kg/m^3; v is the velocity,m/s; $\overline{\overline{\tau}}$ is the stress tensor,Pa; g is the gravitational force, m/s^2; K_{gs} is the gas/solid momentum exchange coefficient, Θ is the granular temperature, m^2/s^2; $\overline{\overline{I}}$ is the stress tensor, $k_{\Theta s}$ is the diffusion coefficient for granular energy, kg/sm; γ_{Θ} is the collision dissipation energy, kg/s^{3}m. Subscripts g and s are gas phase and solid phase respectively.

TABLE I. MOMENTUM EXCHANGE COEFFICIENTS

Syamlal-O'brien drag function

$$f = \frac{C_D \,\mathrm{Re}_s\, \alpha_l}{24 \upsilon_{r,s}^2} \quad \text{for } \alpha_g > 0.8 \tag{6}$$

where the drag function

$$C_D = \left(0.63 + \frac{4.8}{\sqrt{\mathrm{Re}_s / \upsilon_{r,s}}}\right)^2 \tag{7}$$

$$\mathrm{Re}_s = \frac{\rho_l d_s \left|\vec{\upsilon}_s - \vec{\upsilon}_l\right|}{\mu_l} \tag{8}$$

Where f is the drag function, C_D is the drag coefficient, Re is the Reynold Number, $\upsilon_{r,s}$ is the terminal velocity the solid phase, m/s; d_s is the diameter of particles,m; μ is the shear viscosity in kg/sm, subscript l is for the l^{th} fluid phase, s is for the s^{th} solid phase.

The constitutive equations as recommended in FLUENT user guide [26] are needed to close the governing relations. The momentum exchange coefficient can be calculated by specifying drag functions as shown in TABLE I. Despite of rigorous mathematical modeling the drag laws used in the model continue to be semi-empirical in nature. Therefore, it is crucial to use a drag law that correctly predicts the incipient fluidization conditions.

The governing equations are solved by finite volume method. First-order discretization schemes for the convection terms are used. A time step of 0.001s with maximum number of 10 iterations per time step is chosen. This iteration is adequate to achieve convergence. The relative error between two successive iterations is specified by using a convergence criterion of 10^{-3} for each scaled residual component. The phase-coupled SIMPLE algorithm is applied for the pressure-velocity coupling. TABLE II gives a summary of the flow parameters to be used in the simulation of 3D fluidized bed.

TABLE II. SIMULATION MODEL PARAMETERS

Description	**Value**	**Unit**
Height of reactor	1.50	m
Air density, ρ_g	1.2	kg/m^3

Restitution coefficient	0.9	-
Initial volume fraction (river sand)	0.45	-
Initial volume fraction (palm shell)	0.2	-
Superficial velocity, *U*	0.02 - 0.2	m/s
Grid interval spacing	0.2	cm
Inlet boundary conditions	Velocity	-
Outlet boundary conditions	Pressure	-
Time steps	0.001	s
Maximum number of iterations	10	-
Convergence criteria	10^{-3}	-

Bubble properties are determined from void fraction of solids image produced by FLUENT. The diameter of this bubble is calculated as if its shape is assumed circular and centers of all bubble in the bed are recorded. For the simulations comparing the model parameters a void fraction cutoff of 0.3 was used. Average bubble diameters are calculated by averaging the bubble at binned heights above distributor.

The CFD predicted bubble diameter is compared with the Darton's bubble model. The Darton bubble model is a generally accepted semi-empirical model for bubble growth. The model is based upon the preferred paths of bubbles where the distance traveled by two neighbouring bubbles before coalescence is proportional to their lateral separation. The proposed equation is:

$$D_b = 0.54(U_o - U_{mf})^{0.4}(h + 4\sqrt{A_o})^{0.8} / g^{0.2} \quad (9)$$

*w*here D_b is the bubble diameter, m*;* U_o *is the superficial velocity, m/s;* U_{mf} is the m*inimum fluidization velocity, m/s;* h is the height of the bubble above the distributor*, m;* A_o is the 'catchment area' which characterizes the distributor, m^2 and *g* is the gravitational force, m/s^2; 0.54 is the only experimentally determined constant.

III. Results And Discussions

A. *Single Component System Hydrodynamcis Characteristics*

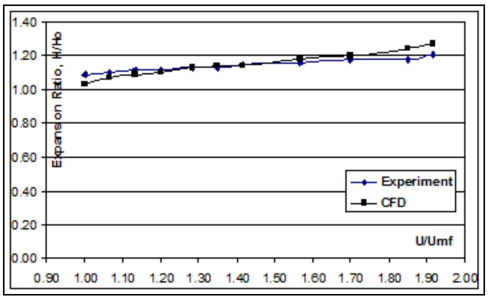

Figure 3. Comparison of CFD Results with Experimental Results at Static Bed at Combustor Side. Height of 0.34 m

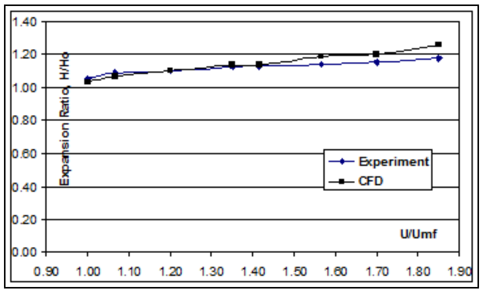

Figure 4. Comparison of CFD Results with Experimental Results at Static Bed at Combustor Side. Height of 0.45 m

The simulations are first performed to examine the bed expansion from onset of fluidization of the combustor side. Figures 3 and 4 show the comparison of CFD results for bed expansion ratio study with the experimental results at combustor side with river sand as the fluidization medium at static bed heights of 0.34m, and 0.45m. The superficial gas velocities are in the range of 1 – 1.9 U_{mf}. The bed of solids becomes more loosen (higher porosity) with increasing superficial gas velocity, thus contributing to steadily increasing bed height for all studies. In all cases, the CFD results show comparatively good agreement with the experimental results for respective static bed heights. Slight discrepancies are observed in the figures, i.e. underestimation of 5.4% to overestimation of 5.8%. Nevertheless, these gaps are in the acceptable error range. Overall the CFD well predicted the bed heights at the combustor side.

Figure 5 and 6 shows the comparison of CFD results for bed expansion ratio study with the experimental results at gasifier side. The superficial gas velocities are in the range of 1 – 2.8 U_{mf}. CFD results demonstrated consistent increase in bed height in both figures and predicted the bed expansion ratio reasonably well as compared to the experimental results.

The discrepancies are relatively greater than the results computed at combustor side. The greatest underestimation is found at 9.9% and overestimation is found at 6.0%. However,

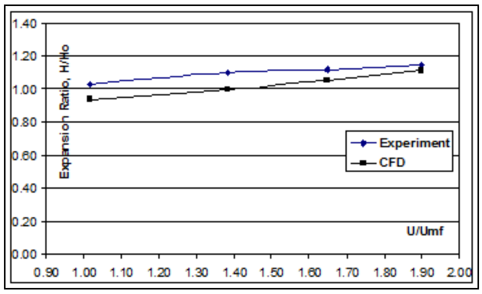

Figure 5. Comparison of CFD Results with Experimental Results at Static Bed at Gasifier Side. Height of 0.34 m

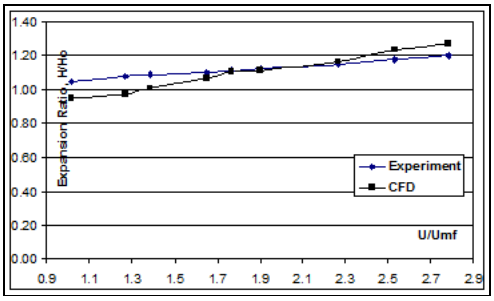

Figure 6. Comparison of CFD Results with Experimental Results at Static Bed at Gasifier Side. Height of 0.40 m

this is still within the acceptable error range for CFD estimation.

CFD initially predicted a lower bed height than the initial static bed height when the bed first fluidized at gasifier side However, this is not observed in combustor side. An underestimation of bed height by around 3% - 16.1% is achieved at gasifier side. This is maybe due to the definition of distributor to CFD. In experiment, a free area of 0.27% perforated plate distributor is used; however this setting is not defined in standard CFD. Also, geometrical effect may contribute to the CFD simulation results. The arrangement of solids inside the bed is different as in combustor. Though same static bed height and initial voidage (as in combustor) is set for the initial condition in CFD but because of the geometrical effect, which in other word, the effective diameter of the column, on solid arrangement, thus the initial solid packing is affected. The wall effect, which has a strong relationship with the effective diameter, may lead to different solid arrangement as compared to combustor side [7]. Werther [27] and other researchers showed that the wall effect becomes progressively more significant as the bed diameter decreases. Hence, the bed height drops slightly during onset of fluidization due to latter voidage compared to initial static bed height.

Figure 7 and 8 shows the comparison of CFD results with Darton's correlation for bubble diameter estimation at static bed heights of 0.34 m and 0.45 m at combustor side. By increasing the gas superficial velocities, the bubbles in a fluidized bed typically increase in size. There are no physical data from experiment in this regards, therefore Darton's correlation is chosen to validate the CFD results. Also, Darton's correlation does not take into considerations the

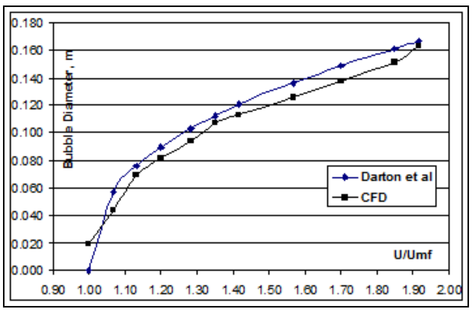

Figure 7. Comparison of CFD Results with Darton's Correlation at Static Bed Height of 0.34m

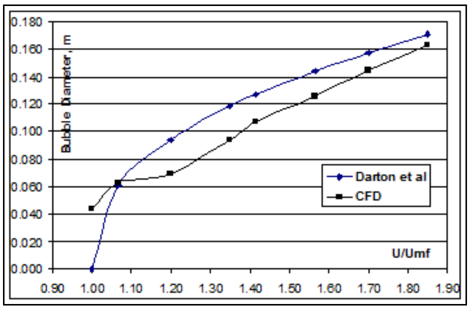

Figure 8. Comparison of CFD Results with Darton's Correlation at Static Bed Height of 0.45 m

effect of coalescence, breaking up and wall effects, the effect of bubbles touching the wall. From the Figures, CFD predicted the same increasing trends with increasing velocity but however it is discernible that smaller bubbles are predicted as compared to Darton's correlation. CFD results are more close to the experimental data at static bed height of 0.34 m.

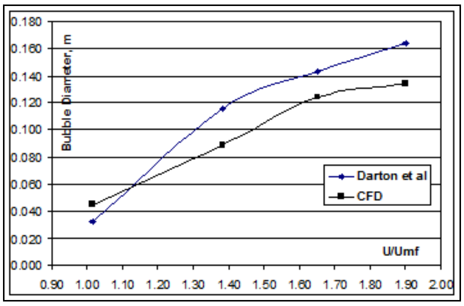

Figure 9. Comparison of CFD Results with Darton's Correlation at Static Bed Height of 0.34m

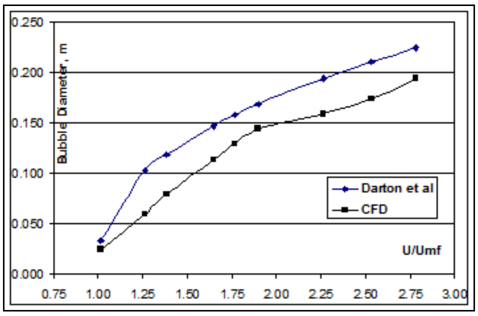

Figure 10. Comparison of CFD Results with Darton's Correlation at Static Bed Height of 0.40m

Figure 9 and 10 shows the comparison of CFD results with Darton's correlation for bubble diameter estimation at various static bed heights, i.e. 0.34 m, and 0.40 m, at gasifier side. Similar results as in combustor are observed here for the bubble diameter estimation by CFD; smaller bubbles are obtained compared to Darton's correlation. The largest discrepancy of under-prediction is found to be 41.5%. At tall bed, the bubble diameter is larger than the bubble diameter at low bed at maximum gas velocity.

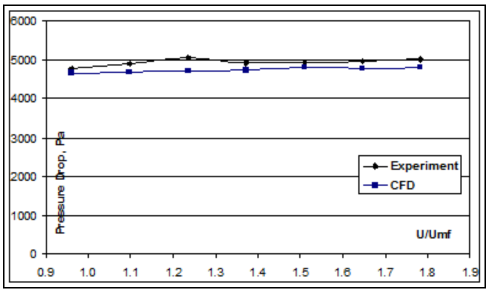

Figure 11. Comparison of experimental and simulated bed pressure drop (combustor, river sand at static bed height of 0.40 m)

Comparisons of the time-averaged bed pressure drop against superficial gas velocity from incipient fluidization onwards for gasifier and combustor sides with different inert used are plotted in Figure 11. From the figure, CFD underestimates the pressure drops. These discrepancies may be attributed to the dominant interparticle frictional forces, which are not considered by the multi-fluid model for simulating gas-solid phases. However, CFD predict results closer to experimental measurements in combustor side manifesting the effect of the chamber geometry (from experimental study [7] it follows that the effective diameter plays an important role in affecting the fluidization quality among the other competing factors, i.e. bed height, distributor free area and etc). The drag model employed does not take into account the effects of complex geometry.

B. Binary Mixture System Hydrodynamcis Characteristics

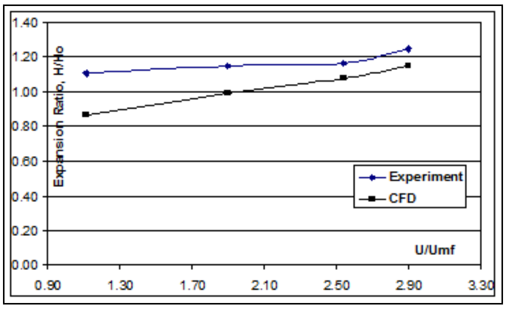

Figure 12. Comparison of CFD Results with Experimental Results at Static Bed Height of 0.445m (10 wt% palm shell, size of 3.32 mm)

Figures 12 - 16 show the comparison of CFD results for bed expansion ratio study with the experimental results in binary system at gasifier side where different sizes and weight percents of palm shell (2.36-4.27mm at 10 wt%, 2.36-4.27mm at 2 wt%, 2.36-4.27mm at15 wt%, 2-2.36mm at 10 wt% and 4.75-9.5mm at 10 wt%) and river sand is used at various static bed heights, i.e. 0.35m, 0.38m, 0.388m, 0.41m and 0.445m. The CFD predicts the same steadily increasing trend as the experimental ones with increasing gas velocities. However, all CFD simulated results underestimate parameters. Maximum error of 23.1% is found in setting of 2.36-4.27mm at 15 wt% at static bed height of 0.388m (Fig 12).

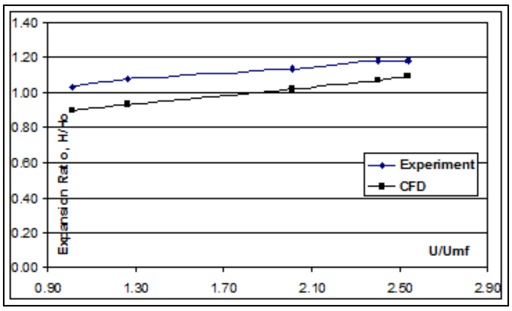

Figure 13. Comparison of CFD Results with Experimental Results at Static Bed Height of 0.35 m (2 wt% palm shell, size of 3.32 mm)

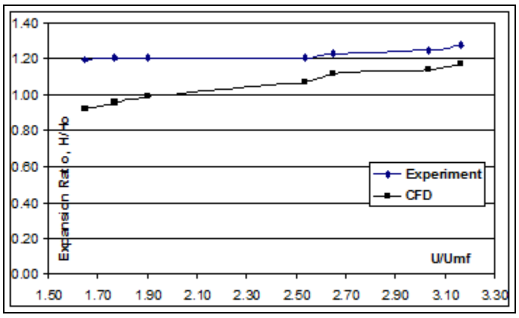

Figure 14. Comparison of CFD Results with Experimental Results at Static Bed Height of 0.388 m (15 wt% palm shell, size of 3.32mm)

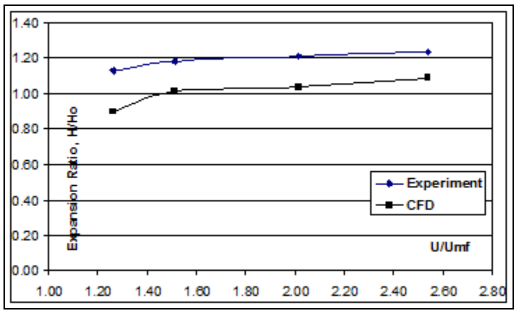

Figure 15. Comparison of CFD Results with Experimental Results at Static Bed Height of 0.38 m (10 wt% palm shell, size of 2.18 mm)

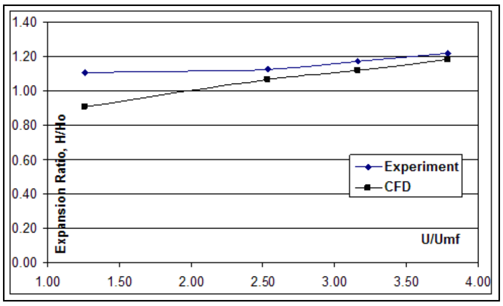

Figure 16. Comparison of CFD Results with Experimental Results at Static Bed Height of 0.41 m (10 wt% palm shell, 7.13 mm)

This discrepancy maybe due to the standard correlations used in CFD software. The correlation is developed based on the single component bed. Thus, the inter-particle drag force between the two different components is considered differently in standard Syamlal O'Brien's drag law as compared to actual situation. Therefore it is far-fetched for standard equations to simulate the binary system accurately.

However, the results obtained are still within the acceptable error range.

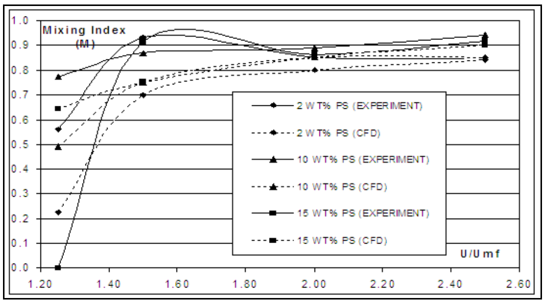

Figure 17. Comparison of Experimental and Simulated Effect of Palm Shell wt% on Overall Mixing Index (M) for dps = 3.315mm in Gasifier at Static Bed Height of 0.35m

Figure 17 shows the comparison of experimental determined effect of palm shell wt% on overall mixing index (M) [10] with CFD simulated results for palm shell size of 3.315mm in gasifier at static bed height of 0.35m. CFD predicted well-matched trends however lower values with the experimental results. Closer CFD results with the experimental results are obtained at $2U_{mf}$ and above. Generally, increasing superficial gas velocity will improve mixing in segregated systems of mixtures with difference densities/ sizes [28]. Shen et al. [29] reported that wake exchange coefficient, which indicates vertical and lateral solid mixing intensity, reduces with the increase in U_{mf}. This corresponds to M decrease. On the contrary, in Figure, M increases with increasing U_{mf}. This is likely due to the increase in bed voidage that leads to greater bed expansion, allowing effective vertical and lateral solid mixing. Good mixing is attainable at $1.5U_{mf}$ for palm shell ranged from 2-15 wt% and no significant changes of M from $1.5U_{mf}$ onwards. Thus, palm shell wt% has no significant effect on M.

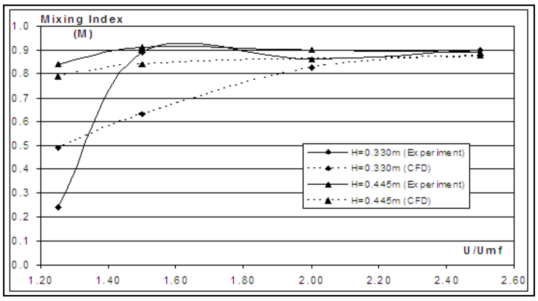

Figure 18. Comparison of Experimental and Simulated Effect of Static Bed Height on Overall Mixing Index (M) of 10 wt% Palm Shell with dps = 3.315mm in Gasifier

Figure 18 shows the comparison of experimental ly determined effect of bed height on overall mixing index (M) with CFD simulated results for 10 wt% palm shell size of 3.315mm in gasifier. The CFD mixing at bed height of 0.445m is in good agreement with the experimental results, however, large error is observed for bed height of 0.330m at lower superficial gas velocity. The error is gradually reduced with increasing gas velocity. For experimental results, M of about 0.9 is achieved at $1.25U_{mf}$ for deep bed, whereas at shallow bed, the same degree of mixing is only achievable at higher velocity ($1.5U_{mf}$). The same trend can be seen for CFD simulated results. Steady M of about 0.85 is attained at $1.5U_{mf}$ for deep bed, but same M is only attainable at $2U_{mf}$. With increasing bed height, the bed pressure drop increases resulting in stable bubbling that uniformly mixes the fluidized bed. Conversely, in shallow bed, where the total bed pressure drop is reduced, the situation is self aggravating and the local preferential channeling formes. This contributes to non-uniformity bubbling at the expense of the rest of the bed [30]. However, increasing superficial gas velocity will improve the mixing in shallow bed.

IV. CONCLUSION

The hydrodynamics of CFBG with single component and binary system have been studied experimentally and computationally. The modeling predictions compared reasonably well with experimental bed expansion ratio measurements and qualitative gas-solid flow patterns. Pressure drops predicted by the simulations are in relatively close agreement with experimental measurements. Furthermore, the simulated bubble diameters show similarities with the Darton et al bubble size equation. Further experimental and modeling efforts are required in a comparable time and space resolutions for the validation of CFD models for gas-solid fluidized bed reactors.

REFERENCES

[1] D. Kunii and O. Levenspiel, "Fluidization Engineering", 2nd Edition, Butterworth-Heinemann, U.S.A, 1991.

[2] T.L.K. Yong, K.T. Lee, A.R. Mohamed, and S. Bhatia, "Potential of hydrogen from oil palm biomass as a source of renewable energy worldwide," *Energy Policy,* 2007, Vol. 35, pp. 5692 – 5710.

[3] B.Bhattacharya, D. Sathiyamoorthy, V. Govardhana Rao, and S.P. Mahajan, "Solid Circulation in a Compartmented Gas Fluidized Bed", Powder Technology, 1999, 101, pp191-204.

[4] B.Bhattacharya, D. Sathiyamoorthy, V. Govardhana Rao, and S.P. Mahajan, "Critical Bed Height for Solid Circulation in a Compartmented Gas-Fluidized Bed", Chemical Engineering Technology, 2000, 23, pp12.

[5] He, Y.H. and Rudolph, V., "Gas-Solids Flow in the Riser of a Circulating Fluidized Bed", Chemical Engineering Science, 1995, Vol.50, No.21, pp3443-3453.

[6] Yan, H.M., and Rudolph, V., "A New Method for Measuring Solid Circulation Rate in Compartmented Fluidized Bed Gasifier", Chemical Engineering Community, 1996, 147, pp. 133-144.

[7] S.K. Wee, V.S. Chok, C. Srinivasakannan, H.B. Chua and H.M. Yan, "Fluidization Quality Study in Compartmented Fluidized Bed Gasifier," Energy & Fuels, vol. 22 (1), 2008, pp. 61-66.

[8] Chok Vui Soon, Wee Siaw Khur, Alexander Gorin, Chua Han Bing, and Yan Hong Ming, "Compartmented Fluidized Bed Gasifier for Syngas Production and Power Generation from Biomass", 12th *Int. Energy Conference and Exibition* '*Energex* 2007', Singapore, November 2007.

[9] S.K. Wee, V.S. Chok, B. Chin, W.W. Tang, A. Gorin, H.B. Chua, and H.M. Yan, "On the Effect of Effective Diameter on Fluidization Quality in Compartmented Fluidized Bed Gasifier", 21st *Symposium of Malaysian Chemical Engineers* (*SOMChE* 2007), Kuala Lumpur, Malaysia, December 2007

[10] V.S. Chok, L.F.B. Chin, S.K. Wee, W.W. Tang, A. Gorin, H.B. Chua and H.M. Yan, "Hydrodynamics Studies of Sand/Palm Shells Binary

Mixture in Compartmented Fluidized Bed Gasifier (CFBG)", 1st *Engineering Conference "Energy & Environment"* (ENCON 2007), Kuching, Malaysia, December 2007.

[11] M.J.V. Goldschmidt, J.A.M. Kuipers, and W.P.M. van Swaaij, "Hydrodynamic modelling of dense gas-fluidised beds using the kinetic theory of granular flow: effect of coefficient of restitution on bed dynamics", Chemical Engineering Science, 2001, 56, pp. 571 – 578.

[12] P.J. Witt, J.H. Perry, and M.P. Schwarz, "A numerical model for predicting bubble formation in a 3D fluidized bed", Applied Mathematical Modelling, 1998, Vol. 22, pp. 1071 – 1080.

[13] Ian Hulme, Eric Clavelle, Loni can der Lee, and Apostolos Kantzas, " CFD Modeling and Validation of Bubble Properties for a Bubbling Fluidized Bed", Ind. Eng. Chem., 2005, 44, pp. 4254 – 4266.

[14] Paola L., Giorgio M., Luca C., and Derek C., "Computational Fluid Dynamics simulations of gas fluidize dbeds: apreliminary investigation of different modelling approaches", Proc. in 10th Workshop on Two-phase Flow Predictions, Meseburg, 2002, pp. 300-309.

[15] R. Krishna and J.M. van Baten, "Using CFD for scaling up gas-solid bubbling fluidised bed reactors with Geldart A powders", Chemical Engineering Journal, 2001, 82, pp. 247 – 257.

[16] Tim Mckeen and Todd Pugsley, "Simulation and experimental validation of a freely bubbling bed of FCC catalyst", Powder Technology, 2003, 129, pp. 139 – 152.

[17] Sebastian Zimmermann and Fariborz Taghipour, "CFD Modeling of the Hydridynamics and Reaction Kinetics of FCC Fluidized-Bed Reactors", Ind. Eng. Chem., 2005, 44, pp. 9818 – 9827.

[18] Scott Cooper and Charles J. Coronella, "CFD simulations of particle mixing in a binary fluidized bed", Powder Technology, 2005, 151, pp. 27 – 36.

[19] Lu Huilin, He Yurong and Dimitri Gidaspow, "Hydrodynamic modelling of binary mixture in a gas bubbling fluidized bed using the kinetic theory of granular flow", Chemical Engineering Science, 2003, 58, pp. 1197 – 1205.

[20] Fariborz Taghipour, Naoko Ellis, and Clayton Wong, "Experimental and computational study of gas-solid fluidized bed hydrodynamics", Chemical Engineering Science, 2005, 60, pp. 6857 – 6867.

[21] Gidaspow D, Huilin L, Mangar E, "Kinetic Theory of Multiphase flow and Fluidization: Validation and Extension to Binaries", XIXth International Congress of Theoretical and Applied Mechanics, Japan 1996.

[22] Mathiesen V, Solberg T, Arastoopour H, Hjertager BH, "Experimental and Computational Study of Multiphase Gas/Particle Flow in a CFB Riser", AIChE, 2001, vol. 45, pp. 2503-2518.

[23] Huilin L, Gidaspow D, Manger E, "Kinetic Theory of Fluidized Binary Granular Mixtures", Phys Rev E 2001, vol. 64, pp. 61301-61309.

[24] Huilin L, Yurong H, Gidaspow D, "Hydrodynamic Modeling of Binary Mixture in a Gas Bubbling Fluidized Bed Using the Kinetic Theory of Granular Flow", Chem Eng Sci 2003, vol. 58, pp. 1197-1205.

[25] Darton R.C., Lanauze R.D., Davidson J.F., Harrison D., "Bubble Growth due to Coalescence in Fluidized Beds", Transaction of Institute of Chemical Engineers, vol. 55, 1977, pp. 274-280.

[26] FLUENT, 2002. Fluent 6.0 User's Guide, 20.4 Eulerian Model. Fluent Inc.

[27] Werther, J. AIChE Symp. Ser. 1968, 141 (70), 53–62.

[28] M. Wirsum, F. Fett, N. Iwanowa, G. Lukjanow, "Particle mixing in bubbling fluidized beds of binary particle systems," Powd. Tech., vol. 120, 2001, pp. 63-69.

[29] L.H. Shen, J. Xiao, F. Niklasson, F. Johnsson, "Biomass mixing in a fluidized bed biomass gasifier for hydrogen production," Chem. Eng. Sci., vol. 62, 2007, pp. 636 – 646.

A.E. Quereshi, D.E Crease, "Fluidized bed gas distributors," Powd. Tech, vol. 22, 1979, pp. 113 –119.