Full Loop Numerical Simulation of Fluidization Characteristics at Cyclone and Furnace of 110 MW CFB

A Wirahadi¹,²,a, P Batubara¹,²,b

¹ Commercial Department, Pembangkitan Jawa Bali, Ketintang Surabaya, Indonesia
² Engineering Department, Prima Layanan Niaga Suku Cadang, Jakarta, Indonesia
a ari@plnsc.co.id, b pesu@plnsc.co.id

Abstract Circulating fluidized bed (CFB) boiler has strong points with its recognition when operating with low quality coal contrast to pulverized coal boiler. Boiler load could be improved by increasing coal input and air ratio combustion adjustment. That is why ratio combustion between secondary and primary air is one of valuable parameters influencing fluidization of CFB Boiler. CFB boiler is consist mainly of a furnace, two cyclones, and forced loop pipes, where cyclones are boiler parts that were indicated to have high velocity fluidization up to 30 m/s. Full loop 3D simulation of CFB with focusing on the furnace and cyclone by using computational fluid dynamic (CFD) program was implemented using Eulerian multiphase model to investigate sand volume of fraction, air, and sand velocity including pressure distribution around Cyclone. Several air ratio combustions between primary and secondary air were simulated such as 55%-45%, 50%-50% with 63%, and 100% load variations. It was shown that operation with air combustion ratio 50%-50% and 55%-45% leads to good fluidization at 63% load, while 100% and 110% fluidization will cause an abundant sand entered to inlet cyclone and induce higher sands and air velocity.

1. Introduction
Circulating fluidized bed (CFB) boiler is one of implementation of the fluidized bed boiler technology. It has acquired recognition, especially in the industrial power-generation users, for its various useful purposes, such as ability to operate with low rank coal combustion and its smaller consequences on the environment where it has low NOx emission. Implementation on the bench trial, prototyping on a commercial scale to get the CFB combustor design with high efficiency and low emissions actually is very costly [1]. Until now, the application of these experiments is still performed, while interest in other approaches through numerical simulations is getting bigger along with the fast development of computing technology including computational fluid dynamics or it was known as computational fluid dynamic (CFD) [2].

Fluidization occurred because solid particles were in a state of suspension through the liquid or gas or when the velocity of the fluid or gas velocity was getting higher. But if there are solid particles in which an air or gas fluid is passed through it at low speed, the solid particle remains undisturbed and fluidization does not take place [3]. Eulerian multiphase and k-standard for turbulence models were employed in one of research regarding fluidization. It was showed that k-ε more accurate and precise that maximum air velocity happened in areas far from wall where air velocity negative [4]. In large-scale CFB simulations the Euler Simulation is commonly implemented. In the two-fluid model or
often called the Eulerian, the gas and the granular phase were considered to be continually fully penetrating. In this case, generalizing the Navier-Stokes Equation for interacting media was the equation that was implemented [5].

The Study examined air area and fluidization of sands could reach up to 30 m/ and damage to the cyclones surrounding has been done. It was found that abrasion arising on the walls of the cyclone by both air or sand potentially happened [6]. But until now, complete understanding of the effect of operating parameters on the mechanism of mixing gas-solid particles is not fully understood. Problems due to insufficient fluidization still arise such as agglomeration which then leads to de-fluidization including the appearance of abrasion on cyclones. With proper operating parameters, such as primary and secondary air distribution that are well understood, it is expected that good fluidization can be achieved. Therefore, proper study regarding primary and secondary air distribution in cyclones is really needed. There were several numerical simulations related to those interests, as one study about 30 MW CFB Boiler that was investigated with various effects of five different primary and secondary air ratios with full load only [7] [8]. Other study used 3D model with 406 Mwe but without load and air ratios variations [9]. 3D model simulation for boiler implemented too in supercritical type, but it only covered furnace area [10]. There was a study that concluded higher erosion rate was influenced by higher fluidizing air velocity [8]. Simulations with various primary and secondary air ratios and 3 different loads were conducted [9]. But it was focused only on the furnace and did not explain anything deeper related to cyclone.

In this simulation, Commercial CFD were used to study the effect of various primary and secondary air ratios with several load variations on fluidization characteristics on cyclone areas. The boiler was modelled as isothermal without a combustion process.

2. Method
This simulation was conveyed using commercial software started from pre-processing, processing, and post-processing processes. At the pre-processing stage, there were several steps that were carried out, such as: boiler test object modelling, meshing in the domain and determining the boundary conditions and parameters.

At the processing stage, the meshing and domain results in numerical simulations were exported to the solver. Some of the arrangements were made including models, materials, boundary conditions, operating conditions, control, and monitoring conditions, and initialize conditions.

Post-processing is the graphical appearance of the results and analysis. It was obtained in the form of qualitative data and quantitative data. Quantitative data in the form of pressure and speed distribution. While qualitative data in the form of flow visualization by displaying path lines, contour plots, and velocity profiles of gas-particle flow with variations in flow velocity then the results were analyzed and compared.

2.1 Governing equation
Gas-solid flow in a fluidized bed was based on the conservation equations of continuity and momentum. So that, in this study the Eulerian multiphase model was executed. Turbulence kinetic energy (k) and its dissipation rate (ε) were the originate model of the standard k-ε model. Below were the transport equations based on these model [10]:

\[
\begin{align*}
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) &= \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{1} \\
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) &= \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 \varepsilon \frac{\varepsilon}{k}(G_k + C_3 \varepsilon G_b) - C_2 \rho \frac{\varepsilon^2}{k} + S_\varepsilon \tag{2}
\end{align*}
\]

Wherein the generation of turbulence kinetic energy related to the gradients of mean velocity is considered to be \(G_k\), the generation of turbulence kinetic energy related to buoyancy is considered to
be by $G_b$, the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate is considered to be $Y_m$. Meanwhile $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are calculated as constants. In this study user-defined source terms where represented by $S_\varepsilon$ and $S_e$ were neglected. Governing equations aforementioned were implemented in a common CFD program.

2.2 Geometry and mesh
In this simulation, it used Wuxi huaguang CFB Boiler which was placed in Nagan, Aceh, Indonesia. This natural-circulation CFB boiler load capacity is 110 MW with its steam generation reaching 382 ton/h. For the main part formed by a furnace, two cyclone separators and Forced loop pipes. The furnace geometry is 3.2m x 14.4m x 36.3m. In this study a Full loop boiler with cyclone and loop seal is simulated without a bypass section. Figure 1 represents simulation domain. It covered 4 rectangular inlets of coal located in front of the furnace as mass flow inlet. There were 9 inlet pipes of secondary air located in front side of the furnace. On the rear side there were another 12 pipes of secondary air inlets. Velocity inlet as boundary conditions were used for secondary air. For simplicity, the primary air entering is assumed to be completely through the bottom of the furnace. This assumption was also used in HPFF (High Pressure Fluidizing air) sections [2]. It used mass flow inlet boundary conditions; Pressure outlet boundary condition was only applied on cyclone outlet. All these settings were resumed on Table 3.

For boiler meshing, it was generally meshed with hexahedron which had previously been divided into 64 volumes. Several parts such as lower cyclones and loop pipes were used tetrahedron for its meshing. It was generated from 445.812 nodes or 497.311 elements where all meshes size scale is below 0.2 m with relative centre and smoothing medium.

![Figure 1. Domains of simulation](image)

2.3 Simulation setting
Sutherland formulation was used to calculate ideal gas formulation and air viscosity [10]. It was summarized on Table 1. As a note the boiler was assumed to operate at constant operational temperature. In the beginning of simulation, sands were patched in with volume of fraction 0.4 and height 2.3m. Table 3 summarized the setting for sands phase properties with reference from Zhang et al [2]. Meanwhile several data were referred to Sudarmanta et al such as density, diameter, and viscosity of sands [11]. The eulerian model was used for the multiphase model to define gas-solid phase and its interactions. Because of its general applicability, robustness, and efficient, the turbulence model used in this simulation was standard k-ε [8]. Only two phases of setting were used, and Coal inlet was assumed as ideal gas. As mentioned earlier, this combustion model was assumed as isothermal.
Table 1. Air input data and its properties

| Cases | Load [MW] | Load [%] | PA-SA (%) | Air Combustion (NM³) | Primary [kg/m³] | Secondary [kg/m³] | Air Viscosity [kg/m-s] | Boiler Temp [°C] | Boiler Pressure [Pa] | Outlet Pressure [Pa] |
|-------|-----------|----------|------------|----------------------|----------------|------------------|-----------------------|-----------------|---------------------|---------------------|
| 1     | 71        | 63       | 50-50      | 154.56               | 154.56         | 0.3063           | 4.51 x 10⁻⁵          | 876             | -303                | -1080               |
| 2     | 71        | 63       | 55-50      | 170.02              | 139.11         | 0.3063           | 4.51 x 10⁻⁵          | 876             | -303                | -1080               |
| 3     | 110       | 100      | 50-50      | 191.13              | 191.14         | 0.2934           | 4.63 x 10⁻⁵          | 928             | -165                | -1480               |
| 4     | 110       | 100      | 55-45      | 211.94              | 170.33         | 0.2934           | 4.63 x 10⁻⁵          | 928             | -165                | -1480               |
| 5     | 121       | 110      | 50-50      | 210.25              | 210.25         | 0.2724           | 4.83 x 10⁻⁵          | 1020            | -165                | -1480               |
| 6     | 121       | 110      | 55-45      | 231.27              | 189.22         | 0.2724           | 4.83 x 10⁻⁵          | 1020            | -165                | -1480               |

2.4 CFD Solver
When it came to solution or solver, for pressure-velocity coupling, the Phase Coupled Semi-implicit method for pressure linked equation (PC-SIMPLE) was used [12]. For the momentum, volume fraction, turbulence kinetic energy, and turbulence dissipation rate were solved by first order upwind. Sequences of Iterations were implemented by transient setting with time step size of 0.2s until maximal iteration/time step of 20s. It finished around 500 number of time steps until the parameter of residual reached below 10⁻³. Standard initialization was applied with based values from inlet Primary Air.

Table 2. Types and phase of boundary condition setting

| Boundary Condition       | Unit type         | Phase |
|--------------------------|-------------------|-------|
| Primary air              | Mass Flow Inlet   | Air   |
| Secondary Air Front      | Velocity Inlet    | Air   |
| Secondary Air Front      | Velocity Inlet    | Air   |
| Coal Inlet               | Mass Flow Inlet   | Air   |
| HPFA Inlet               | Velocity inlet    | Air   |
| Flue Gas Outlet          | Pressure-outlet   | Mixture |

Table 3. Sands input data and its properties

| Properties                  | Input             |
|-----------------------------|-------------------|
| Density [kg/m³]             | 2500              |
| Diameter [m]                | 0.0002            |
| Frictional Pressure         | based-ktgf        |
| Friction Packing limit      | 0.61              |
| Granular Temperature        | Algebraic         |
| Solid Pressure              | lun-et-al         |
| Viscosity [kg/m-2]          | 0.0013            |
| Granular Viscosity          | Gidaspow          |
| Granular Bulk Viscosity     | lun-et-al         |
| Frictional Viscosity        | Schaeffer         |
| Angle of Internal Friction  | 30.00007          |
3. Results and Discussion

3.1 Superficial velocity

Term of Superficial velocity means air velocity only. It has impacted the fluidization of sands caused by higher superficial velocity. Ari et al concluded that greater primary air contributed to superficial velocity more than 10 m/s around central x axes of furnace t=50s at every load variations with larger area [9]. It showed contour of superficial velocity without showing specific values on the furnace superficial velocity. It concluded that superficial velocity would increase when load or primary air were increased.

In this study, the air velocity distribution was analyzed from the z-axis (upward furnace) airspeed plot to the distance from the center point of the furnace at a height of 1 meter above the bottom furnace. Data collection at a height of 1 meter above the bottom furnace was carried out because in this area there was a dense bed so that it can be used to find out how the distribution and value of superficial air velocity where superficial air was the air velocity used for contact with sand particles.

Figure 2a shows the velocity plot in the direction of z (top) where the velocity at each load did not exceed 3 m/s and is generally at 2 m/s which shows the air velocity entering the turbulent transition (1.76 m/s). The plot in figure 2b also shows the same tendency where the velocity towards z did not exceed 3 m/s. So, it can be concluded that the load addition which in this case was the addition of combustion air capacity did not directly affect the superficial air velocity in the bottom furnace. This is slightly different from Wijayanto et al which concluded that the addition of primary air increased superficial speed [11]. This difference could be caused by several things, including the difference in the ratio of air that is not too far away at the same load or is possible due to the simplification of the primary air nozzle where air was considered to flow entirely from the bottom of the furnace without the nozzle.

![Figure 2a: Superficial velocity along furnace with (a) 50-50% ratio and (b) 55-45% Air ratio](image)

3.2 Furnace Pressure

The only parameter could be extracted from boiler operational data is pressure. The other parameters such air superficial velocity and sands fraction were not practical to be extracted in the operation of the boiler. It was shown on Figure 3 that pressure drop of static pressure data for each variation of load and air combustion ratio at center x axes of furnace t=50 along furnace height. Pressure input at t=0 for each variation was not the same, but the final pressure on the top of the furnace had a tendency...
that additional load or primary air would give to higher pressure on the top at t=50s. Identical results appeared from Wijayanto et al for higher primary air input [8].

3.3 Chart of fine solids volume of fraction
These sub-chapters have results for the distribution of volume fractions of sand that are relatively smaller or more commonly called fine particles. Fine particles were different from dense beds which tend to only fluctuate in the lower furnace, which was useful for the process of fluidization. Fine particles tend to be smaller and not lumpy like dense beds. The number of fine particles reaching into the cyclone could cause destructive abrasion in the area. It was necessary to adjust the value of a very small volume fraction range so that the resulting contour is more informative for the analysis of the distribution of sand particles (fine particles) In this simulation, a range of 0 to 0.05 was selected to see areas with high sand volume fractions.

Figure 3. Pressure versus height of center line furnace t=50s
Kinkar et al research was focusing on cyclone areas and it needed to be figured out deeply [6]. So that it was compared in this simulation, where data extraction was carried out at the midpoint of the right cyclone along the 2.4 meter in accordance with Figure 4. It has the purpose to find out the value of the sand volume fraction on those specific areas. Based on data from Figure 5, it was concluded that the higher the load, the higher the volume fraction that entered to the cyclone, as well as with the addition of primary air indicated the higher sand volume fraction appeared. At 70 MW 50-50 showed a very small fraction of sand volume.

3.4 Sands velocity
To get good quantitative data, extraction of the data into a graph to get the x and y direction velocity vector at the cyclone inlet in accordance with the target area in Figure 4 has been done. So that the sand velocity obtained in the X and Y direction as shown in Figure 6 and 7 which showed the speed of sand in the x direction to enter the Cyclone to reach 16.7 m / s while the y direction reached 4.98 at a load of 120 MW 55-45. The conclusion from both graphs is that the increase in load and increase in primary air caused sand velocity to increase.
4. Conclusion

Full loop simulation used CFD gave so much information related to fluidization characteristics that cannot be done through experiments especially when micro study regarding boiler part is needed such as furnace or cyclone area. This numeric simulation has been done at two load variations and three alternative air combustion ratios. The outcomes were shown in the chart of superficial velocity, furnace pressure, fine solids volume of sands fraction and including sand velocity as parameters that affected fluidization. This study has same direction of Ari et al conclusion that operation with air combustion ratio 50%-50% and 55%-45% leads to good fluidization at 63% load, meanwhile fluidization 100% and 110% with all those air combustion ratios will cause a great number of sands entered inlet cyclone also higher sands and air velocities. All these results are essential for power plant engineer’s knowledge of fluidization of CFB boiler components.
References

[1] Bona L., Zhang N., Wang W. and Li J. 2013 3-D Full-Loop Simulation of an Industrial-Scale Circulating Fluidized Bed Furnace. American Institute of Chemical Engineers Journal 59. doi:DOI 10.1002/aic.13917

[2] Zhang N., Bona L., Wang W. and Li J. 2010. 3D CFD simulation of hydrodynamics of a 150 MWe circulating fluidized bed furnace Chemical Engineering Journal 162 821-828

[3] Ganesan T and Lingappan S. 2013 A Survey on Circulating Fluidized Bed Combustion Boilers International Journal of Advanced Research in Electrical, Electronics and Instrumentation Engineering. 2 8

[4] Tanskanen V. 2005 CFD Study of Penetration and Mixing of Fuel in a Circulating Fluidized Bed Furnace. (Lappeenranta: The Department Council of the Department of the Energy and Environmental Technology)

[5] Goldschmidt M. 2001 Hydrodynamic Modelling of Fluidised Bed Spray Granulation (Twente : Twente University Press)

[6] Kinkar A., Dhote G. and Chokkar R. 2015 CFD Simulation On CFBC Boiler International Journal of Scientific & Technology Research 4 02

[7] Wijayanto R., Sudarmanta B., Syaifudin A. and NugrohoA. 2018 CFD Simulation of Circulating Fluidized Bed 30 MW: Effect of Primary and Secondary Air distribution on Fluidization Behavior. AIP Conference Proceedings DOI: 10.1063/1.5046214

[8] Sudarmanta B., Wijayanto R., Syaifudin A. and Nugroho G. 2017 Studi Numerik Pengaruh Diameter Bed Material terhadap Karakteristik Fluidisasi pada Circulating Fluidized Bed. (Proc: Seminar Nasional teknik Mesin 12)

[9] Koksharev M., Gil A. and Lebedev V. 2017 3D CFD simulation of aerodynamics of a 406 MWt CFB boiler In MATEC Web of Conferences 110 01041

[10] Xu L., Cheng L., Ji J., Wang Q. and Fang M. 2019 Comprehensive CFD combustion model for supercritical CFB boilers Particuology

[11] Sudarmanta B., Wijayanto R., Nugroho G., Syaifudin A., Setiyawan A., and Ariateja J. 2017 Numerical Study Effect of Fluidizing Air to Erosion Pattern in Circulating Fluidized Bed Boiler the International Journal of Mechanical Engineering and Sciences 1 17-26

[12] Wirahadi A. and Mirmanto A. 2019 3-D Numerical Study of CFB 110 MW: Fluidization in Furnace and Cyclone with Load and Air Combustion Variations. IPTEK The journal for Technology and Science 30 (2): 2088-2033.

[13] Launder B. and Spalding D. 1972 Lecturers in Mathematical Models of Turbulence. (London: Academic Press)

[14] Kumar R. and Pandey K. 2012 CFD Analysis of Circulating Fluidized Bed Combustion IRACST-ESTIJJ 2 (1) 163-174