

EFFECTS OF THE AIR FLOW ON THE DYNAMIC OF PARTICLES IN THE CIRCULATING FLUIDIZED BED BOILER USING CFD SIMULATIONS

Asyari Daryus¹, Ahmad Indra Siswantara², Didik Sugiyanto¹, Herry Susanto¹, Gun Gun R. Gunadi³, Hariyotejo Pujowidodo⁴, Candra Damis Widiawaty³, Nopryandi¹ and Trisna Ardi Wiradinata¹

¹Department of Mechanical Engineering, Darma Persada University, Jakarta, Indonesia

²Department of Mechanical Engineering, Universitas Indonesia, Depok, Indonesia

³Department of Mechanical Engineering, Politeknik Negeri Jakarta, Depok, Indonesia

⁴Centre for Thermodynamics, Engine and Propulsion, BPP Teknologi, Serpong, Indonesia

Email : asyaridaryus01@gmail.com

ABSTRACT

The optimum distribution of solid phase in fluidized bed system is important thing among others to achieve the best performance. One of the parameter that influences the distribution of solid is air mass flow rate. In this study, simulations of Computational Fluid Dynamics (CFD) is done where Eulerian model is used for solid particle motions and k- ϵ turbulence model for fluid flows. The simulations were carried out using six different air mass flow rates based on the excess air of the reactions. The air mass flow rate influenced the distribution and the velocity of solid phase, and also the pressure difference of gas inside the boiler. The results showed that the excess air between 10 % – 20 % gave the optimum results.

Keywords: *Circulating fluidized bed boiler, distribution of solid, excess air, CFD simulations.*

1 INTRODUCTION

Circulating Fluidized Bed boiler (CFB boiler) is used in many applications, like steam turbine system, drying system, biomass combustion, etc. (Ngoh, 2012; Adamczyk et al., 2014; Bakshi et al., 2017). Some issues have been found in this boiler while diagnostic tools and technique for solutions are still limited (Bakshi et al., 2017). Solve issues through simulations have many advantages, such as time saving and low cost compare to the experiment method (Daryus et al., 2016; Siswantara et al., 2016).

Zhao et.al. has investigated the effect of exit geometry of high-density CFB risers using CFD. He concluded that the type of exit had the significant effects on the bed hydrodynamics while the cavity height of abrupt exit, the curvature-diameter of smooth exit and horizontal tube connecting the exit and the primary cyclone did not. The decrease of the diameter of abrupt exit tube increased the solids holdup significantly (Zhao, Zhou, Wang & Li, 2015). Adamczyk et.al. In their researchs on air-fuel combustion process using simulations found that distribution of pressure and temperature were comparable to measured data (Adamczyk et al., 2015). While Zi et.al. found that the

distribution of solid influenced by slugging formation (Zi et al., 2017). Still many researches have been done regarding with fluidized bed or specifically fluidized bed boiler, such as done by Daryus et.al., Adamczyk et.al. Shi et.al., and others (Daryus et al., 2018; Adamczyk et al., 2018; Shi, Komkrakova & Nikrityuk, 2019). But there are still many subjects needed to be explored to understand more on the phenomena of flow in fluidized bed.

This research aims to investigate the influences of air flow velocity and amount of excess air of CFB boiler on the solid particles dynamics for optimum performances using simulation methods.

2 METHODS

2.1 Combustion Model

The Westbrook-Dryer one-step model is used for combustion. It gives a good estimation of indicator of the expected temperature levels. The Finite Rate and Eddy Dissipation model is used to model the process of the turbulent non-premixed combustion. In this model, the reaction rates are assumed to be controlled by the turbulence, avoid the complex Arrhenius chemical kinetic calculations.

Coal is used in this CFB boiler. The chemical reaction of combustion is:



2.2 Turbulence model

To simplify the complexity of the Navier Stokes equations for fluid flows, the Reynolds Average Navier Stokes (RANS) principle is developed and results 6 additional stresses to the Navier Stokes equations called Reynolds stresses. Some models are then developed for RANS equations to obtain the simpler calculations.

One of the RANS turbulence models is $k-\varepsilon$, where it is simple (only needs the input of boundary condition only), stable, widely validated, and suitable for industrial problems solutions. This model uses the Boussinesq formula to find the Reynolds stresses, i.e. [12]:

$$\tau_{ij} = -\rho \overline{u_i' u_j'} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (2)$$

Where ρ is the density, u' and U is the fluctuating and mean velocity vector respectively, and μ_t is the turbulent or eddy viscosity. The turbulent viscosity determined by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

Where C_μ is constant, k is the transport of kinetic energy, and ε is the transport of dissipation. The transport equation of kinetic energy is formulated as (Versteeg, Malalasekara, 2007):

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div} \left[\frac{\mu_t}{\sigma_k} \text{grad } k \right] + 2\mu_t E_{ij} \cdot E_{ij} - \rho \varepsilon \quad (4)$$

While the transport equation of dissipation is:

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \text{div}(\rho \varepsilon \mathbf{U}) = \text{div} \left[\frac{\mu_t}{\sigma_\varepsilon} \text{grad } \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t E_{ij} \cdot E_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (5)$$

Where \mathbf{U} is the velocity vector, E_{ij} is the mean rate of deformation, if i or $j = 1$ corresponds to the x -direction, and i or $j = 2$ to the y -direction, C_μ , σ_k , σ_ε , $C_{1\varepsilon}$, and $C_{2\varepsilon}$ are constants.

Constants used in the numerical calculations are $C_\mu = 0.09$; $\sigma_k = 1.00$; $\sigma_\epsilon = 1.30$; $C_{1\epsilon} = 1.44$ dan $C_{2\epsilon} = 1.92$.

2.3 Geometry and Meshing

The geometry of CFB boiler and the 2D structured mesh model are shown on Figure 1. The mesh consists of 42x162 cells.

2.4 Boundary Conditions

In this geometry model, the primary air (inlet 1) is located on the bottom of the boiler, while the secondary air (inlet 2 and 3) are located in the right and the left of boiler. The rate of coal assumed to be 14.4 t h^{-1} , equivalent to 25 MW power output of generator, driven by the steam produced by the boiler. Based on the stoichiometry calculations, the air mass flow needed for the combustion is 20.10 kg s^{-1} . The velocity of air for the inlets for various value of excess air can be seen on Table 1.

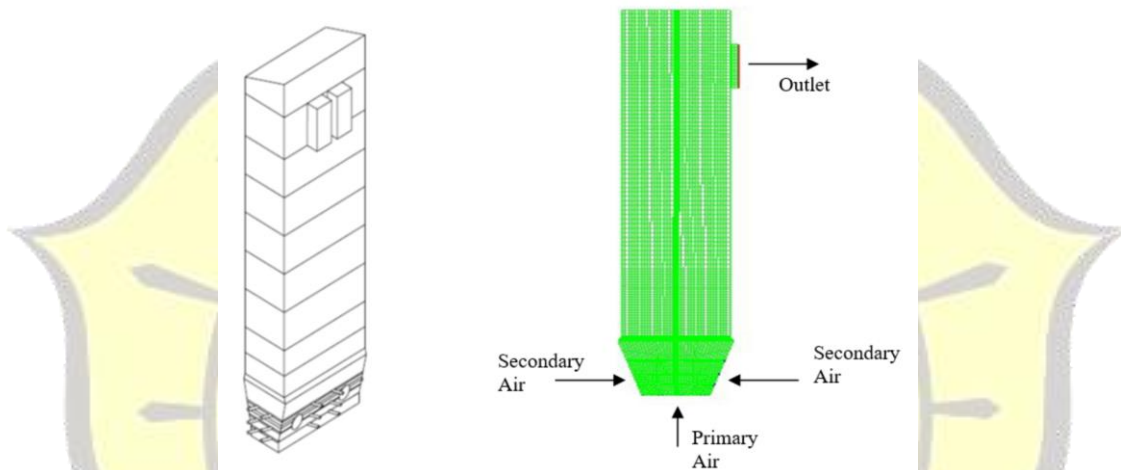


Figure 1: CFB Boiler geometry (left) and 2D mesh model of CFB Boiler (right).

Table 1. The value of inlet velocities.

Excess Air (%)	Air Flow (kg s^{-1})	Inlet 1 (m s^{-1})		Inlet 2 (m s^{-1})		Inlet 3 (m s^{-1})	
		v	u	v	u	v	
0	20.10	12.7	-12.4	-12.4	13.1	-13.1	
10	22.22	12.7	-14.5	-14.5	15.2	-15.2	
15	23.14	12.7	-15.5	-15.5	16.2	-16.2	
20	24.05	12.7	-16.5	-16.5	17.2	-17.2	
25	25.03	12.7	-17.5	-17.5	18.3	-18.3	
30	26.03	12.7	18.6	18.6	19.4	-19.4	

3 RESULTS AND DISCUSSION

It has been conducted the fluid flow simulations in various excess air value, i.e. 0 % (air mass flow 20.1 kg s^{-1}), 10 % (air mass flow 22.22 kg s^{-1}), 15 % (air mass flow 23.14 kg s^{-1}), 20 % (air mass flow 24.05 kg s^{-1}), 25 % (air mass flow 25.03 kg s^{-1}), and 30 % (air mass flow 26.03 kg s^{-1}). The results of simulations presented here are taken on 35

second of operation, because it is assumed that the flow of fluid at this time has reached the stable condition.

To validate the model, the measured and simulation data will compared. The data compared is the pressure difference between inlet boiler (primary inlet) and outlet for various total air flow rate and the result is shown on Figure 2. It can be seen that the simulation results close to the field data.

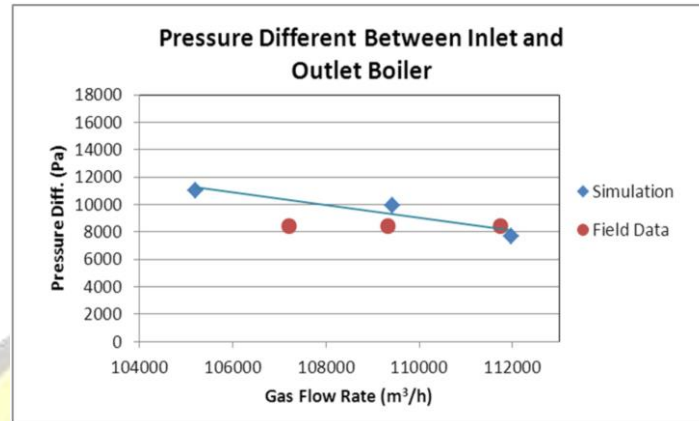


Figure 2: Pressure difference between inlet and outlet boiler.

The contour of solid volume fraction for various excess air values is shown on Figure 3. It can be seen that for the low value of excess air (0 % of excess air), the solid particles are existing around the bottom of boiler, and for the higher value of excess air, the solid particles move upward, proportional with the value of excess air. This upward movement was driven by kinetic energy of air. For the value of excess air of 0 % to 15 %, the height of solid movement does not exceed the height of fire brick (sloping wall or elbow on the figure), but above of 20 % of excess air, its height exceed it. When the solid particle height exceeds the firebrick, it will hit the boiler tube and will cause abrasion and in the long term can create the leak. This situation is not preferable. Hence, the best results of solid particle height are found at 0 % to 20 % of excess air.

Figure 4 shows the contour of solid velocity in various values of excess air. For the excess air below 20 %, the situations are safe, because the solid particle exists in the firebrick area. But for the excess air of 20 % and above, the situations began to worry because the solid particles reach the wall tube area and their velocity is more than 5 m s^{-1} , these conditions can cause the abrasion on the wall tubes. It is also shown that the flow of particles is deflected to the left side, resulting in the left wall tube will experience more abrasion than the right one. The optimum results are achieved by excess air below 20 %.

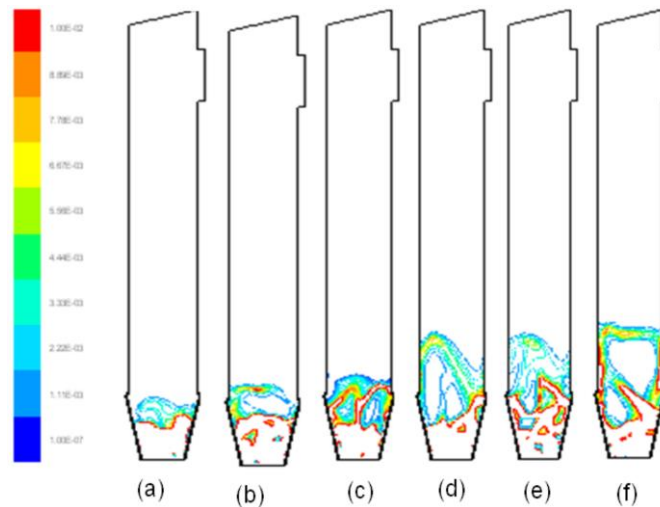


Figure 3: Contour of volume fraction at $t=35$ s for various excess air. (a) 0 %; (b) 10 %; (c) 15 %; (d) 20 %; (e) 25 %; and (f) 30 %.

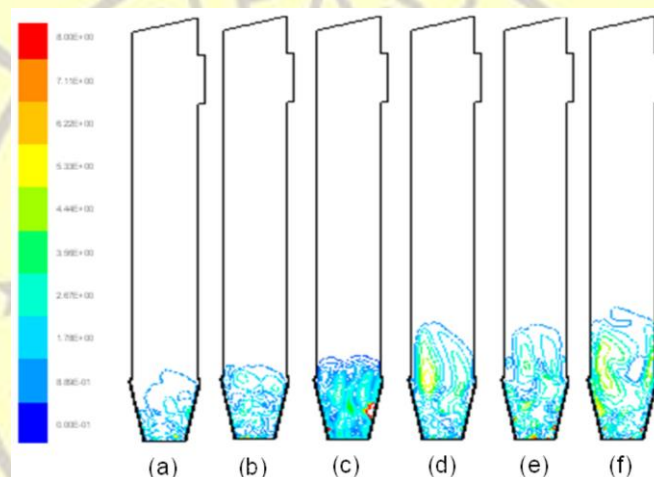


Figure 4: Contour of velocity of particle (m s^{-1}) at $t=35$ s for various excess air. (a) 0 %; (b) 10 %; (c) 15 %; (d) 20 %; (e) 25 %; and (f) 30 %.

Figure 5 shows the contour of solid particle temperature for various values of excess air. For small value of excess air, the distribution of high temperature concentrated on left wall and extended to the near of right wall. The right wall has lower temperature than that of left wall. When the value of excess air increased, the high temperature area moves from the right to the left and from the bottom to the top. But, as shown on Figure 4 and Figure 5, the most of the solid particles exist on the bottom area for all the value of excess air, resulting in the coal will be burnt efficiently (the white colour in the bottom area is the high speed solid particle zone, the contour or colour cannot be seen because its value exceed the limit of velocity scale).

The simulation results for pressure difference between inlet (bottom) and outlet (top) is shown on Figure 6. These values are the average values of pressure difference for 30 s and 35 s. The pressure is constant for the value of excess air of 0 % to 10 % but then decrease and minimum on the value of excess air of 15 %. Above 15 %, its values increase and maximum on the value of the excess air of 20 % and then decrease slowly

afterwards. The best values of the pressure differences exist between the values of excess air of 0 % to 17 %.

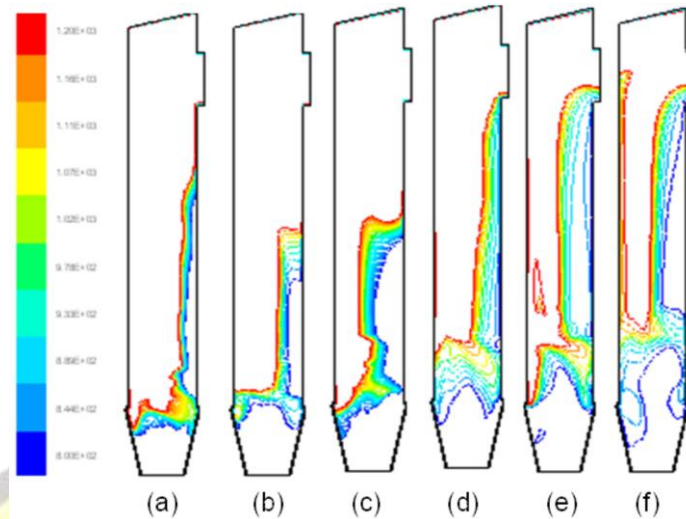


Figure 5: Contour of temperature of particle (K) at t=35 s for various excess air. (a) 0 %; (b) 10 %; (c) 15 %; (d) 20 %; (e) 25 %; dan (f) 30 %.

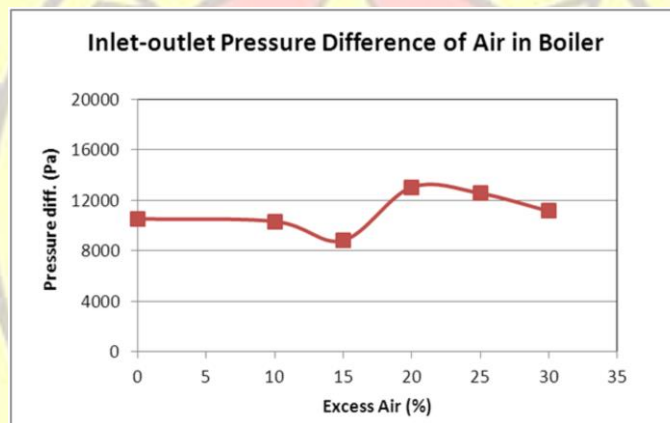


Figure 6: Pressure difference curve in the boiler from simulation results.

Meanwhile, the efficiency changes of wall tube heat absorbing is shown on Figure 7. It measures efficiency changes of heat quantity absorbed by wall tube (negative sign means that the efficiency of heat absorbed by wall tube is drop). It can be seen that the efficiency decreases when the value of excess air increases, except for the value of excess air below 10 %. But this efficiency drop is negligible since less than 1 %.

We come to the conclusion. From the contour of the volume fraction, and the velocity of solid particles, the optimum value of excess air is 0 % to 20 %, while from the contour of the temperature is 0 % to 30 %. From the pressure difference in the boiler, the best results of excess air are found to be 10 % to 17 %, but from wall tube heat absorbing efficiency the results are negligible. The conclusion, the recommended value of excess air for this boiler is 10 % to 20 %.

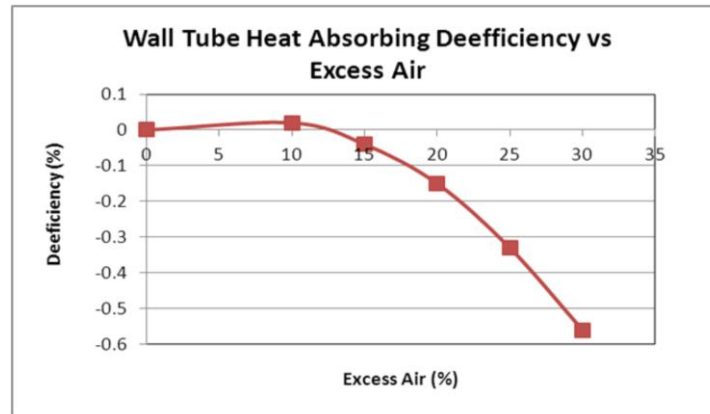


Figure 7: Wall tube heat absorbing deefficiency.

4 CONCLUSION

The simulations of combustion and fluid flow in 25 MW Circulating Fluidized Bed (CFB) Boiler have been done to find the optimum value of excess air using CFD. The simulations were done using 2D geometry. The turbulence model used for fluid flow was $k-\epsilon$, while for the combustion model was Eulerian. From the contour of the volume fraction and the velocity of solid particles, the contour of the temperature of air, the pressure difference of the air in the boiler and from the wall tube heat absorbing efficiency found that the optimum value of excess air was 10 % to 20 %.

5 ACKNOWLEDGEMENT

The authors would like to thanks DRPM University of Indonesia for funding this research through "Hibah Penelitian Disertasi Doktor (PDD) 2018, number 015/KM/PNT/2018" and to PT. CCIT Group Indonesia for CFDSoft® software license.

REFERENCES

- Adamczyk, W.P., G. Węcel, M. Klajny, P. Kozołub, A. Klimanek, R.A. Białecki, 2014, **Modeling of Particle Transport and Combustion Phenomena in a Large-scale Circulating Bed Boiler Using a Hybrid Euler-Lagrange Approach**, *Journal of Particuology*, 16(2014): p. 29-40.
- Adamczyk, W.P., P. Kozołub, A. Klimanek, R.A. Białecki, M. Andrzejczyk, M. Klajny, 2015, **Numerical Simulations of the Industrial Circulating Fluidized Bed Boiler Under Air- and Oxy-fuel Combustion**, *Applied Thermal Engineering*, 87(2015): p. 127-136.
- Adamczyk, W.P., K. Myöhänen, E.U. Hartge, J. Ritvanen, A. Klimanek, T. Hyppänen, R.A. Białecki, 2018, **Generation of data sets for semi-empirical models of circulated fluidized bed boilers using hybrid Euler-Lagrange technique**, *Energy*, 143: p. 219-240.
- Bakshi, A, C. Altantzis, L.R. Glicksman, A.F. Ghoniem. Gas-flow Distribution in Bubbling Fluidized Beds, 2017, **CFD-based Analysis and Impact of Operating Conditions**, *Powder Technology*, 2017.
- Daryus, A., A.I. Siswantara, S. Darmawan, G.G.R. Gunadi, R. Camalia, 2016, **CFD Simulation of Turbulent Flows in Proto X-3 Bioenergy Micro Gas Turbine Combustor Using STD $k-\epsilon$ and RNG $k-\epsilon$ Model for Green Building Application**, *International Journal of Technology*, 7(2): p. 204-211.

6. Daryus, A., A.I. Siswantara, Budiarmo, G.G.R. Gunadi, H. Pujowidodo, 2018, **Simulasi pengaruh kecepatan gas terhadap karakteristik fluidisasi pada fluidized bed menggunakan metode CFD**, [Simulation of the gas velocity effect on the fluidization characteristics of the fluidized bed using the CFD method]. *Jurnal Poros*, 16(1): p. 54-63.
7. Ngoh, J., E.W.C. Lim, 2016, **Effects of Particle Size and Bubbling Behaviour on Heat Transfer in Gas Fluidized Beds**, *Journal of Applied Thermal Engineering*, 105(2016): p. 225-242.
8. Shi, H., A. Komrakova, P. Nikrityuk, 2019, **Fluidized beds modeling: Validation of 2D and 3D simulations against experiments**, *Powder Technology*, 343: p. 479-494.
9. Siswantara, A.I., A. Daryus, S. Darmawan, G.G.R. Gunadi, R. Camalia. (2016). CFD Analysis of Slurry Flow in an Anaerobic Digester. *International Journal of Technology*, 7(2): p. 197-203.
10. Zhao, B., Q. Zhou, J. Wang, J. Li, 2015, **CFD Study of Exit Effect of High-density CFB Risers with EMMS-based two-fluid Model**, *Chemical Engineering Science*, 134(2015): p. 477-488.
11. Zi, C., J. Sun, Y. Yang, Z. Huang, Z. Liao, J. Wang, Y. Yang, G. Han, 2017, **CFD Simulation and Hydrodynamics Characterization of Solids Oscillation Behavior in a Circulating Fluidized Bed with Sweeping Bend Return**, *Chemical Engineering Journal*, 307(2017): p. 604-620.
12. Versteeg, H., W. Malalasekara, 2007, **An Introduction to Computational Fluid Dynamics, the Finite Volume Method, 2nd ed**, Essex, London: Pearson Educational Ltd.