3-D Numerical Study of CFB 110 MW: Fluidization in Furnace and Cyclone with Load and Air Combustion Variations

Ari Wirahadi, Heru Mirmanto

***Abstract*— Circulating Fluidized Bed (CFB) boiler has advantages when uses low rank coal compared to Pulverized Boiler. It will be less operational cost but has greater risk in degradation because of sands material inside the process. Air combustion between primary and secondary air is one important parameter that influent fluidization of CFB. Numerical simulation of commercial CFD was used with Eulerian multiphase model implemented to analyze sand volume of fraction, air and sand velocity including pressure distribution around furnace. It used 55%-45% air combustion ratio of primary and secondary air as reference when boiler operated at 63% and 100% based on operation performance of CFB. Then it simulated with additional variation of air combustion ratio 50%-50%. Simulation of 110% load was added using all air combustion variations. The simulation showed that fluidization with air combustion ratio 50%-50% and 55%-45% executed well when operated at 63%. Meanwhile, fluidization 100% and 110% with all those air combustion ratios would cause a great number of sands entered inlet cyclone and higher sands as well air velocities**

Keywords— Circulating Fluidized Bed, computational fluid dynamics, air combustion, fluidization.

I. INTRODUCTION

Fluidized bed technology makes it possible to burn wet and low quality of fuels with good combustion efficiency. Moreover, the NOx emissions are low, and desulphurization is economical in fluidized bed boilers[1]. During the past 30 years, circulating fluidized bed (CFB) combustion has been developed into a mainstream technology. To achieve successful design of CFB combustor with high efficiency and low emissions, scale-up experiencing trials at the bench, pilot, demonstration, and commercial scale is very costly[2]. To date, experimentation is certainly an approach, while numerical simulation is another, receiving growing interest with the rapid development of computational technologies, especially computational fluid dynamics (CFD)[3].

Fluidization is the process by which the solid particles are brought to a suspended state through gas or liquid. When air or gas is passed upward through the solid particles at low velocity, they remain undisturbed. As the velocity is increased, the particles reach the state of *Fluidization*[4]. One of study about fluidization was used Eulerian multiphase and k-ε standard for turbulence model. The k-ε more applicable and accurate to many flow conditions that showed maximum air velocity occurs in the area far from the wall where on the wall air velocity was negative value[1]. Actually k- ε was applied not only in fluidization but also broad area including mechanical of fluid[5]. Commonly the Eulerian approach was used for the simulation of large scale CFBs. In the Eulerian approach or two-fluid model both the gas and the granular phases are treated as fully interpenetrating continua. The equations used were a generalization of the Navier–Stokes equations for interacting mediums [6].

The study that investigated the potential for abrasion that occurs in the cyclone wall due to erosion by air or sand and informed that air and sands in some areas of the cyclone could reach 30 m/s and it could damage the cyclone has been done[7]. Up to now, the operation parameters that affects the mixing mechanisms of the gas-solid particles were not fully understood. Problems arising from poor fluidization were abrasion on furnace and cyclone walls or an agglomeration that leads to de-fluidization. Good fluidization cannot be achieved except the proper operation parameter, such as primary and secondary air distribution, were well understood. Therefore, proper primary and secondary air distribution is necessary to be investigated. Other study about CFB Boiler 30 MW performed at five different primary and secondary air distribution. It was observed that the primary and secondary air distribution in CFB boiler has a significant effect on fluidization behavior. It was also indicated primary and secondary air distribution in various %PA-%SA at 40-60, 50-50, 60-40,70-30 and 80-20 in full load of boiler[8]. But that study not yet investigate in lower load and higher load. Other study with the same Boiler size indicated that higher fluidizing air velocity will give effect to higher erosion rate[9].

In this study, CFD simulation was conducted using commercial software to investigate effect of various air capacity ratio on fluidization behavior in CFB on several loads mainly focus in the fluidization of gas-solid mixing in full size 3D boiler. The combustion process was not included and the model was assumed to be isothermal.

II. METHOD

A. Governing Equation

The Eulerian multiphase model was utilized to study the fluidization of gas-solid flow in fluidized bed based on the conservation equations of continuity and momentum. The standard k-ε model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε) was obtained from the following transport equations [10].

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \quad (2)$$

Ari Wirahadi and Heru Mirmanto are with Department of Mechanical Engineering, Institute Teknologi Sepuluh Nopember, Surabaya, 60111, Indonesia. E-mail: samir@me.its.ac.id

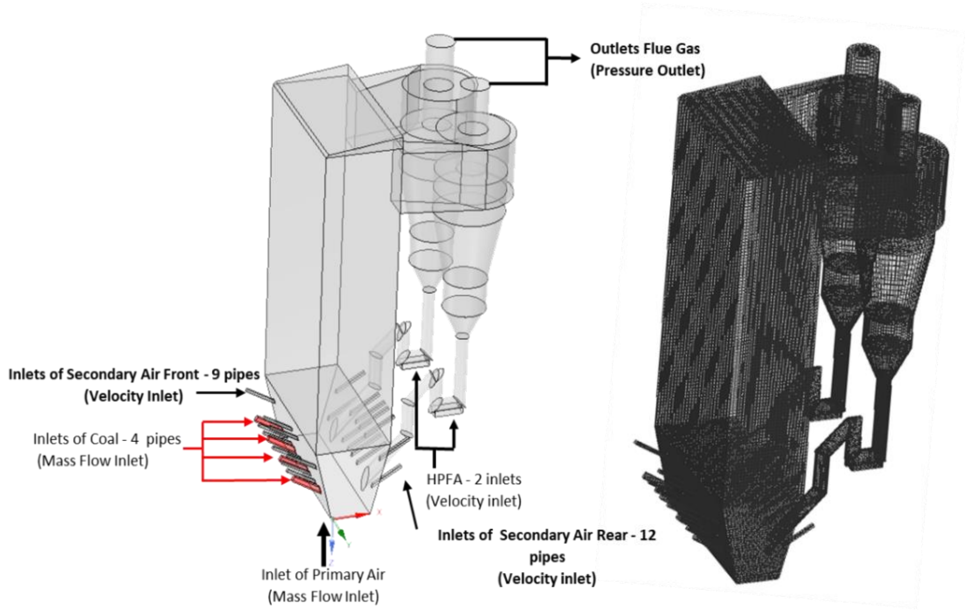

Figure 1. Domains of Simulation and Mesh.

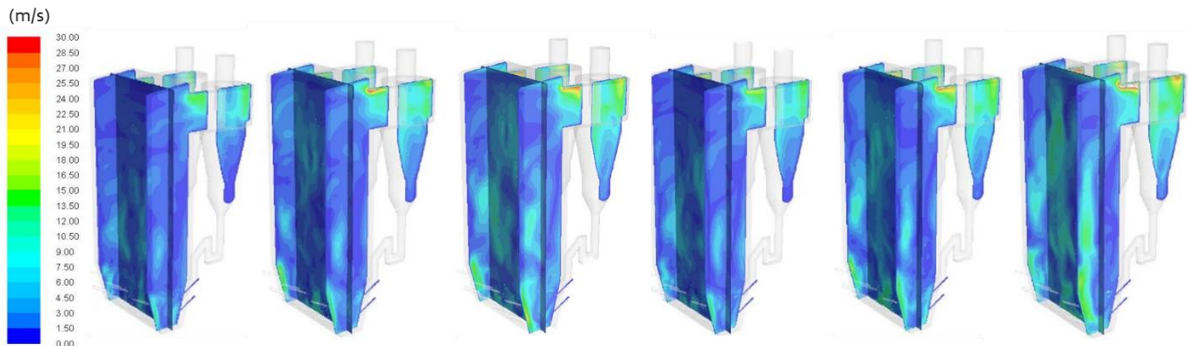

Figure 2. Air Superficial Velocity ISO Center Contours of Furnace and Cyclone t=50s.

where G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, Y_m represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants. S_k and S_ε are user-defined source terms where in this case is neglected. A general CFD software was used to solve governing equations above.

B. Geometry and Mesh

The 110 MW CFB boiler was installed in Nagan, Aceh, Indonesia, designed by Wuxi Huaguang. It was natural-circulation 382 ton/h, as shown in Fig.1, mainly consisting of a furnace, two cyclone separators and Forced loop pipes. This simulation was performed in a 3D model with dimension of furnace is 3.2m x 14.4m x 36.3m without bypass section where cyclone and loop seal were drawn and simulated. Simulation domain as shown in Fig.1 covered 4 rectangular inlets of coal were located on front of furnace as mass flow inlet. There were 9 inlet pipes of secondary air located on the front side of furnace and another 12 pipes on the rear side. Both side used velocity inlet as boundary conditions. For convenience, the primary air was assumed to enter the furnace from the whole bottom including two High Pressure Fluidizing Air[3]. It used mass flow inlet boundary conditions, meanwhile cyclone outlet was defined as pressure outlet boundary condition.

For meshing, the boiler was divided into 64 volumes decomposition where mostly meshed with hexahedron, while the rest used tetrahedron, e.g. lower cyclone and loop pipes as shown on Fig.1. All meshes size scale is below 0.2 m with relative center and smoothing medium were generated 445.812 nodes or 497.311 elements.

C. Simulation Setting

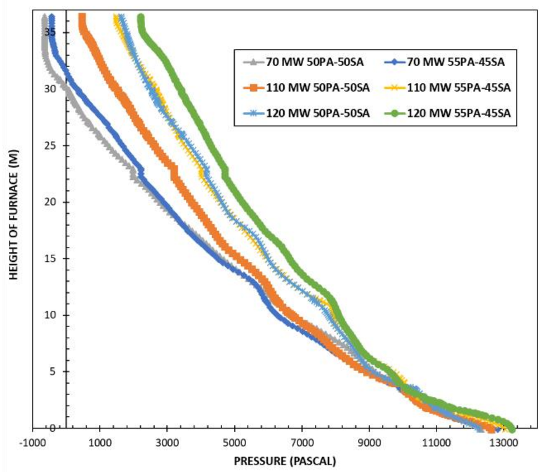

Figure 3. Pressure versus Height of Center Line Furnace t=50s.

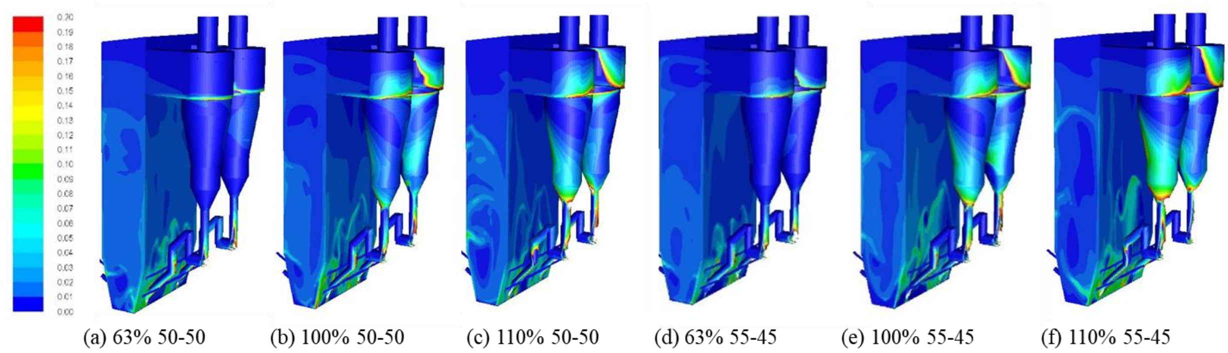

Figure 4. Wall Contours of Solid Volume of Fraction t=50s.

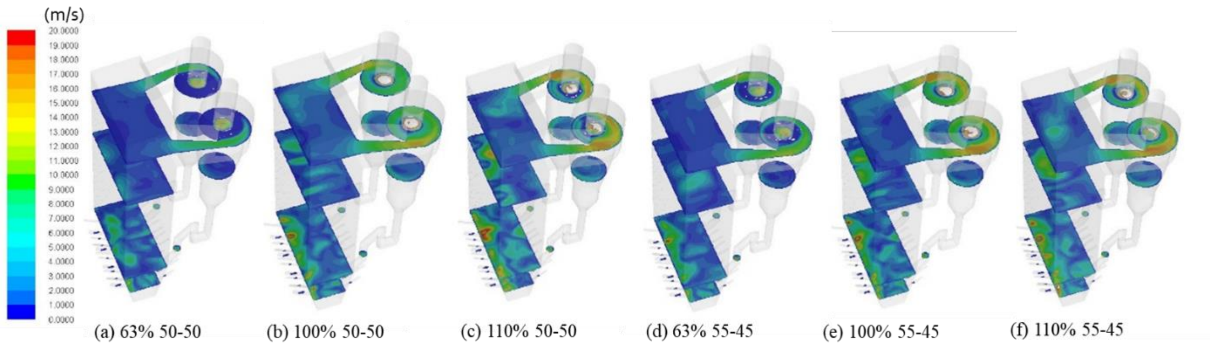

Figure 5. Sands Velocity ISO Contours at Elevation at 3m, 7m, and 32 m t=50s.

TABLE 1.
SIMULATION SETTING AND AIR PROPERTIES

Cases	Load [%]	%PA-%SA	Air Combustion (NM^3)			Air Density [kg/m^3]	Air Viscosity [kg/m-s]	Boiler		Outlet Pressure [Pa]
			Primary	Secondary	Total			Temp [°C]	Pressure [Pa]	
1	63	50-50	154.56	154.56	309.12	0.3063	4.51×10^{-5}	876	-303	-1080
2	63	55-50	170.02	139.11	309.12	0.3063	4.51×10^{-5}	876	-303	-1080
3	100	50-50	191.13	191.14	382.27	0.2934	4.63×10^{-5}	928	-165	-1480
4	100	55-45	211.94	170.33	382.27	0.2934	4.63×10^{-5}	928	-165	-1480
5	110	50-50	210.25	210.25	420.5	0.2724	4.83×10^{-5}	1020	-165	-1480
6	110	55-45	231.27	189.22	420.5	0.2724	4.83×10^{-5}	1020	-165	-1480

TABLE 2.
SANDS PROPERTIES AND SETTNGS

Properties	Settings
Density [kg/m$^{3]}$	2500
Diameter [m]	0.0002
Viscosity [kg/m-2]	0.0013
Granular Viscosity	Gidaspow
Granulal Bulk Viscosity	lun-et-al
Frictional Viscosity	Schaeffer
Angle of Internal Friction	30.00007
Frictional Pressure	based-ktgf
Friction Packing limit	0.61
Granular Temperature	Algebraic
Solid Pressure	lun-et-al
Radial Distribution	lun-et-al

The Boiler was considered operated at operational temperature where air density was calculated by ideal gas formulation and air viscosity based on Sutherland formulation as summarized in Table 1 where load 100% means 110MW[11]. In this study, sands were patched in the beginning of simulation with volume of fraction 0.4 and height 2.3m where the setting for sands phase properties was shown in Table 2 with reference from Zhang, et al[3]. Meanwhile for density, diameter, and viscosity of sands referred to Sudarmanta, et al[12]. Eulerian model was used for multiphase model to define gas-solid phase and its interactions. The turbulence model used in this study was standard k-ε because of its general applicability, robustness, and economy [9]. Coal inlet was modeled as ideal gas, so generally only two phases was implemented and combustion was modeled as isothermal.

D. Simulation Setting

Phase Coupled Semi-implisit method for pressure linked equation (PC-SIMPLE) was implemented for pressure-velocity coupling[13]. The first order upwind was used for momentum, volume fraction, turbulence kinetic energy, and turbulence dissipation rate. Iterations were performed transient for the time step size of 0.2 s with maximal iteration/time step of 20 and around 500 number of time steps until parameter of residual reach below 10-3. Standard initialization was applied with based values from inlet Primary Air.

III. RESULTS AND DISCUSSION

A. Distribution of Superficial Velocity

Superficial velocity in this case was only air superficial velocity. It means that higher superficial velocity will directly impact to the fluidization of sands. Fig.2 showed that increasing primary air contributed to larger area where superficial velocity more than 10 m/s around central x axes of furnace t=50s at every load variation. Load incremental gave the same trends as air combustion incremental. Identical condition also appeared at cyclone contour. It verified Kinkar's research that indicated air velocity around inlet cyclone is 30m/s as shown in red contour[7]. It concluded that superficial velocity will increase when load or primary air were increased.

B. Pressure Distribution

Pressure is the only parameter could be measured from operation of boiler, The others such as volume of sands fraction and air superficial velocity were really difficult to be observed in real boiler operation. Fig.3 showed pressure drop of static pressure trend for each cases at center x axes of furnace t=50 along furnace height. Pressure input setting t=0 for each case were different, but final pressure on the top of furnace gave obvious trends that higher load or primary air would lead to higher pressure on the top at t=50s. Identical result appeared from Wijayanto, et al for higher primary air input[8].

C. Distribution of Fine Solids Volume of Fraction

Dense bed that fluctuated at lower furnace different with fine solid particles that could appear in any sections of boiler. Contour of fine solid volume of fraction that discussed in this section focused on range 0 until 0.2 around cyclone. Fig.4 showed wall contour of solid volume of fraction at max 0.2. It appeared vast area of volume of sand fraction around cyclone wall were accumulated in case of 100% load and higher when it compared at the same primary and secondary air ratio. When the same air combustion was compared, for example 63% load, there were slightly wider area of fine solids volume of fraction around cyclone.

D. Distribution of Sands Velocity

Fig.5 showed four elevation ISO contours of sands velocity less than 20 m/s even at cyclone. The lowest load gave the lowest sands velocity at any elevation and vice versa. For 63% load resulted sand velocity below 10m/s. It means that load on that point is safe for continuing operation, while fluidization of sand above 100% load have greater sand velocities, so that mean has greater risk of abrasion.

E. Conclusion and Discussion

CFD of CFB Boiler provides vast information to enrich knowledge about fluidization phenomena around furnace and cyclone even when simplified with two phases only. The simulation in this study was performed at three different loads and two types of air combustion ratio. The results were presented in volume of sands fraction also air and sand velocity as parameters that affected fluidization. This study concluded 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 engineers knowledge of fluidization of CFB boiler components.

REFERENCES

[1] V. Tanskanen, "CFD study of penetration and mixing of fuel in a circulating fluidized bed furnace," LAPPEENRANTA UNIVERSITY OF TECHNOLOGY, 2005.

[2] B. Lu, N. Zhang, W. Wang, and J. Li, "3-D Full-loop simulation of an industrial-scale circulating," *Am. Inst. Chem. Eng. J.*, vol. 59, 2013.

[3] N. Zhang, B. Lu, W. Wang, and J. Li, "3D CFD simulation of hydrodynamics of a 150 MWe circulating fluidized bed boiler," *Chem. Eng. J.*, vol. 162, no. 2, pp. 821–828, Aug. 2010.

[4] G. Thenmozhi and S. Lingappan, "A Survey on circulating fluidized bed combustion boilers," *Int. J. Adv. Res. Electr. Electron. Instrum. Eng.*, vol. 2, no. 3, 2013.

[5] H. Mirmanto, H. Sasongko, I. Rohmawati, and I. Nurjannah, "The effect of adding FFST on secondary flow characteristics near endwall of asysmmetry airfoil (cas study of different camber)," *Int. J. Appl. Math. Stat.*, vol. 53, no. 5, pp. 67–72, 2014.

[6] M. Goldschmidt, *Hydrodynamic modelling of fluidised bed spray granulation*. Twente University Press, 2001.

[7] A. S. Kinkar, G. M. Dhote, and R. R. Chokkar, "CFD simulation on CFBC boiler," *Int. J. Sci. Technol. Res.*, vol. 4, no. 2, 2015.

[8] R. M. Wijayanto, B. Sudarmanta, A. Syaifudin, and G. Nugroho, "CFD simulation of circulating fluidized bed boiler 30 MW: Effect of primary and secondary air distribution on fluidization behavior," in *AIP Conference Proceedings 1983*, 2018.

[9] B. Sudarmanta, R. M. Wijayanto, G. Nugroho, A. Syaifudin, A. Setiyawan, and J. B. Ariateja, "Numerical study effect of fluidizing air to erosion pattern in circulating fluidized bed boiler," *Int. J. Mech. Eng. Sci.*, vol. 1, no. 2, pp. 17–26, 2017.

[10] B. E. Launder and D. B. Spalding, *Lecturers in mathematical models of turbulence*. London: Academic Press, 1972.

[11] W. Sutherland, "The viscosity of gases and molecular force," *Philos. Mag.*, vol. 36, pp. 507–531, 1893.

[12] B. Sudarmanta, R. M. Wijayanto, A. Syaifudin, and G. Nugroho, "Studi numerik pengaruh diameter bed material terhadap karakteristik fluidisasi pada Circulating fluidized bed boiler beban 28,6 MW," in *Proseding Seminar Nasional Teknik Mesin 12*, 2017.

[13] R. Kumar and K. M. Pandey, "CFD analysis of circulating fluidized bed combustion," *IRACST-ESTIJ*, vol. 2, no. 1, pp. 163–174, 2012.