Simulating the heat transfer behaviour in a fluidized bed by a combined thermal model in the CFD-DEM coupling

Sherko Ahmad Flamarz
Civil Engineering Department, University of Garmian, Kalar, Sulaymaniya
Kurdistan Government Region, Iraq
sherko.flamarz@garmian.edu.krd

ABSTRACT

The exchange of mass and heat transport in fluidized beds is a complicated and extensive topic, making it a rich field for scientific and industrial applications. This numerical research implements the CFD-DEM coupling to investigate the transport of heat energy in a fluidized bed. A numerical test compares the numerical and analytical results for a single particle in a column. Two cases of heating and cooling are compared. The numerical results present well adequacy with the analytical results. The development of fluidized bed units, within fluidized bed units is vital to improving the designs of these units. Therefore, three methods of column heating, including base heating, walls heating, and as well as walls and base heating, have been proposed in the simulations. The results reveal that the fluid inlet velocity increase during the fluidized beds' heating process is invalid. Thus, the fluid's proper inlet velocity and imposed temperature are crucial in the heating process for the fluidized beds since the particles need to provide time for the heat transport process.

KEYWORDS: Fluidized bed, simulation, two phase flow, heat transfer, CFD-DEM

1 INTRODUCTION

Heat transfer in fluidized beds is a complicated and extensive topic, making it a rich field for scientific and industrial applications. The heat transfer investigations in fluidized beds generally focused on three aspects: particle to fluid, fluid to the particle, and wall (cooling or heating) to particles and fluid heat transfer [1-3]. Understanding the mechanisms and methods of exchange of heat energy and mass transport in fluidized bed systems is essential in many applications in the industrial process. Typically, in the systems of agroindustry, food manufacturing processes, powders, combustion, metallurgy, drying, etc. [4, 5]. In the past and
before the development of computers and soft-wares, experimental investigations were the only way to study packed and fluidized beds. Despite the advantages of these experiments, they could not predict and explain the complexity of the fluidization process. The development in computer soft and hard wares are assisted to use the computers in complex calculations tasks in modeling and simulations.

The coupling of CFD-DEM can distinguish the heat exchange between the solid phase particles and the fluid and solid phases. The first model of coupling CFD-DEM was developed by Tsuji et al. [6] to study the behavior of gas fluidized bed. Many studies are proposed the CFD-DEM coupling accompanied by heat transfer [2, 7-9]. Borkink and Westerterp [10] proposed a mechanism to describe heat transfer with a bed of particles. Malone and Xu. [11] are investigating the behavior of heat transfer in fluidized beds on the particle-scale flow level by proposing two convection models in liquid-solid fluidized beds. Stefan et al. [12] compared the numerical results of coupling CFD-DEM with empirical data for a fluidized bed heated by hot air. Chunhua Wang et al. [13] studied the spouted bed by CFD-DEM coupling in the particle cooling process. Other authors [14-20] proposed models of coupling CFD-DEM to investigate heat transport in gas-fluidized beds, pneumatic conveying units, etc.

The novelty of this work is proposing three methods for heating the column of fluidized beds and comparing the three methods to develop, improve, and economize energy consumption in fluidized bed reactors. In this work, a thermal model of heat transfer is integrated into the CFD-DEM coupling to investigate the heat transfer behaviors in a fluidized bed. Subsequently, the numerical implementation is validated by comparing the numerical results with the analytic results.

2. METHODS AND MATERIALS

2.1 FLUID PHASE: COMPUTATIONAL FLUID DYNAMICS (CFD)

The CFD code is based on the Reynolds-Averaged Navier-Stokes (RANS) equations to solve the incompressible flow (continuity and momentum equations) [21];

- Continuity equation:

\[ \frac{\partial \rho_f}{\partial t} + \nabla \cdot (\rho_f \vec{v}) = 0 \]  \hspace{1cm} (1)

- Momentum equation of the quantity of motion:
\[
\frac{\partial \rho_f \vec{v}}{\partial t} + \vec{\nabla} \cdot (\vec{v} \times \vec{\omega}) = -\vec{\nabla}p + \vec{\nabla} \cdot \tau + \rho_f \vec{f}
\]  \tag{2}

Where \( t \) is time, \( \rho_f \), and \( p \) are density, velocity, and pressure of the fluid respectively, \( \tau \) is the viscous stress tensor and \( \vec{f} \) refers to other body forces.

A hypothesis is needed to close Reynolds-Averaged Navier-Stokes (RANS) equations systems. The Boussinesq hypothesis a standard method employs to relate the Reynolds stresses to the mean velocity gradients [22]. The present research uses the standard \( k-\varepsilon \) model, largely used by practical engineering turbulence flow problems and calculations. The standard \( k-\varepsilon \) model is based on model transport equations for the turbulence kinetic energy (\( k \)) and its dissipation rate (\( \varepsilon \)) [23]; the equations are written as:

\[
\rho_f \frac{\partial k}{\partial t} + \vec{\nabla} \cdot \left( \rho_f \vec{v} k - (\mu + \frac{\mu_t}{\sigma_k}) \text{grad} k \right) = P + G - \rho_f \varepsilon
\]  \tag{3}

\[
\frac{\partial \varepsilon}{\partial t} + \vec{\nabla} \cdot \left( \rho_f \vec{v} \varepsilon - (\mu + \frac{\mu_t}{\sigma_\varepsilon}) \text{grad} \varepsilon \right) = C_{\varepsilon 1} \frac{\varepsilon}{k} [P + \{1 - C_{\varepsilon 3} \} G] P - \rho_f C_{\varepsilon 2} \frac{\varepsilon^2}{k}
\]  \tag{4}

\[
v_t = \frac{\mu_t}{\rho_f} = C_\mu \frac{k^2}{\varepsilon}
\]  \tag{5}

where \( \mu \) is viscosity, \( \mu_t \) is turbulent viscosity, \( P \) accounts for the production of the kinetic energy through mean shear stresses, \( G \) is production term related to gravity effects and finally \( \sigma_k = 1 \), \( \sigma_\varepsilon = 1.3 \), \( C_{\varepsilon 1} = 1.3 \), \( C_{\varepsilon 2} = 1.92 \), \( C_\mu = 0.09 \), \( C_{\varepsilon 3} = 0 \) if \( G \geq 0 \) and \( C_{\varepsilon 3} = 1 \) if \( G \leq 0 \) are constants.

### 2.2 SOLID PHASE: DISCRETE ELEMENT METHOD (DEM)

The Discrete Element Method (DEM) is used to model the solid phase (particles); Cundall and Strack [24] developed the Discrete Element Method (DEM) model to study soil mechanics. The DEM model provides the location (coordinates), velocity, and contact force of each particle (solid phase) at each step. The motion of each particle is governed by Newton's second law of motion, and can be expressed, for the \( i \)-particle, by:

\[
m_{i,p} \frac{d \vec{v}_{i,p}}{dt} = m_{i,p} \vec{g} + \sum_{j=1}^{m} \vec{F}_{c,i,j} + \vec{F}_{d,i,p} + \vec{F}_{b,i,p}
\]  \tag{6}
\[ I_{i,p} \frac{d\vec{\omega}_{i,p}}{dt} = \vec{T}_{i,p}, \quad \dot{I}_{i,p} = \frac{2}{5} m_{i,p} r_{i,p}^2 \]  

where \( \vec{v}_{i,p} \) is velocity of the \((i \text{ th})\) particle, \( m_i \) is mass particle, \( g \) is gravity acceleration, \( n_c \) is number of contacts between the particles, \( \vec{F}_{c,ij} \) is contact forces, \( \vec{F}_{D,i,p} \) is drag force, \( \vec{F}_{B,j,p} \) is buoyancy force. \( I_{i,p} \) is moment of inertia, \( \vec{\omega}_{i,p} \) is rotational velocity, \( \vec{T}_{i,p} \) is torque and \( r_{i,p} \) is radius.

The drag force \( \vec{F}_{D,i,p} \) represent the affect between the fluid phase (CFD) and solid phase (DEM) in other word the coupling of CFD-DEM. The drag force is expressed according to relation proposed by Helland et al. [25]:

\[ \vec{F}_{D,i,p} = \frac{C_{D,i,p}}{8} \pi \rho_i d_{i,p}^2 \| \vec{v}_f - \vec{v}_{i,p} \| (\vec{v}_f - \vec{v}_{i,p}) \cdot \epsilon_f^2 \cdot f(\epsilon_f) \]  

where \( C_{D,i,p} \) is particle drag coefficient, \( d_{i,p} \) is particle diameter, \( \rho_i, \vec{v}_f, \epsilon_f \) are density, velocity and local porosity of the fluid, and \( m,n \) is a parameter. \( f(\epsilon_f) \) is porosity function [26]:

\[ f(\epsilon_f) = \epsilon_f^{-n}, \quad \text{and} \quad f(\epsilon_f)^m = \epsilon_f^{-m,n} \]  

The drag coefficient on a particle is function of Reynolds number \( (\text{Re}_{i,p}) \), and \( C_{D,i,p} \) is given [23]:

\[ C_{D,i,p} = \begin{cases} \frac{24}{\text{Re}_{p,i}} (1 + 0.15 \text{Re}_{p,i}^{0.687}) \text{ if } \text{Re}_{i,p} < 1000, \\ \frac{\epsilon_f \rho_i d_{i,p} \| \vec{v}_f - \vec{v}_{p,i} \|}{\mu_f} \text{ if } \text{Re}_{i,p} \geq 1000 \end{cases} \]  

\[ \text{Re}_{i,p} = \frac{\epsilon_f \rho_i d_{i,p} \| \vec{v}_f - \vec{v}_{p,i} \|}{\mu_f} \]  

\[ m C_{p,i} \frac{dT}{dt} = \Phi_{i,Conduction} + \Phi_{i,Convection} \]  

\[ (14-208) \]
where \( m \) is fluid mass, \( C_{p,i,j} \) is heat capacity, \( \Phi_{i,j, \text{Conduction}} \) is heat transferred between the particles \( \Omega_i \) and \( \Omega_j \), and \( \Phi_{i,j, \text{Convection}} \) is heat transport by convection.

\[
\Phi_{i,j, \text{Conduction}} = H_{i,j,C} (T_j - T_i), \quad \text{and} \quad H_{i,j,C} = 2k_ia = 2k_i \left( \frac{3F_ia^*}{4E^*} \right)^{\frac{1}{3}}
\]

(12)

where \( H_{i,j,C} \) is the contact conductance between the particles \( \Omega_i \) and \( \Omega_j \) with \( j \) varies from 1 to \( N \) (contact number), \( k_i \) is particle thermal conductivity, \( F_n \) is normal force, \( a \) is hertzian radius, \( a^* \) is equivalent radius , and \( E^* \) is an equivalent Young’s modulus between the contacted particles.

\[
\Phi_{i,j, \text{Convection}} = h_iS_j (T_j - T_\infty)
\]

(13)

where \( h_i \) is heat transfer coefficient, \( T_s \) temperature of particle surface, \( T_\infty \) temperature of fluid and \( S_j \) is surface area of contact particle with the fluid [2].

Many of empirical and theoretical investigations are settled equations to calculate the value of heat transfer coefficient from the Nusselt number \( (Nu) \), where the Nusselt number is the ratio of convection heat transfer in the fluid in motion (through the Prandtl number \( \nu = \frac{C_{p,i,j} \mu_f}{k_f} \) ) to the conduction heat transfer in the flow regime (through the Reynolds number). The Nusselt number is founded from the set of equations proposed by Li & Mason [27] as followings:

\[
Nu_{i,p} = \frac{h_{i,p} d_{i,p}}{k_f} = \begin{cases}
2 + 0.6\varepsilon_{i,p}^{m,n} \text{Re}_{i,p}^{0.5} \text{Pr}^{0.3} & \text{Re}_{i,p} < 200 \\
2 + \varepsilon_{i,p}^{m,n} (0.5 \text{Re}_{i,p}^{0.5} + 0.02 \text{Re}_{i,p}^{0.8}) \text{Pr}^{0.3} & 200 < \text{Re}_{i,p} < 1500 \\
2 + 0.000045\varepsilon_{i,p}^{m,n} \text{Re}_{i,p}^{1.8} & \text{Re}_{i,p} > 1500
\end{cases}
\]

(14)

where \( k_f \) is fluid thermal conductivity, \( \varepsilon_{i,p} \) is porosity of particle and \( m.n \) equals 4.7. Then, substituting the equations 12 and 13 in equation 11 yields.

\[
mC_{p,i} \frac{dT_i}{dt} = \sum_{j=1}^{N} H_{i,j,C} (T_j - T_i) + h_iS_j (T_s - T_\infty)
\]

(15)

Finally, the temperature of the particle \( \Omega_i \) is calculated by integrating equation 14 into the Discrete element method (DEM) code, as follow:
\[
T_i^{t+\Delta t} = T_i^t + \frac{\Delta t}{mC_{p,i}} \left[ \sum_{j=1}^{N} H_{ij,C}(T_j^t - T_i^t) \right] + \Delta T_i + \sum_{i \in \text{h}} S_i(T_i - T_{\text{eq}})
\]

(16)

3. RESULTS AND DISCUSSIONS

The numerical approaches and modelling have been developed to investigate and simulate the difficult phenomena that accompanied the transport of heat and energy.

3.1 VALIDATION OF THERMAL APPROACH WITH THE ANALYTICAL CALCULATION

The integration of the thermal approach in the CFD–DEM coupling is tested. A single particle (sphere) in a column is tested numerically, comparing the numerical results with the analytical results. The comparison has been made for two cases (heating and cooling) between the temperatures of the sphere, which is calculated by the model with the analytical calculation from the equation below for pure convection:

\[
T_p = T_f + \left( T_{p,o} - T_f \right) \text{EXP} \left( \frac{-hS_p t}{mC_p} \right)
\]

(17)

where \( T_{p,o} \) is initial temperature, \( T_p \) is final temperature of the particle respectively, \( T_f \) is temperature of the fluid, \( S_p \) is the surface area of the particle, and \( t \) is the time.

A spherical particle is put in the center of a column (\( D = 51 \) mm, \( H = 0.75 \) m), under the effect of an upward flow of water, with a flow rate of 0.08 m/s. The physical properties of the aluminum particle and water are detailed in Table 1. For this test, two cases are checked: the heating and the cooling of the sphere. In the case of heating, the initial temperature of the particle and the fluid in the column is; 20 °C. The boundary condition for the heated surface is 80 °C. In the case of cooling, this temperature becomes; 80 °C, and the boundary temperature for the cooling surface becomes 20 °C.

The Comparison of the simulation and analytic results for the heating and cooling cases are depicted in Figure 1. The heating and cooling curves show that the numerical results present good adequacy with the analytical results. Thus, the proposed thermal model integrated into the CFD-DEM can deal with a natural and complex system in fluidization units.
Table 1. Physical properties and dimensions of simulated system.

<table>
<thead>
<tr>
<th>Property</th>
<th>Aluminium</th>
<th>Water</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Particle diameter, $d_p$</td>
<td></td>
<td></td>
<td>1.5 mm</td>
</tr>
<tr>
<td>Density, $\rho_p$</td>
<td></td>
<td></td>
<td>2700 kg/m$^3$</td>
</tr>
<tr>
<td>Heat capacity, $C_p$</td>
<td></td>
<td></td>
<td>900 J/kg.K</td>
</tr>
<tr>
<td>Thermal conductivity, $k_p$</td>
<td></td>
<td></td>
<td>237 W/m.K</td>
</tr>
<tr>
<td>Number of particles</td>
<td></td>
<td></td>
<td>2244</td>
</tr>
<tr>
<td>Young modulus, $E$</td>
<td></td>
<td></td>
<td>69 GPa</td>
</tr>
<tr>
<td>Poisson ratio, $\nu$</td>
<td></td>
<td></td>
<td>0.35</td>
</tr>
<tr>
<td>Friction coefficient, $\mu$</td>
<td></td>
<td></td>
<td>0.4</td>
</tr>
<tr>
<td>Viscosity, $\mu_f$</td>
<td></td>
<td></td>
<td>0.001 kg/m.s</td>
</tr>
<tr>
<td>Density, $\rho_f$</td>
<td></td>
<td></td>
<td>1000 kg/m$^3$</td>
</tr>
<tr>
<td>Heat capacity, $C_f$</td>
<td></td>
<td></td>
<td>4200 J/kg.K</td>
</tr>
<tr>
<td>Thermal conductivity, $k_f$</td>
<td></td>
<td></td>
<td>0.6 W/m.K</td>
</tr>
<tr>
<td>Bed height, $H$</td>
<td></td>
<td></td>
<td>75 cm</td>
</tr>
<tr>
<td>Bed width (diameter)</td>
<td></td>
<td></td>
<td>5.1 cm</td>
</tr>
<tr>
<td>Bed thickness</td>
<td></td>
<td></td>
<td>0.15 cm</td>
</tr>
<tr>
<td>Time step, $\Delta t$</td>
<td></td>
<td></td>
<td>5x10$^{-6}$ sec</td>
</tr>
</tbody>
</table>

3.2 THERMAL MODELLING OF THE FLUIDIZATION PROCESS

The development of fluidized bed units, need to understand what goes on inter these processes within fluidized bed units is vital to improving designs for these units. For this, three methods of column heating have been proposed as following; a) heating the base (inlet), b) heating the walls, c) heating the base and the walls (see Figure 2). The system consist of spherical particles of Aluminium are placed in a column. The properties of the particles and water, and the dimensions of geometry are presented in Table 1. The height of the bed at rest ($h = 9.9$ cm) and the thickness of bed is one particle diameter ($d_p=1.5$ mm). The bed particles were fluidized over a range of inlet
water velocities (0.08 m/s, 0.12 m/s and 0.16 m/s) for heating temperatures (boundary condition for the heating surface or the temperature impose) of 60 and 100 °C. The initial temperature for the particles and the inlet water were set at 20 °C.

![Diagram of column heating methods](image)

Figure 2: Sketch of three methods of column heating; a) base heating, b) wall heating, c) wall and base heating.

### 3.2.1. MEAN TEMPERATURE OF PARTICLE

The mean temperatures of particle are plotted as a function of simulation time as shown in Figures 3 and 4. They depict the mean temperature of particle with three different water fluidization velocities 0.08 m/s, 0.12 m/s, and 0.16 m/s, for three methods of heating with imposed temperatures (\(T_{\text{impose}}\) of 60 °C and 100 °C). The development of the temperature curves for base heating and base plus wall heating are increase with the time. They are reached steady state after 3 seconds from start of simulation, as depicted in Figure 3 and Figure 4. In other hand, the value of mean temperature for wall heating is not progress and still near the initial temperature, and the heating temperature not passing 5 °C (mean temperature didn’t pass 25 °C).

This can be explained from the fact that the flow of water and particles are prevented the wall, whereas the stresses are concentrated and the collisions time of contact between the particles and heating walls is very short that is not sufficient for exchanging heat and energy. The results of heated wall method were consistency with the results of Malone, and Xu [11]. Thus, the two
heating methods of base and base plus wall heating are more efficacious rather than the wall heating method in which the particles exchange the heat energy with the heating surfaces. From the Figures 3, 4, for the two methods base column heating and base plus wall column heating are almost matching results and the temperature are developed until it has reached a steady state for the two case of heating surfaces with (T\text{impose} = 60 \, ^\circ\text{C} \text{ and } 100 \, ^\circ\text{C}).

In comparing between the two methods of heating; base column heating and base plus wall column heating, the first method will take the advantage, because it is less complex in terms of engineering in industrial applications and less costly for energy consumption.

The inlet velocity is essential in fluidization process; Figure 5 reveals a comparison between the results of the three inlet water fluidization velocities 0.08 m/s, 0.12 m/s and 0.16 m/s of solid phase (particles) heating process. The Figure 5 a, b reveals the mean particle temperature of base column heating for first four seconds from the start of simulation of fluidization process. It is obvious that the water velocity (v_f = 0.12 m/s) is fully fluidized the bed and reaches the thermal steady state before two others water velocities 0.08 m/s and 0.16 m/s (see Figure 5 a, b).

The results in Figure 5 reveals that the increase of inlet velocity of the fluid in heating process in fluidized beds it doesn't have to be useful. In fact, the high velocity of the fluid didn’t give the time to transport of heat and energy between particle–particle, particle–heating surface and particle–fluid. In other hand, the low velocity of the fluid in heating process may cause overheating to the particles. Thus, the proper inlet velocity of fluid and imposed temperature is crucial in heating or cooling process in fluidized beds.
Figure 3: Mean temperature distribution of particle for: a) Base heating, b) Wall heating, c) Wall and base heating. ($T_{\text{impose}}=60 \, ^\circ\text{C}$).
Figure 4: Mean temperature distribution of particle for: a) Base heating,  b) Wall heating, c) Wall and base heating. ($T_{\text{impose}}=100$ °C).

Figure 5: Mean temperature distribution of particle for: a) heating base for $T_{\text{impose}} = 60$ °C, b) heating base for $T_{\text{impose}} = 100$ °C.
Figure 6 a-g, is depicted the snapshots of temperature development as a function of time in the fluidized of base heated method for inlet water velocity \( v_f = 0.12 \) m/s and the base heated to 100 °C (\( T_{\text{impose}} = 100°C \)). The snapshots (a-d) in Figure 6, shows the starting of the fluidization under the effect of drag force, when water passes through the interstitial clearance between the particles upward from the bottom of the column, the drag force between the water and the static bed trend the bed (particles) to move upward. The drag force will push the bed to overcome the gravity force of particles. After about the three seconds the fluidized bed (particles) trend to be uniform and stable. The good mixing of the particles with the water, increase the contact between the solid phase (particle-particle), solid phase with the walls (particle-heated surface and walls), the particles with the water, and fluid phase with the heated surfaces are developed the heat exchange between the two phases as shown in Figure 6 e-g, (the temperature profile is in Kelvin). The Figure 6 reveals the well integration of the thermal model with the CFD-DEM coupling.

![Figure 6: Snapshots of temperature development in the fluidized bed. \( v_f = 0.12 \) m/s and \( T_{\text{impose}} = 100°C \).](image_url)
4. CONCLUSIONS

This study investigates the heat transfer behavior in a fluidized bed by implementing a thermal model of conduction-convection with CFD-DEM coupling. The thermal approach considers almost all the known mechanisms of heat transfer, including conduction between particle-particle and pipe wall–particle, as well as convection between particle–fluid, pipe wall–particle, fluid–pipe wall, and fluid with the heated surfaces. All these mechanisms are applied in this study. A numerical validation has been done for the thermal approach, comparing the simulation and analytic results for the heating and cooling cases for a single particle in a column. The numerical results present excellent adequacy with the analytical results. Then, three methods of column heating have been proposed as follows, heating the base (inlet), heating the walls, and heating the base and the walls have been proposed in the simulations. The development of the temperature curves for base heating and base plus wall heating increases with time. They reached a steady state after 3 seconds from the start of the simulation. On the other hand, the mean temperature value for wall heating is not progress and is still near the initial temperature, and the heating temperature does not pass 5 °C. The simulations results show that the method of the heating wall was not reached the thermal steady state, which can be explained by the fact that the flow of water and particles prevented the wall, where stresses are concentrated, and the collusions time of contact between the particles and heating walls is a short time. It is not sufficient for heat and energy exchange to occur. The results of the heated wall method were consistent with the results of Malone and Xu [11]. The results reveal that; the increase of inlet velocity of the fluid in the heating fluidized beds process was not helpful. The high velocity of the fluid did not give the time to transport heat and energy between particle-particle, particle–heating surface, and particle–fluid. Thus, the proper inlet velocity of the fluid and imposed temperature is crucial in the heating process in fluidized beds since the particles need to provide time for the heat transport process.

5. REFERENCES


