



COB-2021-2056

STUDY OF BED FLUIDIZATION THROUGH CFD ANALYSIS AND SIMULATION

Fernando Léo Bueno de Oliveira e Silva

Diego Carneiro de Oliveira

Universidade Federal de Itajubá, UNIFEI, Itajubá 37500-903, Brazil

fernando.leo17@unifei.edu.br

diego.oliveira@unifei.edu.br

Letícia de Oliveira Silva Della Colleta

Universidade Federal de Itajubá, UNIFEI, Itajubá 37500-903, Brazil

leticia.oliveira@unifei.edu.br

Electo Eduardo Silva Lora

Universidade Federal de Itajubá, UNIFEI, Itajubá 37500-903, Brazil

electo@unifei.edu.br

Rubenildo Vieira Andrade

Universidade Federal de Itajubá, UNIFEI, Itajubá 37500-903, Brazil

ruben@unifei.edu.br

Abstract. Computational fluid dynamics (CFD) is a useful tool for solving complex problems involving topics like thermodynamics, fluid mechanics, heat transfer, etc. It can reduce the cost and the need for construction and tests of many types of equipment via process simulation in many operational conditions that help engineers developing new technologies and processes. Gasification is an emerging technology able to process solid combustible materials, like biomass (forestry and agricultural residues), anthropogenic residues (sewage sludge and municipal solid waste), and others. The products from the gasification process are electricity, heat, chemicals, biofuels, etc. Although, the gasification process has several challenges to be overcome, in the case of fluidized bed gasifiers, the main difficulty remains in understanding the bed behavior during fluidization. The fluidization is a complex phenomenon, hard to observe and measure in experimental apparatus, especially when dealing with turbulence topics. For this reason, the main objective of this work is to present the hydrodynamic behavior of a 2-D gas-solid fluidized bed, using computational fluid dynamics software. The simulation output data were validated by comparing them with the experimental data obtained from the literature. The simulations were carried out by using bed material having 300 μm of particle diameter and a density of 2400 kg/m^3 at different inlet gas velocities. The Multiphase Eulerian-Eulerian model approach along with the kinetic theory of granular flow is used for simulating the gas-solid fluidization behavior. The minimum fluidization velocity conditions were calculated and inserted as an initial parameter for the CFD simulation. The momentum exchange coefficients were calculated using Syamlal-O'Brien functions. In the present work, the effects of inlet gas velocity, pressure drop, and bed expansion ratio are discussed. The results obtained will be used to develop an experimental fluidized bed cold reactor.

Keywords: Gasification, Fluidized Bed, Fluidization, CFD

1. INTRODUCTION

Gasification is an emerging technology able to process solid combustible materials, like biomass (forestry and agricultural residues), anthropogenic residues (sewage sludge and municipal solid waste), and others (Basu 2013). The main product from biomass gasification is the Syngas. This gaseous mixture is mainly composed of H_2 , CO , CH_4 , and other gaseous species (in minor concentrations), being a very useful product, able to produce electricity, heat, chemicals, biofuels, etc (Sikarwar et al. 2016). Despite this, the gasification process itself has several challenges to be overcome (Fouilland, Grace, and Ellis 2010). In the case of fluidized bed gasifiers, one of the main difficulties remains in understanding the bed behavior during the fluidization process. Fluidization is the operation by which solid particles are transformed into a fluidlike state via suspension in a gas or liquid (Kunii and Levenspiel 1993) and as a complex process, many studies are aided by simulation software, such as Computational Fluid Dynamics from ANSYS®.

The computational fluid dynamics (CFD) from ANSYS® is a tool useful to solve complex problems involving topics like thermodynamics, fluid mechanics, heat transfer, etc. It can reduce the cost and the need to construct and test physically

a desired equipment through a simulation of the processes in many operational conditions, demonstrating the main characteristics to be focused on the experimental steps of a given project (Wendt et al. 2009). CFD models are mainly categorized into Eulerian-Lagrangian and Eulerian–Eulerian approaches for a gas-solid fluidized bed (Pain, Mansoorzadeh, and De Oliveira 2001; Sahoo and Sahoo 2015). Eulerian–Eulerian model is considered the most appropriate approach in fluidized bed simulation because of its faster potential to formulate constitutive equations when compared to the Eulerian-Lagrangian method (Gera et al. 1998).

In this work, the main objective is to present the hydrodynamic behavior of a 2-D gas-solid fluidized bed using computational fluid dynamics software. The simulation output data were validated by comparing them with the experimental data obtained from the literature. The simulations were carried out by using bed material having 150-300 μm of particle diameter and a density of 2000-2500 kg/m^3 at different inlet gas velocities. The Multiphase Eulerian-Eulerian model approach along with the kinetic theory of granular flow is used for simulating the gas-solid fluidization behavior. The minimum fluidization velocity conditions were calculated and inserted as an initial parameter for the CFD simulation. The momentum exchange coefficients were calculated using Syamlal-O'Brien functions. In the present work, the effects of inlet gas velocity, pressure drop, and bed expansion ratio are discussed. The results obtained in this work will be used for the development of an experimental cold flow fluidized bed and thus, will be presented in a future paper.

2. MODELLING METHODOLOGY

2.1 Gas-Solid Flow equations

In this work, the Eulerian-Eulerian multiphase flow model was used to calculate the transient process gas-solid for granular solids fluidization. The different phases of the mentioned model were considered constants in the mathematical approach. The volumetric fraction between phases was considered to draw attention to the occupied volume by each phase by a function of space and time, showing that even the considered phases have interacted between themselves, the volume of one phase would not be filled by another (Mehrabadi 2012). The conservation equations were derived for all phases and converged by constitutive relations, defined by the application of kinetic theory (Khezri et al. 2019). The gaseous phase was modeled from turbulence equations of (RNG $k-\varepsilon$ e $k-\omega$) model, while the particle phase was modeled by using granular flow theory. General equations for mass conservation of gas and solid are presented following (Debnath, Nath, and Chauhan 2020).

The conservation of gas and solid mass is presented by Eq. (1) and (2).

$$\frac{\partial}{\partial t} \cdot (\varepsilon_g \rho_g) + \nabla \cdot (\varepsilon_g \rho_g v_g) = 0 \quad (1)$$

$$\frac{\partial}{\partial t} \cdot (\varepsilon_s \rho_s) + \nabla \cdot (\varepsilon_s \rho_s v_s) = 0 \quad (2)$$

Where ε is the volume fraction, ρ is the density (kg/m^3) e v is the velocity (m/s).

The moment conservation of gas and solids are presented by Eq. (3) and Eq. (4).

$$\frac{\partial}{\partial t} \cdot (\varepsilon_g \rho_g v_g) + \nabla \cdot (\varepsilon_g \rho_g v_g^2) = -\varepsilon_g \nabla P + \nabla \tau_g + \varepsilon_g \rho_g g + k_{gs}(v_g - v_s) \quad (3)$$

$$\frac{\partial}{\partial t} \cdot (\varepsilon_s \rho_s v_s) + \nabla \cdot (\varepsilon_s \rho_s v_s^2) = -\varepsilon_s \nabla P + \nabla \tau_s + \varepsilon_s \rho_s g + k_{gs}(v_g - v_s) \quad (4)$$

Where P represents the pressure (Pa); τ is the stress-strain tensor, in Pascal; g is the gravity acceleration (m/s^2) and k_{gs} is the heat transfer coefficient for fluid-solid and solid-solid. The expression $[-\varepsilon_s \nabla P + k_{gs}(v_g - v_s)]$ represents the interaction force, therefore, the drag and buoyant forces that act in momentum transfer between the gas and solid particles.

2.2 Gas-Solid Interaction

The gas-solid momentum exchange coefficient k_{gs} is defined by Eq. (5) (Dhrioua et al. 2019):

$$k_{gs} = \frac{\varepsilon_s \rho_s f}{\tau_s} \quad (5)$$

Where the particle relaxing time τ_s is defined by Eq. (6)

$$\tau_s = \frac{\rho_s d_s^2}{18 g} \quad (6)$$

And d_s is the particle size in the solid phase.

2.3 Drag Model

In an Eulerian-Eulerian approach, the drag force is a significant term for the fluidized bed hydrodynamic description. Its effects are observed in the interaction of gas and solid phases. The drag model used in this work is the Syamlal-O'Brien (Syamlal and O'Brien 1989) And is used for gas-solid momentum exchange coefficient calculation, presented by Eq. (7).

$$K_{gs} = \frac{3}{4} \cdot \frac{\alpha_s \alpha_g \rho_g}{v_{r,s}^2 d_s} \cdot C_D \cdot \left(\frac{Re_s}{v_{r,s}} \right) \cdot |\vec{v}_s - \vec{v}_g| \quad (7)$$

Where

$$C_D = \left(0.63 + \frac{4.8}{\sqrt{Re_s/v_{r,s}}} \right)^2 \quad (8)$$

And

$$v_{r,s} = 0.5 \cdot (A - 0.06 \cdot Re_s + \sqrt{((0.06 \cdot Re_s)^2 + 0.12 \cdot Re_s \cdot (2 \cdot B - A) + A^2)}) \quad (9)$$

With

$$A = \alpha_g^{4.14}, B = 0.8\alpha_g^{1.28} \text{ to } \alpha_g \leq 0.85 \text{ or } A = \alpha_g^{4.14}, B = \alpha_g^{2.65} \text{ to } \alpha_g > 0.85$$

2.4 Turbulence Models

Between the available turbulence models, k-ε and e k-ω are used for the simulations. Eq. (10) and (11) are the transport equations for the models RNG k- ε presented below, where the buoyancy effect is not considered.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k + \rho \varepsilon \quad (10)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon 1} \frac{\varepsilon}{k} P_k + C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k} \quad (11)$$

Where $C_{\varepsilon 2}^* = C_{\varepsilon 2} + \frac{C_\mu \eta^3 (1 - \frac{\eta}{\eta_0})}{1 + \beta \eta^3}$, $\eta = \frac{S k}{\varepsilon}$, $S = (2S_{ij}S_{ij})^{\frac{1}{2}}$ and P_k being the turbulence production rate.

The Eq. (12) and (13) of transport model k-ω are described below (Wilcox 1991):

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta \rho k \omega + P_{kb} \quad (12)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + P_{\omega b} \quad (13)$$

The coefficients for RNG k-ε and k-ω models are extracted from literature (Perini et al. 2017). The RNG model was developed using Re-Normalisation Group (RNG) methods by Yakhot et al. (1992) to renormalize the Navier-Stokes equations, to account for the effects of smaller scales of motion. In the standard k-epsilon model the eddy viscosity is determined from a single turbulence length scale, so the calculated turbulent diffusion is that which occurs only at the specified scale, whereas in reality, all scales of motion will contribute to the turbulent diffusion. The RNG approach, which is a mathematical technique that can be used to derive a turbulence model similar to the k-epsilon, results in a modified form of the epsilon equation which attempts to account for the different scales of motion through changes to the production term.

2.5 Minimal Fluidization Velocity

The minimal fluidization velocity is calculated from Eq. (14). This value depends exclusively on the fluidization agent and bed material characteristics (Kunii and Levenspiel 1993).

$$v_{mf} = \frac{g(\rho_s - \rho_g)d_p^2 \epsilon_{mf}^3 \phi^2}{150\mu(1 - \epsilon_{mf})} \quad (14)$$

3. COMPUTATIONAL MODEL

The fluidized bed simulation was performed by using the Ansys FLUENT software package. To observe the hydrodynamic behavior of the bed, a thin and inert material was considered (300 μm of diameter and 2400 kg/m^3 specific mass). Definitive model validation is expected by comparing simulation results with experimental data from an experimental apparatus that is still in the development phase, presented in figure 1. Therefore, to advance in computational model development and refining, a 2D geometry was created in FLUENT. The mesh was created from a square elements structure and for the bed fluidization simulation, 4150 cells were selected from a total of 20750, illustrated in figure 2.

Two simulations were considered in this work and they differ only by the step number (1500 and 5000), with 0.002s of time step size used for both cases. The convergence range of numerical simulation was 10^{-3} , to monitoring the residual mass fraction. The hydrodynamic simulation was performed under different injection velocities between 0.15m/s – 0.60 m/s. The Eulerian multiphase model was chosen to modeling the bed and its efficiency of the approach depends on the proper description of the gas-solid interface, particle-particle, and particle-wall collision and friction. In the computational domain, the gas was considered as a continuous phase meanwhile the solid was treated as a dispersed phase inside the reactor, to interpenetrate and interact itself.

The gas-solid fluidized bed simulation was considered non-reactive, transient, without buoyant force, neither mass transfer between phases, also assuming pressure and density as constants in each step. The injection velocity was considered uniform along the bottom of the bed.

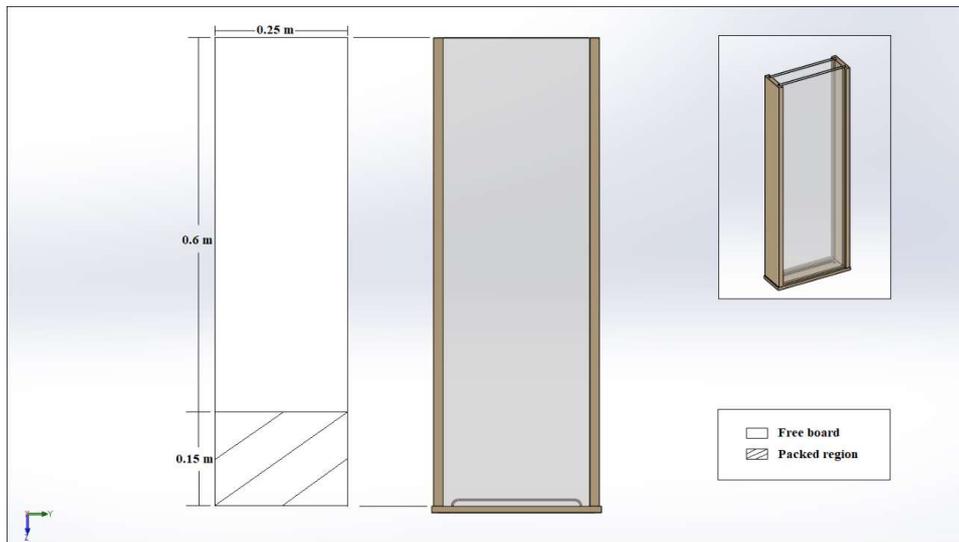


Figure 1 - Experimental Cold Flow Reactor

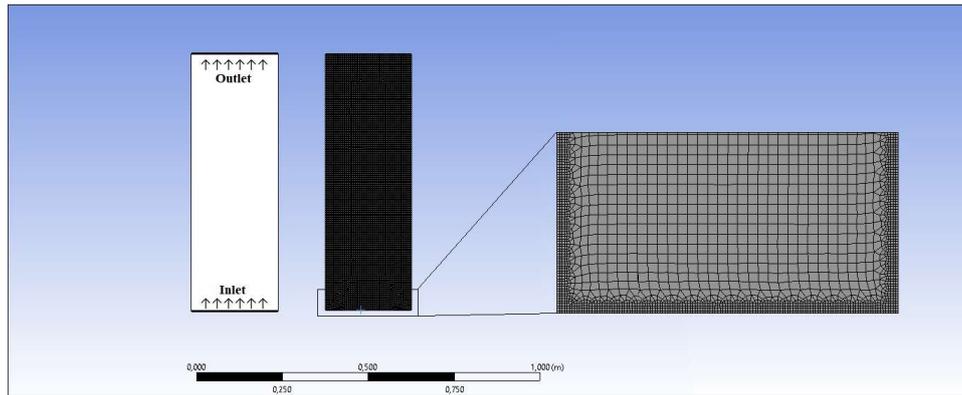


Figure 2 - Mesh and Geometry

The second-order implicit was chosen for transitory formulation. The drag model by Syamlal and O'Brien was chosen only for the volumetric fraction of the solid matter under different inlet velocities. The remaining operational parameters are presented in table 1.

Table 1 - Description of bed material and simulation settings

Descriptions	Base Case Settings	Comments
Granular Viscosity	Syamlal and O'Brien	Fixed Value
Drag Model	Syamlal and O'Brien	Fixed Value
Particle density	2400 kg/m ³	Fixed Value
Particle Diameter	300 μm	Uniform Distribution
Bed Width	0.25 m	Fixed Value
Bed Height	0.75 m	Fixed Value
Static Bed Height	0.15 m	Fixed Value
Initial Solids Packing	0.15 m	Fixed Value
Minimum Fluidization Velocity	0.136 m/s	Specified Value
Inlet Velocity	0.15, 0.35, 0.55 m/s	Specified Value
Outlet Boundary Condition	Pressure	Fully Developed Flow

4. RESULTS AND DISCUSSION

The computational analysis was performed under different inlet gas velocities (from 0.15 m/s to 0.60 m/s). The results analyzed in this work are pressure drop (values extracted from measuring points), bed expansion rate, and bubbles formation (estimated from the simulation contour graphs). Results observations have indicated that bubbles formation and growth were intensified by the increase of inlet velocity. Bed material volumetric fraction has diminished with the growth of inlet gas velocity, indicating fluidization behavior, that has a direct influence on bed expansion.

4.1 Effects of inlet gas velocity

The variation of inlet gas velocity for different ranges in CFD simulation was performed to better understand the fluidization phenomena and determination of the minimal fluidization velocity of the bed. For the 0.15 m/s case (figure 3), little bubble formation was observed and the bed expansion height is almost constant along the elapsed time, with a floating surface behavior due to the low inlet velocity applied to the gas-solid bed. For this case, the bed particles are moving towards the bed surface that hits a peak and reduces after stabilization, remaining almost static. Figures 4, 5 and 6 demonstrate cases of 0.30 m/s, 0.45 m/s, and 0.60 m/s, respectively. For those cases, the inlet velocity has a considerable impact on fluidization phenomena, once the pressure drop and bubble formation are intimately linked with gas inlet velocity. For both cases, the bed height has increased with the rise of gas inlet velocity. At high velocities, the bed material moves towards the top with a turbulent and lean fluidization behavior (voids can be observed). For both cases, the pressure drop in the bottom of the bed is high and diminishes as the height is increased and bed density reduces.

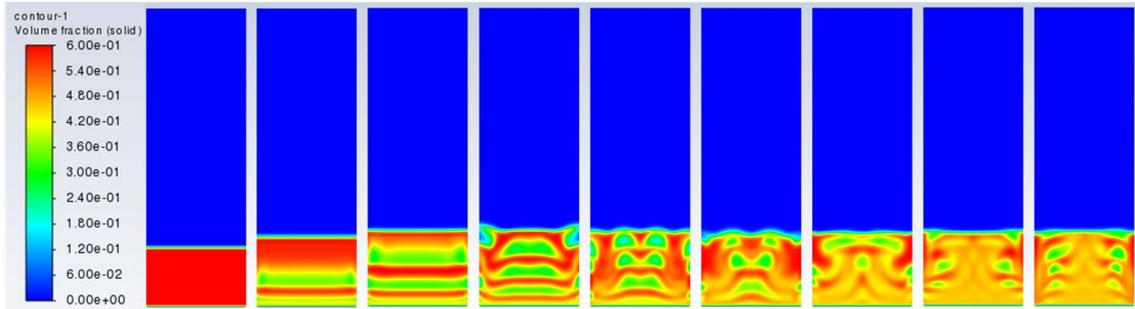


Figure 3 - Bed behavior with an inlet velocity of 0.15m/s

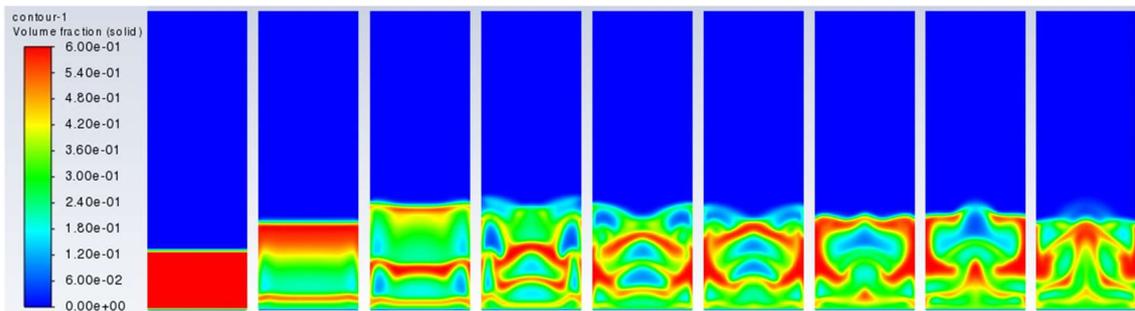


Figure 4 - Bed behavior with an inlet velocity of 0.30m/s

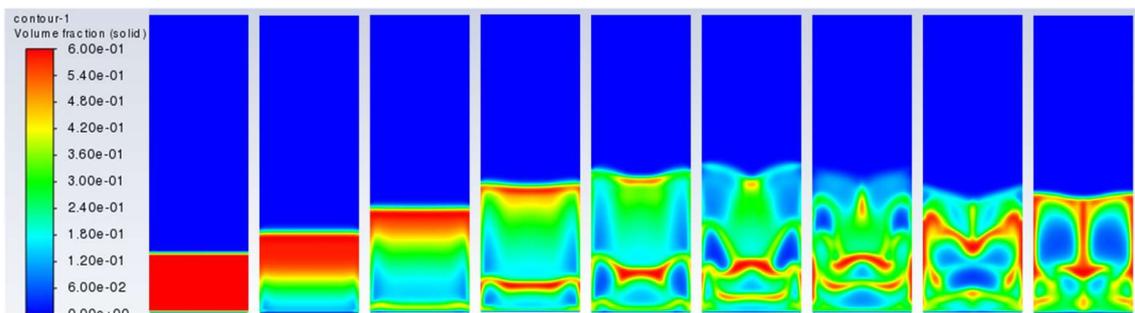


Figure 5 - Bed behavior with an inlet velocity of 0.45m/s

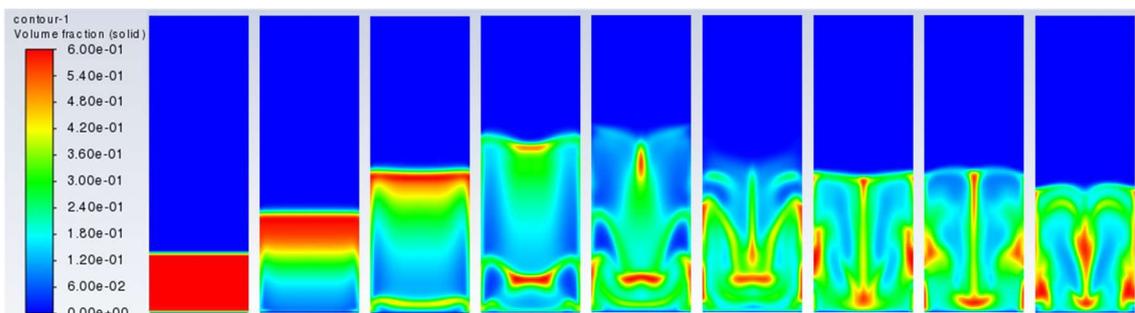


Figure 6 - Bed behavior with an inlet velocity of 0.60m/s

From the result analysis, the velocity of 0.15m/s can be considered as the minimal fluidization velocity, where bed expansion is lower and the weight of particles is balanced by the drag force (Kunii and Levenspiel 1993). The bed can be considered fluidized when using inlet velocities above this minimum value. The condition of the bed using inlet velocities from 0.30 m/s to 0.60 m/s seemed to have a good mixture between the bed material and the gas. For velocities above 0.60

m/s, there must be a caution about elutriation of the bed material, caused by the increase of the gas velocity (Santana, Rodríguez, and Macías-Machín 1999).

4.2 Pressure Drop

The pressure drop inside the reactor occurs mainly due to the solid-gas interaction. The value hits its maximum at the reactor bottom and the minimum at the top, influenced by injection velocity and material density. It can be observed that the pressure drop turns almost constant when all the solid material inside the bed is floating, hitting the fluidization condition. This behavior is shown in figure 7.

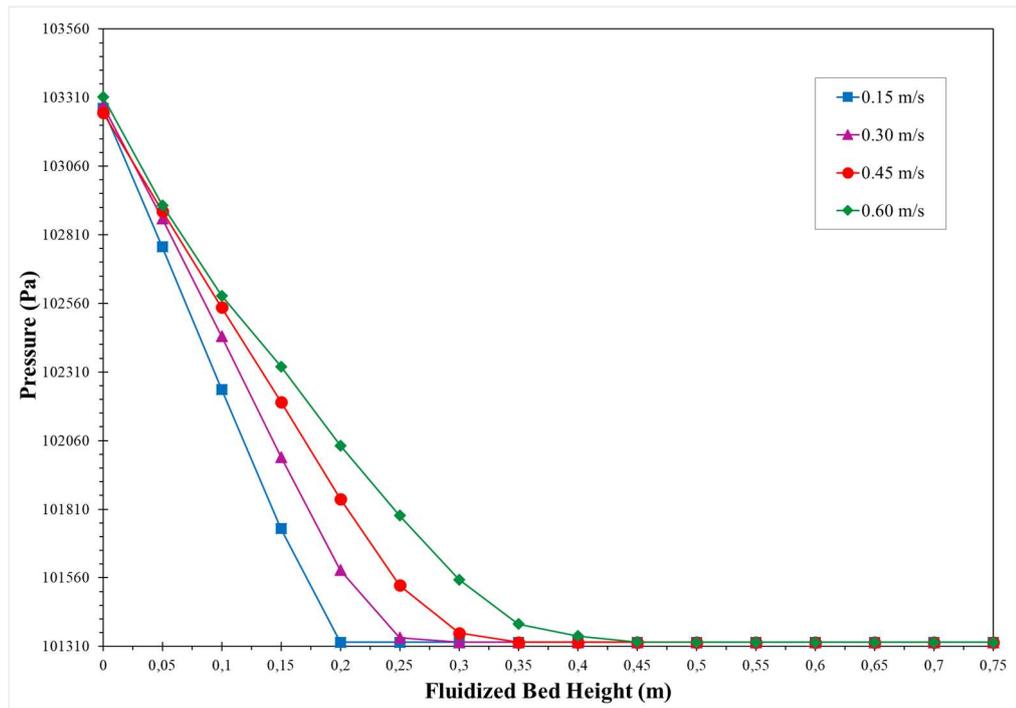


Figure 7 - Variation of bed pressure drop for air velocities at 0.15 m/s, 0.30 m/s, 0.45 m/s and 0.60 m/s

4.3 Bed Expansion

As mentioned before, in contour graphs, initially the bed is expanded and bubbles are generated, due to the growth of gas volumetric fraction. The expanded bed heights considered in this work are not necessarily the maximum heights observed during the simulations. This occurs because the first bubbles generated are compressed by the static bed, concentrating energy, causing the first bubble explosion and material projection at higher heights. After the fluidization condition, the bed height stabilizes below the maximum reached, remaining almost constant during the rest of the simulation (Debnath, Nath, and Chauhan 2020). Therefore, was observed that under higher inlet velocities, the volumetric fraction is high and leads to a high bed expansion. The results of bed expansion are presented in table 2.

Table 2 - Comparison of bed height under different inlet velocities

Inlet Velocities	Static Bed Height (m)	Expanded Bed Height (m)	Percentage of expansion (%)
0.15	0.15	0.186	24
0.30	0.15	0.223	48.6
0.45	0.15	0.281	87.36
0.60	0.15	0.335	123.43

5. CONCLUSION

It was shown that the results obtained through the simulation as bubble formation and growth, volumetric fraction of the bed material, and bed expansion were greatly affected by the different inlet velocities, which was the key parameter investigated in this work. Was observed that under lower inlet velocities (Minimum Fluidization Velocity - U_{mf}) the bed

material behavior was almost constant, observed through the bed expansion which was minimal with low height alterations. Increasing the inlet velocity to double and triple of the U_{mf} , it can be observed that an increase of the bed expansion, bubble formation and size, and pressure drop inside the reactor had occurred.

For the highest inlet velocity, the bed expansion and pressure drop were very distinct if compared with the lower inlet velocities, this was also observed in different works on the literature. The bubble formation was slightly reduced, but its size was increased. This velocity almost caused the elutriation of the bed material on the first expansion. In addition, the mixing of volumetric fractions of the bed material was severely increased with the increase of the inlet velocity.

The main objective of this work was to develop a computational model of a cold flow fluidized bed, the results produced will be used to improve the construction of an experimental apparatus that is under construction. The data of the present work were validated by comparing them with similar works present in technical literature. This contributes to a better understanding of the software as well as model refining (mesh improvement, adequate turbulence models, etc.). The aim is to reach a definitive model and validate it by experiments performed in the experimental apparatus, where different bed materials and inlet velocities can be studied.

6. REFERENCES

- Basu, Prabir. 2013. Biomass Gasification, Pyrolysis and Torrefaction: Practical Design and Theory Biomass Gasification, Pyrolysis and Torrefaction: Practical Design and Theory.
- Debnath, Shubhankar, Harjeet Nath, and Vishal Chauhan. 2020. "CFD Modeling of a Typical Fluidized Bed Column." *Materials Today: Proceedings* (xxxx). <https://doi.org/10.1016/j.matpr.2020.04.079>.
- Dhrioua, Maryem et al. 2019. "Gas Distributor and Bed Material Effects in a Cold Flow Model of a Novel Multi-Stage Biomass Gasifier." *Biomass and Bioenergy* 126(September 2018): 14–25. <https://doi.org/10.1016/j.biombioe.2019.05.001>.
- Fouilland, Thomas, John R. Grace, and Naoko Ellis. 2010. "Recent Advances in Fluidized Bed Technology in Biomass Processes." *Biofuels* 1(3): 409–33.
- Gera, D. et al. 1998. "Computer Simulation of Bubbles in Large-Particle Fluidized Beds." *Powder Technology* 98(1): 38–47.
- Khezri, Ramin et al. 2019. "Computational Fluid Dynamics Simulation of Gas-Solid Hydrodynamics in a Bubbling Fluidized-Bed Reactor: Effects of Air Distributor, Viscous and Drag Models." *Processes* 7(8): 1–16.
- Kunii, Daizo, and Octave Levenspiel. 1993. *74 Powder Technology Fluidization Engineering: Second Edition*.
- Mehrabadi, Mohammad. 2012. "Fluid-Phase Velocity Fluctuations in Gas-Solid Flows."
- Pain, C. C., S. Mansoorzadeh, and C. R.E. De Oliveira. 2001. "Study of Bubbling and Slugging Fluidized Beds Using the Two-Fluid Granular Temperature Model." *International Journal of Multiphase Flow* 27(3): 527–51.
- Perini, Federico, Kan Zha, Stephen Busch, and Rolf Reitz. 2017. "Comparison of Linear, Non-Linear and Generalized RNG-Based k-Epsilon Models for Turbulent Diesel Engine Flows." *SAE Technical Papers 2017-March*(March).
- Sahoo, Pranati, and Abanti Sahoo. 2015. "A Comparative Study on Effect of Different Parameters of CFD Modeling for Gas-Solid Fluidized Bed." *Particulate Science and Technology* 33(3): 273–89.
- Santana, D., J. M. Rodríguez, and A. Macías-Machín. 1999. "Modelling Fluidized Bed Elutriation of Fine Particles." *Powder Technology* 106(1–2): 110–18.
- Sikarwar, Vineet Singh et al. 2016. "An Overview of Advances in Biomass Gasification." *Energy and Environmental Science* 9(10): 2939–77.
- Syamlal, M., and T. J. O'Brien. 1989. "Computer Simulation of Bubbles in a Fluidized Bed." *Fluidization and Fluid Particle Systems: Fundamentals and Applications. Papers Presented At Aiche Annual Meeting, Washington, (270)* }edited by L. S. Fan], New York, U.S.A., American Institute Chemical Engineers, 1989, pp.22-31. (IS).
- Wendt, John F. et al. 2009. *Computational Fluid Dynamics Computational Fluid Dynamics: An Introduction*.
- Wilcox, David C. 1991. "A Review Of The K-w Model." 29th AIAA Aerospace Sciences Meeting.
- Yakhot, V. et al. 1992. "Development of Turbulence Models for Shear Flows by a Double Expansion Technique." *Physics of Fluids A* 4(7): 1510–20.

7. RESPONSIBILITY NOTICE

The author is the only responsible for the printed material included in this paper.