A CFD study on the effect of bed thickness in an internally circulating fluidized bed

Q. Zhou,1 Y.Q. Feng,2 J.W. Wang,1 P.J. Witt,2 C. Doblin,3 S. Lim3 and M.P. Schwarz2
1Institute of Process Engineering, Chinese Academy of Science, Beijing, China
2CSIRO Mathematics, Informatics and Statistics, Melbourne, Australia
3CSIRO Process Science and Engineering, Melbourne, Australia

Keywords: computational fluid dynamics, two fluid model, gas-solid flow, internally circulating fluidized bed, solid recirculation rate

Abstract—This paper presents a comparison between two dimensional (2-D) and three dimensional (3-D) computational fluid dynamics (CFD) model predictions of an internally circulation fluidized bed (ICFB). A two fluid model (TFM) based on the kinetic theory of granular flow is used with the commercial CFD software package, ANSYS Fluent. Simulations were conducted for a 2-D geometry and several 3-D geometries at various bed thicknesses. Their differences were quantified in terms of solids recirculation rate at difference superficial velocities and explained in terms of gas-solid flow dynamics.

INTRODUCTION

Gas fluidization technology has been widely used in industry with applications ranging from its original application in catalytic cracking of oil to various process industries.1 The gas-solid flow in gas fluidized beds demonstrates complex behaviour, which is particularly so at an industrial scale where different types of fluidization equipment co-exists. The study of fluidization technology has been an area of intensive research for decades, but is still very active to meet the increasing demands for improving process efficiency of existing equipment and to apply it to new processes.2 Detailed investigation of gas-solids flow in industrial scale equipment is very difficult, if not impossible, because they are often operated at high temperatures, and mostly involved chemical reactions and/or combustion. Furthermore, the ability to trial unusual operating conditions is constrained by the need to maintain control over the unit operation. Physical and numerical models provide the ability to trial changes to operating conditions and geometrical configurations without risk, hence are often used to gain a better understanding of the process for achieving an optimized design and operating conditions of the process unit.

Physical modeling traditionally played a major role in developing a better understanding of the process. This was achieved with the application of advanced measurement techniques, such as positron emission particle tracking (PEPT)3, particle image velocimetry (PIV)4, and radiotracer particles5. Physical modeling is often conducted using cold models. Such cold model data can not be directly used for designing plant scale operation, as cold models often use different solid and gas properties from the hot system. Pilot scale trials are often an intermediate step before a commercial scale unit can be built, but scale-up problems can still be experienced.

Computational fluid dynamics (CFD) modeling can account for these complexities and use the actual gas and solid properties as well as include chemical reactions. Over the past two decades, with advances in computing speed, parallelization technology, improved software
and multiphase algorithms, CFD has progressed substantially to a point where it can be used for prediction of complex gas-solid flows such as those encountered in industrial fluidization systems. By using CFD models, a wide range of variations in physical design and operational parameters can be tested and refined until a design that gives optimum performance is identified. Today, CFD models play an increasingly important role in process design, control and/or optimization in complex multiphase flow systems. Depending on the application and information required, gas-solid flow can be modeled at different length scales: at the particle scale or at the macro level by local averaging. The particle scale approach tracks the motion of individual particles using a discrete element method (DEM), and the flow of gas using a continuum based CFD model with local properties averaged over a number of computational cells. This approach results in what is commonly referred to as a DEM-CFD model in the literature. The macro level approach treats both solid and gas phase as interpenetrating continua, the motion of each phase is solved using a continuum based CFD model with suitable closure terms. This method is known as the Eulerian-Eulerian model. Both models are widely used to study the behaviour of various gas-solid flows and are areas of ongoing research at CSIRO.

Often there are tight timelines for the design of industrial fluidization systems making full three dimensional (3-D) simulations of large numbers of potential designs and operating parameters impractical. To reduce the time and computing costs, numerical investigation of a wide range of design and/or operating parameters are often conducted using two-dimensional (2-D) geometry before 3-D simulations of the final design is often performed. However, 2-D simulations may not fully represent the real system, which is actually 3-D. This uncertainty also applies to 2-D physical experiments where the bed thickness, or the distance between front and back bounding walls, is small for the convenience of construction, operation and analysis. There is a risk that such 2-D data is directly used in practical applications without appreciation of its limitations.

The difference between 2-D and 3-D models has been a topic of research in the past. Saxena and Vadivel compared the wall effect on incipient fluidization in a gas-fluidized bed. The minimum fluidization velocity \( (U_m) \) increases as the bed thickness reduces, and the bed thickness effect varies based on particle diameters. Liu et al drew a similar conclusion using a micro fluidized bed. Sanchez-Delgado et al proposed a correlation for the ratio of \( U_m \) between 2-D and 3-D beds as a function of the mean particle size and bed thickness. The wall effects on \( U_m \) can be neglected when the bed thickness to particle size ratio is larger than 100. Using an Eulerian-Eulerian CFD model, Chalermsinsuwan et al compared the bed expansion ratio, solid volume fraction, granular temperature and dispersion coefficients of FCC particles in a thin bubbling fluidized bed with 2-D and 3-D computational domains. The granular temperatures obtained from the 3-D simulation were slightly lower than that from the 2-D simulation. Using a DEM-CFD model, Feng and Yu compared the segregation behaviour of bi-sized particles in a gas fluidized bed. They found that 2-D and 3-D simulations can produce results that are qualitatively similar. However, due to the effect of walls, the 2-D models cannot fully reproduce the experimental observations in areas such as the velocity ranges corresponding to segregation, the particle kinetics and particle contacts. Therefore, 3-D modelling was found necessary to produce quantitative results that can be used in an engineering application.

Recently, an internally circulation fluidized bed (ICFB) was designed for a reacting process at CSIRO. Both physical and numerical modelling are adopted to gain a better understanding of the process. In numerical modeling, a 2-D Eulerian-Eulerian model was used to investigate the likely effect on solid recirculation rate of design and/or operating parameters such as the gas injection velocity, gas distributor inclination angle, the presence of a tube bundle. It is now of interest to know how closely the 2-D simulations are to 3-D simulations of the real geometry. This paper aims to clarify this issue. In addition, the 3-D simulations were conducted at different bed thicknesses to check the effect of bed thickness. Their differences were measured in terms of solid recirculation rate and explained in terms of gas and solid flow dynamics.
Model description

The study is based on an Eulerian model, which describes both solid and gas phases as interpenetrating continua. The motion of each phase is solved using a continuum based CFD model with suitable closure terms. The governing equations are conservation of mass and momentum for each phase, which are given as:

**Phase mass conservation equations**

\[
\frac{\partial (\alpha_g \rho_g)}{\partial t} + \nabla \cdot \left( \alpha_g \rho_g \mathbf{u}_g \right) = 0
\]  

(1)

\[
\frac{\partial (\alpha_s \rho_s)}{\partial t} + \nabla \cdot \left( \alpha_s \rho_s \mathbf{u}_s \right) = 0
\]  

(2)

with \( \alpha_g + \alpha_s = 1 \).

**Momentum conservation equations**

\[
\frac{\partial (\alpha_g \rho_g \mathbf{u}_g)}{\partial t} + \nabla \cdot \left[ \alpha_g \rho_g \mathbf{u}_g \otimes \mathbf{u}_g - \alpha_g \mu_g \left( \nabla \mathbf{u}_g + \left( \nabla \mathbf{u}_g \right)^T \right) \right] = -\alpha_g \nabla p + \beta (\mathbf{u}_g - \mathbf{u}) + \alpha_g \rho_g \mathbf{g}
\]

(3)

\[
\frac{\partial (\alpha_s \rho_s \mathbf{u}_s)}{\partial t} + \nabla \cdot \left[ \alpha_s \rho_s \mathbf{u}_s \otimes \mathbf{u}_s \right] = -\alpha_s \nabla p - \beta (\mathbf{u}_g - \mathbf{u}) + \alpha_s \rho_s \mathbf{g} + \nabla \sigma_s,
\]

(4)

where \( \alpha, \rho, \mathbf{u}, \mu, t \) stand for volume fraction, density, velocity vector, viscosity and time respectively. \( P \) and \( \beta \) are, respectively, pressure and inter-phase momentum transfer coefficient. \( \sigma \) is solids stress tensor and \( \mathbf{g} \) is gravitational acceleration. The subscripts \( g \) and \( s \) stands for gas and solid phase respectively.

Equation (3) is actually an extension of the Navier-Stokes equation for a single gas phase to a multiphase system, with an extra term to consider the momentum exchange between the gas and solid phases. For the solid phase, as described in equation (4), a similar formula to the gas phase is used, but the closure model to describe the solid stress term (includes solid pressure and solid stress tensor) is not straight forward. Borrowing the concept of the thermodynamic temperature for gases, the kinetic theory of granular flow is used for these closures, which is the most popular and commonly accepted approach. The governing equation for granular kinetic energy is expressed as:

**Granular kinetic energy conservation equation**

\[
\frac{3}{2} \frac{\partial (\alpha_g \rho_g \theta)}{\partial t} + \nabla \cdot \left( \alpha_g \rho_g \mathbf{u}_g \theta \right) = \kappa \cdot \nabla \theta - \kappa \nabla \cdot \left( \frac{1}{\kappa} \nabla \theta \right) - \gamma
\]

(5)

In equation (5), \( \theta \) stands for granular temperature, \( \kappa \) for granular conductivity and \( \gamma \) is the collisional dissipation of solid fluctuating kinetic energy.

Gidaspow described the kinetic theory of granular flow and how it is applied to the solids phase to enable calculation of solids phase pressure and viscosity. A summary of the sub-models used in this work and additional equations required to close the governing equations are given in Table 1. Values used for parameters in the models are: maximum solids volume fraction \( (\alpha_{s,max}) \) is set to 0.63, a value of 0.9 is used for the restitution coefficient \( (\varepsilon) \) and for the frictional stress model an internal friction angle \( (\phi) \) of 30\(^\circ\) is used.

ANSYS/Fluent is used to solve the model equations (1) to (5) by a finite volume method using the “phase couple SIMPLE” algorithm to handle pressure-velocity and phase coupling. A second order discretisation scheme is used for convection terms in the momentum equations while the QUICK scheme is used for the volume fraction equations. The tangential contact between particles and wall is treated as half-slip boundary condition. A time step of 0.001 seconds is used to advance the solution in time, with a second order implicit time integration scheme. The model is well documented in the literature and fully implemented in ANSYS/Fluent and has been widely used in fluidized bed studies, readers wanting further details of the numerical model are referred to the various references listed.
Table 1. Constitutive relationships for Eulerian-Eulerian model for gas-solid flow

<table>
<thead>
<tr>
<th>Constitutive relationships</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inter-phase momentum transfer</td>
</tr>
<tr>
<td>( \beta = 150 \left( 1 - \alpha_g \right) \frac{\mu_g}{d_p} + 1.75 \frac{d_p}{d_p} \left( 1 - \alpha_g \right) ) for ( \alpha_g &lt; 0.8 )</td>
</tr>
<tr>
<td>( \beta = \frac{3}{4} C_d \alpha_g \rho_u \left( \mathbf{u}_s - \mathbf{u}_g \right) \left( 1 - \alpha_g \right) \alpha_g^{2.65} ) for ( \alpha_g \geq 0.8 )</td>
</tr>
</tbody>
</table>
| Where \( C_d = \begin{cases} 
\frac{24 \left( 1 + 0.15 \text{Re}^{0.67} \right)}{\text{Re}} & \text{Re} \leq 1000 \\
0.44 & \text{Re} > 1000
\end{cases} \) |
| Re = \( \frac{\rho_s \left( \mathbf{u}_s - \mathbf{u}_g \right) \alpha_g d_p}{\mu_s} \) |
| Solids stress \( \sigma_s = -P_s + \zeta \nabla \cdot \mathbf{u}_s + 2 \mu_s S_s \) |
| Solids pressure \( P_s = \rho_s \alpha_s \left[ 1 + 2 \left( 1 + e \right) \alpha_s g_0 \right] + P_c \) |
| Solids shear viscosity \( \mu_s = \mu_{s,c} + \mu_{s,h} + \mu_{s,f} \) |
| \( \mu_{s,c} = \frac{4}{5} \zeta \alpha_s^2 \rho_u d_p g_0 \left[ 1 + \left( \frac{\theta}{\pi} \right)^{2} \right] \) |
| \( \mu_{s,h} = \frac{\alpha_s \rho_u d_p \sqrt{\theta \pi}}{6 \left( 3 - e \right)} \left[ 1 + \frac{3}{2} \left( 1 + e \right) \left( 3 e - 1 \right) \alpha_s g_0 \right] \) |
| \( \mu_{s,f} = \frac{P_c \sin \phi}{2 \sqrt{2} \left( S_s - S_s \right)} \) |
| Solids bulk viscosity \( \zeta_s = \frac{4}{3} \zeta \alpha_s^2 \rho_u d_p g_0 \left[ 1 + \left( \frac{\theta}{\pi} \right)^{2} \right] \) |
| Granular conductivity \( \kappa = \frac{15 \alpha_s \rho_u d_p \sqrt{\pi \theta}}{4 \left( 41 - 33 \eta \right)} \left[ 1 + \frac{12}{5} \eta \left( 4 \eta - 3 \right) \alpha_s g_0 + \frac{16}{15 \pi} \left( 41 - 33 \eta \right) \eta \alpha_s g_0 \right] \) |
| where \( \eta = \frac{1}{2} \left( 1 + e \right) \) |
| Collision energy dissipation \( \gamma = \frac{12 \left[ 1 - e^2 \right] \rho_s g_0 \alpha_s^2 \theta^{3/2}}{d_p \sqrt{\pi}} \) |
| Radial distribution function \( g_0 = \left[ 1 - \left( \frac{\alpha_s}{\alpha_{s,max}} \right)^{1/5} \right]^{-1} \) |

Simulation conditions

The investigation is based on an ICFB geometry which was recently used to investigate some design and operating parameters on solid recirculation rate \( \text{Re} \). Figure 1 shows the 2-D geometry used in the simulations. The bed has two inter-connected fluidized bed chambers, with the left chamber representing the “reaction chamber” (RX), and right chamber representing the heat exchanger chamber (HEX). The two chambers are separated by a vertical baffle with a slot in the lower section to enable the return of solids from the HEX to RX. The slot height is 25 mm and the height of the top of the baffle from the distributor is 275 mm. The baffle thickness is 8 mm. For the 3-D simulation, the geometry is simply extruded in the third direction perpendicular to the front and back bounding walls.

Table 2 lists the properties of gas and solid particles used in this study. Table 3 summarizes the simulation conditions. For the comparison between 2-D and 3-D, bed thickness of 3-D simulation is set to 150 mm, the \( U_m \) is fixed at 0.12 ms\(^{-1}\), four simulations were performed with different \( U_f \) that are 0.24, 0.30, 0.40 and 0.60 ms\(^{-1}\) respectively. For the comparison of bed thickness effect, four bed thicknesses are used, being 10 mm, 20 mm, 50 mm and 150 mm.
respectively. One set of superficial velocities is used for different bed thicknesses, with $U_f$ being 0.60 ms$^{-1}$ and $U_m$ 0.12 ms$^{-1}$.

Figure 1. Schematic diagram of the 2-D ICFB geometry

Table 2. Gas and solid properties

| Solid density | 2760 kg/m$^3$ | Gas density | 0.585 kg/m$^3$ |
| Solid diameter | 0.0003 m | Gas viscosity | 5.12x10$^{-5}$ Pa·s |
| $U_{mf}$ | 0.028 m/s | Initial gas porosity | 0.40 |

Table 3. Simulation conditions

<table>
<thead>
<tr>
<th></th>
<th>$U_f$ (m/s)</th>
<th>$U_m$ (m/s)</th>
<th>Initial solid packing height (mm)</th>
<th>Bed thickness for 3-D geometry (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D versus 3-D</td>
<td>0.24</td>
<td>0.12</td>
<td>275</td>
<td>150</td>
</tr>
<tr>
<td>Effect of bed thickness</td>
<td>0.60</td>
<td>0.12</td>
<td>275</td>
<td>10, 20, 50, 150</td>
</tr>
</tbody>
</table>

RESULTS AND DISCUSSION

Solid recirculation rate

One important concern in ICFB design and operation is the solid recirculation rate, which controls the heat exchange between the two chambers and the rate of reaction in RX chamber in some cases. Figure 2 shows the variation of solid recirculation rate for both 2-D and 3-D simulations when the $U_f$ is 0.60 ms$^{-1}$ and $U_m$ is 0.12 m/s. The solid recirculation rate is calculated as the total solid passing through the slot divided by horizontal cross area of the HEX chamber. Positive values signify normal solid recirculation behaviour with solids passing through the slot from the HEX chamber to the RX chamber, and are returned back to the HEX chamber above the baffle. Right after the introduction of gas, a reverse recirculation occur for a short period before normal recirculation. The solid flux varies strongly with time, but it mainly fluctuates around a certain value once a dynamically stable state is reached, usually shortly
after gas injection commences. Occasionally, reverse recirculation occurs in 2-D simulations. Note that in 3-D simulation, the averaged value over bed thickness direction is used and thus a net reverse circulation cannot be observed. Results in Figure 2 show that the 2-D model predicts larger fluctuation in solids flux than is predicted by the 3-D model. In this work we have chosen to use the mean value of solid recirculation rate as the parameter to quantitatively compare the difference between 2-D and 3-D simulations.

Figure 2. Comparison between 2-D and 3-D simulation on solid recirculation rate when the $U_f$ is 0.6 m/s and $U_m$ is 0.12 m/s.

The difference in mean solid recirculation rate between 2-D and 3-D simulations is shown in Figure 3, where the solid flux is plotted as a function of different ratios of $U_f$ to $U_m$, expressed as $(U_f / U_{mf})/(U_m / U_{mf})$. A time-averaged value of the last 15 seconds of the total 20 seconds of simulation time is used for this analysis. The solid flux increases as the ratio increases for both 2-D and 3-D simulations. At any given ratio, the 3-D simulation predicts a solid recirculation that is about 30% lower than that of 2-D simulation.

Figure 4 plots the mean solid recirculation rate for different bed thicknesses in the 3-D model. The solid recirculation rate for the narrow, 10 mm thick, bed is substantially lower than the other three bed thicknesses modeled. For the 20-mm and thicker beds solids recirculation rate decreases slightly bed thickness is increased. When the bed is too narrow, the front and back bounding wall constrains the gas and solid flow due to wall friction, which is probably the main reason for low recirculation rate in the narrowest bed (i.e. 10 mm). As the bed thickness increases, there is less effect of wall friction, thus the solid recirculation rate is higher. Additionally in thicker beds bubbles are able to develop more complex 3-D shapes and behavior but this effect does not further increase the recirculation rate. A detailed understanding of the resulting gas and solid dynamics is required to give a better explain of this trend.

Gas pressure

The driving force for the solid recirculation is the difference in pressure between the HEX and RX chamber, this was analysed in detail in our previous work. Here only the averaged value is plotted to compare the difference between 2-D and 3-D simulations. Figure 5 shows the pressure drop over the HEX and RX chambers at different $U_f$. For both 2-D and 3-D simulations, as the $U_f$ increases, the pressure drop increases in the HEX chamber and decreases in the RX chamber. The pressure difference between the HEX and RX chamber in 2-D simulations is larger than that in 3-D simulations.

The relation between solid recirculation rate and pressure difference is plotted in Figure 6. The relation between pressure difference and solid recirculation rate is almost linear. For the same pressure difference between the HEX and RX chambers the 2-D simulations predict higher solid recirculation rates than the 3-D models do. It is interesting to know whether the contribution of each unit of pressure difference between the HEX and RX has an influence on solids circulation rate. Figure 7 plots the solid recirculation rate divided by pressure difference.
at different $U_f$. As the $U_f$ increases, the contribution of each unit of pressure difference to the recirculation rates decreases. Interestingly for a given $U_f$, the contribution of each unit pressure difference to the recirculation is almost the same for the 2-D and 3-D models. Pressure is generally a function of solid holdup; hence this would indicate that the reason for different recirculation rates between the 2-D and 3-D models is due to different solid holdup being predicted by the 2-D and 3-D models.

Figure 3. Comparison between 2-D and 3-D simulation on solid recirculation rate at different $U_f$ when the $U_{in}$ is 0.12 m/s.

Figure 4. Effect of bed thickness on solid recirculation rate for the simulation parameters: $U_f = 0.60$ m/s, $U_{in} = 0.12$ m/s.
Figure 5. Gas pressure in the HEX and RX chamber at different $U_f$.

Figure 6. Solid recirculation rate versus pressure difference between the HEX and RX chamber.

Figure 7. Contribution of per unit pressure difference between the HEX and RX chambers to solid recirculation rate at different $U_f$. 
Solid flow patterns
To further investigate the difference in solids recirculation rate between 2-D and 3-D simulations, the solids flow patterns are investigated by plotting the instantaneous spatial distributions of solid volume fraction in Figure 8. For the 3-D cases, the plane plotted is at the midpoint across the bed thickness. As shown in Figure 8, the flow demonstrates bubbling fluidization behaviour. In both 2-D and 3-D cases fluidization is much stronger in the RX chamber than that in HEX chamber, as is demonstrated by bubble size and bed expansion height. Generally, for a given $U_f$, there is greater solids holdup below the baffle height in 3-D simulations (Figure 8b) than there is in the 2-D simulations (Figure 8a). From these results it is hard to quantify the difference between the models and cases because only one time frame for each case is plotted and the flow exhibits strong transient behaviour.

For a fixed bubble size, the ratio of the bubble surface area to bubble diameter is different between 2-D and 3-D simulations. In a 2-D bed this leads to higher bed expansion. Consequently, there will be less solids holdup below the baffle height, thus, more solids above the baffle to promote solids recirculation in 2-D simulation. This may explain why there is a larger solids recirculation rate in 2-D simulations than in 3-D simulations.

The effect of bed thickness is also analysed by investigating the solids distribution. The bed thickness can affect the solids flow in two ways. Firstly when the bed is narrow, the front and back bounding walls exert a strong wall friction force to the particles near the walls, which impedes solids and hence bubble motion. Secondly in very narrow beds the bubbles are constrained to have effectively a 2-D shape as the bed thickness is of the order of bubble diameter. When bed thickness increases, the wall effect reduces and also bubbles are able develop 3-D shapes and 3D flow fields. Figure 9 shows the solid concentration at one time instant for each of the different 3-D models with different bed thicknesses. Again, it is quite difficult to identify a big difference just based on visual observations of a single frame of the distribution, as the gas and solids flow changes rapidly with time.

To confirm and quantify these effects, further analysis based on time and spatial averaging as well as their statistical distribution in both the HEX and RX chambers is required. This represents part of our on-going effort to fully clarify this issue.

CONCLUSIONS
By applying an Eulerian-Eulerian CFD model and using an internally circulation fluidized bed as the testing geometry, the effect that 2-D versus 3-D model setups and different bed thicknesses in a 3-D model have on gas-solids flow are evaluated.

2-D simulations predicted a 30% higher solids recirculation rate than the 3-D simulations did for the investigated conditions. The effect of bed thickness is significant when the bed thickness is small. For a very narrow 3-D bed the recirculation rate was significantly lower than for the thicker beds and is likely to be as a result of significant wall effects. For wider beds the recirculation rate decreased slightly as the bed thickness increased, as a combined effect of reduced wall effect and the development of 3-D bubbling phenomena.

There is almost a linear correlation between the solid recirculation rate and pressure difference between the RX and HEX chambers. The contribution to solids recirculation rate per unit pressure difference is almost identical for both the 2-D and 3-D simulations. With the contribution decreasing as $U_f$ is increased.

An initial investigation of solid flow patterns based on a single time instant cannot provide an explanation for difference in recirculation rates predicted by the 2-D and 3-D simulation domains. All simulated cases qualitatively capture the key flow phenomena, such as bubbling fluidization behaviour, and stronger fluidization in the RX chamber than in the HEX chamber. Further analysis of solid flow patterns, by time and spatial averaging of the results and their statistical distribution in the HEX and RX chambers, as well as solid mixing kinetics, is necessary to clarify this issue in detail.
Figure 8. Snapshots of solid volume fraction distribution after 20 seconds of simulation time at different gas superficial velocities on the RX side.
Figure 9. Snapshots of solid volume fraction distribution after 20 seconds of simulation time for the 3-D model at different bed thicknesses: (a) 10 mm; (b) 20 mm; (c) 50 mm and (d) 150 mm.

ACKNOWLEDGMENTS

The work is financially supported by Ministry of Science and Technology of China through the China-Australia Special Fund with the grant No. 2007DFA41320 and the startup fund awarded to Junwu Wang from the “hundred talents program” of Institute of Process Engineering, Chinese Academy of Sciences.

Dr Feng is grateful for the international travel grant from the Australia Academy of Science through the International Scientific Linkage program.

Support of CSIRO’s Light Metals Flagship is also acknowledged.

NOMENCLATURE

\[ d_p \] particle diameter
\[ e \] coefficient of restitution
\[ g_0 \] radial distribution function
\[ g \] gravity vector
\[ I \] identity tensor
\[ p \] pressure
\[ P_c \] critical solids pressure
\[ P_s \] solids pressure
\[ Re \] particle Reynolds number
\[ t \] time
\[ u \] velocity vector
\[ \alpha \] volume fraction
\[ \alpha_{s,max} \] maximum solids volume fraction
\[ \beta \] interphase momentum transfer coefficient
\[ \gamma \] collisional dissipation of solid fluctuating kinetic energy.
\[ \phi \] internal friction angle
\[ \mu \] shear viscosity
\[ \kappa \] granular conductivity
\[ \theta \] granular temperature
\[ \rho \] density
\[ \sigma_s \] solids stress tensor
\[ \zeta \] bulk viscosity
subscripts

\text{g} \quad \text{gas phase}

\text{s} \quad \text{solids phase}

\text{s,c} \quad \text{solids collision}

\text{s,k} \quad \text{solids kinetic}

\text{s,f} \quad \text{solids frictional}

REFERENCES


24. ANSYS Fluent 12.1 User manual, ANSYS Inc.