JMES The International Journal of
Mechanical Engineering and Sciences
http://iptek.its.ac.id/index.php/jmes/index

Numerical Study Effect of Fluidizing Air to Erosion Pattern in Circulating Fluidized Bed Boiler

Bambang Sudarmanta[1] *, Rizki Mohammad Wijayanto[2], Giri Nugroho[3], Achmad Syaifudin[4], Atok Setiyawan[5], Julendra B. Ariateja[6]

Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia

Received: 28 June 2016, Revised: 20 July 2017, Accepted: 3 August 2017

Abstract

Bed material particles in a Circulating Fluidized Bed (CFB) boiler which entrained in the flue gas may cause material degradation due to abrrasive and high velocity impact of particles to wall surface. In this study, Computational Fluid Dynamic (CFD) commercial software with Eulerian multiphase is used to study the erosion pattern in several different fluidizing air velocity. The result obtained from simulation in terms of particles volume fraction and particles velocity in selected area was utilised to predict the erosion rate in several different fluidizing air velocity to achieve the optimal value of fluidizing air velocity. The results obtained in this study are helpful to understand how erosion pattern in CFB boiler, how effect fluidizing air velocity to erosion rate, and also helped to know the potential areas occur erosion so helped to choose suitable material in different region.

Keywords: Circulating Fluidized Bed Boiler, Computational Fluid Dynamics, Erosion, Eulerian Multiphase, Fluidizing Air Velocity, Solid Particles.

1. Introduction

Circulating Fluidized Bed (CFB) boiler is a type of steam generator which is introduced in the 1970s [1]. A CFB boiler operated under a special hydrodynamic condition, where fine solid are transported through the furnace due to fluidizing air which injected from nozzle in the beneath of bed material. The solid leaving the furnace is captured be a gas-solid cyclone separator and recirculated back to the furnace. Compared to other types of solid fuel fired boilers (e.g., bubbling fluidized bed and pulverized coal combustion), the CFB has a number of unique features which make it more attractive, which is fuel flexibility, high combustion efficiency, good gas-solid mixing, efficient sulfur removal, low NOx emission produce, and suitability for supercritical operation [2].

While the use of CFB boiler has grown greatly, CFB boiler faces some types of problem like blockage of feed line, loss in efficiency, fouling, solid leakage, material degradation, and ash deposition. The most likely problem is material degradation due to erosive effect from solid particles [2]. Erosion in a coal-fired power plant is recognized as the main cause of the downtime due to outage accounting up to 50-75 % of the total forced outage and costing up to 54% of the total production cost [2].

Erosion will occur when solid particles (sand from bed material and fuel ash) are entrained in the flue gas along the furnace and cyclone and then impact on the wall surfaces with a certain angle and relatively high velocity [3]. The impact of solid particles to the surface will result in two force component: a force normal, which causing material deformation and a force parallel to the surface, which causing material removal. The impact angle is very affect to the erosion rate. Erosion rate also depends on impact velocity, solid particles concentration, shape and size of solid particles, temperature and properties of wall surface material.

Due to the increasing of electricity demand, the operation and maintenance issue causing outage shutdown and production lost like material abrasion must be resolved. So, the erosion pattern in the CFB boiler is very necessary to study as the information and recommendation to power plant operator or as the consideration in the improvement of the CFB boiler operation and also redesign. Computational Fluid Dynamic (CFD) simulation is conducted in this study to observe the erosion pattern in the CFB boiler under different operating conditions. The difference of operating conditions in this study lies in the primary air velocity. The investigation through the experimental study would be costly and taking much time. The Eulerian multiphase in combination with $k-\epsilon$ standard turbulence model is utilized to conduct hydrodynamic phenomena of gas-solid inside the CFB boiler simulation [4].

However, it is still difficult to consider effect from all parameters which causing erosion as shown before. So in this study, the attempt to identify erosion pattern and to identify those areas inside the furnace which may be

*Email:sudarmanta@me.its.ac.id
Phone/Fax:+62(31)5922941

prone to erosion only limiting on the measureable parameter from the CFD software we use. There are fraction volume and velocity of solid particles. The detailed geometry, model, and input parameter are described in the following section.

2. Literature Review

Simulation of fluidization behavior in CFB boiler was performed by Tanskanen [5]. In his study, Eulerian multiphase was used for multiphase model and $k-\epsilon$ standard was used for turbulence model. This is because compared to other available turbulence models such as $k-\epsilon$, realizable $k-\epsilon$, and turbulence mixture, $k-\epsilon$ standard is more applicable and accurate to many flow conditions. The study showed that bed density decrease with increasing height. While the maximum air velocity occurs in the area far from the wall and on the wall air velocity is negative value. It is also known that relatively large sand or coal particles tend to be in the lower region. Mass of the bed material decrease with the increasing time, but the decrease is not too significant so it can be ignored.

Kinkar et al. [6] investigated the potential for erosion that occurs in the cyclone wall due to erosion by air or sand. The SST k-w model is selected to ensure revolving the flow gradients that are expected in the cyclone separator. The study informed that in some areas of the cyclone, the air and sand velocity can reach 30 m/s which can damage the cyclone if ignored. The results also help to understand the flow of flue gases in the CFB loop and actual target velocities which is practically difficult to get from the operating boiler.which isThe most common ares of refractory erosion and failure are bull nose cyclone, cyclone inlet duct, and loop seal. Installation of the refractory with a stronger material will prevent erosion in that areas so as to extend the life of cyclone.

Gandhi et al. [3] conducted Computational Fluid Dynamics to understand the flow field and identify the areas likely to be subjected to erosion under various operating conditions in a pulverised coal fired boiler. An Eulerian-Langrangian approach was used to analyse the continuum phase and particle tracking for individual coal particles. The data obtained on particle velocities, particles concentration, and temperature have been utilised to predict the extent of erosion in selected areas of the boiler. This results also provide a platform for the development of an erosion tool which could assist power utilities in avoiding unnecessary shutdowns.

2.1. Erosion Model

Erosion is removal of materials from a surface by a stream of solid abrasive particles impacting on it [2]. The process of erosion is similar to metal cutting Figure 1 shows the process when a solid abrasive particle impacts a surface at an angle, α, and with a velocity, Vp. Two components of impact force will be result, a force normal to the surface causing deformation and a force parallel causing material removal.

Erosion is affected by three major factors, flow and enviromental conditions, particle properties, and nature of target materials [7]. Flow and enviromental condition factor included by angle of implingement, particles concentration, and particle velocity (momentum). The erosion rate of ductile material increase with the angle of implingement. The rate peaks at around 45° and decrease again, reaching a minimum value at 90°. Higher particle velocity impacting wall surface will be increase erosion rate. Operating temperature does not exert much influence on erosion, especially in CFB boiler where the CFB boiler operation temperature is not enough to make the boiler wall materials being soft.

Particles properties like mechanical properties, equivalent size, and sphericity also affect to erosion rate. For example, a soft particles material could have a considerably lower erosion rate than a hard particles material. The abrasiveness of the particles also plays a major role. Increasing particles size also will increase the erosion rate. Nonspherical angular particles cause greater erosion than rounded ones [7].

Nature of target materials such as surface hardness, impact strength and ductility are a major factor influencing the erosion rate.

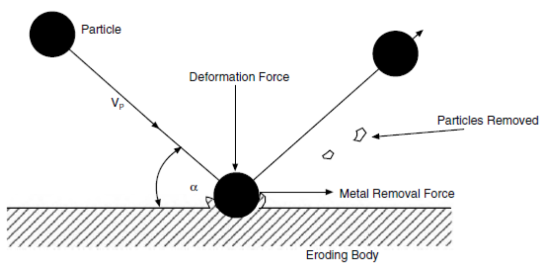

Figure 1. Mechanism of erosion due to solid particle impact.

However, it is difficult to predict the relative contribution of each factors propesed erosion [3]. An empirical equation for an overall erosion rate by Mbabazi et al. [8]. This equation included all major parameters which are responsible to metal erosion process.

$$\varepsilon = \frac{3.5x^{4.95}\rho_m\rho_p^{0.5}V^3\sin^3\beta}{\sigma_y^{1.5}} \tag{1}$$

where ϵ is erosion rate, x is particle concentration, ρ_g is gas density, ρ_p is particle density, V is particle velocity, β is angle of implingement, dan σ_y is yield stress of target material. From Equation 1 shows that the attempt to identify erosion rate can be identified from particle concentration and velocity of solid particles which the only measurable parameter from the CFD simulation.

Erosion in CFB boiler can reduce with some action such as use of flow disrupters, protective coating or surface treatment, and changes in fluidization condition. Computational Fluid Dynamic (CFD) simulation result from this study can use as basic to reducing erosion rate. Instalation flow disrupter and protective coating in the area known as area potentially occur erosion from simulation is necessary to prevent erosion. And also the effect of operation parameter, in this study fluidization air velocity also will be known. So with the most optimal fluidization air velocity, good fluidization conditions which has the lowest erosion rate can be reach.

3. Simulation Approach

The simulation geometry in this study involves furnace and cyclone of CFB boiler, as shown in Figure 2. The boundary conditions used in this study are mass-flow inlet for two pieces primary air inlet, sixteen pieces secondary air inlet, three pieces coal inlet, pressure-outlet for two pieces flue gas outlet, and the rest set as wall.

This simulation was performed in a 3D model with dimension 11.66 m x 8.50 m x 31.78 m. The air distribution grid, also known as nozzle, is located at a height of 5 m. This simulation was based on a finite volume approach where computational domains are divided into control volume through computational mesh [9], which has a total 2,273,622 of nodes, as shown in Figure 3.

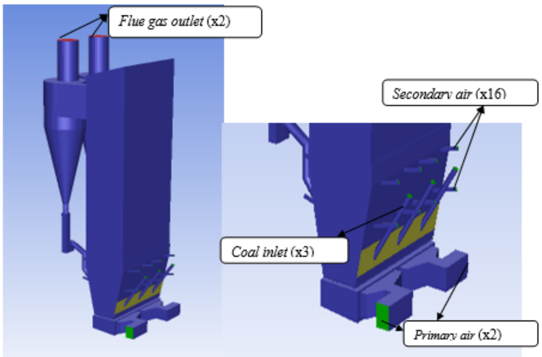

Figure 2. Geometry of CFB boiler.

In this study, Eulerian model is used for multiphase model to define gas-solid phase and its interactions. The turbulence model used in this study is standard $k-\epsilon$ because of its general applicability, robustness, and economy [10]. Moreover, the Syamlal O'Brien and the Luen et al. were chosen for granular viscosity and solid granular bulk viscosity, respectively [4]. The drag model in interaction between sand and air used gidasprow model. The gidasprow model is recommended for dense fluidized beds [11]. The collisions value is 0.1 because lower value of that coefficient predicts the behavior of gas-solid mixing near the wall more precisely [4]. To observe the hydrodynamic of gas-solid mixing, the combustion process is not needed to be considered in the simulation and the model is assumed to isothermal.

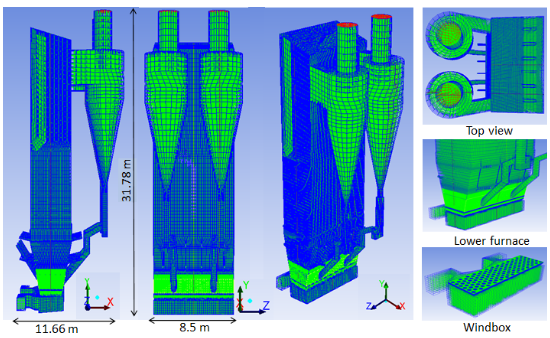

Figure 3. Meshing of CFB boiler.

The simulation parameters used in this study are resumed in Table 1. Sand used for bed has the diameter of 200 μm and density of 2500 kg/m^3 [12, 13]. The initial static bed height is 0.4 m from nozzle base with solid volume fraction of 0.5. This data is adjusted to the 30 MW CFB boiler at one of the power plants in Indonesia.

In this study, variation of operation parameter will be done on the fluidizing air velocity when enter primary air's boundary condition. The variation of fluidizing air velocity is shown in Table 2.

Table 1. Simulation parameters

Parameters	Value
Sand diameter	200 μm
Sand density	2500 kg/m3
Initial static bed height	0.4 m
Initial solid volume fraction	0.5
Temperature combustion air	470 K
Outlet pressure (gauge)	-1 kPa

Table 2. DATA VARIATION OF FLUIDIZING AIR VELOCITY

Case	Volumetric Flow (m^3/h)	Area of Boundary Condition (m2)	Velocity (m/s)
1	46400	1.254	5.137
2	54000	1.254	6.422
3	69600	1.254	7.706
4	81200	1.254	8.991

3.1. Governing Equations

The Eulerian multiphase model was utilized to study the hydrodynamic of gas-solid flow in fluidized bed based on the conservation equations of continuity and momentum [4, 12, 13]. Since there is no mass transfer, the conservation equation of continuity for gas (g) and solid (s) phases can be reduced to

$$\frac{\partial}{\partial_t}\left(\varepsilon_g\rho_g\right)+\nabla\left(\varepsilon_g\rho_g\vec{V_g}\right)=0 \tag{2}$$

$$\frac{\partial}{\partial_t}\left(\varepsilon_s\rho_s\right)+\nabla\left(\varepsilon_s\rho_s\vec{V_s}\right)=0 \tag{3}$$

where ϵ, ρ and $\vec{V}$ are the volume fraction, density and local velocity, respectively. The conservation equation of momentum for gas and solid phase are given by

$$\frac{\partial}{\partial_t}\left(\varepsilon_g\rho_g\vec{V_g}\right)+\nabla\left(\varepsilon_g\rho_g\vec{V_g}\vec{V_g}\right)=-\varepsilon_g\nabla_p+\quad\nabla_{\tau_g}+\quad\varepsilon_g\rho_g g\quad+K_{gs}\left(\vec{V_s}-\vec{V_g}\right) \tag{4}$$

$$\frac{\partial}{\partial_t}\left(\varepsilon_s\rho_s\vec{V_s}\right)+\nabla\left(\varepsilon_s\rho_s\vec{V_s}\vec{V_s}\right)=-\varepsilon_s\nabla_p+\quad\nabla_{\tau_s}+\quad\varepsilon_s\rho_s g\quad+K_{gs}\left(\vec{V_g}-\vec{V_s}\right) \tag{5}$$

where K_{gs} is the gas-solid momentum exchange coefficient, p_g is gas phase static pressure, p_s is solid pressure, g is the gravitational acceleration and τ is and stress tensor.

A general CFD software was used to solve governing equations above. Phase Coupled SIMPLE was used to consider the pressure-velocity coupling [10]. The first order upwind discretization was used for momentum, mixture fraction, turbulence kinetic energy and turbulence dissipation rate. Iterations were performed transient for the time step size of 0.01 s with 500 number of time steps.

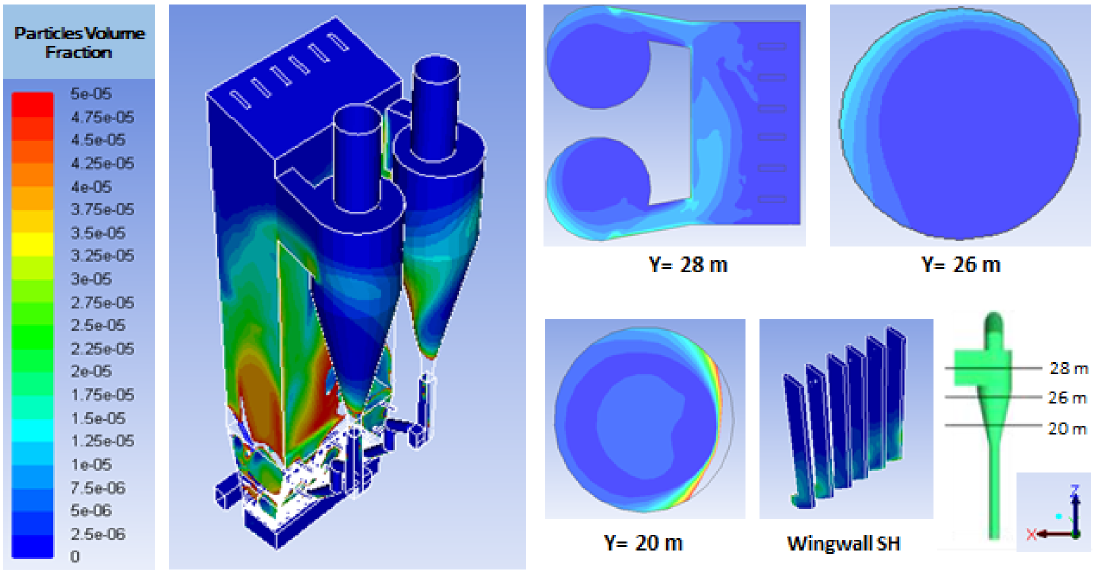

Figure 4. Contour of Particles Volume Fraction for Case 1.

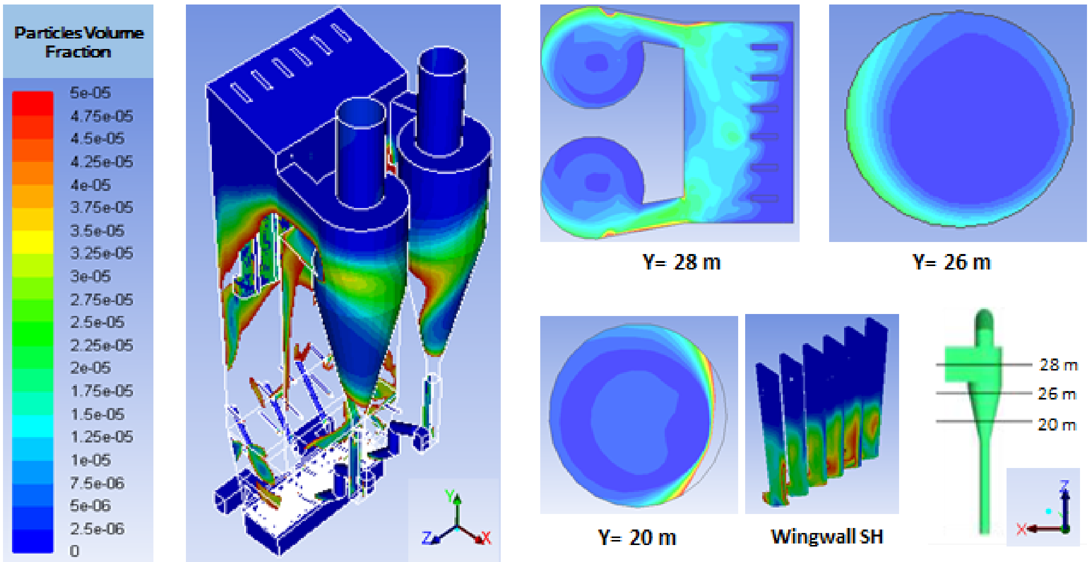

Figure 5. Contour of Particles Volume Fraction for Case 2.

4. Results and Discussions

According to Basu [2, 7], it is known that the areas with the most potential erosion are cyclone, wingwall superheater, lower furnace, wall tubes, and nozzle grid. Therefore, the discussion of erosion pattern will be more focused on that areas. Analysis will be done by comparing the erosion pattern on four variations of fluidizing air velocity.

4.1. Particle concentration

In this section will be discussed the distribution of particle concentration which is depicted with particles volume fraction contour. Figure 4,5,6,7 are the contours of particles volume fraction in isometric view, cyclone elevation at 28 m, 26 m, and 20 m, and also wingwall superheater for case 1, case 2, case 3, and case 4, respectively.

In case 1 the value of the particles volume fraction used is 0.5x10^{-5}. From Figure 4 it known that the particles that reach the cyclone are only a very small number about 1.5x10^{-5} – 2x10^{-5} (15-20 ppm). Then for the superheater wingwall area, sand particles found only at the base of the base only with a volume fraction value of about 1.5x10^{-5} (15 ppm). The sand particles tend to remain in the furnace, so the potential for erosion in the cyclone section in this case is relatively small.

Then for case 2 as shown in Figure 5 it can be seen that there is an increase in the concentration of sand particles reaching the wingwall superheater and cyclone. In the cyclone the maximum sand volume fraction can reach a value of $4.5\text{x}10^{-5}$ (45 ppm) in the inlet duct region. Then for wingwall superheater area the concentration of sand particle is higher than in case 1, that can reach value about $4.2\text{x}10^{-5}$ (42 ppm).

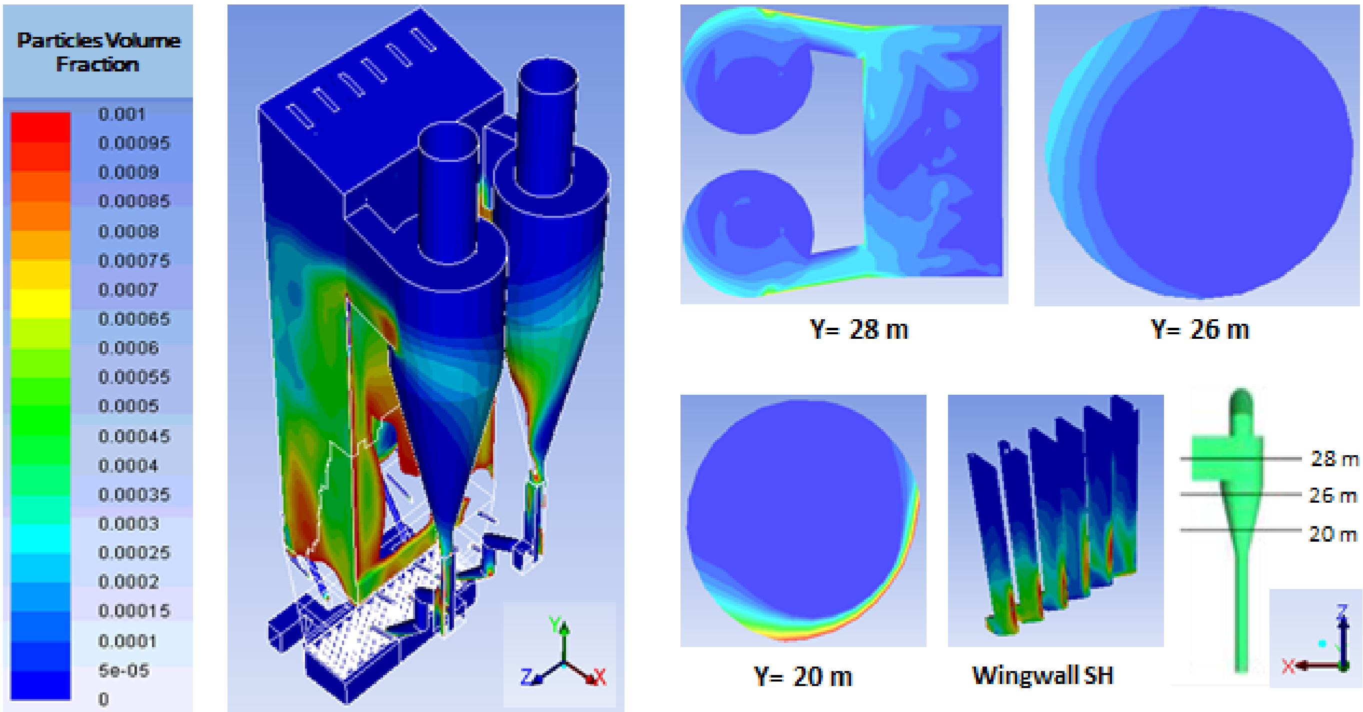

Figure 6. Contour of Particles Volume Fraction for Case 3.

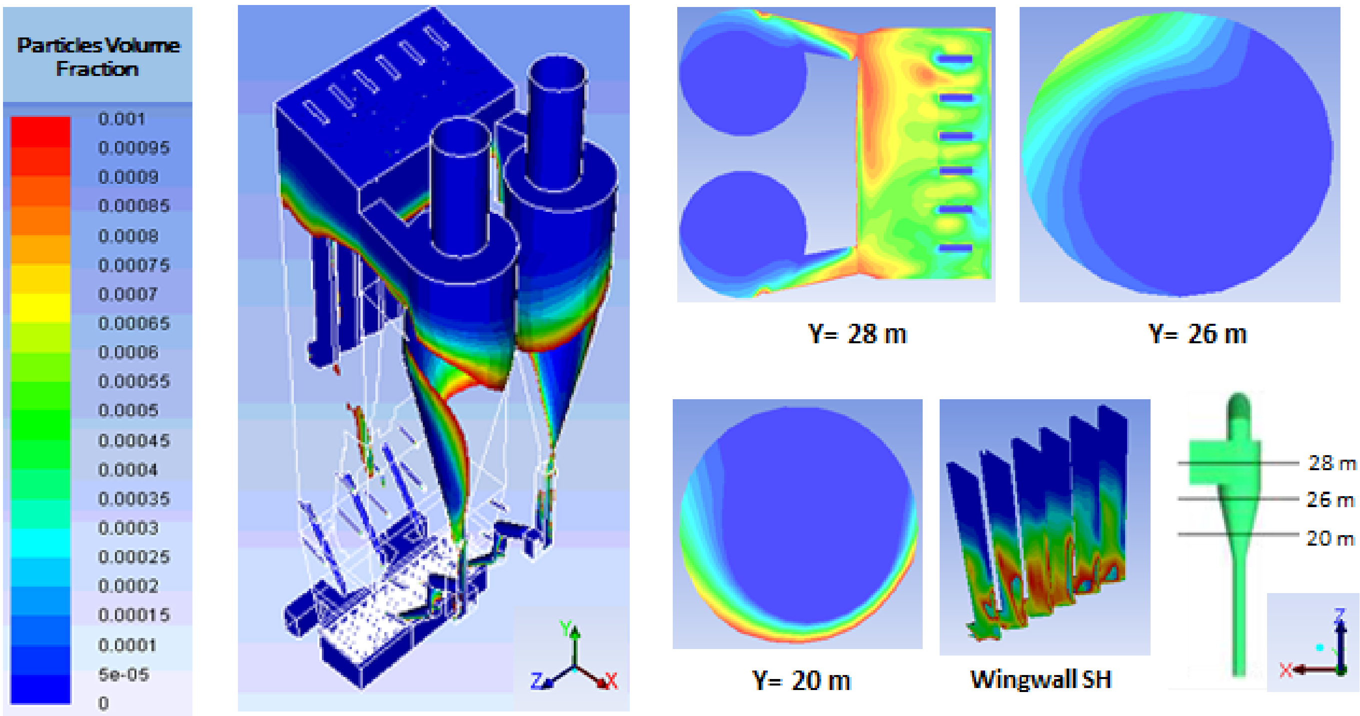

Figure 7. Contour of Particles Volume Fraction for Case 4.

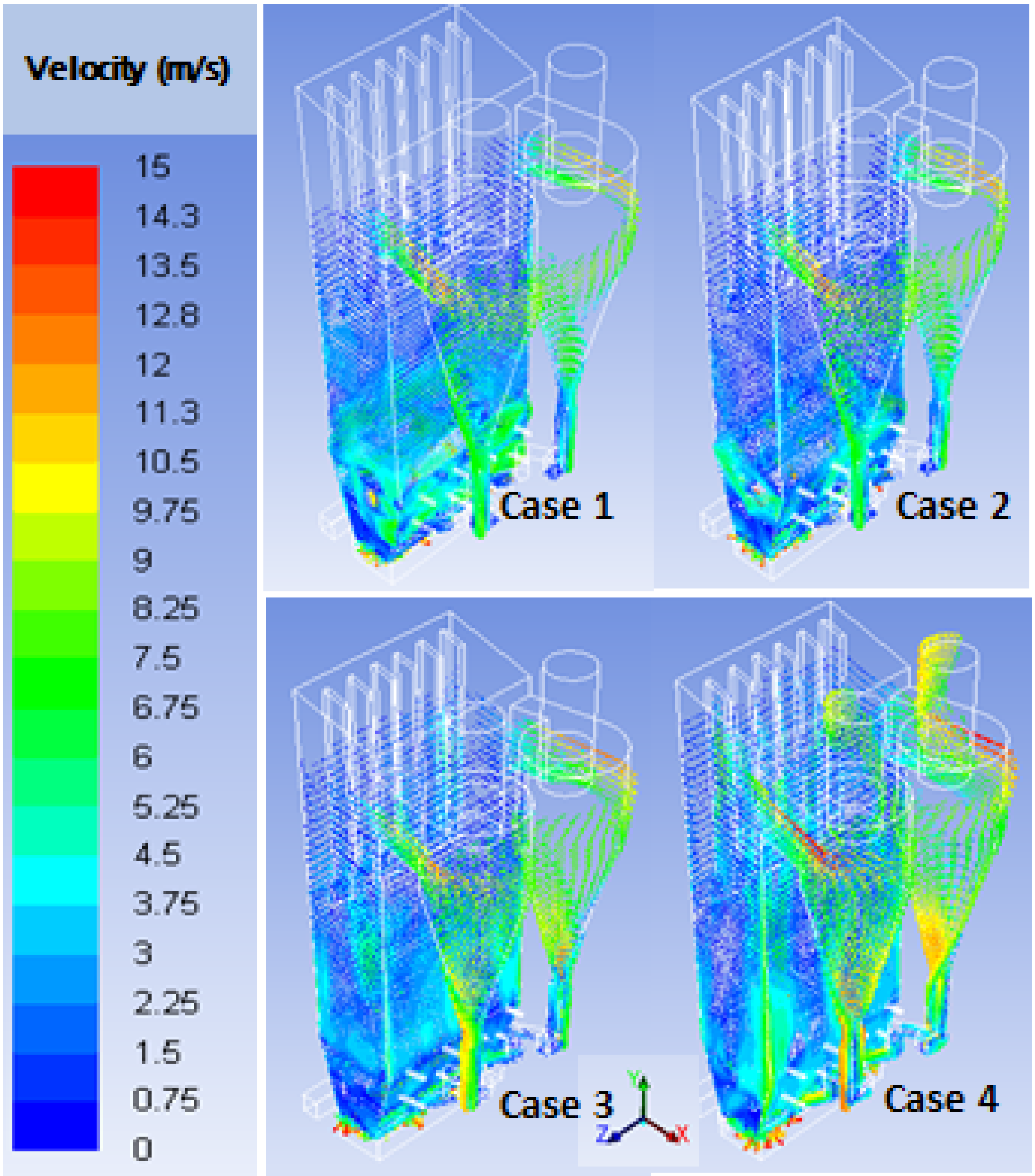

Figure 8. Vector of Particles Velocity in Isometric View for All Cases.

The distribution of the volume fraction of the sand particles in Case 3 can be seen in Figure 6. In the case 3, the value of the volume fraction range is made smaller than case 1 and case 2, (0-0001) because as was known that in case 3 the probability of the number of particles entering the cyclone is more than case 1 and 2. From the figure it can be seen that the sand particles that reach the cyclone are about $3x10^{-4}$ – $4x10^{-5}$ (300-400 ppm) and centered on the inlet duct region. While on wingwall superheater the distribution of sand volume fraction is almost the same as previous case that is more concentrated at the bottom of the base and the value reaches 0.001 (1000 ppm).

Then in case 4 the value of the fraction of the volume of sand reaching the wingwall superheater and cyclone increased. In the cyclone the maximum volume of sand fraction is in the inlet duct region which reaches a value of $5x10^{-4}$ – $1x10^{-4}$ (500-1000 ppm). Then in the superheater wingwall area the value of the sand volume reaches $1.1x10^{-4}$ (1100 ppm).

From the discussion of the distribution of the particles volume fraction it can be seen that with the increasing fluidizing air velocity, the concentration of particles in the wingwall superheater and cyclone will increase which can also increase the erosion rate. While in the lower furnace erosion that occurs in all cases is not too much different from the indication of the high particles volume fraction in that area in all variations of fludizing air velocity.

4.2. Particle Velocity

Figure 8 shows the vector of particles velocity on the overall cross section of the CFB for all cases. While Figure 9 is a detailed view of vector of particles velocity in some areas shows clearly velocity magnitude and its direction.

From both figures, it can be seen that in the lower furnace all cases show irregular vector direction which can cause erosion in that area. So the refractory installation with a stronger material in the lower furnace is absolutely necessary because the erosion potential in the lower furnace area occurs in all operating patterns. While for the wall tubes area in the upper furnace all cases show the direction of the vector pointing down parallel to the wall tubes. It indicated that the potential for erosion in this area tends to be small. Meanwhile, the bottom of the wingwall superheater is very vulnerable to erosion because it is the first area to collide with sand particles. It can be seen also that the vector of particles velocity entering the cyclone will increase as the velocity of fluidizing air increases. The bull nose area of cyclone also experiences a high velocity of sand with the vector angle to the surface due to changes in the drastic cross-sectional area of the area. For inlet duct area also experience high speed of sand, but because the direction of vector tend to be parallel to surface then the potential of erosion in the area is smaller.

Case 4 is the case with the highest number of particle vector velocities that enter the cyclone region because the amount of sand entering the cyclone in case 4 is also the greatest. Then it is also known that in case 4 is seen a vector of particles velocity of the sand that leads to the top toward the convective zone. This is because the sand particles entering the cyclone can no longer be captured optimally because the amount has exceeded the capture capacity of this cyclone. This is very dangerous to the heat exchanger circuit in the convective zone.

In order to obtain a more complete particle velocity analysis in the cyclone and wingwall superheater areas is shown Figure 10. Figure 10 shows vector of particles velocity vector taken at 28 m, 26 m, and 20 m cyclone elevations, as well as wingwall superheater. From the figure it can be seen that in all cases it is shown that the maximum sand velocity in the cyclone occurs in the inlet duct and also at a height of 20 m (loop seal) due to the reduction of the cross section. However, the direction of the vector in the inlet duct tends to be parallel to the surface of the wall so that its erosion potential tends to be small. In contrast to the target zone it is apparent that in that area the sand vector pounds at a certain angle and high velocity to the surface which may lead to erosion.

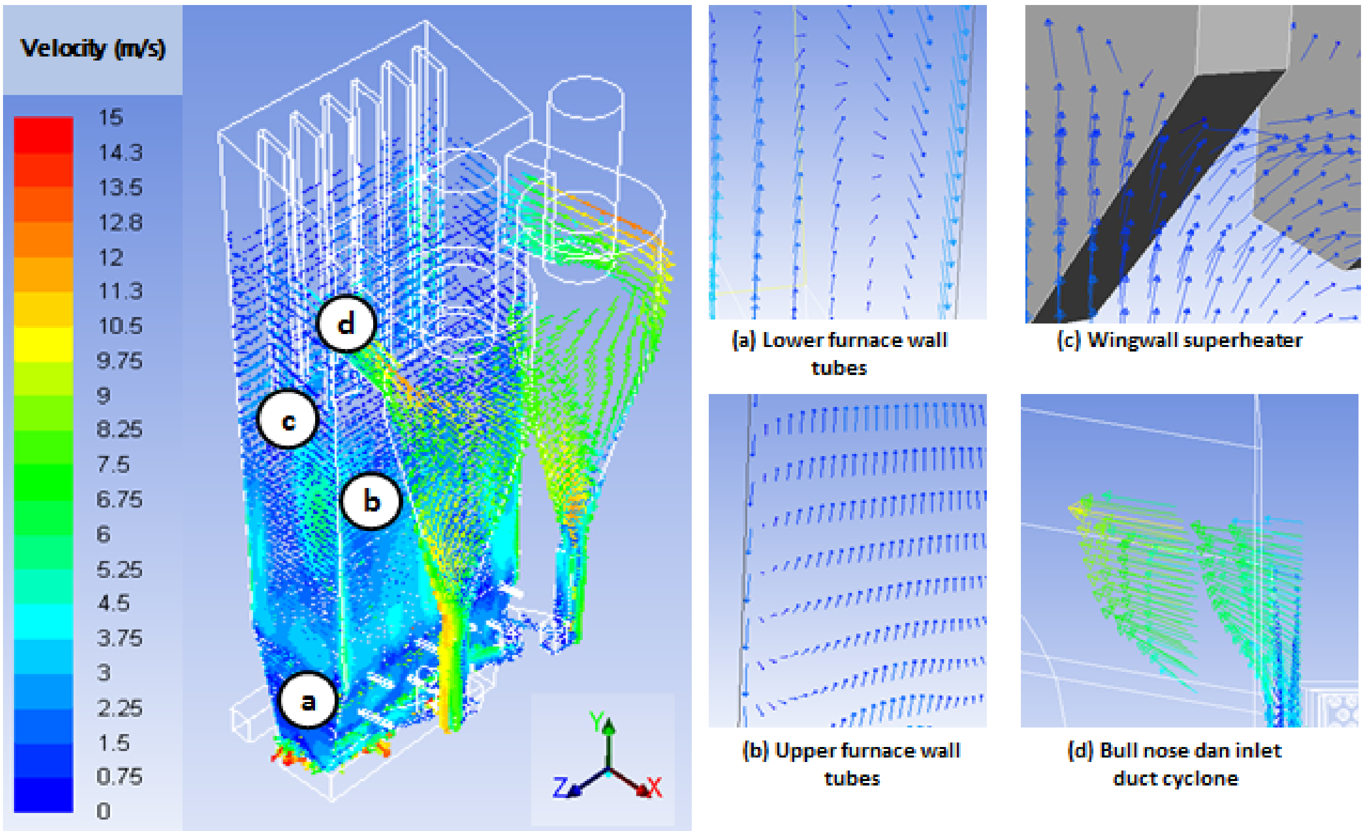

Figure 9. Detailing of Vector of Particles Velocity.

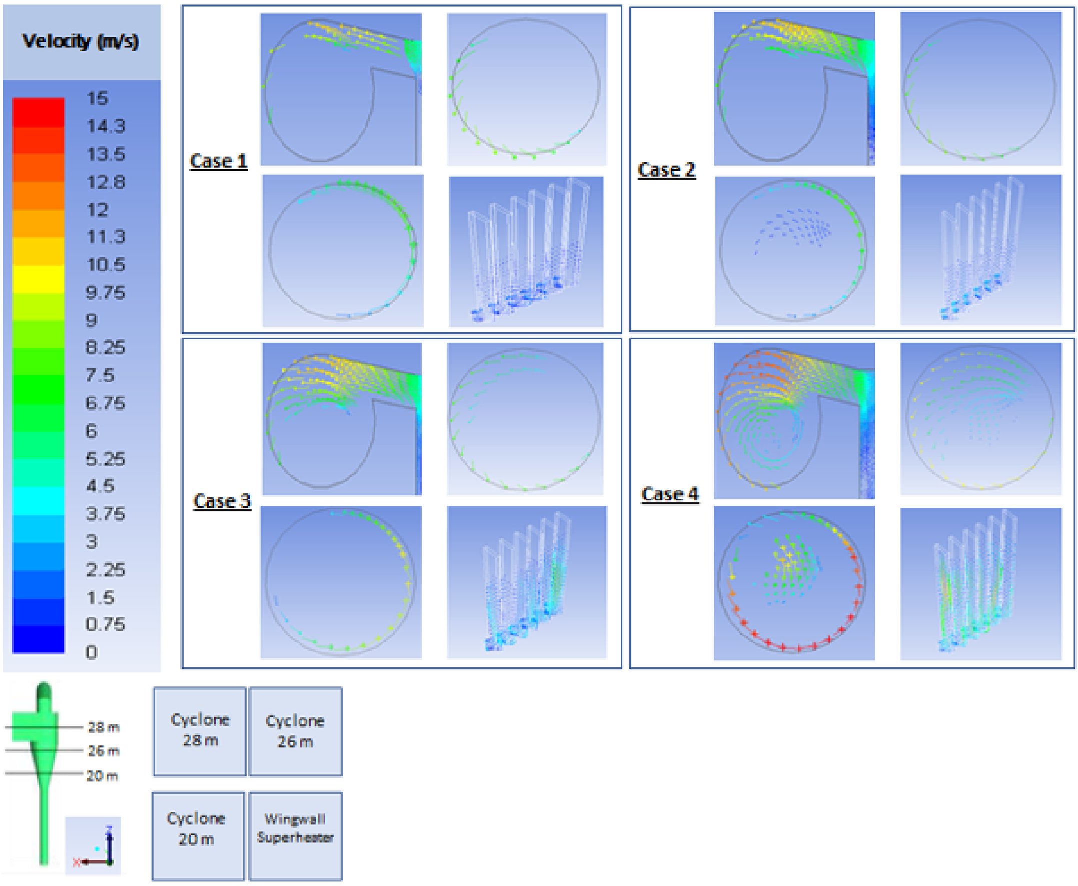

Figure 10. Vector of Particles Velocity in Wingwall Superheater and Cyclone Section.

It is known that the number of vectors and the velocity magnitude value of particles in cyclone and wingwall superheater is increasing as the fluidizing air velocity increases. This indicates that with the higher fluidizing air velocity the erosion rate will increase.

5. Conclusion

Computational Fluid Dynamics simulation provides useful information to improve knowledge about erosion pattern in CFB boiler due to abrasive bed material particles. The simulation in this study is performed at four different fluidizing air velocity. The simulation results are presented in the particles volume fraction and velocity which is one of the most influential parameters to erosion rate. It is observed that the fluidizing air velocity in a CFB boiler has a significant effect on erosion pattern. It is known that increasing fluidizing air velocity will effect to increasing erosion rate indicated from the increasing of particles volume fraction and particles velocity. The most common areas erosion and failure are lower furnace, bottom of the wingwall superheater, cyclone bull nose, cyclone target zone, and loop seal. Installing refractory with material which has better abrasion resistant is required to commit expected life for the boiler. All this result is vital for power plant engineer to improve realibility of power plant especially for CFB boiler equipment.

Acknowledgment

This paper is part of research activities cooperation between ITS with PT. PJB Services (Pembangkitan Jawa Bali Services) and funded by the PT. PJB Services in 2017. On this occasion, the authors would like to thank PT. PJB Services and ITS for the funds and trust given to us. The authors are also grateful to the CFB Boiler Team, Mechanical Engineering Department who has helped complete this work.

References

[1] M. Ozkan, “Simulation of circulating fluidized bed combustors firing indigenous lignite,” Master’s thesis, Dept. Chemical. Eng., Middle East Technical Univ, 2010.

[2] P. Basu, “Hydrodynamics,” in *Circulating Fluidized Bed Boilers*, pp. 17–47, Springer, 2015.

[3] M. B. Gandhi, R. Vuthaluru, H. Vuthaluru, D. French, and K. Shah, “Cfd based prediction of erosion rate in large scale wall-fired boiler,” *Applied Thermal Engineering*, vol. 42, pp. 90–100, 2012.

[4] I. Afrooz, C. Sinnathambi, S. Karuppanan, and D. Ching, “Cfd simulation of bubbling fluidized bed:

Effect of distributor plate orifice pattern configuration on hydrodynamics of gas-solid mixing," *ARPN J. Eng. Appl. Sci*, vol. 11, pp. 11954–11959, 2006.

[5] V. Tanskanen *et al.*, "Cfd study of penetration and mixing of fuel in a circulating fluidized bed furnace," 2005.

[6] A. S. Kinkar, G. Dhote, and R. Chokkar, "Cfd simulation on cfbc boiler," *Int. J. Sci. Technol. Res*, vol. 4, no. 2, pp. 34–39, 2015.

[7] P. Basu, *Combustion and gasification in fluidized beds*. CRC press, 2006.

[8] J. Mbabazi, T. Sheer, and R. Shandu, "A model to predict erosion on mild steel surfaces impacted by boiler fly ash particles," *Wear*, vol. 257, no. 5-6, pp. 612–624, 2004.

[9] B. Sudarmanta *et al.*, "Simulasi numerik sistem injeksi bertingkat pada ruang bakar mesin diesel caterpillar 3406," in *Proceeding Seminar Nasional Tahunan Teknik Mesin XI (SNTTM XI) Thermofluid IV Universitas Gadjah Mada (UGM), Yogyakarta, 16-17 Oktober 2012*, 2015.

[10] R. Kumar and K. Pandey, "Cfd analysis of circulating fluidized bed combustion," *Engineering Science and Technology*, vol. 2, no. 1, pp. 163–174, 2012.

[11] D. Gidaspow, "Hydrodynamics of fiuidizatlon and heat transfer: Supercomputer modeling," *Applied Mechanics Reviews*, vol. 39, no. 1, pp. 1–23, 1986.

[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 *Proc. Seminar Nasional Teknik Mesin XII, Surabaya*, pp. 5–9, 2017.

[13] B. A. Muliawan, B. Sudarmanta, A. Syaifudin, and G. Nugroho, "Studi numerik variasi volume pasir terhadap efisiensi boiler circulation fluidized bed combustion," in *Proc. Seminar Nasional Teknik Mesin XII, Surabaya*, pp. 10–14, 2017.