Computational Simulation of Gas-Solid Flow to Reduce Erosion in a Circulating Fluidized Bed (CFB) Boiler

M K Wisyaldin\textsuperscript{1,4}, P Y Akshinta\textsuperscript{1}, A Barkati\textsuperscript{1}, H Pariaman\textsuperscript{1,2}, and M Hisjam\textsuperscript{3}

\textsuperscript{1} PT. Pembangkitan Jawa Bali, Surabaya, Indonesia
\textsuperscript{2} Institut Teknologi PLN, Jakarta, Indonesia
\textsuperscript{3} Department of Industrial Engineering, Faculty of Engineering, Universitas Sebelas Maret, Surakarta, Indonesia
\textsuperscript{1} Corresponding author. E-mail address: m.kamalw@ptpjb.com

Abstract. Circulating fluidized bed (CFB) is one type of coal combustion technologies that plays an essential role in the power generation. CFB boilers use the principle of the thermal inertia of material deliberately inserted into the furnace (known as bed materials) which is usually sand particles. The existence of materials in the form of particles with high speed can lead to erosion on the wall furnace. The amount of erosion in the furnace wall is influenced by the speed and flow pattern as well as the particle shape of the particle. This paper performed three-dimensional simulations of a CFB boiler using CPFD software to determine the behavior and location of erosion on furnace modelling and simulation. A computational particle fluid dynamics model of a pilot-scale circulating fluidized bed (CFB) was used to simulate its gas-solid flow characteristics numerically. The accuracy of the model was confirmed by a comparison of the simulation results with the operation data. Decreasing the operating parameters according to the calculation of stoichiometry reactions shows a decrease in erosion, thereby making CFB boiler more reliable. Furthermore, the simulation result provide a reference for further comprehension of the flow characteristics in CFBs.

1. Introduction
Circulating fluidized bed (CFB) is one type of coal combustion technologies that plays an essential role in the power generation plant and coal gasification industry [1]. Large thermal capacity and high steam pressure is a tendency for the improvement and development of CFB boilers [2,3,4]. High efficiency of gas-solid separation is a key to achieving high combustion efficiency, reducing limestone consumption and NOx emission to meet the demands for high steam parameters and sizeable thermal capacity [5].

A detailed overview of the fluidization characteristics in the large-scale CFB boiler was provided by using Computational Fluid Dynamics (CFD) [1]. The use of a CFD approach to investigate the three-dimensional gas-solid flow in CFB boilers was proposed for research[6,7,8, 9]. There are two types of theoretical approaches that one can use in computational fluid dynamics (CFD) to describe gas and solid flows: a continuum approach for both phases, which is known as Eulerian–Eulerian or
two-fluid model (TFM) \cite{10,11} and a continuum approach for the fluid and a discrete approach for the particle phase also known as Eulerian-Lagrangian or discrete particle method (DPM).

The Eulerian – Eulerian or TFM model treats solids as a continuous phase that interacts with the gas phase through an exchange of momentum. The TFM model has been widely used in simulating multiphase flows. However, it has limitations, such as not applicable to the distribution of particle size and interparticle forces \cite{12}. The DPM model describes a discrete phase by tracking the many-particle trajectories that exchange mass, momentum, and energy with the gas phase across the simulation field. The DPM model takes into account the particle size distribution and the interactions between particles. However, it is difficult to simulate a solid-gas flow with a solid volume fraction of more than 5% due to a large number of particle numbers. In general, particle numbers are in the order of 2 x 10^5 in the DPM model and are often used for two-dimensional simulation. Recently, an Eulerian-Lagrangian model called CPFD (Computational Particle Fluid Dynamics) was used to model gas-solid flow in a fluidized bed \cite{13,14}.

In the CPFD calculation, numerically calculated particles are not physical particles but are composed of several particles with similar properties (e.g. composition, size, density, temperature). After such treatment, a gas-solid system containing hundreds of millions of particles can be simplified into a system containing only millions of particles. Simulation of the full-size distribution of any solid particle can be done by such a calculation mode \cite{21}. This provides a feasible means to capture the dynamic characteristics of the gas-solid flow within a fluidized bed quickly and accurately. Chen \cite{14} used CPFD to simulate the CFB riser and Wang et al. \cite{22-25} used CPFD to simulate the CFB’s gas-solid flow characteristics with a U-type loop seal. Both studies have produced reasonable results, demonstrating the effectiveness of CPFD in modelling gas-solid flow in a complex system. Therefore, this paper performed three-dimensional simulations of a CFB using CPFD software. This paper has focused on simulating the gas-solid flow characteristics to predict the erosion area that occurs so that it can be optimized for better CFB boiler reliability.

2. Literature Review

2.1. Fluidization and CPFD Model

In the boiler there is flow when a substance enters the boiler. Liquid or gas that flows into the boiler in sufficient quantities and will pass through the bed particles will later form a particle suspension. In the end a flow will be formed. The drag force acting on the particles of the gas flow must be at least in the same range as the gravitational force to obtain fluidization \cite{26}. The lowest speed at which all the particles are suspended by the gas is called the minimum fluidization velocity. If the speed is greater, it will cause different behavior. This fluidized by a combination of the behavior of the two forces on the particle (drag force), \( F_d \), which is caused by the flow of fluidizing air and the force of gravity, \( F_g \). \( F_d \) is linearly dependent on the area of the particle, as explained in equation (1) (Wolfram Alpha). \( F_g \) is linearly dependent on the volume of the particle, as explained by equation (2), according to Newton’s second law of motion.

\[ F_d = \frac{1}{2} C_d \rho u^2 A \]  
\[ F_g = \rho V g \]  

where \( C_d \) is the drag coefficient, \( \rho \) is the particle density, \( u \) is the difference in velocity between the gas (air) and the particle, \( A \) is the particle area, \( V \) is the particle volume and \( g \) is the gravitational constant. As the particle size increases, the force of gravity increases more than the drag force increases. Therefore larger particles tend to survive while smaller particles are more likely to move along with the gas flow. For CFB, this results in an average particle size at the top of the riser being smaller than the average particle size at the bottom. The difference in particle size along the riser heights can be measured by the Johnsson method of the CFB riser wall layer \cite{4}.
The Wen-Yu create a model for gas-solid flow with solid volume fraction lower than 0.61. The Ergun create model with range 0.47-0.7. The interphase drag function is defined by Wen-Yu model [29].

\[ D_p = C_d \frac{3 \rho_g |v_g - v_p|}{6 \rho_p (\frac{V_p}{4\pi})^{1/3}} \]  

where \( C_d \) is the drag coefficient. It depends on the Reynolds number, with each of the conditions shown in equation (4).

\[ C_d = \begin{cases} \frac{24}{Re} (1 + 0.15Re^{0.687})\theta_g^{-2.65} & \text{for } Re < 1000 \\ 0.44\theta_g^{-2.65} & \text{for } Re \geq 1000 \end{cases} \]  

The Reynolds number is given by

\[ Re = \frac{2\rho_g |v_g - v_p| (\frac{3V_p}{4\pi})^{1/3}}{\mu_g} \]  

where \( \rho_g \) is the gas density, \( \mu_g \) is the gas viscosity, \( v_g \) and \( v_p \) represent the gas and particle velocity respectively, and \( V_p \) is the volume of a particle.

CPFD calculates particle forces as a gradient on the grid and maps it back to particles. The present study adopts the particle normal stress model by [30].

\[ \tau_p = \frac{P \rho_p \theta_p^2}{\max[\theta_{cp} - \theta_p(1-\theta_p)]} \]  

where \( \epsilon \) is a constant suggested to be on the order of \( 10^{-7} \), \( \theta_{cp} \) is the solids volume fraction at close packing. \( P \), has pressure units and \( \gamma \) is supposed to be 2 \( \leq \gamma \leq 5 \) [31].

### 2.2. Computational Particle Fluid Dynamics (CPFD) Software

Barracuda software was founded in 2003 by Dean Drako. Barracuda is a computational particle fluid dynamics (CPFD) software for the analysis and optimization of fluidized bed [15,16,17] reactors and other particulate fluid systems [18]. The CPFD method is essentially a numerical calculation based on the multiphase particle-in-cell (MP-PIC) method, which can effectively solve the problem of coupling the fluid and a large number of particles in three-dimensional space. The MP-PIC method was first proposed by Andrews [19] and Snider [20]. The most distinctive feature of this method is its ability to solve the coupled three-dimensional particle and fluid momentum equations. The fluid momentum equation is the Navier–Stokes equation; the Eulerian method handles this phase. On the other hand, the particle phase is coupled with the fluid phase equation and is handled by the Lagrangian method.

### 3. Material and Method

#### 3.1. Computational Method and Model

The study was carried out by simulating the flow patterns of the particles in the CFB using CPFD software. The 3D CFB model for simulation is based on the geometry and size of the CFB available in technical drawings with 3D CAD software, and then the 3D models are exported in *.stl format. Further adding meshing with the total grid cell that gives a chance for 550000 meshing results can be seen in Figure 1. The parameters used in the simulations are based on data drawn from the operational condition of CFB legible on the Distributed Control System (DCS) in the control room and boundary conditions of the stoichiometric reactions calculation results. Combustion processes that involve chemical reactions are ignored and the combustion chamber temperature is considered isothermal.
3D models of CFB and locations for boundary conditions are illustrated in Figure 1. The simulated CFB has a height of 34 m, a length of 12 m, a top width of 6.5, and a bottom width of 4.8 m. The cyclone diameter of 5.5 m and return leg diameter of 1 m.

3.2. Simulation Setup and Parameters
The parameters used for the simulation boundary conditions are shown in Table 1 and Table 2. The simulation is carried out by assuming isothermal conditions, where the temperature value is considered constant at 1070K (797 °C). Gas viscosity and density also affect cross-phase drag values, impacting the velocity profile and particle distribution in simulations. The effect of changes in temperature on the viscosity value is shown in Figure 3. Air is assumed to have a composition that is N2 = 78.084%, O2 = 20.9476%, Ar = 0.9365%, CO2 = 0.0319 and has a molecular mass of 28.9652 g/mol.

![Figure 1. 3D Model of CFB and boundary condition](image1)

The sand particles used in the simulation have a density of 2200 kg/m3 and a molecular weight of 60.0843 g/mol. The particle size distribution (PSD) of the sand is represented by the graph in Figure 3. This sand initially occupies a small part of the space at the bottom of the furnace and a small portion in the loop seal and is in a close packing condition (the maximum volume fraction the particle can reach) at 0.6. The term static bed height is often used to determine the value of the height of a particle entered when starting from operational in CFB. In this case, the static bed height is set at 860 mm, and the particles in the loop seal are 300 mm in height.

![Figure 2. Particle Size Distribution (PSD) on model simulation](image2)

| No | Parameter Input | Mass flow (Nm3/h) | Pressure (Pa) |
|----|-----------------|-------------------|--------------|
| 1  | Actual based on operation actual data 93 MW |  |  |
Table 2. Input parameters optimization based on calculation

| No | Parameter Input                  | Mass flow (Nm3/h) | Pressure (Pa) |
|----|----------------------------------|-------------------|---------------|
| 1  | Mass Flow PA Fan                 | 186394.77         | 14500         |
| 2  | Mass Flow Sealpot A (right)      | 1417.31           | 39000         |
| 3  | Mass Flow Sealpot A (left)       | 934.99            | 39000         |
| 4  | Mass Flow Sealpot B (right)      | 881.69            | 37700         |
| 5  | Mass Flow Sealpot B (left)       | 747.66            | 38190         |
| 6  | Mass Flow SA Front Upper         | 23100             | 12000         |
| 7  | Mass Flow SA Front Bottom        | 122297            | 12000         |
| 8  | Mass Flow SA Rear Upper          | 70811             | 13000         |
| 9  | Mass Flow SA Rear Bottom         | 57527             | 13000         |
| 10 | Outlet Cyclone A                 |                   | -2000         |
| 11 | Outlet Cyclone B                 |                   | -1790         |

4. Result and Discussion

4.1. Gas-Solid Flow Pattern Distributions

The pattern of solid (sand) particle flow is shown in Figure 5(a). From the pattern of particle distribution, it appears that the particles dominate in the refractory region. This shows that the particle is back down and not circulated through the cyclone. Solids generally move upward in a dispersed phase through upright beds, where the slip velocity will be the order of the speed of individual particles. Near the wall, the gas velocity is much lower, and may even drop in some cases. When solid particles drop along the wall, they don’t always find very high slip speeds [32]. Direct measurements can show the speed of local gasses and particles with slip velocity higher towards the wall. The magnitude of the local slip velocity is higher than the particle velocity.
Most of the combustion heat produced in the CFB furnace must be transferred to the water wall coverings or panels maintained in the furnace to maintain the average furnace temperature in the range of 800-900°C. Thus, knowledge of heat transfer in the CFB furnace is fundamental. The effectiveness of heat transfer through particle convection depends on the homogeneity of the particle, the more homogeneous the more effective the heat transfer. The transfer of gas heat to the particles also regulates the transient response of the CFB furnace to operational changes. The heat transfer through the water pipe by particle convection is only effective at about 5% because the water pipe is placed above the refractory at the height of 12 m.

Simulations performed by changing the working parameters of CFB according to the calculation of coal stoichiometric reactions (ideal conditions) show different particle flow patterns. The number of particles rising to the top decreases significantly (Figure 4a). With this condition, erosion occurs only around the refractory wall (Figure 6), there is no erosion on the cyclone. This can also be seen from the pattern of air velocity in the decreasing cyclone area (Figure 4b). This condition is good, but the effectiveness of heat transfer through particle convection can decrease.

4.2. Erosion Intensity and Location

The upper furnace area is often designed based on superficial fluidization or furnace speed. The fluidization rate, calculated by dividing the total flue gas volume at the furnace temperature by the top furnace section, is typically around 5 m/s. To avoid erosion, it generally does not exceed 6 m/s [32].
Erosion caused by the flow of solids can be seen in Figure 5. From the analysis, it can be seen that the greatest erosion occurs in the refractory area and inlet cyclone. This is due to particles moving and hitting walls at an angle where erosion is most likely to occur and at a relatively high speed. The speed of the particles can reach up to 30 m/s. The magnitude of these particles speed is in line with the air velocity, which is also high.

Figure 5. Erosion intensity on CFB with operation actual conditions

Figure 6. Erosion intensity on CFB with ideal conditions

5. Conclusion
In this paper, CPFD was used to predict the flow characteristics in a practical CFB Boiler system considering the fluidizing air velocity in a large-scale gas-solid system. The pattern of particle distribution that dominates in the refractory region indicates that most of the particles descend into the bed and are not circulated through the cyclone. Erosion occurs a lot in several places, including refractory walls, return legs, and some circulation areas. Decreasing the parameters of working conditions according to the calculation of coal stoichiometric reactions shows a decrease in erosion, but can reduce the effectiveness of thermal transfer in the furnace. This simulation only shows the
location of erosion. It does not calculate the amount of erosion; therefore, it is necessary to analyze further the erosion amount occurring.

Acknowledgments
The authors gratefully acknowledge the support provided by colleagues of PT PJB UBJOM Kaltim Teluk.

References
[1] Reh L 2003 Development potentials and research needs in circulating fluidized bed combustion. Particulology 1, p 185–200.
[2] Lv J F, Yu L, Zhang Y J, Yue G X, Li Z Y, Wu Y X 2007 A 600 MW super-high circulating fluidized bed boiler. J. Power Eng. 27, 497–501.
[3] Fan Z, Goidich S, Robertson A 2008 800 MWe circulating fluidized bed boiler with 1300 JC superhigh steam. In: Proceedings of the 33rd International Technical Conference on Coal Utilization and Fuel Systems June 1–5, 2008.
[4] Chen Y M, Zhou X L, Zheng C H, Wang L H, Fang M X, Shi Z L, Luo Z Y, Cen K. F 2008 Development of large-scale circulating fluidized bed boiler. J. Power Eng. 28, 817–826.
[5] Koornneef J, Junginger M, Faaij A, 2007 Development of fluidized bed combustion— an overview of trends, performance, and cost. Prog. Energy Combust. Sci. 33, 19–55.
[6] Zhang N, Lu B N, Wang W, Li J H, 2008 Virtual experimentation through 3D full-loop simulation of a circulating fluidized bed. Particulography 6, 529–539.
[7] Zhang N B L, Wang W, Li J H, 2010 3D CFD simulation of hydrodynamics of a 150 Mwe circulating fluidized bed boiler. Chem. Eng. J. 162, 821–828.
[8] Ahuja G N, Patwardhan A W, 2008 CFD and experimental studies of solids hold- up distribution and circulation patterns in gas-solid fluidized beds. Chem. Eng. J. 143, 147–160.
[9] Hartge E U, Ratschow L, Wischniewski R, Werther J, 2009 CFD-simulation of a circulating fluidized bed riser. Particulology 7, 283–296.
[10] Gidaspow D, Ettedadieh B, 1983 Fluidization in two-dimensional beds with a jet: Part II. Hydrodynamic Modeling Ind. Eng. Chem. Fund. 22, 193–201.
[11] Lun C K K, Savage S B, Jeffrey D J, Chepurniy N, 1984 Kinetic theories for granular flow: inelastic particles in coquette flow and slightly inelastic particles in a general flow field. J. Fluid Mech. 140, 223–256.
[12] Makkawi Y T, Wright P C, Ocone R, 2006 The effect of friction and inter-particle cohesive forces on the hydrodynamics of gas-solid flow: a comparative analysis of theoretical predictions and experiments. Powder Technol. 163, 69–79.
[13] Abbasi A, Ege P E G, de Lasas H I, 2011 CPFD simulation of a fast fluidized bed steam coal gasifier feeding section. Chem. Eng. J. 174, 341–350.
[14] Chen C, Werther J, Heinrich S, Qi H Y, Hartge E U, 2013 CPFD simulation of circulating fluidized bed risers. Powder Technol. 235, 238–247.
[15] S Karimipour, T Pugsley 2012 Application of the particle in cell approach for the simulation of bubbling fluidized beds of Geldart A particles, Powder Technol. 220, 63–69.
[16] Y Liang, Y Zhang, T Li, C Lu 2014 A critical validation study on the CPFD model in simulating gas-solid bubbling fluidized beds, Powder Technol. 263, 121–134.
[17] V Vivacqua, S Vashisth, G Hébrard, J R Grace, N Epstein 2012 Characterization of fluidized bed layer inversion in a 191-mm-diameter column using both experimental and CPFD approaches, Chem. Eng. Sci. 80, 419–428.
[18] L D Zhang, Z J Wang, Q Wang, H Qin, X D Xu 2016 Simulation of oil shale semi-coke particle cold transportation in a spouted bed using CPFD method. Powder Technology 301, 360–368.

[19] Andrews M J, O'Rourke P J 1996 The multiphase particle-in-cell (MP-PIC) method for dense particulate flows. Int. J. Multiph. Flow 22, 379–402.

[20] Snider D M 2001 An incompressible three-dimensional multiphase particle-in-cell method for dense particle flows. J. Comput. Phys. 170, 523–549.

[21] J Parker, K. LaMarche, W Chen 2013 CFD simulations for the prediction of scaling effects in pharmaceutical fluidized bed processors at three scales. Powder Technology. 235, 115-120.

[22] Q G Wang, H R Yang, P N Wang, J F Lu 2014 Application of CPFD method in the simulation of a circulating fluidized bed with a valve Part I—Investigation of solids circulation. Powder Technology. 253, 814–821.

[23] Q G Wang, H R Yang, P N Wang, J F Lu 2014 Application of CPFD method in the simulation of a circulating fluidized bed with a loop seal Part II—Investigation of solids circulation. Powder Technology. 253, 822–828.

[24] Q G Wang, T Niemi, J Peltola, S Kallio, H R Yang 2015 Particle size distribution in CPFD modeling of gas-solid flows in a CFB riser. Particuology. 4, 107–117.

[25] Q Wang, J B Xiao, H P Liu 2013 CFD simulation of hydrodynamics on the dense zone on a 65 t/h oil shale-fired high–low bed CFB boiler. Powder Technology. 249, 339–352.

[26] P Basu, C Kefa, L Jestin 2000 Boiler and burners, Design and theory. New York.

[27] A Kullendorff, B Leckner 1991 Circulating fluidized bed boilers, Report A 91-194, Chalmers University of Technology, Department of Energy Conversion, Gothenburg, Sweden, pp. 37.

[28] F Johnsson, C Breitholtz, B Leckner 1998 Solids segregation in a CFB-boiler furnace. In Proceedings of Fluidization IX, (Eds. Fan, L-S., Knowlton, T.) Engineering Foundation. New York, pp. 757-764.

[29] Wen C Y, Yu Y H 1966 Mechanics of fluidization. Chem. Eng. Prog. Symp. Ser. 62, 100–111.

[30] Harris S E, Crighton D G 1994 Solitons, solitary waves, and voidage disturbances in gas-fluidized beds. J. Fluid Mech. 266, p 243–276.

[31] Auzerais F M, Jackson R, Russel W B 1988 The resolution of shocks and the effects of compressible sediments in transient settling. J. Fluid Mech. 195, 437–462.

[32] P Basu 2015 Circulating fluidized bed boilers. Springer International.