Simulation on Air- Fly Ash Particles Using Discrete Phase Method

N.M. Zahari\textsuperscript{1*}, Mastika Falsha Harun\textsuperscript{1}, Mahyun Zainoodin\textsuperscript{1}, Daud Mohamad \textsuperscript{1}, Agusril Syamsir\textsuperscript{1}, M. Z. Ramli\textsuperscript{1}, M. H. Zawawi\textsuperscript{1}, and Aizat Abas\textsuperscript{2}

\textsuperscript{1}Department of Civil Engineering, Universiti Tenaga Nasional (UNITEN), Kajang 43000, Selangor Darul Ehsan, Malaysia
\textsuperscript{2}School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Pulau Pinang, Malaysia

\*Email: mubinzahari@gmail.com

Abstract. In order to suggest for development of a better fluidized bed combustor designs, numerical simulations could be used to model the flow behavior inside the boilers. However, the complex nature of the multiphase flow of particles and combustion air makes the modelling very challenging. In this study the relatively recently introduced approaches for fly ash particles distribution model in CFD setting are studied. The applicability of the approaches is examined, and the computational requirements of the approaches are compared. Euler-Lagrange approach is used in order to track the particles in the simulation. The simulations results obtained are compared between different shape of simple models, square and cylindrical.

1. Introduction
Coal is widely available and less expensive compared to oil-based fuels, thus coal is broadly used in coal combustion by-products in power industry. Coal combustion also has become one of the prominent technologies in environment-friendly biomass combustion. In pulverized coal fired boiler the solid fuel particles are suspended on upward-blowing air resulting in a turbulent mixing of gas and solids. Rapid cooling in the post-combustion zone results in the formation of spherical, non-crystalline particles. The particles then may be collected in the flue gas particle equipment, or may be escaped and deposited to the environment, or also can be deposited on the furnace wall and gradually become sintered and harden. Therefore, manual cleaning and removal of the deposition need to be made and costly [1]. The fly ash particles in the flue gas stream experience forces from fluid drag and lift, gravity, and intermittent collisions with the duct walls and other particles. The higher density of fly ash particles approximately around 2500 kg/m\textsuperscript{3} while the flue gas approximately around 1 kg/m\textsuperscript{3}. These differences densities cause the fly ash particles to diverge from the flue gas flow and separate from the flue gas at select locations depending on local flow conditions and, particle size and density [2].

Particles impacting an existing packed bed may transferred their kinetic energy to the fluid medium and be assimilated in the bed after. The fly ash particles typically have a range of sizes and densities and, a higher concentration of the heavier particles will almost always be seen near the bottom horizontal surfaces of the duct [3].

Recently, many studies have been done upon the fly ash utilization but still less in fly ash behavior in the combustion vessel in term of their characteristics. A study carried by Kaer et al developed a
dynamic model to predict the ash deposition formation and investigated the ash deposition rate by different deposition mechanism [4]. García Pérez et al. modeled the deposit growth of fume particles based on the thermophoretic force, Brownian motion and inertial impaction [5]. However, the details on particle impaction and the movement patterns are not highlighted in the previous study.

Before that, there are many researchers using ANSYS-Fluent software to simulate few kinds of case study [6-9]. Thus, this paper will study on the fly ash transport by introducing particulates into the scale model in simulations by using discrete phase method in ANSYS-Fluent software. Two difference shape of scale model will be generated in SolidWorks and exported to the ANSYS for simulations. The simulations can give out varies result based on setting chosen in the processor such as the pressure, velocity, kinetic energy and others. Conclusion of the minimum flue gas velocity required for ensuring no drop out of transported particles and the minimum gas velocity required to re-entrain existing deposits of fly ash are made. Moreover, particle parameters and model scaling will need to be appropriate to ensure similar particle behavior.

2. Methodology

2.1. Discrete Phase Model

Simulation techniques commonly used in engineering analysis which account for coupling between the carrier gas phase and particulates, can be classified as Lagrangian dispersed phase simulation or Eulerian multiphase simulation. The Eulerian approach is currently commonly used in fluidized bed simulations and it is a relatively fast and usable approach. In the Eulerian-Lagrangian approach on the other hand, the particles are treated as separate, solid objects and their behavior is modeled using Newtonian equations of motion. Depending on the implementation, the Eulerian-Lagrangian methods can give very accurate and detailed results in wide range of different settings. However, the computational costs of these approaches are often very demanding especially in case of dense suspensions and their application is mainly limited to smaller scale computations and research use. In these approaches the fluid phase is still modeled with Navier-Stokes equations, but the particle phase is modeled using a large amount of individual particles obeying the Newtonian equations of motion [10].

The relative ability of the particles to follow the changes in the flue gas flow field can be determined by the particle Stokes number, \( S_{tp} \) given Equation 1.

\[
S_{tp} = \frac{t_p}{t_f}
\]  

\( t_p \) = the ratio of the particle relaxation time scale  
\( t \) = the applicable fluid dynamic time scale, expressed as Equation 2

\[
t_p = \frac{\rho_p d^2}{18 \mu_f}
\]  

\( \mu_f \) = dynamic viscosity of the carrier fluid.

An ANSYS flow process is simplified in Figure 1,
2.2. Geometry
There are two different models in this study which are square and cylindrical in shape to be used in all simulations. Besides that, the input parameter for the simulations is the same in all simulations except for the inlet velocity. The models are displayed in Figure 2:

Figure 1. Simplified flow diagram.

Figure 2. Front view of square (left side) and cylindrical (right side) shape of simple model for simulations
2.3. Meshing
Meshing generated for both models are shown as below:

Figure 3. Meshing of square (left side) and cylindrical (right side) shape of simple model for simulations.

2.4. ANSYS-Fluent solver settings
The Ansys-Fluent solver settings are summarized in table below:

Table 1. Formatting sections, subsections and sub subsections.

| Phase Properties |  |
|------------------|--|
| **Phase 1 (air)** |  |
| Density, kg/m³   | 1.225       |
| Viscosity, kg/m³ | 1.7894 x 10⁵ |
| **Phase 2**      |  |
| Diameter, m      | 0.001       |
| Density, kg/m³   | 1000        |
| Boundary Conditions | Inlet |
|---------------------|------|
| Turbulent intensity, % | 5 |
| Hydraulic diameter, m | 0.006 |
| Phase 1 velocity, m/s | Case 1: 1 m/s, Case 2: 10 m/s |
| Phase 2 velocity, m/s | - |

| Outlet |
|--------|
| Turbulent intensity, % | 4.6 |
| Hydraulic diameter, m | 0.006 |
| Gauge pressure, Pa | 0 |

All simulations are run as a transient with a small step of 0.01 m/s and the initialization is done. Simulations are run until the initial transient had vanished and it reached a pseudo steady-state. Calculation of the time average will take place. The time averaging is mostly straightforward. During calculating time averages, to obtain more representative results it is assumed that a longer time is needed. Since the computations are very time consuming, the sampling time has to be limited.

3. Results and discussion

3.1. Particle trajectories between the models on different inlet velocity 1m/s and 10m/s

![Figure 4. Particles impact at 1 m/s inlet velocity: (a) square model and (c) cylindrical model, Particle impact at 10 m/s: (b) square model and (d) cylindrical model.](image)

The Particle trajectories that are formed show in Figure 4 to make better visualization for both models. It can find the differences in Particle trajectories to make comparison in different models (Square and Cylinder) and different velocity (1 m/s and 10 m/s).
3.2. Comparison of Pressure, Turbulence kinetic energy, and Turbulence eddy dissipation for both models.

![Graphs showing comparison of Pressure, Turbulence kinetic energy, and Turbulence eddy dissipation for both models.](image)

**Figure 5.** Comparison of Pressure, Turbulence kinetic energy, and Turbulence eddy dissipation for both models: (a) at inlet boundary, and (b) at outlet boundary.

Simulations results that are obtained graphs are formed to make a better evaluation for both models. The graph will be compared between the square and cylindrical models at a velocity of 1 m/s and 10 m/s. The maximum value of each result used to plot the graphs. The graph shows in Figure 5(a) the pressure of the square model at input 1 m/s velocity (2.66 Pa) is slightly higher (2.35 Pa) than the pressure in the cylindrical model. Inlet 10 m/s also shows that the pressure in the square (128 Pa) is slightly higher (127 Pa) than the cylindrical model. For the turbulence kinetic energy in the square model is (0.17 m²s⁻²) higher than the cylindrical model (0.1 m²s⁻²) for velocity input 1 m/s. Meanwhile, at input velocity 10 m/s the turbulence kinetic energy in the square model shows large different values compare to the cylindrical model. The difference value between the models at 10 m/s is 1.8 Pa while at 1m/s the pressure difference is 1 Pa. Referring to the turbulence eddy dissipation in the cylindrical model is greater than the square model at two inlet values, 1 m/s and 10 m/s. The turbulence eddy dissipation value difference between models shows a big difference which is (0.24 m²s⁻³) at input velocity 1 m/s, and (6.3 m²s⁻³) at input velocity 10 m/s.

This graph illustrates in Figure 5(b) the pressure of the square model at input 1 m/s velocity is 0.013 Pa less than the pressure in the cylindrical model (0.06). However, at inlet 10 m/s the pressure reading in the square is 2.7 Pa higher than the cylinder model which is 2.1 Pa. The graph shows that the turbulence kinetic energy in the square model is 0.2 m²s⁻² higher than the cylindrical model (0.16 m²s⁻²) for velocity input 1 m/s. Meanwhile, at input velocity 10 m/s the turbulence kinetic energy in the square model shows 4.7 m²s⁻² different values compared to the cylindrical model is only 4 m²s⁻². A bar graph shows that the turbulence eddy dissipation in the cylindrical model is greater than the square model at two inlet values, 1 m/s and 10 m/s. The turbulence eddy dissipation value difference between models shows big difference which is around (2.7 m²s⁻³) at input velocity 1 m/s, and (1.6 m²s⁻³) at input velocity 10 m/s.
Results obtained showed in Figure 5 are comparison of pressure, turbulence kinetic energy, and Turbulence Eddy Dissipation. The cases were for both models (Cylinder and Square) at inlet boundary and at outlet boundary also compare with different velocity (1 m/s and 10 m/s). Clearly, it shows that between both models in the pressure and turbulence kinetic energy square model got high value to compare with the cylinder model either in the inlet boundary or in outlet boundary. It also compares with different velocity, where the value is higher in 10 m/s compared within 1 m/s. For turbulence eddy dissipation it shows that the cylinder model is higher than the square model and if comparing in different velocity, inlet boundary is higher in 10m/s and at the outlet, velocity is higher in 1m/s.

4. Conclusions
In the study presented in this paper single phase CFD flow been used for the analysis. All the phase has been tracking by Lagrange approach of the particulate optionally used for visualization purposes. The flow in these models depends strongly on particle–particle interaction and gas–particle interaction. For this reason, proper closure relations for these two interactions are vital for reliable predictions on the models. Gas–particle interaction can be studied with the use of the lattice Boltzmann model, while the particle–particle interaction can appropriately be studied with a discrete particle model. In this work it is shown that the discrete particle model has the capability to generate insight and finally closure relations in several processes. Finally, examples were shown demonstrating the capabilities of the discrete particle model in ANSYS-Fluent for the prediction of fly ash patterns in different models.

Acknowledgement
This research was supported by the internal grant under Universiti Tenaga Nasional (UNITEN). The authors acknowledge Civil Engineering Department, College of Engineering (UNITEN) for the facilities. Special thanks to those who contributed to this project directly or indirectly.

References

[1] M. U. Garba, D. B. Ingham, L. Ma, M. U. Degereji, M. Pourkashanian, and A. Williams, “Modelling of deposit formation and sintering for the co-combustion of coal with biomass,” Fuel, vol. 113, pp. 863–872, 2013.

[2] F. Goodarzi, “Characteristics and composition of fly ash from Canadian coal-fired power plants,” vol. 85, pp. 1418–1427, 2006.

[3] S. Thipse, M. Schoenitz, and E. Dreizin, “Morphology and composition of the fly ash particles produced in incineration of municipal solid waste,” Fuel Process. Technol., vol. 75, pp. 173–184, 2002.

[4] S. K. Kær, L. A. Rosendahl, and L. L. Baxter, “Towards a CFD-based mechanistic deposit formation model for straw-fired boilers,” Fuel, vol. 85, no. 5–6, pp. 833–848, 2006.

[5] M. García Pérez, E. Vakkilainen, and T. Hyppänen, “2D dynamic mesh model for deposit shape prediction in boiler banks of recovery boilers with different tube spacing arrangements,” Fuel, vol. 158, pp. 139–151, 2015.

[6] Zahari, N. M. et al. 2018. “Introduction of Discrete Phase Model (DPM) in Fluid Flow: A Review.” AIP Conference Proceedings, Vol. 2030, No. 1, p. 020234, 2018.

[7] Zawawi, M. H., A. Saleha, et al. 2018. “A Review: Fundamentals of Computational Fluid Dynamics (CFD).” AIP Conference Proceedings, Vol. 2030, No. 1, p. 020252, 2018.

[8] Zawawi, M. H., W. N. Yusairah, et al. 2018. “Computational Fluid Dynamic Analysis for Solar Powered Water Treatment Device.” AIP Conference Proceedings, Vol. 2030, No. 1, p. 020256, 2018.

[9] Zawawi, M. H., N. H. Hassan, et al. 2018. “Fluid-Structure Interactions Study on Hydraulic Structures: A Review.” AIP Conference Proceedings, Vol. 2030, No. 1, p. 020244, 2018.

[10] T. Niemi, “Particle Size Distribution in CFD Simulation of Gas-Particle Flows,” Diss, p. 79, 2012.