Experimental and computational studies to investigate flow dynamics of Geldart A and Geldart B particles in a Circulating Fluidized Bed, CFB

Subham Kandel*, Niroj Koirala*, Nirajan Raut, Rajan Jaiswal**, Sunil Prasad Lohani*, Britt M.E. Moldestad****

**Department of Mechanical Engineering, Kathmandu University, Nepal*

*(*nepal.subham, ern.koirala)@gmail.com, *splohani@ku.edu.np*

***Department of Process, Energy and Environmental Technology, University College of Southeast Norway, Norway*

***nirajannnraut@gmail.com, (**rajan.jaiswal, **britt.moldestad)@usn.no*

Abstract: Circulating fluidized beds is one of the emerging technologies to convert waste to energy and an attractive method on a large scale. Key components such as the loop seal, gas distributor and cyclone separator play pivotal roles in facilitating solid recirculation and heat transfer within the system. This study focuses on the design and optimization of a CFB reactor using data derived from Barracuda Virtual Reactor software (CPFD). Initially, data from a small scale CFB reactor with main dimensions of 84 mm diameter and a loop seal diameter of 34 mm was utilized for simulation validation. By comparing simulation results with experimental data, the accuracy and reliability of the computational model were ensured. Subsequently, different reactor models were constructed and analyzed to explore various configurations and operating conditions. The results obtained from simulation based design and optimization provided valuable insights into achieving the optimal performance of the CFB system. By refining geometry, efficiency was increased by 32%. Overall, this study contributes to advancing the understanding, application and design modification of CFB technology in waste to energy conversion and large-scale industrial processes.

Keywords: *Circulating Fluidized Bed, Fluidization, Minimum fluidization velocity, CFB, simulation, CPFD, Grid Size*

.

1. INTRODUCTION

In the contemporary energy landscape, the escalating demand for sustainable energy solutions has accelerated efforts to harness energy from renewable sources. Among various technologies, fluidized beds stand out for their efficiency in biomass gasification and combustion, attributed to superior mixing, enhanced heat transfer and uniform temperature distribution (Moradi *et al.,* 2020).

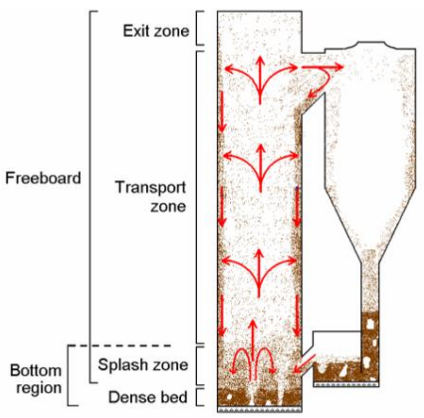

Fig. 1. Schematics of CFB (Pallarès, 2008).

Fluidization is a process wherein solid particles in a loosely packed bed exhibit fluid-like behavior when gas is blown upwards through them. In gas-solid systems, gas is introduced at the bottom of a column containing particles, causing them to vibrate and spread out to balance the drag force from the gas. As the gas velocity increases, this drag force equals the weight of the particles, resulting in a fluidized bed. Various flow patterns can emerge depending on the gas velocity, as illustrated in Fig. 2.

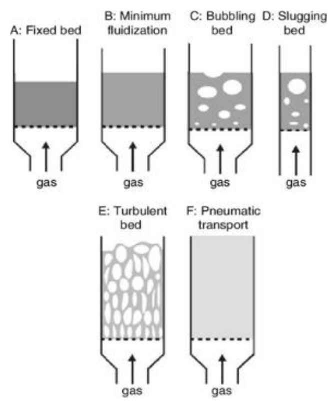

Fig. 2. Schematic of fluidized bed in different regimes (Soomro *et al.*, 2012).

These regimes include fixed, bubbling, slugging, turbulent and pneumatic conveying. The transition from a fixed bed to a fluidized bed occurs at the minimum fluidization velocity, the gas velocity needed to suspend the solids in the gas stream. At very high gas flow rates, a substantial portion of solid is ejected into the space above the bed, leading to significant particle loss through elutriation and entrainment (Huang, 2006).

To mitigate this issue, gas-solid separators are employed to capture and return most particles to the bed (Soomro *et al.*, 2012). Maintaining stable and continuous solid circulation in a gas-solid system has led to the development of Circulating Fluidized Beds (CFB's) as shown in Fig. 1, for large-scale processes, particularly in the petroleum and power generation industries (Huang, 2006). CFB's function similarly to bubbling beds but with substantially higher fluid flow velocities, resulting in more intense mixing and enhanced gas-solid contact. The high relative velocity between gas and solid particles leads to exceptionally high rates of heat and mass transfer. However, the increased gas velocities and recirculation of solids make CFB systems more costly in terms of power consumption and investment compared to conventional fluidized bed reactors (Moradi *et al.*, 2020).

The efficiency of CFBs is highly dependent on flow behavior making it crucial to understand this behavior for scaling, designing and optimization. Over the past decades, Computational Fluid Dynamics (CFD) has emerged as a valuable tool for predicting flow behavior in fluidized bed processes. However, further model development and validation are necessary (Pallarès, 2008).

The experiment utilized sand particles as the bed material and air as the fluidizing fluid. Sand is commonly used in fluidized bed reactors to enhance the mixing of fuel with the fluidizing gas, improving mass and energy transfer. In a biomass CFB reactor, biomass rapidly reacts with the gas to produce synthesis gas and char. The unreacted char circulates with the sand particles through the system and back to the reactor, with sand primarily controlling the circulation behavior rather than the char (Niven, 2002). Various design modifications were implemented on the cyclone, adjustments to the recirculating pipe inlet angle and the height of the recirculating pipe. Among the different design variants, the one exhibiting the maximum particle recirculation rate was selected.

This work underscores the importance of precise modeling and experimentation in optimizing fluidized bed reactors for biomass gasification and combustion, paving the way for more efficient and sustainable energy production.

2. MP PIC MODEL DESCRIPTION

The gas phase mass and momentum conservation can be modelled by the volume averaged Navier-Strokes equation and are used as a continuum on a Eulerian grid (Chutima Dechsiri, 2004).

$$\frac{\partial(\theta_f\rho_f)}{\partial t} + \nabla.(\theta_f\rho_f u_f) = 0 \tag{1}$$

$$\frac{\partial(\theta_f\rho_f u_f)}{\partial t} + \nabla.(\theta_f\rho_f u_f) = \nabla p - F + \theta_f\rho_f g + \nabla.(\theta_f\tau_f) \tag{2}$$

$$F = \iint f m_s \left[D_s(u_f - u_s) - \frac{1}{\rho_s}\nabla p\right] dm_s du_s \tag{3}$$

where θ_f, ρ_f, u_f, τ_f are fluid phase volume fraction, density, velocity and stress tensor and m_s, u_s are the mass and velocity of the particle. F is the total momentum exchange with particle phase per volume, g is the acceleration due to gravity and p is the pressure.

The solid phase can be modelled by a particle distribution function given by equation 4 (Snider, (2001)). Considering the time rate of change of above equation the Liouville equation is obtained. This equation assumes that there are no direct collisions or particle breakup.

$$f(x, m_s, u_s, t)dm_s du_s$$

$$\frac{\partial f}{\partial t} + \nabla_x.(f u_s) + \nabla u_s.(fA) = 0 \tag{4}$$

The particle acceleration, A as a function of aerodynamics drag, buoyancy, gravity and interparticle normal stresses can be expressed as,

$$A = D_s(u_f - u_s) - \frac{1}{\rho_s}\nabla p + g - \frac{1}{\theta_s\rho_s}\nabla\tau_s \tag{5}$$

The particle volume fraction, θ_s and the particle stress τ_s, which are used to calculate the interparticle collisions and are expressed as

$$\theta_s = \iint f\frac{m_s}{\rho_s}\, dm_s du_s \tag{6}$$

$$\tau_s = \frac{10P_s\theta_s^{\beta}}{max[(\theta_{cp} - \theta_s), \epsilon(1 - \theta_s)]} \tag{7}$$

Here, P_s, β, θ_{cp} are the constant term related with pressure, is a constant, particle volume fraction equals the close pack volume (Andrews *et al.*, 1996).

3. METHOD AND COMPUTATIONAL MODEL

3.1 Experimental Model

The experiment was conducted at ambient temperature using sand particles with diameter ranging from 63 to 200 µm and density of 2650 kg/m³. Particle size distribution analysis was performed prior to the experiment revealed a mean particle size diameter of 116 µm. The gas flow rate was varied from 0 to 650 SLPM in increments of 50 SLPM. Pressure transducers were installed at various locations, and data acquisition was facilitated using LabVIEW software.

3.2 Computational Model:

Following the measurement of the dimensions of the CFB at the University of South-Eastern Norway, a CAD geometry was created using SOLIDWORKS 2020. Various simulation models with different grid sizes were analyzed before finalizing the computational model. Experimental pressure transducer data were primarily used to validate the model (Figs. 4 and 5). The gas inlet of the riser and the loop seal were configured as flow boundary conditions, while the top of the cyclone was set up as a pressure boundary condition. The simulation time step was set to 0.0005 seconds (Bandara *et al.*, 2018), with a total simulation duration of 45 seconds. The maximum momentum from particle collision redirection was assumed to be 40%, and the default values of 0.85 were used for normal and tangential wall collisions.

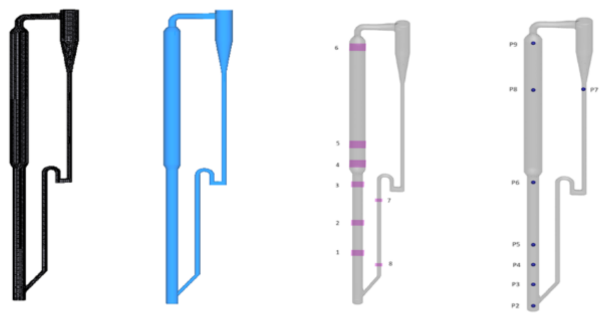

Fig. 3. (a) Grid (b) CAD Geometry (c) Flux Plane (d) Pressure reading Points.

4. RESULT AND DISCUSSION.

4.1 Model Validation:

After conducting multiple simulations with varying grid sizes and drag models, the Wen-Yu Ergun drag model was selected. As illustrated in Figs. 4 and 5 *(experiment with trim means first 180 sec data has been excluded on an experiment of 3600 sec*), the minimum deviation was observed for a grid size of 80,000 cells. Increasing the grid size beyond this point resulted in deviations, as the grid size became smaller than the particle size. The relationship between cells to computational particles is shown in Table 1.

Table 1. Computational Particle to Cell Ratio

Grid	Cells	Computational Particles	Computational Particle to Cell Ratio
01	40000	35640	0.891
02	60000	56400	0.940
03	80000	77792	0.972
04	80000	77800	0.973
05	100000	96768	0.968
06	120000	111384	0.928

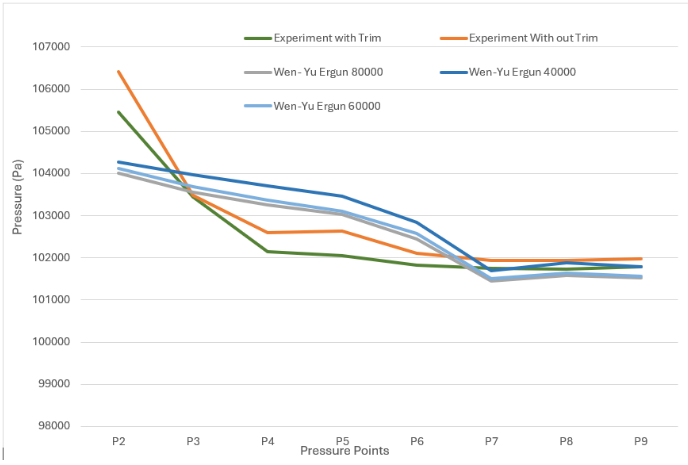

Fig. 4. Pressure Variation with Pressure point (Different grid size).

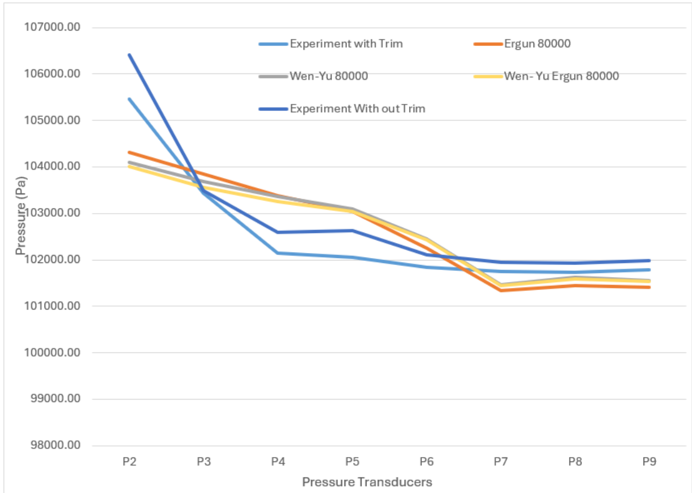

Fig. 5. Pressure Variation with Pressure point (Different drag model).

4.2 Impact of changing the diameter of Cyclone Keeping height constant

The diameter of the cyclone was varied while keeping the height constant to investigate its impact on particle circulation rate. Adjusting the diameter of a cyclone separator in a circulating fluidized bed (CFB) yielded various outcomes in particle circulation rate. In the design, with a height to diameter (H/D) ratio of 2.78, it was observed a 1.6% increase in particle circulation rate as shown in Figs. 7 and 8 and Table 2. The CAD models varying diameter is shown in Fig. 6.

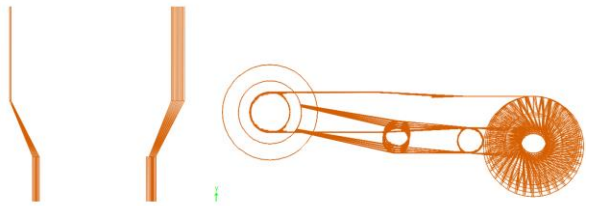

Fig. 6. Variation in diameter keeping height constant.

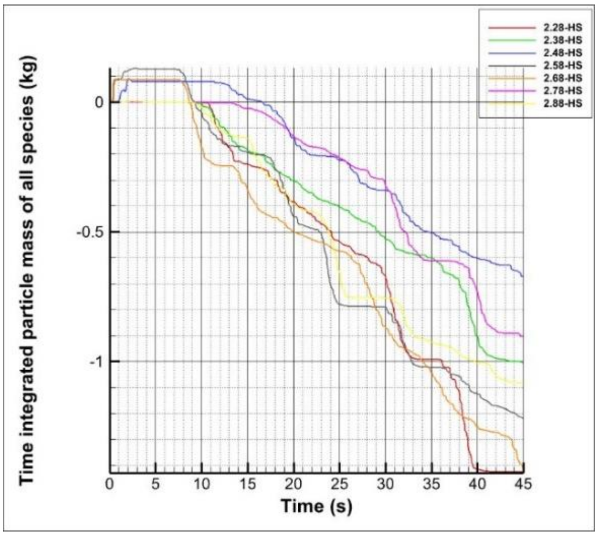

Fig. 7. Time Integrated particle mass of all Species (Seventh Plane).

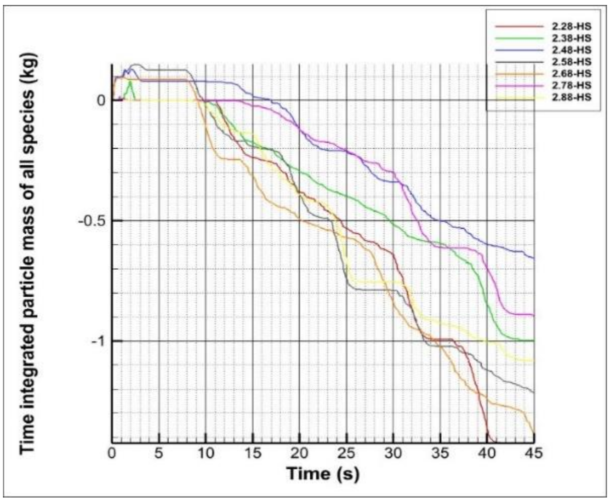

Fig. 8. Time Integrated particle mass of all Species (Eighth Plane)

Table 2. Change in circulation rate keeping height constant (305 mm)

H/D	Diameter (mm)	Circulation Rate (kg)	Change in Circulation Rate (%)
2.28	133.77	1.02	-16.4
2.38	128.15	1.17	-4.1
2.48	122.98	0.80	-34.4
2.58	118.21	1.22	0.0
2.68	113.80	0.79	-35.2
2.78	109.71	1.24	1.60
2.88	105.90	0.75	-38.5

4.3 Impact of changing the height of cyclone keeping diameter constant:

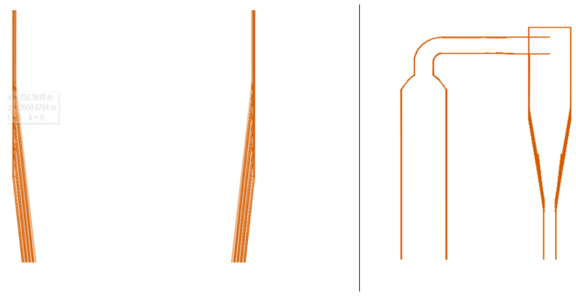

Fig. 9. Variation in height keeping diameter constant.

Changing the height of a cyclone can have a direct impact on both gas and particle residence times. The longer the residence time, the more opportunity particles have to be separated from the gas stream by centrifugal force. A taller cyclone can enhance both the separation efficiency and the rate of circulation, as particles have more time to move towards the cyclone wall under the influence of centrifugal force.

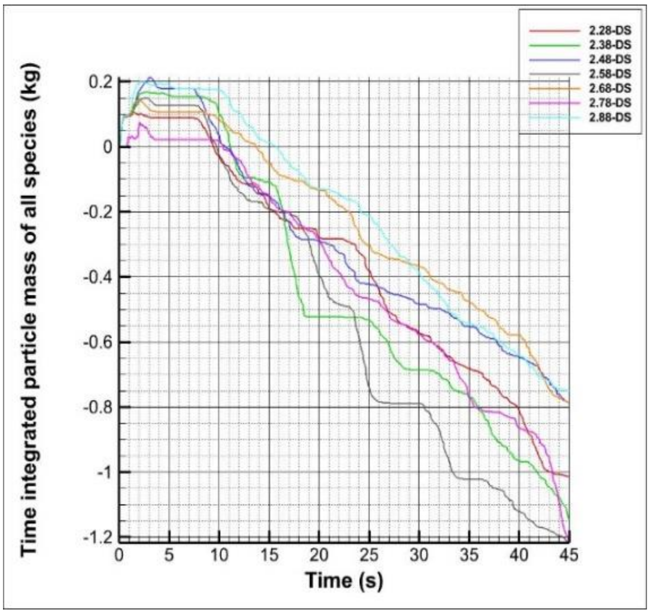

Fig. 10. Time Integrated particle mass of all species (Seventh Plane).

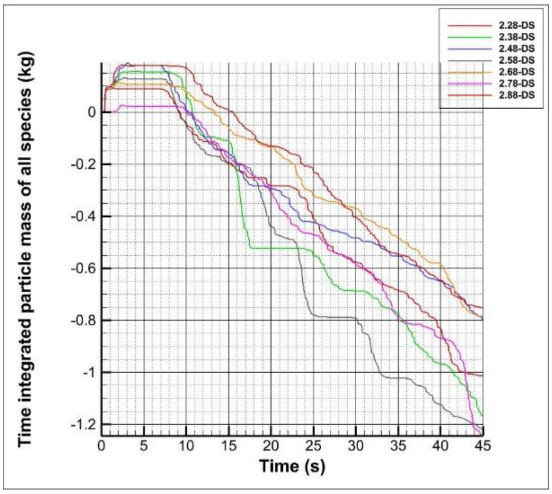

Fig. 11. Time Integrated particle mass of all species (Eighth Plane).

Table 3. Change in circulation rate keeping diameter constant (118 mm)

H/D	Height (mm)	Circulation Rate (kg)	Change in Circulation Rate (%)
2.28	269.04	1.40	14.75
2.38	280.84	1.00	-18.03
2.48	292.64	0.65	-46.72
2.58	304.44	1.22	0.00
2.68	316.24	1.42	16.39
2.78	328.04	0.89	-27.05
2.88	339.84	1.08	-11.48

In the design modification, with an H/D ratio of 2.68, it was observed a 16.39% increase in the particle circulation rate. Figs. 10 and 11 shows particle circulation rate for 45 sec on flux plane 7 and 8. *Several factors influence the particle circulation rate, and a design optimized for a particular particle size may not be effective for a wide range of particle sizes.*

4.4 Impact of changing the angle of recirculation pipe (Return Leg /Downcomer)

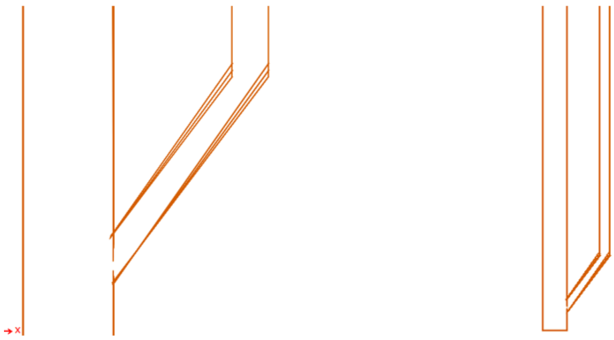

Fig. 12. Variation in inlet angle of return leg.

The angle of the recirculation pipe significantly affects particle movement to the riser in a Circulating Fluidized Bed (CFB). Typically, increasing the angle of the downcomer enhances gravitational forces, aiding particle movement. This improvement results in a higher particle recirculation rate, as particles flow more smoothly and quickly to the riser. Additionally, the risk of blockage decreases with steeper angles, as particles are more likely to settle and move smoothly within the downcomer.

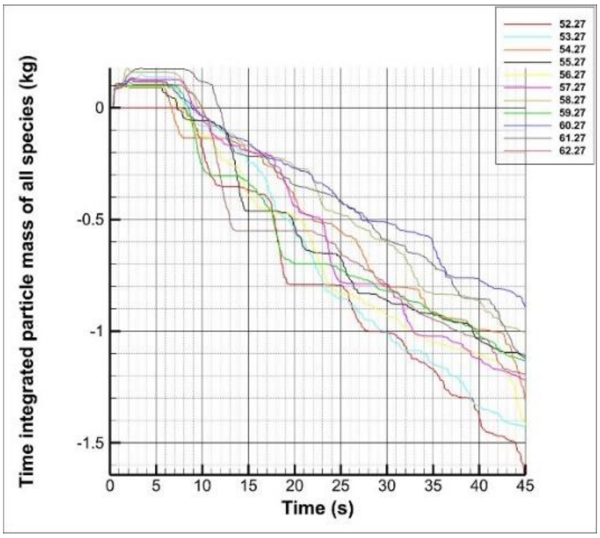

Fig. 13. Time Integrated particle mass of all species (Seventh Plane).

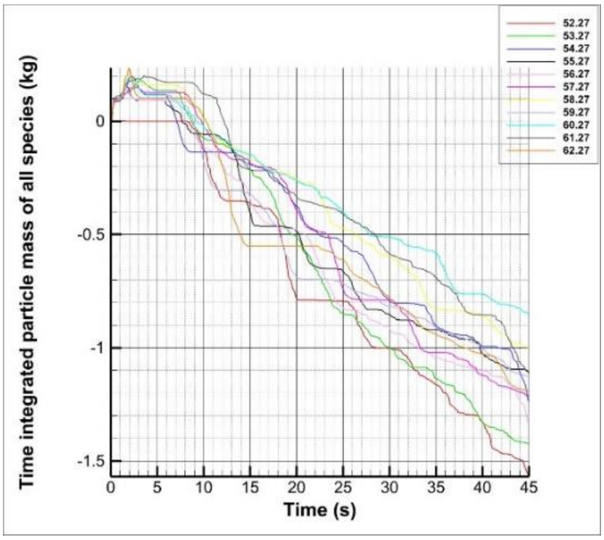

Fig. 14. Time Integrated particle mass of all species (Eigth Plane).

To assess the impact on particle recirculation rate, 11 design variations were implemented, altering the angle from 52.27° to 62.27°. The angle of the return pipe used in the experiment was 57.27°. In the observation, the maximum circulation rate occurred at an angle of 52.27°. Although this result appears unusual, it can be attributed to the backflow of particles from the riser to the loop seal, causing blockages and pushing particles backward towards the standpipe rather than the downcomer. As long as there is backpressure from particles inside the riser, the circulation rate will decrease. Therefore, the optimal angle was found to be 52.27 degrees, despite it being a shallower return angle. The circulation rate varying return leg angle is shown in Figs. 13 and 14.

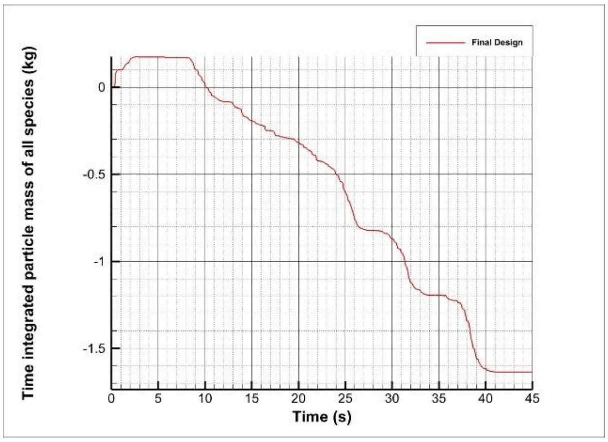

Fig. 15. Final Design.

In the final design, each individual component was optimized to maximize circulation rate, resulting in a 32% increase in particle circulation rate as shown in Fig. 15.

5. CONCLUSIONS

This study examines the influence of design parameters on the particle circulation rate in a circulating fluidized bed with a mixture of Geldart A and B particles. Key design modifications included varying the cyclone diameter while keeping the height constant, varying the height while keeping the diameter constant and adjusting the angle of the recirculating pipe. An initial H/D ratio of 2.58 was optimized to 2.78 (height same, diameter 109.7 mm) and 2.68 (diameter same, height 316.2 mm) for improved particle circulation. The optimal recirculation angle was found to be 52.27 degrees. The results are influenced by multiple factors, including particle size distribution, particle to particle and particle to wall collisions, particle breakdown after collisions and velocity profile irregularities. Consequently, a design suitable for one particle size may not be valid for a wide range of particles.

REFERENCES

Andrews, M. J. and P. J. O'Rourke (1996). "The multiphase particle-in-cell (MP-PIC) method for dense particulate flows." International Journal of Multiphase Flow 22(2): 379-402.

Bandara, J. C., Nielsen, H. K., Moldestad, B. M. E., and Eikeland, M. S. (n.d.). Sensitivity Analysis and Effect of Simulation parameters of CPFD Simulation in Fluidized Beds (2018).

Chutima Dechsiri (2004) Particle Transport in Fluidized Beds: Experiments and Stochastic Models. https://pure.rug.nl/ws/portalfiles/portal/9807341/thesis.pdf

Huang, Y. (2006). Dynamic model of circulating fluidized bed. https://researchrepository.wvu.edu/etd/2749

Moradi, A., Samani, N. A., Mojarrad, M., Sharfuddin, M., Bandara, J. C., and Moldestad, B. M. E. (2020). Experimental and computational studies of circulating fluidized bed.

Pallarès, David., and Chalmers tekniska högskola. Department of Energy and Environment. Division of Energy Technology. (2008). Fluidized bed combustion : modeling and mixing. Chalmers University of Technology.

R. K. Niven, Physical insight into the Ergun and Wen and Yu equations for fluid flow in packed and fluidised beds, Chemical Engineering Science, vol. 57, no. 3, pp. 527–534, Feb. 2002.

Soomro, A., Samo, S. R., and Hussain, A. (2012). Fluidization in cold flow circulating fluidized bed system. In Energy, Environment and Sustainable Development, pp. 161–173. Springer-Verlag Vienna.

Snider, D. M. (2001). An Incompressible Three-Dimensional Multiphase Particle-in-Cell Model for Dense Particle Flows. Journal of Computational Physics 170(2), 523-549.