Study of Sound Waves in Fluidized Bed using CFD-DEM

Simulations

H. A. Khawaja
hassan.a.khawaja@uit.no
Process and Gas Group
Department of Engineering & Safety
UiT-The Arctic University of Norway
9037, Langnes, Tromsø, Norway

Abstract

The speed of sound waves is investigated using CFD-DEM numerical simulations. Appropriate initial and boundary conditions are applied to capture the phenomenon. The effect of varying the height of the bed is also studied. The results of the simulations matched those from literature. The pressure and particle velocity profiles from the simulation showed the oscillatory behavior. Functions (based on a damped standing wave) were fitted to these, which allowed them to be stated in time and space variables. These fitted functions were substituted to the linearized governing equations for the two-phase flow. Using these assumed solutions allowed a new relationship to be derived for the speed of sound and damping in the system. It is concluded that the damping in the system is due to the effective bulk viscosity of the solid phase, which arises from the particle viscosity.

1 Introduction

The presence of particles in a gas phase (as in a fluidized bed) is known to affect the propagation of sound waves through the continuous phase. Cahan (1990) studied sound waves by sprinkling lycopodium seeds into an oscillating column of air within a tube to identify the nodes of a standing wave. It was found that the sound waves diminished in the presence of particles and that the speed of sound measured changed from its theoretical value in air. Later, Mallock (1910) studied the velocity of sound in liquid-gas mixtures such as froths. The results also showed that the speed of sound differed from the value in gas in a similar manner to that of the gas-particle mixture studied by Cahan (1990). Similarly, Roy, Davidson, and Tuponogov
(1990) studied the speed of sound in a gas-fluidized bed. They cross-correlated the pressure signal at different heights of the bed to detect the speed of the moving disturbance, as well as measuring the frequency of the standing wave after a disturbance had been introduced to infer wave speed. It was found that the speed of sound is significantly lower in the gas-particle medium.

The velocity of a sound wave in a continuous compressible medium is given by Lamb (1963) as shown in Equation (1),

$$u_s = \sqrt{\frac{dp}{d\rho}} \quad (1)$$

where $u_s$ is the speed of sound and $dp/d\rho$ is the rate of change of pressure with bulk density.

To apply the given relationship (Equation (1)) to a two-phase mixture of gas and particles, a number of assumptions need to be made, as provided by Roy et al. (1990) and later acknowledged by H. T. Bi, Grace, and Zhu (1995) and Hsiaotao T. Bi (2007). These assumptions are also given by Mallock (1910), Tangren, Dodge, and Seifert (1949) and Campbell and Pitcher (1958).

1. The particles and gas move together (i.e. homogenous rather than separated flow),
2. The gas is compressible and obeys the ideal gas law,
3. The particles are incompressible,
4. The particulate matter and gas are isothermal.

The assumption that the gas and particles are in an isothermal state can be justified by computing the time required for solid and gas to attain the same temperature as discussed by Roy et al. (1990). This assumption might not be valid in fluidized beds with larger particles because increasing the size of particle increases the time constant value, hence, increasing the time taken by the system to reach thermal equilibrium. A similar conclusion is reached by Turton, Fitzgerald, and Levenspiel (1989) and Kunii and Levenspiel (1991).

Roy et al. (1990) derived an expression for the speed of sound in a homogenous two-phase medium as shown in Equation (2),
where $\rho_s$ is the density of solids, $\rho_g$ is the density of gas, $\varepsilon$ is the void fraction, $T_g$ is the absolute gas temperature and $R$ is the specific gas constant.

It is to be noted that Equation (2) is only valid when the value of voidage is less than one ($\varepsilon < 1$). Roy et al. (1990) demonstrated experimentally that the speed of sound in a fluidized mixture of sand and air is typically $1/30$ of the speed of sound in air. Similar results are reported by T. Bi et al. (1995), who found the speed of sound to be $10$ m/s in a fluidized mixture of air and fine particles (50 µm diameter with a density of 1580 kg/m$^3$).

Roy et al. (1990) also suggested a theoretical damping time relationship, derived by assuming a system of a mass attached to a spring with viscous damping as shown in Equation (3),

$$\tau = \frac{2g}{\omega^2 U_{mf}}$$

where $\tau$ is the damping time, $g$ is the gravity constant, $\omega$ is the angular frequency of the oscillations and $U_{mf}$ is the minimum fluidization velocity.

In this work, the speed of sound in the fluidized medium is verified through experiment and CFD-DEM (Computational Fluid Dynamics – Discrete Element Modelling) numerical simulations. The results are also analyzed analytically, revealing the importance of particle viscosity in the damping of sound waves in the fluidized bed.
2 Experimental verification of speed of sound in a fluidized medium

An experiment was set up to demonstrate the standing wave which can be created in a fluidized medium. Roy et al. (1990) associated standing waves in the fluidized medium with the speed of sound. Their explanation takes into account a case analogous to an organ pipe, with one end closed and the other open. A simple experiment was set up to observe the same behavior, as shown in Figure 1.

![Diagram of experiment setup](image)

**Figure 1: Experiment setup to study the standing wave in a fluidized bed**

The experimental setup consists of a Perspex® tube with an internal diameter of 5 cm and an external diameter of 6 cm. The two ends of the tube are sealed with rubber plugs. The tube is filled with alumina silicate particles (diameter ≈ 50 µm). These particles are fluidized by rotating the tube vertically for a few 360-degree rotations. As the result of rotation, the particles are exposed to centrifugal force, building a relative velocity between gas and particles. This causes the particles to fluidize, i.e. the powder gas mixture becomes free-flowing, and a horizontal level ‘free surface at meniscus’ forms, regardless of the tilt of the tube. In addition, a significant expansion in the fluidized bed is noted before and after fluidization. Once
fluidized, an impact load is induced in the fluidized medium by striking the tube on the ground. This induces vertical oscillations in the fluidized medium, corresponding to the standing wave in the medium. The frequency of these oscillations is noted by making a video of the meniscus of the fluidized medium at 30 Hz using Sony Handycam DCR-HC14E. A .wmv clip was captured and converted into image files; the captured images were observed to measure the wave frequency. Figure 2 shows the oscillations in the fluidized medium as captured using the above-described experimental setup.

Figure 2: The top meniscus of the fluidized bed captured at 30 Hz; green arrows indicate the direction of oscillation on the free surface; time is shown in the bottom of each image.

It was found that the average time period for a single oscillation in the experiments was 0.286 s, which corresponds to a frequency of 3.50 Hz. The length of the wave can be found from the height of the fluidized medium in the tube. It was noted that the height after fluidization was 80 cm. This corresponds to a quarter of a standing wave in a tube with one end closed and the other open; therefore, the complete wavelength of the standing wave is 320 cm. Therefore, the speed of sound is 11.2 m/s.

The results found through experimentation were compared with the speed of sound given by Equation (2). The values of variables used are as follows: the density of alumina silicate particles $\rho_s$ is 3500 kg/m$^3$, the density of air $\rho_g$ is 1.24 kg/m$^3$, the temperature of the air $T_g$ is 298 K, (void fraction) $\epsilon$ is estimated to be 0.4, and the specific gas constant of the air is 287
J/kg/K. This resulted in the speed of sound in the fluidized medium $u_s$ being equal to 11.23 m/s. Hence, a good agreement between experimental and theoretical results was found.

### 3 CFD-DEM numerical simulation of speed of sound in a fluidized medium

A computational fluid dynamics-discrete element methods (CFD-DEM) numerical simulation was set up to study the speed of sound in a fluidized medium. The simulated setup is shown below in Figure 3.

![Figure 3: CFD-DEM numerical simulation domain for the study of speed of sound in the fluidized medium; the particles are simulated in three dimensions, whereas fluid is simulated in two dimensions.](image)

In this setup, the particles are simulated using a discrete element model as in H. Khawaja (2011); HA Khawaja and Scott (2011), and H. Khawaja (2015). The CFD-DEM model is based on the volume-averaged continuity and momentum equations (Equations (4) and (5)), which
are solved using the CFD (computational fluid dynamics) density driven method as discussed in T. B. Anderson and Jackson (1967); Crowe, Sommerfeld, and Tsuji (1998),

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_k)}{\partial x_k} = 0 \]  

(4)

\[ \frac{\partial (\rho \varepsilon u_i)}{\partial t} + \frac{\partial (\rho \varepsilon u_i u_k)}{\partial x_k} = \frac{\partial}{\partial x_k} \rho \varepsilon p + \frac{\partial}{\partial x_k} \varepsilon \tau_f + \rho \varepsilon g_i \]  

(5)

where \( \varepsilon \) is the voidage, \( \rho \) is the density fluid, \( u_k \) is the velocity of the fluid, \( p \) is the pressure of the fluid, \( \vec{F}_i \) is the interaction force felt by the fluid due to the particles, \( g_i \) is the gravity constant and \( \tau_f \) is the fluid stress tensor. Note that \( k \) and \( i \) subscripts are Einstein notations (T. B. Anderson & Jackson, 1967). Voidage \( \varepsilon \) is the ratio of the volume of fluid (excluding the particles) to the total volume of a fluid cell. It needs to be accurately computed in a cuboidal domain of CFD with moving spherical particles, as given by (HA Khawaja, Scott, Virk, & Moatamedi, 2012).

The finite volume discretization technique is applied to Equations (4) and (5). This technique is based on conservation of variables; therefore, it ensures that the physical quantities are conserved over the chosen control volumes and the domain as a whole (J. D. Anderson, 1995; Patankar, 1980).

The stability and sensitivity of the solution depend on the time step and cell size, whose values are determined by the Courant-Friedrichs-Lewy (CFL) condition (Courant, Friedrichs, & Lewy, 1928; Hirsch, 2007) as shown in Equation (6),

\[ CFL \text{ Number} > \frac{a \Delta t}{\min(\Delta x, \Delta y, \Delta z)} \]  

(6)

where \( \Delta t \) is the time step size, \( \Delta x, \Delta y, \Delta z \) are the dimensions of the fluid cell and \( a \) is the speed of sound in the gas medium.

The interaction force correlations are given by (Beestra, van der Hoef, & Kuipers, 2007; Di Felice, 1994; Ergun, 1952; Wen & Yu, 1966). By conducting fluidized bed experiments, Müller et al. (2008) compared these correlations and found that the correlation from Beestra et al.
(2007) is the most promising in the voidage range of $0.3 < \epsilon < 0.5$. This correlation is shown in Equation (7),

$$
\beta = A \frac{(1-\epsilon)\mu_f}{\epsilon d_p^2} + B \frac{\mu_f(1-\epsilon)Re}{d_p^2}
$$

(7)

where $\beta$ is the drag coefficient, $A$ is shown in Equation (8) and $B$ is shown in Equation (9),

$$
A = 180 + \frac{18\epsilon^4}{1-\epsilon} \left(1 + 1.5\sqrt{1-\epsilon}\right)
$$

(8)

$$
B = \frac{0.31(\epsilon^{-1} + 3\epsilon(1-\epsilon) + 8.4Re^{-0.343})}{1 + 10^{3(1-\epsilon)Re^{2\epsilon-2.5}}}
$$

(9)

In CFD-DEM simulations, the interaction force felt by the fluid due to the particles is the sum of the drag on the particles in the particular fluid cell as shown in Equation (10),

$$
\vec{F}_i = \frac{1}{(1-\epsilon)V_{cell}} \sum_{i=1}^{n_p} \vec{f}_i
$$

(10)

where $V_{cell}$ is the volume of the cell, $n_p$ is the number of particles in the cell, $\vec{f}_i$ is the drag force on $i$ particle as shown in Equation (11),

$$
\vec{f}_i = V_p \beta(\vec{u}_f - \vec{u}_p)
$$

(11)

where $\vec{f}_i$ is the force vector felt by the particle due to the fluid drag, $V_p$ is the volume of the particle and $\beta$ is the drag coefficient computed using the correlation given in Equation (7).

Discrete element modelling (DEM) is based on a Lagrangian approach where each particle’s motion is governed by Newton’s second law. The linear momentum equation for each particle is,

$$
m_p \ddot{d}_p = \vec{f}_i + \sum_{\text{contacts}} \vec{f}_{\text{contacts}} + m_p \vec{g}
$$

(12)
where $m_p$ is the mass of the particle, $\vec{a}_p$ is the linear acceleration vector, $\vec{f}_i$ is the force on the particles due to the fluid, $\vec{f}_{\text{contact}}$ is the force due to the contact with other particles.

The third-order Adams-Bashforth time stepping scheme (Gear, 1971; Hairer, Nørsett, & Wanner, 1993), as shown in Equation (13), is used to advance the fluid as well as the particle variables forward in time.

$$P_{t+1} = P_t + \Delta t \left( \frac{23}{12} dP_{t+1} - \frac{4}{3} dP_t + \frac{5}{12} dP_{t-1} \right)$$  \hspace{1cm} (13)

where $\Delta t$ is the time step size, $P_{t+1}$ is the value of the physical property stepping forward in time, $P_t$ is the value of the physical property before stepping forward in time, and $dP$ is the change in the property. The subscript $t$ in Equation (13) refers to the time step.

Particle-particle contact is solved using soft sphere contact models (Crowe et al., 1998; van der Hoef, Annaland, Deen, & Kuipers, 2008). In the soft contact model, the contact forces are based on a simple linear spring-dashpot model. These models have been used in DEM by (Crowe et al., 1998; Cundall & Strack, 1979; Third, Scott, Scott, & Müller, 2010; Tsuji, Kawaguchi, & Tanaka, 1993; Tsuji, Tanaka, & Ishida, 1992; van der Hoef, van Sint Annaland, & Kuipers, 2004). The contact forces can be divided into normal and tangential forces. The normal contact model is based on the non-linear spring model given by Hertz (Hertz, 1882). The tangential contact model is given by Mindlin and Deresiewicz (1953) and simplified by Tsuji et al. (1992) for DEM. Both models are tested for their suitability for DEM by Ha Khawaja and Parvez (2010). The most computationally intensive operation in the CFD-DEM simulation is the search for particle-particle contacts. H. Khawaja (2015) has undertaken a study on the optimization of this algorithm.

The setup is three-dimensional for the particles and two-dimensional for the fluid. The particles are allowed to move in three dimensions, but, due to the narrow domain in the $z$-direction, as shown in Figure 3, the fluid flow is modeled in the $x$ and $y$ dimensions (this is achieved by setting a single fluid cell in the $z$-direction). There are 14 cells in the $x$-direction, of which 12 are computing cells and two are boundary cells. The cells in the $y$-direction are varied based on the size of the simulation. The physical parameter values set for the CFD-DEM numerical simulation are given in Table 1.
Table 1: Physical parameters set for CFD-DEM numerical simulations

<table>
<thead>
<tr>
<th>Physical Parameters</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Temperature</td>
<td>298.15K</td>
</tr>
<tr>
<td>Fluid density</td>
<td>1.13 Kg/m³</td>
</tr>
<tr>
<td>Fluid viscosity</td>
<td>$1.8 \times 10^{-5}$ Pa s</td>
</tr>
<tr>
<td>Time step size</td>
<td>$3.25 \times 10^{-7}$ sec</td>
</tr>
<tr>
<td>Number of CFD cells in x-direction</td>
<td>14 (12 computing and 2 boundary cells)</td>
</tr>
<tr>
<td>CFD cell size in x-direction</td>
<td>0.45 mm</td>
</tr>
<tr>
<td>CFD cell size in y-direction</td>
<td>0.6 mm</td>
</tr>
<tr>
<td>Width of domain in z-direction</td>
<td>1.25 mm</td>
</tr>
<tr>
<td>Diameter of particles</td>
<td>$0.15 \pm 0.00625$ mm</td>
</tr>
<tr>
<td>Density of solid particles</td>
<td>1000 Kg/m³</td>
</tr>
<tr>
<td>Minimum fluidization velocity</td>
<td>0.0085 m/s</td>
</tr>
<tr>
<td>Speed of sound in the two-phase medium,</td>
<td>from Equation (2)</td>
</tr>
<tr>
<td></td>
<td>20.7 m/s</td>
</tr>
<tr>
<td>Young modulus of solid particles</td>
<td>$1.2 \times 10^8$ Pa</td>
</tr>
<tr>
<td>Poisson ratio of solid particles</td>
<td>0.3</td>
</tr>
<tr>
<td>Coefficient of normal restitution for solid particles</td>
<td>0.986</td>
</tr>
<tr>
<td>Coefficient of friction</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Four different CFD-DEM numerical simulations were set up with various heights of bed as shown in Table 2.

Table 2: Different sets of CFD-DEM numerical simulations

<table>
<thead>
<tr>
<th>Test case</th>
<th>Number of particles</th>
<th>Number of cells in y-direction</th>
<th>Height of the particles after fluidization (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>107800</td>
<td>208</td>
<td>49</td>
</tr>
<tr>
<td>2</td>
<td>215320</td>
<td>312</td>
<td>98</td>
</tr>
<tr>
<td>3</td>
<td>322480</td>
<td>416</td>
<td>147</td>
</tr>
<tr>
<td>4</td>
<td>430360</td>
<td>520</td>
<td>196</td>
</tr>
</tbody>
</table>
The simulation is initialized by randomly placing the particles in the domain. The particles are allowed to fall under gravity to settle down as shown in Figure 3. Then the particles are fluidized to approximately 1.1 times the minimum fluidization velocity ($U_{mf}$). This is achieved by specifying a rate of change of mass flow rate in the $y$-direction $\dot{m}_y$ in the guard (inlet) cells, as shown in Equation (14),

$$
\Delta u_y = \frac{\dot{m}_y \Delta t}{\rho_g \Delta x \Delta z} u_y \leq 1.1 U_{mf}
$$

$$
\Delta u_y = 0 \quad u_y > 1.1 U_{mf}
$$

where $u_y$ is the velocity of fluid in the $y$-direction, $\Delta u_y$ is the change in fluid velocity in the $y$-direction, $\Delta x$ is the dimension of the fluid cell in the $x$-direction, $\Delta z$ is the dimension of the fluid cell in the $z$-direction, $\rho_g$ is the density of the fluidizing gas and $\Delta t$ is the time step size. The boundary conditions are specified by specifying walls for the particles in the $x$, $y$ and $z$ planes as shown in Figure 3 (walls in the $z$-plane are not visible in Figure 3). The CFD boundary conditions are specified by first setting full slip boundary conditions for the fluid in the cells on either side of the domain in the $x$-direction. This is achieved by setting the $y$-velocity in the guard cell equal to one in the closest cell in the $x$-direction, as shown in Equation (15),

$$
\begin{align*}
  u_y(1,y) &= u_y(2,y) \\
  u_y(14,y) &= u_y(13,y)
\end{align*}
$$

where bracketed numbers indicate the position of the cell in the domain and the $y$ coordinate means that it is applicable in all corresponding $y$-cells except the corner cells. Characteristic boundary conditions are applied to the outlet cells to avoid reflection in pressure signals as discussed in Chung (2010).

Initial attempts to introduce a disturbance into the bed, in which the boundary fluid inflow was perturbed, were unsuccessful. The perturbation quickly damped and did not perturb the relatively massive particles. The given CFD-DEM numerical simulations were performed very close to the minimum fluidization velocity of the particles. Therefore fluid velocity was neither so high that the particles would flow with the fluid nor so low that particles would not be affected at all. Therefore, it was challenging to introduce an appropriate disturbance in such a
case. After few trials, it was found that, to introduce an appropriate disturbance, the particles need to be perturbed rather than the fluid. Therefore, the disturbance is introduced by raising the particles in the y-direction by 1.5 mm (10 times the diameter of the particles) and then allowing them to drop under gravity. The sequence of steps taken to introduce this disturbance is illustrated in Figure 4.

![Figure 4: The sequence of steps to generate a disturbance in the fluidized medium](image)

With the drop, the fluidized medium behaves in the same way as discussed earlier in Section 2. This behavior of various physical parameters was recorded and analyzed as discussed in Section 4.

### 4 Results of CFD-DEM numerical simulation

The results of standing waves from the CFD-DEM numerical simulations in the two-phase medium were investigated by plotting the relevant oscillating physical parameters over time. Fluid pressure and particle velocity were averaged width-wise. The results were averaged in order to reduce the impact of other phenomena in the two-phase medium such as formations of bubbles, their coalescence, their eruption, etc.

It can be observed that the maximum fluctuations in gas pressure occur at the bottom of the bed, whereas the maximum fluctuations in particle velocity occur at the top of the bed (supplementary material). Roy et al. (1990) observed the same trends as shown in Figure 7. These trends can be explained by the fact that particle motion is more constrained at the bottom...
of the bed in comparison to the particles at the top of the bed. Figure 7 also shows that the
highest value of pressure fluctuation occurs at the bottom of the bed, which is in agreement
with CFD-DEM numerical simulation results. It can also be observed from Figure 5 that
pressure fluctuation is at its peak when the disturbance is introduced at time zero, whereas the
particle velocity fluctuation is zero at time zero, as shown in Figure 6. This difference indicates
that fluctuations in particle velocity is out of phase by $\frac{\pi}{2}$ from pressure fluctuations.

Figure 5: Pressure is plotted against time at 100 mm height in the bed for 430360 particles
fluidized bed (test case 4); the red curve shows the fitted function and the blue circles
represents the CFD-DEM numerical simulation results.
Figure 6: Particle velocity fluctuation plotted against time at 100 mm for 430360 particles fluidized bed (test case 4); the red curve shows the fitted function and the blue circles represents the CFD-DEM numerical simulation results.
Figure 7: The variation in amplitude of particle velocity and pressure fluctuations of a standing wave in two-phase medium with respect to the height of the bed; amplitude of particle velocity fluctuation is shown on the left, where $A$ is the maximum amplitude and $H$ is height of the bed; oscillation in particle motion is illustrated in the middle; variation in the amplitude of pressure fluctuations is shown on the right [Roy et al. (1990)].

In order to study the interaction between the physical parameters, appropriate functions were fitted in the CFD-DEM numerical simulation results of pressure fluctuation and particle velocity fluctuation. From the fitted equations, it can be seen that the fluctuations in pressure and particle velocity all have the same form, i.e. a sinusoidal variation in time which is damped, multiplied by sinusoidal variation in space; therefore, we can assume a generic function for fluctuation variables as shown,

$$p'(y,t) = P_o e^{-\frac{t}{\tau}} \cos(c y) \cos(\omega t + \phi_p) \tag{16}$$

$$u'_p(y,t) = U_{p_o} e^{-\frac{t}{\tau}} \sin(c y) \cos(\omega t + \phi_u) \tag{17}$$

Here $P_o$, $U_{p_o}$ are the (initial) amplitudes of the pressure and particle velocity fluctuations, respectively; $\phi_p$ and $\phi_u$ are the temporal phase shifts, $\tau$ is the damping time constant, $c = \frac{\pi}{2h}$, where $h$ is the height of the fluidized medium in the bed, and $\omega$ is the angular frequency.
It was found from the fitted equations that the damping time and angular frequency of the oscillations in pressure fluctuation and particle velocity fluctuation are consistent; however, they are out of phase by $\pi/2$. Differences can also be noted in the amplitudes and the time phase angles in the fitted equations.

The values of fitted constants for CFD-DEM cases are given in Table 3.

<table>
<thead>
<tr>
<th>Test case</th>
<th>Number of particles</th>
<th>Height of the particles after fluidization – m</th>
<th>Pressure fluctuation amplitude $P_o$ – Pa</th>
<th>Particle velocity fluctuation amplitude $U_{po}$ – m/s</th>
<th>Damping time period $\tau$ – s</th>
<th>Angular frequency $\omega$ – rad/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>107800</td>
<td>0.049</td>
<td>3893</td>
<td>0.298</td>
<td>0.0055</td>
<td>640.5</td>
</tr>
<tr>
<td>2</td>
<td>215320</td>
<td>0.098</td>
<td>1962</td>
<td>0.153</td>
<td>0.0129</td>
<td>339.6</td>
</tr>
<tr>
<td>3</td>
<td>322480</td>
<td>0.147</td>
<td>1371</td>
<td>0.107</td>
<td>0.0211</td>
<td>222.4</td>
</tr>
<tr>
<td>4</td>
<td>430360</td>
<td>0.196</td>
<td>1025</td>
<td>0.0804</td>
<td>0.0397</td>
<td>167.1</td>
</tr>
</tbody>
</table>

5 Analytical study of waves in a fluidized medium

Taking a volume averaged view of the behavior of the fluidized bed, the system can be described by four equations: (1) the volume averaged fluid continuity equation, (2) the volume averaged fluid momentum equation, (3) the volume averaged dispersed phase continuity equation, and (4) the volume averaged dispersed phase momentum equation. These equations are discussed in (H. A. Khawaja, 2015), where (1) and (2) are used as part of the DEM simulation, with the volume averaged particle equations (3 and 4) replaced by a detailed Lagrangian simulation. Here, the volume averaged equations are linearized, and a phasor analysis is applied in an attempt to describe the behavior of the standing waves in the bed, seen both experimentally and in the CFD-DEM simulations. The system is taken to be essentially one-dimensional (i.e. the fluctuations exist in the vertical dimension only), and scaling analysis is used to simplify the equations to a tractable form.

Both the CFD-DEM numerical simulations and experiments (e.g. Roy et al. (1990)) have demonstrated that the presence of particles can alter the speed of sound waves in a two-phase medium. This was also highlighted by Roy et al. (1990) in their derivation of a theoretical
relationship for the speed of sound in a two-phase medium. They attributed this behavior to the fluidized phase having not only a large momentum (owing to the motion of the particles), but also a high compressibility (due to the gas). Therefore, any analysis must take into account the particle momentum equation in the y-direction (i.e. vertical). The volume averaged momentum equation for the particle phase (written here on a per particle basis rather than on the per unit volume basis, given by Jackson (2000)) is

\[
m_p \frac{\partial u_p}{\partial t} + m_p u_p \frac{\partial u_p}{\partial y} = -k_p (u_p - u_g) - v_p \frac{\partial p}{\partial y} + m_p g + f \tag{18}
\]

where \(m_p\) is the mass of the particle, \(u_p\) is the velocity of the particle, \(k_p\) is the coefficient of drag force from the fluid, which is a function of local voidage (HA Khawaja et al., 2012), \(v_p\) is the volume of the particle, \(p\) is the fluid pressure, \(g\) is the gravity constant and \(f\) is the net force arising from particle contacts. It should be noted that interaction forces here have been explicitly split into terms proportional to the difference between the particle and gas phases. The term \(f\) represents the force on the particles from the stresses in the solid phase arising from particle contacts, which, if written on a per unit volume basis, would be equal to the gradient of the solid phase stress tensor. On a per particle basis, this term can be re-written, and in one dimension, as \(v_p \nabla S_y\), where \(S_y\) is the y component of the particle stress tensor. One of the difficulties in solving the volume averaged equations lies in being able to specify closure relationships for this stress tensor. The simplest closure is used here, which is an analogous form of the stress tensor for the fluid, with a particle pressure, and an effective viscosity for the particle phase [Harris and Crighton (1994)], i.e.

\[
f = v_p \frac{\partial}{\partial y} \left( -p_p + \mu_p \frac{\partial u_p}{\partial y} \right) \tag{19}
\]

where \(\mu_p\) is the effective particle viscosity and \(p_p\) is the particle pressure. Harris and Crighton (1994) suggested the particle pressure could be modeled by,

\[
p_p = A \left( \frac{1 - \epsilon}{\epsilon - \epsilon_{cp}} \right) \tag{20}
\]
where $A$ is a constant, $\epsilon$ is the voidage (void fraction) and the subscript $cp$ denotes ‘close packing’. This form of equation ensures that the particle pressure becomes infinite when particles are closely packed and reduces to zero when the particles are fully separated. Thus, the particle momentum equation is taken here to be

$$m_p \frac{\partial u_p}{\partial t} + m_p u_p \frac{\partial u_p}{\partial y} = -k_p (u_p - u_g) - v_p \frac{\partial p}{\partial y} + m_p g$$

$$+ v_p \frac{\partial}{\partial y} \left( -p_p + \mu_p \frac{\partial u_p}{\partial y} \right)$$

This equation can be linearized by writing each of the variables ($u_p, u_g, p, \epsilon$) as the sum of the static (i.e. steady state) value, taken here to be at incipient fluidization, plus a small fluctuation. The resulting terms can be substituted in Equation (21). The resulting equations can be linearized and scaled, as discussed in H. A. Khawaja (2013). This analysis results in the correlation of speed of sound and damping time period, as shown in Equations (22) and (23),

$$u_s = \frac{p_o}{(1 - \epsilon) \rho_p u_{po}}$$

$$\tau = \frac{\rho_p}{\mu_p c^2}$$

The speed of sound is computed using Equation (22) and compared with those obtained from the CFD-DEM simulations and the theoretical expression given in Equation (2).

Table 4: Speed of sound for test cases given in Table 2; theoretical speed of sound values using Equation (2), CFD-DEM speed of sound values from fitted functions, and CFD-DEM speed of sound values using Equation (22)

<table>
<thead>
<tr>
<th>Test case</th>
<th>Theoretical speed of sound using Equation (2) - m/s</th>
<th>CFD-DEM speed of sound from fitted functions angular velocity - m/s (percentage difference from theoretical value)</th>
<th>CFD-DEM speed of sound using Equation (22) – m/s (percentage difference from theoretical value)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20.7</td>
<td>20.0 (3.4 %)</td>
<td>21.8 (5.3 %)</td>
</tr>
<tr>
<td>2</td>
<td>20.7</td>
<td>21.1 (1.9 %)</td>
<td>21.4 (3.4 %)</td>
</tr>
<tr>
<td>3</td>
<td>20.7</td>
<td>20.8 (0.5 %)</td>
<td>22.3 (7.7 %)</td>
</tr>
<tr>
<td>4</td>
<td>20.7</td>
<td>20.8 (0.5 %)</td>
<td>21.3 (2.9 %)</td>
</tr>
</tbody>
</table>
The results given in Table 4 show that Equation (22) agrees well with both the CFD-DEM fitted function values and the theoretical expression (Equation (2)). It is also observed that the value of $\frac{R_0}{u_{po}}$ is constant. This can be justified by combining Equation (22) with Equation (2). Equation (23) shows that the damping time is a function of the height of the bed, density of particles and bulk particle viscosity. This correlation was used to compute particle viscosity in a fluidized medium for both the simulations here and also in the experiments of Roy et al. (1990). The results are shown in Table 6.

The damping time results of the CFD-DEM test cases are compared using the theoretical relationship given in Roy et al. (1990), as shown in Table 5.

<table>
<thead>
<tr>
<th>Test case</th>
<th>Height of the bed - mm</th>
<th>Theoretical damping time from Equation (3) - s</th>
<th>CFD-DEM fitted function Damping t - s</th>
<th>Percentage difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>49</td>
<td>0.0056</td>
<td>0.0035</td>
<td>37.5 %</td>
</tr>
<tr>
<td>2</td>
<td>98</td>
<td>0.0200</td>
<td>0.0129</td>
<td>35.5 %</td>
</tr>
<tr>
<td>3</td>
<td>147</td>
<td>0.0467</td>
<td>0.0211</td>
<td>54.8 %</td>
</tr>
<tr>
<td>4</td>
<td>196</td>
<td>0.0827</td>
<td>0.0397</td>
<td>52.0 %</td>
</tr>
</tbody>
</table>

The comparison from Table 5 shows a significant difference between the theoretical damping time and the damping time computed via CFD-DEM numerical simulations. Similarly, significant difference is found when this relationship is used against damping time data provided in Roy et al. (1990). The reason that this correlation did not prove to be effective is the fact that the effect in damping due to the particles’ contacts was not taken into account in Roy et al. (1990). In contrast, the expression given in Equation (23) includes an effective particle viscosity, which takes into account the damping effect due to particle contacts. Therefore, it is proposed that the damping in a two-phase medium such as a fluidized bed is mainly due to the particles’ contacts; however, further study is required in this area.
Table 6: The value of particle viscosity from experimental (Roy et al. (1990)) and CFD-DEM results; $U_{mf}$ is the minimum fluidization velocity and $u_s$ is the speed of sound in the fluidized medium.

<table>
<thead>
<tr>
<th>Catalyst; dia. = 70µm, particle density = 1250 kg/m$^3$, $U_{mf} = 0.01$ m/s, $u_s = 15.4$ m/s</th>
<th>Heights (m)</th>
<th>Time period (s)</th>
<th>Damping time period (s)</th>
<th>Particle dynamic viscosity (Pa s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0.11</td>
<td>0.06</td>
<td>1350.9</td>
<td></td>
</tr>
<tr>
<td>0.6</td>
<td>0.15</td>
<td>0.1</td>
<td>1823.8</td>
<td></td>
</tr>
<tr>
<td>0.8</td>
<td>0.21</td>
<td>0.25</td>
<td>1296.9</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.26</td>
<td>0.35</td>
<td>1447.4</td>
<td></td>
</tr>
<tr>
<td>1.2</td>
<td>0.3</td>
<td>0.43</td>
<td>1696.5</td>
<td></td>
</tr>
<tr>
<td>1.4</td>
<td>0.36</td>
<td>0.55</td>
<td>1805.4</td>
<td></td>
</tr>
<tr>
<td>1.6</td>
<td>0.43</td>
<td>0.56</td>
<td>2315.9</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Glass beads; dia. = 100µm, particle density = 2900 kg/m$^3$, $U_{mf} = 0.05$ m/s, $u_s =$11.0 m/s</th>
<th>Heights (m)</th>
<th>Time period (s)</th>
<th>Damping time period (s)</th>
<th>Particle dynamic viscosity (Pa s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0.15</td>
<td>0.08</td>
<td>2350.7</td>
<td></td>
</tr>
<tr>
<td>0.6</td>
<td>0.24</td>
<td>0.17</td>
<td>2488.9</td>
<td></td>
</tr>
<tr>
<td>0.8</td>
<td>0.29</td>
<td>0.23</td>
<td>3270.5</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.38</td>
<td>0.24</td>
<td>4897.2</td>
<td></td>
</tr>
<tr>
<td>1.2</td>
<td>0.44</td>
<td>0.29</td>
<td>5836.1</td>
<td></td>
</tr>
<tr>
<td>1.4</td>
<td>0.48</td>
<td>0.39</td>
<td>5906.8</td>
<td></td>
</tr>
<tr>
<td>1.6</td>
<td>0.53</td>
<td>0.41</td>
<td>7338.6</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Vermiculite; dia. = 220µm, particle density = 384 kg/m$^3$, $U_{mf} = 0.025$ m/s, $u_s =$23.3 m/s</th>
<th>Heights (m)</th>
<th>Time period (s)</th>
<th>Damping time period (s)</th>
<th>Particle dynamic viscosity (Pa s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4</td>
<td>0.07</td>
<td>0.05</td>
<td>498.0</td>
<td></td>
</tr>
<tr>
<td>0.6</td>
<td>0.1</td>
<td>0.08</td>
<td>700.3</td>
<td></td>
</tr>
<tr>
<td>0.8</td>
<td>0.14</td>
<td>0.1</td>
<td>996.0</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0.17</td>
<td>0.13</td>
<td>1197.1</td>
<td></td>
</tr>
<tr>
<td>1.2</td>
<td>0.21</td>
<td>0.16</td>
<td>1400.7</td>
<td></td>
</tr>
<tr>
<td>1.4</td>
<td>0.23</td>
<td>0.21</td>
<td>1452.5</td>
<td></td>
</tr>
<tr>
<td>1.6</td>
<td>0.28</td>
<td>0.24</td>
<td>1660.0</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>CFD-DEM; dia. = 150µm, particle density = 1000 kg/m$^3$, $U_{mf} = 0.0085$ m/s, $u_s =$20.7 m/s</th>
<th>Heights (m)</th>
<th>Time period (s)</th>
<th>Damping time period (s)</th>
<th>Particle dynamic viscosity (Pa s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.049</td>
<td>0.00981</td>
<td>0.0055</td>
<td>278.0</td>
<td></td>
</tr>
<tr>
<td>0.098</td>
<td>0.0185</td>
<td>0.0129</td>
<td>301.7</td>
<td></td>
</tr>
<tr>
<td>0.147</td>
<td>0.02825</td>
<td>0.0211</td>
<td>415.1</td>
<td></td>
</tr>
<tr>
<td>0.196</td>
<td>0.0376</td>
<td>0.0397</td>
<td>503.9</td>
<td></td>
</tr>
</tbody>
</table>
The computed particle viscosities are much higher than those reported by Hagyard and Sacerdote (1966). However, Hagyard and Sacerdote (1966) showed that the particle viscosity increases asymptotically when close to minimum fluidization; therefore, the found values are reasonable, considering that the tests were performed very close to minimum fluidization. Particle viscosity is because of particles contacts hence its value rises as hydrostatic pressure rises in the fluidized bed.

### 6 Conclusions

In this work, sound waves were studied in a fluidized medium using CFD-DEM simulations. The following conclusions can be drawn from this study:

- The theoretical relationship for speed of sound in a two-phase medium given by Roy et al. (1990) was validated by the CFD-DEM numerical simulations.
- The linearized equations were used to show that the speed of sound in a two-phase medium can be linked to physical properties of the particles and the amplitudes of fluctuations in pressure and particle velocity. Since the speed of sound in a two-phase medium is constant (Roy et al. (1990)), it was also shown that the ratio of the amplitude of the fluctuations in pressure and particle velocity is also constant. This was also observed in CFD-DEM simulations.
- The most significant effect in terms of damping was the particle viscous term. Previous work by Roy et al. (1990) had neglected this effect, with the consequence that they were not able to describe the damping accurately. Using the expressions derived from the linear analysis, it was possible to compute the particle dynamic viscosity for the experiments from Roy et al. (1990) and CFD-DEM test cases.

### 7 Acknowledgements

The author acknowledges the support of Dr. Stuart A. Scott, Department of Engineering, University of Cambridge, UK and Prof. John F. Davidson, Department of Chemical Engineering, University of Cambridge, UK.
List of references


