

PAPER • OPEN ACCESS

Effects of the superficial gas velocity on the distribution of solid particles in the circulating fluidized bed using CFD simulations

To cite this article: A Daryus *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1098** 062112

View the [article online](#) for updates and enhancements.



240th ECS Meeting

Digital Meeting, Oct 10-14, 2021

We are going fully digital!

Attendees register for free!

REGISTER NOW



Effects of the superficial gas velocity on the distribution of solid particles in the circulating fluidized bed using CFD simulations

A Daryus^{1,2,*}, A I Siswantara², B Budiarto², H Pujowidodo^{2,3} and G G R Gunadi^{2,4}

¹ Department of Mechanical Engineering, Universitas Darma Persada, Jl. Radin Inten II, Pondok Kelapa, Jakarta Timur, Indonesia 13450

² Department of Mechanical Engineering, Universitas Indonesia, Depok, Indonesia 16424

³ Center for Thermodynamics, Engine, and Propulsion, BPP Teknologi, Serpong, Indonesia 15314

⁴ Department of Mechanical Engineering, Politeknik Negeri Jakarta, Depok, Indonesia 16424

*asyari_darius@ft.unsada.ac.id

Abstract. The purpose of the study is to investigate the effects of various air flow rate or superficial gas velocity in term of excess air to obtain the optimum distribution of solid phase in the Circulating Fluidized Bed (CFB). The Computational Fluid Dynamics (CFD) method used for simulations uses the Multi-fluid Eulerian approaches incorporating the Kinetic Theory of Granular to calculate the motion of particles and the standard $k-\varepsilon$ turbulence model to simulate the fluid flows. The simulations were carried out using five different air mass flow rates, based on the air consumption in the combustion of coal used in real CFB boiler operation. Since this simulation is the preliminary study of the particle dynamics in CFB, the combustion process had not simulated yet. In this work, the complex flow has been resolved by applying the CFDSOF commercial CFD software. The pressure on bed, the distribution of pressure, the particle velocity and volume fraction in the CFB were investigated to obtain the optimum results of the fluidization. The values of the pressure found from experiments at the bottom of bed was around 4,500 Pa, and the results from simulations were not different significantly with this values. At the excess air of 90%, the velocity of particles at the surface of bed is highest that can lead to the erosion on the firewall in long term operations, meanwhile at the lower excess air values, their velocity were relatively small. The optimum of excess air that found to be safe for boiler to operate was around 60% - 75% for average particle size 1 mm.

1. Introduction

Circulating Fluidized Bed boiler (CFB boiler) is a boiler that uses solid particles to enhance the combustion process. Because of its superior heat and mass transfer, this type of boilers are used in wide range of applications, such as power generations, combustion, gasification and combustion of biomass, drying of solids, incineration, etc. [1-5]. Design and performance optimization of this type of boiler is still challenging because there is still the limitation for diagnostic tools and techniques in the harsh condition where the boiler installed [2]. Recently, many researches have been conducted the fluid



simulations using Computational Fluid Dynamics (CFD), which can accurately enough calculate and simulate thermodynamics and fluid properties [4,6-8].

Using CFD to simulate the combustion processes and fluid flow has many advantages, such as low cost and time efficient, because it does not require expensive, and complex experiments tools to collect data, but rather a set of equipment that used to compute the numerical calculations [9-11]. It also can calculate and provide data faster because there is no time to be spent to build the complex experiments apparatus.

CFD simulation gives detail fluid flow description and receives great attention in recent years [4]. Benzarti et al. [12] had conducted the research on simulation of gas-solid turbulent fluidized bed hydrodynamics using Gidaspow, Syamlal and O'Brien, and Wen and Yu and McKeen drag models and STD $k-\epsilon$ turbulence model. Their results showed that the Gidaspow drag model and the no slip boundary condition gave a good prediction of experimental data. Jiménez et al. [13] had conducted the simulation and experiment researches of two-dimensional gas-solid fluidized bed. They used two drag models for CFD simulations, i.e., Gidaspow and Syamlal and O'Brien. The results showed that the Gidaspow model provided the best approach to experiments and the bubble probability profiles seemed to be best predicted by the Syamlal and O'Brien drag model. Iswara et al. had [14] investigated the minimum fluidization velocity and bubbling of fluidized bed using CFD simulations. They used the STD $k-\epsilon$ for turbulence model. The results showed that the value of bed pressure drop between simulations and analytic using Ergun equations were not different significantly.

Meanwhile, Adamczyk et.al. [15] have investigated the measured and numerical results of air-fuel combustion process with large scale industrial CFB boiler. The results showed that the comparison of the pressure and temperature distributions gave comparable trends in contrary to measured data. Zi et.al. [16] have conducted the CFD simulation of solid oscillation in two-dimension CFB. Their results showed that the solid oscillation circulation was influenced by the formation of the slugging in the downner and resulted the pressure drop in the downner and sharp increase of the riser bypass gas velocity. It also showed that the lateral solids volume distributions were uniform in the riser and in the downner. Sudarmanta et al. [17] in their study on circulating fluidized bed (CFB) boiler investigated the effects of bed diameter on characteristic of fluidization. The studies were conducted on 28.6 MW CFB boiler by CFD method and STD $k-\epsilon$ model used as turbulence model. The simulation results gave the information that in the inlet duct and the roof were the areas where the erosion took place.

This aim of this research is to find the optimum of air mass flow rate or excess air on the industrial 100 MW CFB boiler by investigating their effects on solid particle behaviours using CFD simulations using operational conditions.

2. Computational model

2.1. Governing equations and closures

To solve the governing equations of mass, momentum and granular energy for both gas and solid, the Multi-fluid Eulerian model incorporating the Kinetic Theory of Granular Flow (KTGF) is used, where already available in the commercial software package CFDSOF. The continuity equations for gas and solid phases without mass transfer between them, respectively, are:

$$\frac{\partial}{\partial t}(\epsilon_g \rho_g) + \nabla \cdot (\epsilon_g \rho_g \mathbf{u}_g) = 0 \quad (1)$$

$$\frac{\partial}{\partial t}(\epsilon_s \rho_s) + \nabla \cdot (\epsilon_s \rho_s \mathbf{u}_s) = 0 \quad (2)$$

Where the volume fraction for each phases are related with:

$$\epsilon_g + \epsilon_s = 1 \quad (3)$$

The momentum equations for gas and solid phases is described by:

$$\frac{\partial}{\partial t}(\varepsilon_g \rho_g u_g) + \nabla \cdot (\varepsilon_g \rho_g u_g u_g) = -\varepsilon_g \nabla P_g + \nabla \cdot \tau_g + \varepsilon_g \rho_g g + \beta(u_g - u_s) \quad (4)$$

$$\frac{\partial}{\partial t}(\varepsilon_s \rho_s u_s) + \nabla \cdot (\varepsilon_s \rho_s u_s u_s) = -\varepsilon_s \nabla P_g - \nabla P_s + \nabla \cdot \tau_s + \varepsilon_s \rho_s g + \beta(u_s - u_g) \quad (5)$$

And for the granular energy is:

$$\frac{3}{2} \left(\frac{\partial(\varepsilon_s \rho_s \Theta)}{\partial t} + \nabla \cdot (\varepsilon_s \rho_s u_s \Theta) \right) = (-P_s I + \tau_s) : \nabla u_s - \nabla \cdot q - \gamma - J \quad (6)$$

To solve the equations, closure laws are required. The closure models of KTGF using in the calculations are presented on table 1.

Table 1. Closure models.

Drag model	Gidaspow [18]
Solid viscosity	Gidaspow [18]
Solid bulk viscosity	Lun et al. [19]
Frictional viscosity	Schaeffer [20]
Solid pressure	Lun et al. [19]
Radial distribution function	Lun et al. [19]

The standard k - ε turbulence model was used to calculate the fluid flow. The transport of kinetic energy (k) and dissipation (ε) of k - ε are modeled respectively as follow [21]:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_K \quad (7)$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (8)$$

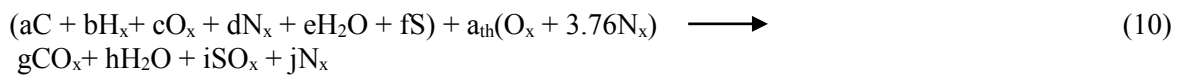
And the turbulent viscosity μ_t is calculated by:

$$\mu_t = \left(\rho C_\mu \frac{k^2}{\varepsilon} \right) \quad (9)$$

Where C_μ , σ_k , σ_ε , $C_{1\varepsilon}$, and $C_{2\varepsilon}$ are constants and their default values are: $C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$ and $C_{2\varepsilon} = 1.92$.

2.2. Excess air calculations

The excess air is the amount of air exceeds the need of air to burn of the fuel. To know the value of excess air, first we need to know the amount of air ideally to burn the fuel to perform the perfect combustion based on the stoichiometry reaction of fuel with the air. Theoretically, the ideal stoichiometry combustion reaction of coal is:



where C is carbon, H is hydrogen, O is oxygen, N is nitrogen, S is sulfur and H₂O is water. The amount of a, b, c, d, e and f constants are obtained from the ratio of mass over its molecular weight for each

constituents. The amount of mass are obtained from the calculations of the percentage of constituent contents from ultimate analysis of coal.

Practically, the excess air (XpA) can be found from the following equation [22]:

$$XpA = \frac{100DVpO2(MoDPc + 0.7905MoThACr)}{(MoThACr \times (20.95 - DVpO2))} \quad (11)$$

This formula needs the operational data and some additional calculations for its variables. Based on the operational condition, the excess air is calculated and the result together with other excess air values are used in simulations.

2.3. Numerical simulation procedures and boundary condition

The simulations are based on experimental data of Circulating Fluidized Bed (CFB) boiler of 100 MW electrical power generation. The actual size of CFB is 14.37 m length \times 6 m width \times 35.8 m height and 3.5 m width at the bottom. The geometry of the CFB boiler is shown on figure 1. The simulations were done on 3 dimensional grid. The grid used for simulations is only half size, from the front side to the middle section, because the geometry of boiler is symmetric from the front to the back, assumed that the flows are symmetric, and this will decrease the simulation time. Meanwhile the high of grid is cut off at 17 m of 35.8 m, because there was no much information obtained on the top section. The Simulations were done in 3D geometry with 32 \times 86 \times 61 or total 167872 cells using rectangular mesh (figure 2(a)). The schematic drawing of CFB boiler is shown on figure 2(b). The mesh size of 32 \times 86 \times 61 cells was chosen after performing the grid dependency tests on 3 different mesh sizes, i.e. 32 \times 51 \times 61; 32 \times 86 \times 61 and 32 \times 143 \times 61.

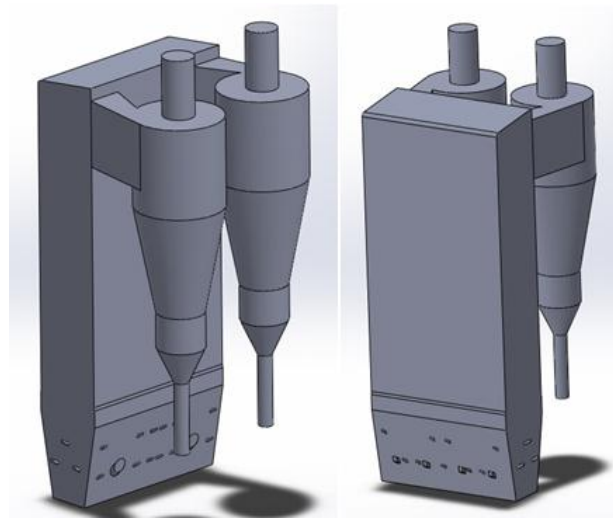


Figure 1. The geometry of CFB boiler.

The transient CFD simulations was carried out with a time step of 0.005 s and the convergent criteria was set at 10^{-3} for every residues. Maximum iterations per time step was 300. The velocity inlet boundary condition was used for the gas input and the outlet for the output. The superficial gas velocity was set uniform in all inlet section. The no-slip boundary condition was used at wall for gas phase. This condition was to guarantee the experiment conditions were met. The Gidaspow model was used for drag function. The main simulation parameters are listed on table 2. The calculations were done by commercial CFDSOF® application. The combustion process was not included in the simulations, because this study was limited only to observe the hydrodynamic of fluid flows and assumed that the combustion process did not influence the hydrodynamics of flow significantly. The complete combustion process in the simulations of CFB boiler will be studied in the next research. So, the inlet

only consisted of the primary and secondary air flow, no coal as fuel and cyclone air back flow accounted for the calculations. The simulations were done in the various excess air values, based on the air needed for the combustion of coal, since this boiler used coal for its fuel.

From the experimental data, to produce around 100 MW electric power, the coal mass rate was 68 t/h, and the total air supply was 316,139 m³/h. Based on this data, the excess air of combustion found to be around 27%. Then the excess air values for simulations were specified at 27%, 45%, 60%, 75% and 90%. Table 3 shows the different air flow for different excess air values, where the quantity of primary air flow is taken 60% of the total air supply.

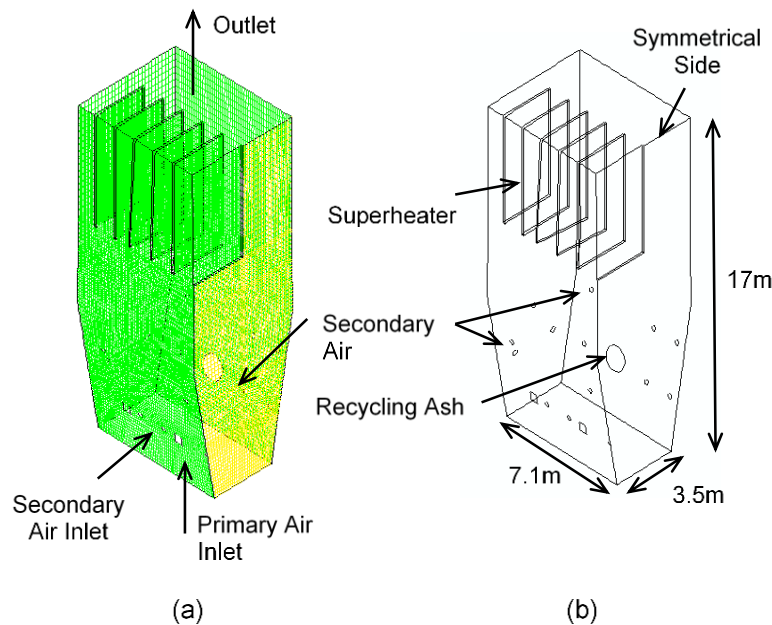


Figure 2. (a) Computational grid. (b) Schematic drawing.

Table 2. Simulations and model parameters.

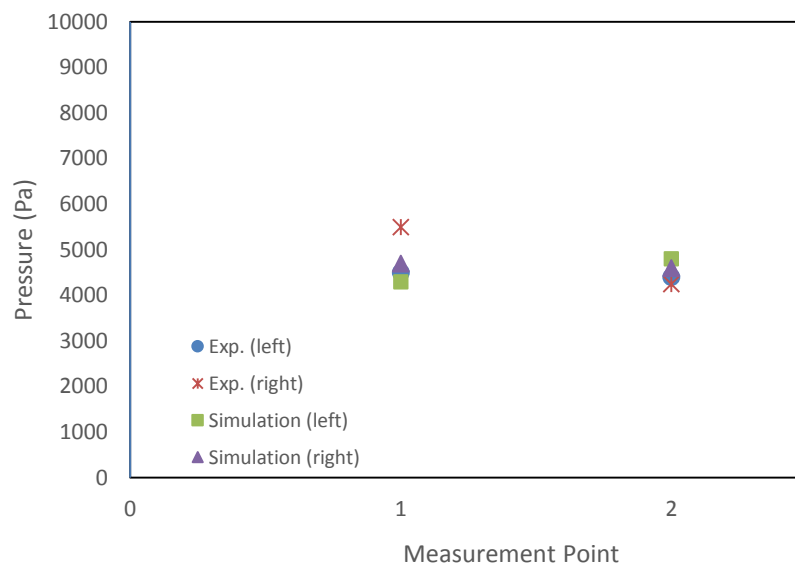
Description	Value	Comment
Particle density, ρ_s	2,500 kg/m ³	Silica sand
Primary air inlet temperature	486 K	Fixed value
Secondary air inlet temperature	498 K	Fixed value
Particle diameter, d_p	1.04 mm	Uniform distribution
Particle temperature	488 K	Fixed value
Restitution coefficient, e	0.80	Fixed value
Initial solid volume fraction, ε_s	0.60	Fixed value
Static bed height	800 mm	Fixed value
Wall boundary condition	<i>no-slip</i>	Fixed value
Step time	0.005 s	Specified
Primary air velocity	Variable, based on excess air values	Specified
Air pressure	1 atm	Specified

Table 3. The air flow for different excess air value.

Excess Air	Primary Air Flow (m ³ /h)	Secondary Air Flow (m ³ /h)	Primary Air Velocity (m/s)
27% (actual)	184946	131193	1.00
45%	216604	144403	1.075
60%	239012	159341	1.15
75%	261419	174279	1.23
90%	283826	189218	1.32

3. Results and discussion

The pressure of gas measured around the bottom of the bed was used to validate the simulation results. The figure 3 shows the graphic of pressure around 1 m from the bottom of fluidized bed for experiments and simulations, point 1 for the left side and 2 for the right. The values of the pressure found to be around 4,500 Pa. The results between experiments and simulations are not different significantly, means the simulation results represent the actual conditions.

**Figure 3.** Pressure in the bottom area for operational excess air (27%).

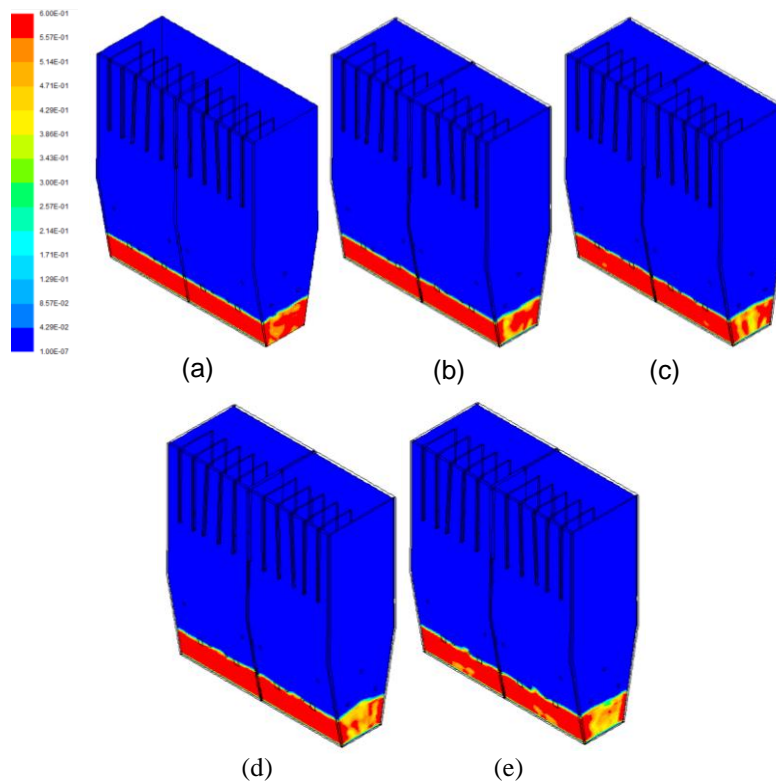


Figure 4. Distribution of solid volume fraction on external boiler walls at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

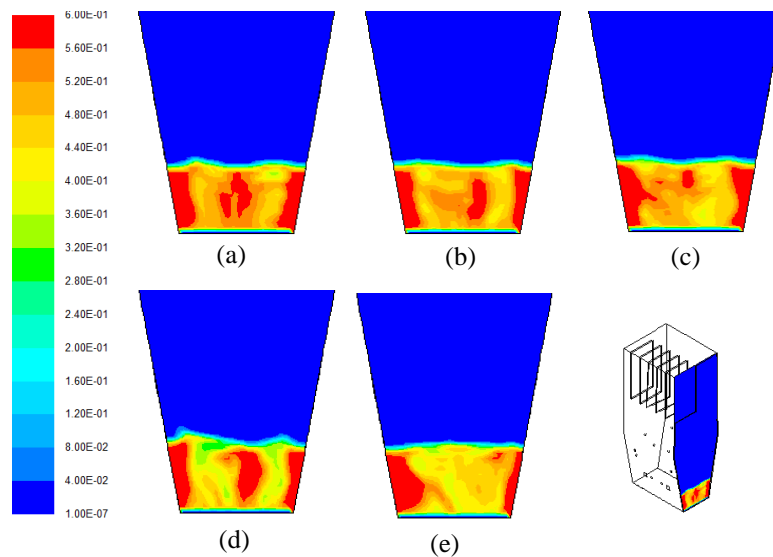


Figure 5. Distribution of solid volume fraction in the middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

Figure 4 shows the simulation results of distribution of solid volume fraction of boiler on the wall side for various excess air values, i.e. 27%, 45%, 60%, 75% and 90%, after 10s of boiler operation. It can be seen that even in the high excess air value, the solid particles are remain at the bottom of fluidized bed, no particles are going up to the outlet section. From their distributions, the higher the excess air value the more uniform the distributions. This distribution in 2 dimensions are presented on figure 5 for middle section of boiler. It is clearly shown that at excess air of 90, the distribution is most uniform, as seen the

red colour that represents the high particles concentration, is reduced. This is resulted by more kinetic energy of air from both sources that increases the drag force. Also seen that the concentration of the particles in the wall area is higher. This is resulted from the downward flow of particle that comes from the middle upward flow.

The figure 6 shows the distribution of solid volume fraction in the longitudinal middle section after 10s boiler operation. The dense of particles are decreasing whenever the excess air is increasing, since more particles are lifted up as the results of more kinetic energy. The turbulent fluidization regimes are found at the excess air values of 75% and above, even though in small scale, because no clear surface observed on this pictures, especially at excess air of 90%. Meanwhile at the excess air of 60% and below, the fluidized beds are still in the bubbling conditions. The regime of the fluidization occurred, beside affected by the superficial velocity, it is also affected by the particle diameter, where the bigger the diameter the slower the transition of fluidization from one regime to another. The average particle diameter used in this boiler is around 1 mm, bigger than usual size found in the CFB boiler, i.e. around 0.2 mm [1, 4, 5]. That is why to achieve the turbulent fluidization, more air supply or higher excess air values needed. In one side, this is an advantage because there is no particles lifted up by the air to the water tube so the water tube will be free from erosion and the leakage can be avoided. But in the other side there will be more energy needed by the blower to supply the air to the boiler.

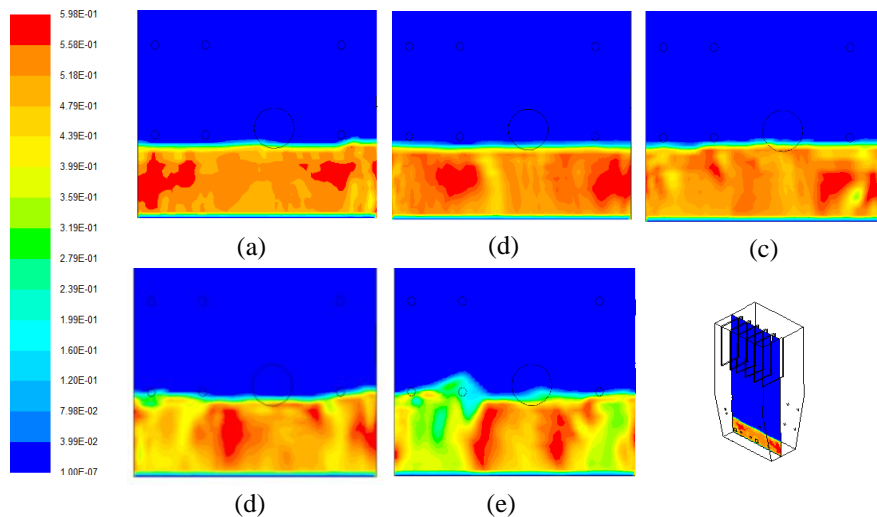


Figure 6. Distribution of solid volume fraction in the longitudinal middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

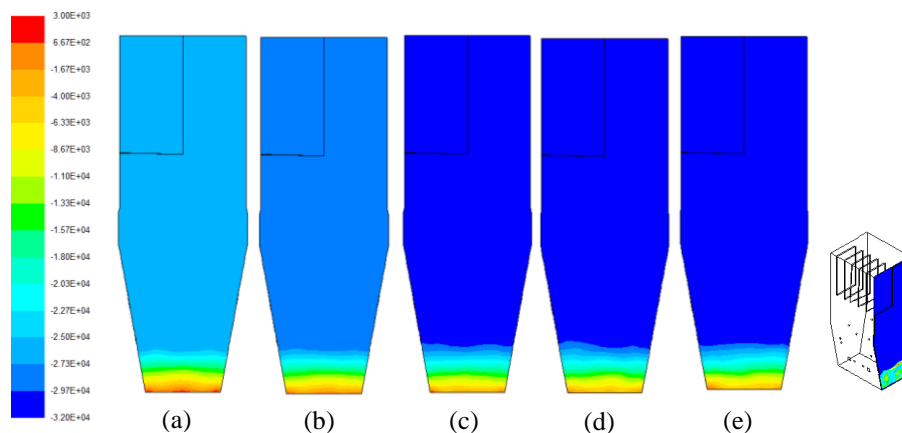


Figure 7. Distribution of relative static pressure in the middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

Figure 7 shows the distribution of relative static pressure in the middle section of boiler after it runs for 10 s. From the colour in the freeboard, it can be seen that at the excess air values of 27% and 45% the relative static pressure are higher than that of 60% and above, or in other word, the pressure drop on bed is relatively lower. This is possibly due to that at the excess air of 45% and below, the fluidized bed is still near or could be below the minimum fluidization velocity, means that the fluidized bed has not entered the fluidization state. At the excess air of 60% and above, the static pressures are relatively constant, so the fluidized bed might enter the stable fluidization state already.

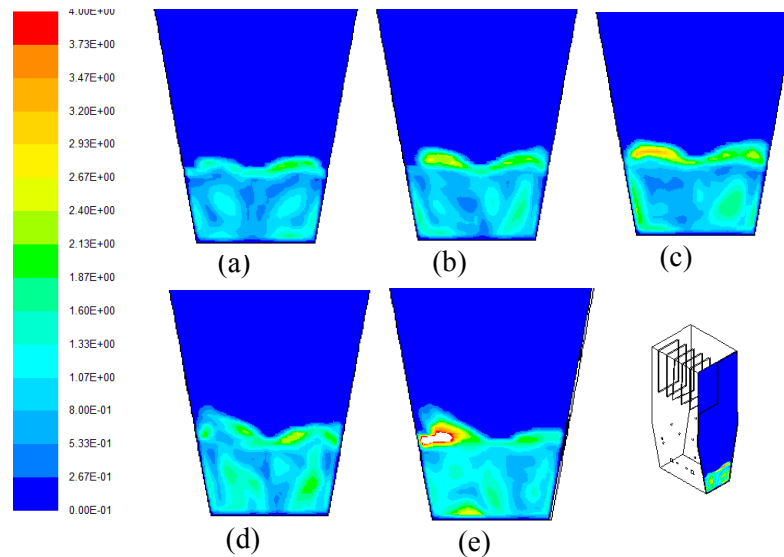


Figure 8. Distribution of solid velocity in the middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%. (white region indicates the velocity is above the scale).

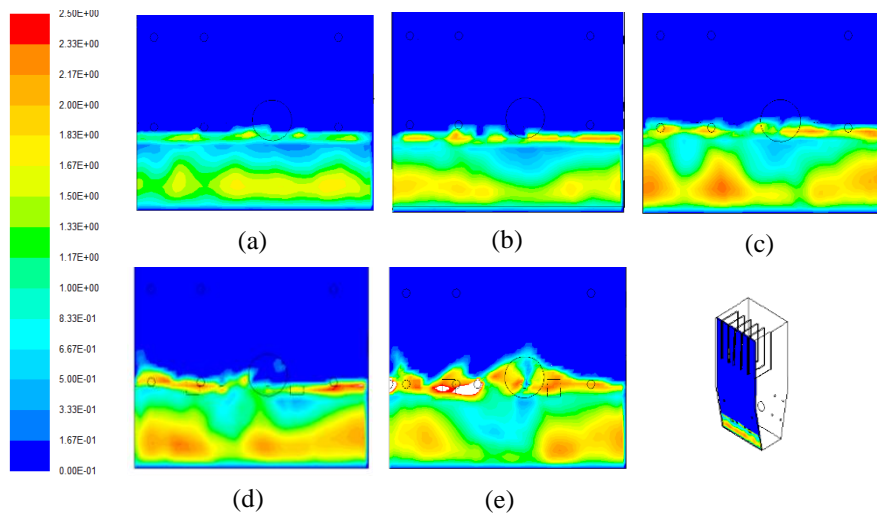


Figure 9. Distribution of solid velocity in the left wall section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90% (white region indicates the velocity is above the scale).

Figure 8 and Figure 9 show the distribution of particle velocity in the middle and the left wall section of boiler respectively. The higher the excess air values the higher the velocity of particles, especially at the top of the bed as seen on figure 8. The high velocity in the left top area, clearer seen on the excess air of 90%, possibly caused by the secondary air flow coming in from the left since its hole is near from this section. On figure 9 also seen that the velocity of particles are uniform on the higher excess air values than that of lower ones. This phenomena are the same with the volume fraction distributions, indicate that at the higher excess air values the distribution of particle are more uniform. Also can be seen from

these figures that the particles exist in the furnace, no particles fly to the water tube area, even on the highest excess air values. For the excess air of 90%, the velocity of particles at the surface of bed is highest, and this can erode the firewall in long term operations.

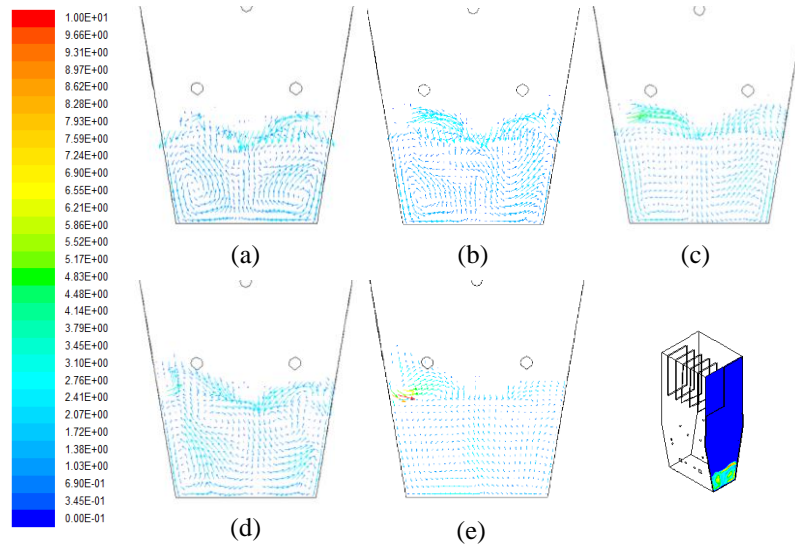


Figure 10. The particle velocity vector in the middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

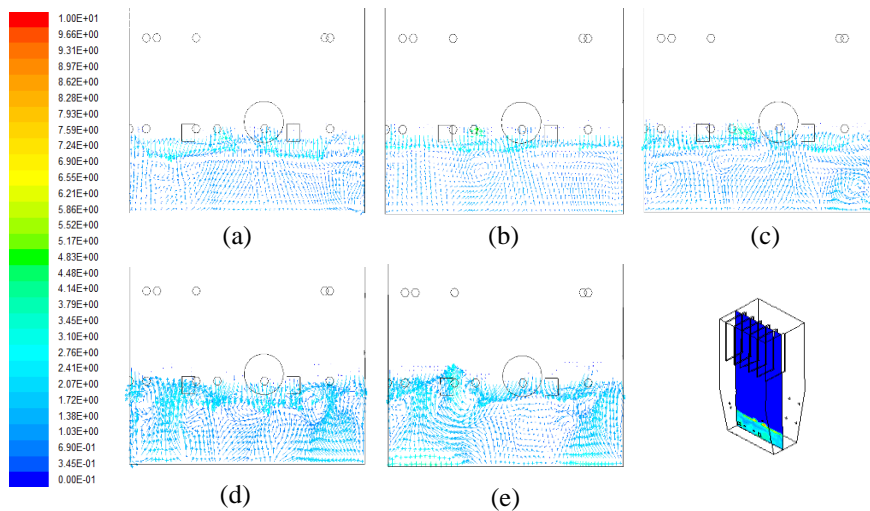


Figure 11. The particle velocity vector in the longitudinal middle section at 10 s for the excess air of: (a) 27%, (b) 45%, (c) 60%, (d) 75%, and (e) 90%.

Figure 10 shows the velocity vector of particles in the middle section after 10s for different excess air values. From this vectors, seen that there are circulating flows of particles in the left and right zones. These flows caused by the downward flow of particles in annulus and upward flow in the core zone. These circulating flows will give the good effects in mixing among of air, particles and fuel. The obvious difference observed among the various excess air values is the height of bed, where the bigger the excess air value the higher the bed. Meanwhile the velocity vector of particles in the longitudinal section is shown on figure 11. The patterns on this figures are not much different with the figure 10. So, from these 2 figures, observed that the different of excess air values do not affect much the pattern of the velocity vector, accept the height of bed.

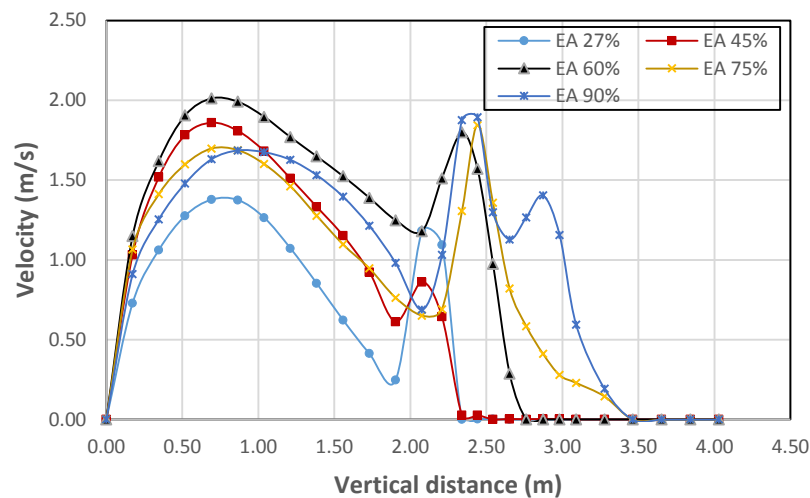


Figure 12. Particle velocity in the left wall (vertical axis) at various excess air (EA) values.

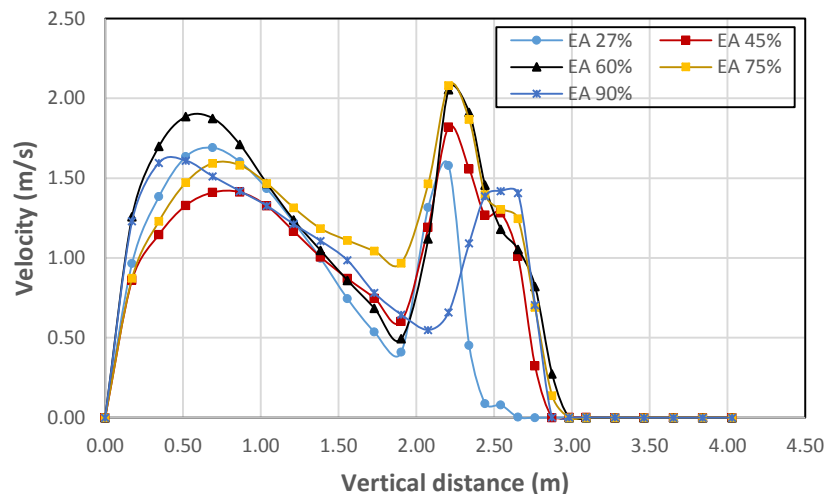


Figure 13. Particle velocity in the right wall (vertical axis) at various excess air (EA) values.

The particle velocities near the wall can be seen on figure 12 for the left wall, and figure 13 for the right wall. The maximum velocity is around 2 m/s, still safe for boiler. From these figures observed that in the higher values of the excess air, the curves shifted to the right, means that the particles move higher in the fluidized bed. Overall, the maximum particles movements not more than 3.5 m, or still in the furnace area, safe for water tube.

4. Conclusion

The simulations of complex fluid flow in 100 MW Circulating Fluidized Bed (CFB) boiler have been done using Computational Fluid Dynamics (CFD) method to find the optimum value of excess air. The simulations were done using 3D geometry with mesh size of 32×86×61 cells. The CFD used the Multi-fluid Eulerian approaches incorporating the Kinetic Theory of Granular to calculate the motion of particles and the standard $k-\varepsilon$ turbulence model to simulate the fluid flows. The values of the pressure drop found from experiments were between 15,000 Pa to 16,000 Pa, and the results from simulations were not different significantly with this values. From the distributions of volume fraction of solid from simulations, found that the higher the excess air value the more uniform the distributions, but the particles did not reach the water tube area. From the distribution of static pressure, at the excess air of 27% and 45%, the static pressure is higher than that of excess air of 60% and above. It means that the pressure drop on bed of these excess air are lower than the others. It could be the fluidized bed at this

excess air values did not reach the fluidization state yet. The velocity of particles increased when the excess air values increased and at the excess air of 90%, the velocity of particles at the surface of bed is highest that can lead to the erosion on the firewall in long term operations, meanwhile at the excess air values lower, their velocity were relatively small. From the velocity vector of solid, found that there were the circulating flow for all values of excess air, but the different of excess air values do not affect much the pattern of the velocity vector, accept the height of bed. The maximum particle velocity near the wall was around 2 m/s, and at the higher values of the excess air, the velocity curves shifted to the vertical direction. From this study found that the optimum amount of excess air safe for boiler to operate was around 60% - 75%. These values could be minimized by reducing the solid particle size to around 0.2 – 0.3 mm.

Table 4. Nomenclature.

g	Gravitation	XpA	Excess air
G_k	Generation of turbulence kinetic energy due to mean velocity gradients	$C_\mu, C_{1\varepsilon}, C_{2\varepsilon}$	Constants
G_b	Generation of turbulence kinetic energy due to buoyancy	<i>Greek letters</i>	
Y_M	Fluctuation dilatation in compressible turbulence to the overall disipation rate	β	Interphase exchange coeeficient (kg/m ³ s)
I	Unit tensor	ε	Volume fraktion, disipation transport
J	Granular energy transfer (kg/m/s ³)	γ	Disipation of fluctuating energy (kg/m/s ³)
P	Pressure (kPa)	Θ	Granular temperatur (m ² /s ²)
q	Difusion of fluctuating energy (kg/s ³)	μ	Shear viskosity (Pa.s)
$MoDPc$	Moles dry product (actual SO2 produced)	μ_t	Turbulent viskosity (Pa.s)
$MoThACr$	Moles of theoretical air required (corrected)	ρ	Density (kg/m ³)
$DVpO2$	Percent O ₂ in dry flue gas at Air Heater inlet	τ	Shear stress tensor (N/m ²)
S_k, S_ε	Source term	$\alpha_k, \alpha_\varepsilon$	Constants
u	Velocity (m/s)	<i>Subscripts</i>	
k	Kinetic energy	g	Gas
		s	Solid

Acknowledgments

The authors would like to thanks “Kementrian RISTEK DIKTI Republik Indonesia” for funding this research through “Hibah Penelitian Disertasi Doktor (PDD) 2018” and to PT. CCIT Group Indonesia for CFDSOF® software license.

References

- [1] Adamczyk W P, Węcel G, Klajny M, Kozolub P, Klimanek A and Bialecki R A 2014 Modeling of particle transport and combustion phenomena in a large-scale circulating fluidized bed boiler using a hybrid Euler–Lagrange approach *Particuology* **16** 29-40
- [2] Bakshi A, Altantzis C, Glicksman L R and Ghoniem A F 2017 Gas-flow distribution in bubbling fluidized beds: CFD-based analysis and impact of operating conditions *Powder Technology* **316** 500-511
- [3] Ngho J and EWC Lim 2016 Effects of particle size and bubbling behavior on heat transfer in gas fluidized beds *Applied Thermal Engineering* **105** 225-242
- [4] Zhang N, Lu B, Wang W and Li J 2010 3D CFD simulation of hydrodynamics of a 150 MWe circulating fluidized bed boiler *Chemical Engineering Journal* **162** 821-828
- [5] Adamczyk W P, Myöhänen K, Hartge E U, Ritvanen J, Klimanek A, Hyppänen T and Bialecki R A 2018 Generation of data sets for semi-empirical models of circulated fluidized bed boilers using hybrid Euler-Lagrange technique *Energy* **143** 219-240

- [6] Adanta D, Budiarmo W and Siswantara A I 2018 Assessment of turbulence modelling for numerical simulations into pico hydro turbine *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* **46** 21-31
- [7] Ramdhan G G G, Siswantara A I, Daryus A and Pujowidodo H 2016 Turbulence model and validation of air flow in wind tunnel *International Journal of Technology* **7** 1362-1371
- [8] Daryus W E Y A, Siswantara A I, Gunadi G G R, Darmawan S and Camalia R 2016 CFD simulation in annular combustion chamber of proto X-3 bioenergy micro gas turbine *Journal of Engineering and Applied Sciences* **11** 2066-2071
- [9] Daryus A, Siswantara A I, Darmawan S, Gun G, Gunadi R and Camalia R 2016 CFD simulation of turbulent flows in proto X-3 bioenergy micro gas turbine combustor using std k- ϵ and rng k- ϵ model for green building application *International Journal of Technology* **7** 204-211
- [10] Siswantara A I, Daryus A, Darmawan S, Gunadi G G R and Camalia R 2016 CFD analysis of slurry flow in an anaerobic digester *International Journal of Technology* **7** 197-203
- [11] Shi H, A Komrakova and P Nikrityuk 2019 Fluidized beds modeling: Validation of 2D and 3D simulations against experiments *Powder Technology* **343** 479-494
- [12] Benzarti S, Mhiri H, Bournot H and Occelli R 2013 *Simulation of gas-solid turbulent fluidized bed hydrodynamic in The 14th International Conference on Fluidization - From Fundamental to Products* 2013
- [13] Hernández-Jiménez F, Sánchez-Delgado S, Gómez-García A and Acosta-Iborra A 2011 Comparison between two-fluid model simulations and particle image analysis & velocimetry (PIV) results for a two-dimensional gas–solid fluidized bed *Chemical engineering science* **66** 3753-3772
- [14] Iswara M A I, T Nurtono and S Winardi 2016 Penentuan kecepatan minimum fluidisasi dan bubbling batu bara di dalam fluidized bed dengan simulasi CFD *Jurnal ESDM* **8** 51-57
- [15] Adamczyk W P, Kozolub P, Klimanek A, Białecki R A, Andrzejczy M and Klajny M 2015 Numerical simulations of the industrial circulating fluidized bed boiler under air-and oxy-fuel combustion *Applied Thermal Engineering* **87** 127-136
- [16] Zi C, Sun J, Yang Y, Huang Z, Liao Z, Wang J and Han G 2017 CFD simulation and hydrodynamics characterization of solids oscillation behavior in a circulating fluidized bed with sweeping bend return *Chemical Engineering Journal* **307** 604-620
- [17] Sudarmanta B, et al *Studi numerik pengaruh diameter bed material terhadap karakteristik fluidisasi pada circulating fluidized bed boiler beban 28,6 MW in Seminar Nasional Teknik Mesin 12, Universitas Kristen Petra Surabaya* 2017 Surabaya, Indonesia
- [18] Gidaspow D 1994 *Multiphase flow and fluidization: Continuum and kinetic theory descriptions* (Boston: Academic Press)
- [19] Lun C K K, Savage S B, Jeffrey D J and Chepurniy N 1984 Kinetic theories for granular flow: inelastic particles in Couette flow and slightly inelastic particles in a general flowfield *Journal of fluid mechanics* **140** 223-256
- [20] Schaeffer D G 1987 Instability in the evolution equations describing incompressible granular flow *Journal of Differential Equations* **66** 19-55
- [21] Anonymous 2006 *Fluent 6.3 User's Guide* (Lebanon: Fluent Inc)
- [22] Anonymous 2008 *Sumatera Barat 2x112 MW Power Plant Project, Performance Test Report for Unit No 1, Boiler Section* (China National Technical Import and Export Corporation)