Study of turbulent two-phase gas-solid flow in horizontal channels

S Lain*

Fluid Mechanics Group, Energetics and Mechanics Department, Universidad Autonoma de Occidente, Santiago de Cali, Colombia

Received 3 February 2012; accepted 29 November 2012

The present study deals with pneumatic conveying of spherical particles in a six meter long horizontal channel with rectangular cross-section from a numerical perspective. Calculations are done for spherical glass beads of different sizes with a mass loading of 1.0 kg particles/kg gas. Additionally, different wall roughnesses are considered. Air volume flow rate is kept constant in order to maintain a fixed gas average velocity of 20 m/s. The numerical computations are performed by the Euler/Lagrange approach in connection with a Reynolds stress turbulence model accounting for two-way coupling and inter-particle collisions. For the calculation of the particle motion, all relevant forces (drag, slip-shear and slip-rotational lift and gravity), inter-particle collisions and particle-rough wall collisions are considered. The agreement of the computations with the findings or earlier experiments are found to be satisfactory for mean and fluctuating velocities of both phases as well as for the normalized particle mass flux.

Keywords: Euler-Lagrange approach, Gas-solid flow, Inter-particle collisions, Pneumatic conveying, Wall roughness

Pneumatic conveying of solid particles in channel or pipe flows is of great technical importance and therefore found in numerous areas of process engineering such as chemical, pharmaceutical, food, power generation (conveying of pulverised coal) and air conditioning technology. The size of the conveyed particles typically ranges from a few micrometers to several millimetres. The wide application of pneumatic conveying is connected to numerous advantages for powder transport such as dust free transportation, high flexibility and low cost as well as maintenance. Similar wall-bounded two-phase flow systems are found in cyclone separators, classifiers and dilute fluidised beds. Due to the presence of the confinement in these systems, collisions of solid particles with the walls play an essential role in the particle transport process. The wall collision frequency is directly responsible for the additional pressure drop due to the solids as a result of the momentum and energy loss involved in the deformation process. Additionally, wall roughness considerably affects the wall collision process and causes an enhancement of the wall collision frequency in pipes or channels for inertial particles. On the other hand, the wall collision frequency might also be reduced for smaller particles (τp < 50 ms) as a consequence of wall roughness.

Particle inertia and the effect of gravity have a pronounced effect on powder conveying and will cause a segregation of the mixture, whereby rather dense ropes of solids may be formed near the bottom of horizontal conveying lines. The increasing local solids concentration supports the occurrence of inter-particle collisions. This again results in a dispersion of particles out of a dense rope due to the transfer of momentum from the main stream direction to the transverse component. Hence, collisions between particles may cause a destruction of dense ropes and therefore also will have an effect on particle transport, wall collision frequency and pressure drop in such two-phase systems.

For the numerical treatment of two or multiple interpenetrating phases, two approaches are commonly used, namely the two-fluid (or often also referred to as Euler/Euler) approach and the hybrid Euler/Lagrange approach. The first approach is based on the assumption of two or more interacting continua and therefore results in similar conservation equations for the continuous and dispersed phase, which can be solved with the same discretisation schemes. Therefore, this method is preferred by industry, since the considered problem can be numerically solved rather fast. The drawback of the continuum assumption for the dispersed phase, however, is the sophisticated modelling of the relevant elementary processes affecting particle motion and behaviour. Additionally, the consideration of a particle size distribution, which is inherent for many processes in powder technology, requires the solution of a set of conservation equations, one for each particle size fraction, by accounting for their interaction through collisions. The Euler/Lagrange approach is only
applicable to dispersed two-phase flows and is based on a coupled computation of fluid flow (Eulerian) and particle phase (Lagrangian). The dispersed phase is modelled by tracking a large number of representative point-particles through the previously computed flow field by solving the equations of motion accounting for all relevant forces acting on the particles and other important elementary processes. Local average properties such as dispersed phase density or velocities are obtained by ensemble averaging for each control volume in the computational domain. The effect of the dispersed phase on the continuous phase has to be accounted for by appropriate source terms in the Eulerian conservation equations, which also need to be sampled during the Lagrangian tracking. A converged solution is achieved by sequential calculation of Eulerian and Lagrangian part. An essential advantage of this method is that the discrete nature of the dispersed phase particles is retained. Moreover, physical effects influencing the particle motion (i.e. elementary processes), such as particle-turbulence interaction, particle-wall collisions, inter-particle collisions or agglomeration, can be modelled on the basis of physical principles. Additionally, a particle size distribution may be easily considered by sampling the size of the injected particles from a given distribution function.

Essential in the application of both approaches is the modelling of the elementary processes affecting particle motion, which considerably influences the temporal and spatial distributions of the dispersed phase in the process of consideration. For operational conditions where the influence of the particle phase on the fluid flow, i.e. the so-called two-way coupling, becomes important also the fluid flow is strongly affected by the elementary processes. As a consequence, also integral parameters are influenced, for example the pressure drop in pneumatic conveying. As in the processes considered here, the particle size distribution is of eminent importance, the Euler/Lagrange-approach is used in this study.

For the numerical prediction of pneumatic conveying the Euler/Lagrange approach was further developed to account for all relevant elementary processes, such as transverse lift forces, wall roughness, inter-particle collisions and coupling between the phases. The present study deals with the 3D study of the particle behaviour in a particle-laden six meter long horizontal channel with rectangular cross-section from a numerical perspective. Calculations are carried out for spherical glass beads with different sizes and a mass loading of 1.0 kg particles/kg gas. Additionally, different wall roughnesses are considered. Air volume flow rate is constant in order to maintain a fixed gas average velocity of 20 m/s. The numerical computations are performed by the Euler/Lagrange approach in connection with a Reynolds stress turbulence model accounting for two-way and inter-particle collisions. For the calculation of the particle motion all relevant forces (drag, slip-shear and slip-rotational lift and gravity), inter-particle collisions and wall collisions with wall roughness are considered. The agreement of the computations with the findings of Sommerfeld and Kussin is found to be fully satisfactory for the gas and particle velocities.

**Experimental Procedure**

The test facility has been described in detail previously, hence only brief details of the main characteristics of the system are given here. The main component of the experimental rig is a horizontal channel of 6 m length which has a height of 35 mm and a width of 350 mm, hence almost two-dimensional flow conditions can be established. The upper and lower channel walls were made of stainless steel plates which could be exchanged in order to study the effect of wall material and wall roughness on the particle behaviour. The measurements were performed close to the end of the channel at a distance of 5.8 m from the entrance. A Phase-Doppler anemometer (PDA) was used to determine gas and particle velocities. In all the experiments the volume flow was kept constant in order to maintain a fixed averaged gas velocity of 20 m/s and measurements have been conducted for spherical glass particles $\rho_p = 2450 \text{ kg m}^{-3}$ with different diameters (60, 130, 195 and 625 µm), variable mass loadings of up to 1.5 kg particles/kg gas, and also different wall roughness. Full details about the experimental facility as well as the main experimental findings can be found in the study reported by Kussin and Sommerfeld and Sommerfeld and Kussin.

**Numerical approach**

The numerical scheme adopted to simulate the dispersed two-phase flow along the channel has been the fully coupled stationary and three-dimensional Euler/Lagrange approach.

The fluid flow was calculated based on the Euler approach by solving the Reynolds-averaged conservation equations in connection with the Reynolds Stress turbulence model (version Jones and Musonge and Jones) which were extended in order to account for modulation by the dispersed phase, i.e. two-way
coupling\textsuperscript{17}. The time-dependent three-dimensional conservation equations for the fluid may be written in the general form (using tensorial notation), as shown below:

\[ (\rho \phi)_t + (\rho U_i \phi)_x = (\Gamma_{ik} \phi_{k})_x + S_{t\phi} + S_{\phi\phi} \]  \quad \text{(1)}

where \( \rho \) is the fluid density; \( U_i \), the Reynolds-averaged velocity components; and \( \Gamma_{ik} \), the effective transport tensor. The usual source terms within the continuous phase equations are summarised in \( S_{t\phi} \), while \( S_{\phi\phi} \) represents the additional source term due to phase interaction. For near wall treatment the logarithmic law of the wall has been employed in conjunction with an explicit setting of the Reynolds stress anisotropy at the wall based on experimental data.

The simulation of the particle phase by the Lagrangian approach requires the solution of the equation of motion for each computational particle. This equation includes the particle inertia, drag, gravity-buoyancy, slip-shear lift and slip-rotational lift forces. The Basset history term, the additional source term due to phase interaction. For near wall treatment the logarithmic law of the wall results mainly from wall collisions but also the viscous interaction with the fluid, i.e. the torque \( T \). Hence, the equations of motion for the particles are:

\[ \frac{dx_{pi}}{dt} = u_{pi} \]  \quad \text{(2)}

\[ m_p \frac{du_{pi}}{dt} = 3 \frac{\rho}{4 D_p} m_p c_p (u_i - u_{pi})\left| \vec{u} - \vec{u}_p \right| + \]

\[ m_p g \left( 1 - \frac{\rho}{\rho_p} \right) + F_{mi} + F_{li} \]  \quad \text{(3)}

\[ I_p \frac{d\omega_{pi}}{dt} = T_i \]  \quad \text{(4)}

where \( x_{pi} \) is the coordinates of the particle position; \( u_{pi} \), its velocity components; \( u_i = U_i + u^i \), the instantaneous velocity of the gas; \( D_p \), the particle diameter; and \( \rho_p \), the density of the solids. \( m_p = \frac{\pi}{6} \rho_p D_p^3 \) is the particle mass and \( I_p = 0.1 m_p D_p^2 \) is the moment of inertia for a sphere. The drag coefficient is obtained using the following standard correlation:

\[ c_D = \begin{cases} 24 \text{Re}_p^3 (1 + 0.15 \text{Re}_p^{0.687}) & \text{Re}_p \leq 1000 \\ 0.44 & \text{Re}_p > 1000 \end{cases} \]  \quad \text{(5)}

with \( \text{Re}_p = \rho D_p \sqrt{\vec{u} - \vec{u}_p} / \mu \), the particle Reynolds number.

The slip-shear force is based on the analytical result of Saffman\textsuperscript{18} and extended for higher particle Reynolds numbers according to Mei\textsuperscript{19}:

\[ \vec{F}_s = 1.615 D_p \mu \text{Re}_s^{1/2} c_s \left[ (\vec{u} - \vec{u}_p) \times \vec{\omega} \right] \]  \quad \text{(6)}

where \( \vec{\omega} = \nabla \times \vec{u} \) is the fluid rotation; \( \text{Re}_s = \rho D_p^2 |\vec{\omega}| / \mu \) is the particle Reynolds number of the shear flow; and \( c_s = F_s / F_{s,Saff} \) represents the ratio of the extended lift force to the Saffman force, as shown below:

\[ c_s = \begin{cases} (1 - 0.331 \sqrt{\beta}) e^{-\text{Re}_s^{10}} & \text{Re}_s \leq 40 \\ 0.331 \sqrt{\beta} \frac{\text{Re}_p}{\text{Re}_s} & \text{Re}_s > 40 \end{cases} \]  \quad \text{(7)}

Here \( \beta \) is a parameter given by \( \beta = 0.5 \text{Re}_s / \text{Re}_p \).

The applied slip-rotational lift force is based on the relation given by Rubinow and Keller\textsuperscript{20}, which was extended to account for the relative motion between particle and fluid. Moreover, several authors allowed an extension of this lift force to higher particle Reynolds numbers. Hence, the following form of the slip-rotation lift force has been used:

\[ \vec{F}_l = \frac{\pi}{8} D_p^3 \rho \text{Re}_p c_l \left[ \vec{\Omega} \times (\vec{u} - \vec{u}_p) \right] \]  \quad \text{(8)}

where \( \vec{\Omega} = 0.5 \nabla \times \vec{u} - \vec{\omega}_p \) and the Reynolds number of particle rotation is given by \( \text{Re}_r = \rho D_p^2 |\vec{\omega}| / \mu \). The lift coefficient according to Oesterlé and Bui Dinh\textsuperscript{21} for \( \text{Re}_p < 2000 \) is given below:

\[ c_l = 0.45 + \left( \frac{\text{Re}_r}{\text{Re}_p} - 0.45 \right) e^{-0.05684 \text{Re}_r^{0.4} \text{Re}_p^{0.3}} \]  \quad \text{(9)}

For the torque acting on a rotating particle the expression of Rubinow and Keller\textsuperscript{20} was extended to account for the relative motion between fluid and particle and higher Reynolds numbers:

\[ \vec{T} = \frac{\rho}{2} \left( \frac{D_p}{2} \right)^5 c_k |\vec{\Omega}| \vec{\Omega} \]  \quad \text{(10)}
where coefficient of rotation is obtained from Rubinow and Keller\textsuperscript{20} and direct numerical simulations of Dennis \textit{et al.}\textsuperscript{22} in the following way:

\[
c_R = \begin{cases} 
\frac{64\pi}{Re_R} & \text{if } Re_R \leq 32 \\
\frac{12.9}{\sqrt{Re_R}} + \frac{128.4}{Re_R} & \text{if } 32 < Re_R < 1000 
\end{cases} \quad \ldots (11)
\]

The above equations [Eqs (2) – (4)] to calculate the particle motion are solved by the integration of differential equations. For sufficiently small time steps and assuming that the forces remain constant during this time step, the new particle location, and the linear and angular velocities are calculated. The time step for the particle tracking was chosen to 50% of the smallest of all relevant time scales, such as the particle relaxation time, the integral time scale of turbulence and the mean inter-particle collision time. This choice guarantees the stability of the numerical integration scheme.

When a particle collides with a wall, the wall collision model provides the new particle linear and angular velocities and the new location in the computational domain after rebound. The applied wall collision model, accounting for wall roughness, is described by Sommerfeld and Huber\textsuperscript{3}. The wall roughness seen by the particle is simulated assuming that the impact angle is composed of the particle trajectory angle plus a stochastic contribution due to wall roughness ($\Delta\gamma$), which depends on the structure of wall roughness and particle size. In sampling the instantaneous roughness angle from a normal distribution with standard deviation ($\Delta\gamma$), the so-called shadow effect\textsuperscript{1} was accounted for. Such effect enhances the effective transfer of horizontal to transverse particle momentum, especially for shallow angles of attack, and it is associated with a strong increase of particle fluctuation velocity and as a consequence wall collision frequency.

Inter-particle collisions are modelled by the stochastic approach as described by Sommerfeld\textsuperscript{25}. This model relies on the generation of a fictitious collision partner and the calculation of the collision probability according to kinetic theory of gases, accounting for a possible correlation of the instantaneous velocities of colliding particles in turbulent flows. The advantage of this model is that it does not require information on the location of surrounding particles and hence it is also applicable if a sequential tracking of the particles is adopted, as usually done when applying the Euler/Lagrange approach to stationary flows. During each time step of the trajectory calculation of the considered particle, a fictitious second particle is generated. The size and velocity of this fictitious particle are randomly sampled from local distribution functions.

**Influence of particles on carrier flow**

The source terms for the momentum equations resulting from the exchange between particles and fluid are obtained on the basis of the Particle-Source-In-Cell (PSI Cell) concept. Hence, the momentum exchange is calculated by averaging over all parcels traversing a given control volume during one Lagrangian calculation. Instead of summing up all fluid dynamic forces acting on the particles, which is quite cumbersome, the momentum exchange is calculated from the velocity change of the parcels when traversing the control volume. In this procedure, however, the external forces have to be subtracted yielding the momentum source in the following form:

\[
\overline{S_{i,p}} = -\frac{1}{V_{cv}} \sum_k m_k N_i \sum_s \left\{ \left[ u_{i,s} \right]^{+i} - \left[ u_{i,s} \right]^{-i} \right\} - g_s \left( \frac{1}{\rho_p} \Delta t_l \right) \quad \ldots (12)
\]

where the sum over $n$ indicates averaging along the particle trajectory (time averaging) and the sum over $k$ is related to the number of computational particles passing the considered control volume with the volume $V_{cv}$. The mass of an individual particle is $m_k$ and $N_k$ is the number of real particles in one computational particle. $\Delta t_l$ is the Lagrangian time step which is used in the solution of Eqs (2)-(4). The source term in the conservation equation of the Reynolds stress components ($R_{ij}$) are expressed in the Reynolds average procedure as shown below:

\[
S_{R_{ij,p}} = u_j \overline{S_{i,p}} + u_i \overline{S_{j,p}} - \left( U_j \overline{S_{i,p}} + U_i \overline{S_{j,p}} \right) \quad \ldots (13)
\]

while the source term in the $\varepsilon$-equation is modelled in the following standard way:

\[
S_{\varepsilon,p} = c_{\varepsilon3} \frac{1}{2} \overline{S_{R_{ij,p}}} \quad \ldots (14)
\]

with $c_{\varepsilon3} = 1.0$ is a model coefficient and the sum is implicit in the repeated sub-index $i$.

All the calculations have been performed with a mesh of 240,000 control volumes for the channel with a length of
6 m, width of 350 mm and height of 35 mm. A converged solution of the coupled two-phase flow system is obtained by successive solution of the Eulerian and Lagrangian parts, respectively. Initially, the flow field is calculated without particle phase source terms until a converged solution is achieved. Thereafter, a large number of parcels are tracked through the flow field (typically 480,000) and the source terms are sampled for each control volume. In this first Lagrangian calculation inter-particle collisions are not considered, since the required particle phase properties are not yet available. Hence, for each control volume the particle concentration, the local particle size distribution and the size-velocity correlations for the mean velocities and the rms values are sampled. These properties are updated by each Lagrangian iteration in order to allow correct calculation of inter-particle collisions. Additional particle phase properties and profiles may be sampled for each transverse cell when the computational particle crosses a pre-defined location. From the second Eulerian calculation, the source terms of the dispersed phase are introduced using an under-relaxation procedure. For the present calculation, typically about 25 – 35 coupling iterations with an under-relaxation factor between 0.5 and 0.1 are necessary in order to yield convergence of the Euler-Lagrange coupling.

Results and Discussion

Three dimensional numerical calculations are compared with experimental data obtained in the horizontal channel facility described previously. Spherical glass beads with two diameters ($D_p = 130 \, \mu m$ and $195 \, \mu m$) are considered with a material density ($\rho_p$) of $2450 \, \text{kg m}^{-3}$. In the computations the particles are assumed to be mono-disperse due to the narrow size distribution and the resulting small value of the standard deviation normalised by the mean particle diameter. This implies that the particle response characteristics are quite similar for the largest and smallest particles in the distribution. The simulations have been performed using the Reynolds Stress turbulence model mentioned in the previous section and accounting for particle-wall as well as inter-particle collisions, i.e. considering the so-called four-way coupling, as it has been previously mentioned. The gas density ($\rho$) and the dynamic viscosity ($\mu$) are $1.25 \, \text{kg m}^{-3}$ and $1.8 \cdot 10^{-5} \, \text{kg m}^{-1} \text{s}^{-1}$ respectively.

Two particle diameters ($D_p = 130 \, \mu m$ and $195 \, \mu m$) are considered with a mass loading ratio of $\eta = 1.0$ kg particles/kg gas and two wall roughness, the so-called cases R0 and R2 (Table 1). The main trends shown by the experiments are reproduced by the calculations. The comparison of numerical results with experiments are carried out at the section located at 5.8 m downstream the channel inlet.

For the small roughness case (R0), a value of $\Delta \gamma = 1.5^\circ$ has been chosen for $130 \, \mu m$ particles. With this choice, the agreement between calculations and measurements is very good for the mean velocities (Fig. 1a) of both phases and very reasonable for the fluctuating velocities (Fig. 1b) of gas and particles. Figure 1 shows the obtained results for the single-phase flow, denoted “clean” in the graphics, which compare well with the experiments. Regarding the stream-wise mean velocity (Fig. 1a), the RSM turbulence model provides results which compare very well with the experimental points. Regarding the fluctuating velocities predicted by the RSM (Fig. 1b), the rms values are in good agreement with the experimental values, especially in the centre of the channel. When approaching the wall, the stream-wise velocity fluctuation falls below the experiments. The calculated vertical fluctuation is in very good agreement with the measurements, except in the near-wall region. This behaviour is typical for the application of the RSM in channel flow.

Figure 1 also summarises the results for the two-phase flow condition obtained with four-way coupling (i.e., considering inter-particle collisions). Figure 1a shows the mean velocity of both phases and Fig. 1b shows the fluctuating velocities. For the gas phase variables, the same comments as for the single-phase flow can be applied, showing that the computations are in good agreement with the measurements. Both components of the fluid mean fluctuating velocity, horizontal $u'$ and vertical $v'$, are reduced in the two-phase case regarding the single-phase flow, which is consistent with the observation that small particles tend to suppress turbulence. This turbulence modulation is correctly predicted by the model described above, except near the bottom of the channel where the vertical fluid fluctuation is damped slightly stronger in the experiment. However, the predicted profiles of the vertical mean fluctuation of the fluid are becoming asymmetric with increasing mass loading due to the higher particle concentration near the channel bottom. It is necessary to point out that with the

<table>
<thead>
<tr>
<th>Degree of roughness</th>
<th>Roughness in stream-wise direction</th>
<th>Roughness in lateral direction</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Mean roughness</td>
<td>Maximum roughness</td>
</tr>
<tr>
<td>Case R0</td>
<td>2.32</td>
<td>2.86</td>
</tr>
<tr>
<td>Case R2</td>
<td>6.83</td>
<td>8.32</td>
</tr>
</tbody>
</table>

Table 1 — Data for considered mean roughness cases
[All the values are given in micron]
use of the Reynolds-stress turbulence model the agreement between computations and measurements is reasonably good and much better than that obtained with an isotropic turbulence model such as the $k\varepsilon$ model.

Regarding the comparison with experiments, the computations are pretty good in all particle variables, especially for the mean velocity. However, a higher value of the stream-wise fluctuating velocity component in the lower half of the channel than in the upper half is predicted in these three-dimensional computations. The agreement with the measurements is very good for most parts of the profile, but under-predicted near the upper wall. Nevertheless, the computational results seem to be more logical since the particle concentration near the channel bottom is high (Fig. 3) and hence the wall collision frequency, yielding a broader particle velocity distribution and, hence, higher fluctuations. Nevertheless, the asymmetry of the profiles due to higher wall collision frequency with the channel bottom is captured very well by the computations. The vertical component of the particle fluctuation, on the other hand, is in perfect agreement with the measurements.

Figure 2 shows the same variables as in Fig. 1 for 195 $\mu$m particles and small roughness (R0) case with the wall roughness parameter chosen as $\Delta\gamma = 1.3^\circ$. Results are presented for an average conveying velocity of 20 m/s and mass loading of $\eta = 1.0$. The predicted mean gas and particle velocities match very well the experimental data (Fig. 2a). Most interesting is the behaviour of root mean square (rms) values of velocities. The fluid fluctuating velocities and turbulence suppression by the particles with increasing mass loading are satisfactorily reproduced by the computations. It should be noted that for 195 $\mu$m case turbulence modulation by the particles is lower than for the smaller 130 $\mu$m particles (Fig. 1) due to the larger particle Stokes number and lower particle number density. The agreement between computations and experiments for the particle variables is quite good, especially for the mean velocities. However, in this case the computed particle fluctuating velocities do not follow the experimental points as close as in the smaller particles case. Also, the profiles of the stream-wise component for the larger more inertial particles show no pronounced asymmetry as found for 130 $\mu$m particles.
Figure 3 shows the normalized particle mass flux, at 5.8 m downstream the inlet, for both particle sizes. Also the two-dimensional results for 195 µm diameter particles are included for comparison. As it can be appreciated from the experiments, the particle mass flux is very similar for both particle sizes, showing that it is well captured by the computations. In this case of rough walls the effect of inter-particle collisions on the profiles of the normalised particle mass flux is less pronounced as for smooth walls and only a slight reduction is observed near the bottom of the channel. Hence, the particle dispersion is mainly governed by wall roughness in this case. Without wall roughness, the re-dispersion the particles due to inter-particle collisions is much better visible. Moreover, the agreement of the computations with the measurements is very good for both particle diameters, showing only small differences near the walls, which can be attributed to the applied wall collision model because it produces somewhat higher rebound velocities at very shallow impact angles. This fact might be associated with multiple wall bouncing of the particles, which is not accounted for in the present model yet. Let us also note that in this case of low roughness the gravitational settling is still remarkable.

The performance of the wall roughness model may be also checked by comparing the velocity profiles obtained for two different wall roughness in the case of \( D_p = 195 \mu m \) and loading ratio \( \eta = 1.0 \) (Fig. 4). For the higher roughness case (R2 in Table 1), a value of \( \Delta \gamma = 4.5^\circ \) has been chosen for the 195 µm particles. With this choice, the agreement between calculations and measurements is pretty good for the mean velocities (Fig. 4a) of both phases and very reasonable for the fluctuating velocities (Fig. 4b) of gas and particles.

It is then clearly demonstrated that the calculations reproduce satisfactorily the main trends suggested by the experiments. In the case of R2, the roughness considerably promotes re-dispersion of the particle phase, reducing the gravitational settling and the particle mass flux becomes more even across the channel. However, a low wall roughness of only 2.32 µm is very important for the behaviour of inertial particles and only a slight increase of the mass flux towards the bottom is observed. As a consequence of wall roughness, the particles will bounce from wall to wall causing an increase of the particle – wall collision frequency and accordingly a reduction in...
particle mean velocity, due to the higher momentum loss. Also, as a consequence of the two-way coupling, the increasing drag force yields a slight reduction of the fluid mean velocity (Fig. 4a). The computations, however, overpredict slightly the particle mean velocity for the R2 roughness compared to the experiments. Regarding the rms velocities, the values for R2 case are considerably higher than those for the R0 case. This trend is captured satisfactorily by the model and the agreement with the measurements is very good for the particle horizontal fluctuating velocity (Fig. 4b) and the vertical rms values for both roughness cases. The measured fluid rms velocity values in the stream wise and vertical direction for the R2 case show a rather strong scatter compared to the R0 data. For explaining this, it should be mentioned that the R2 measurements were done some time before the more recent R0 measurements using an older PDA system with a covariance processor less accurate than the FFT processor used for the R0 measurements.

Figure 5 shows the computational results for 130 µm diameter particles in the R2 case (for which, unfortunately there are no experimental measurements) compared with the experiments and computations for the 195 µm particles. In this case, a tentative parameter $\Delta \gamma = 5.5^\circ$ has been chosen according to the correlations $\Delta \gamma - D_p$ presented in Lain et al.\textsuperscript{14}. As in the other figures, Fig. 5a presents the mean velocities and Fig. 5b shows the fluctuating velocities. The 130 µm mean particle velocity is remarkably higher than that of the 195 µm and less flat, indicating that the particle-wall collision frequency is lower for the smaller particles. On the other hand, the influence of particles into the fluid variables is quite similar in both cases. As happened in the R0 case, the 130 µm particles have a slightly stronger effect on turbulence modulation than the 195 µm particles (Fig. 5b). Accordingly, the particle rms velocities are somewhat lower for the smaller particles than for the larger particles, owing to the lower particle-wall collision frequency.

Figure 6 shows the normalized particle mass flux, at 5.8 m downstream the inlet, for both particle sizes and high roughness R2. As it can be readily seen, both profiles are very similar and flat revealing that the particle motion is, in this case, governed by wall-to-wall bouncing as the particle fluctuating velocities suggested.
Conclusion

Three-dimensional numerical computations of dilute phase pneumatic conveying in a long horizontal channel with a rectangular cross-section have been presented and compared with experimental data. Computations have been carried out by means of the Euler-Lagrange approach using a Reynolds Stress turbulence model for the gas phase and accounting for two-way coupling. Particle-rough wall interactions and inter-particle collisions have also been taken into account. Predicted mean gas and particle velocities agree very well with the measurements. Turbulence attenuation by the particles is reasonably well captured for the 130 and 195 µm glass beads by the applied Reynolds stress model with standard coupling terms. Moreover, the computed particle velocity fluctuations capture satisfactorily the main tendencies shown by the experiments for different roughness cases which indicate that the considered model is appropriate to simulate pneumatic conveying of spherical solid particles in a horizontal channel. The present results clearly emphasise that numerical predictions of wall bounded flows, such as pneumatic conveying, cannot be performed properly when neglecting wall roughness effects.

References

1. Adam O, Untersuchungen über die Vorgänge in feststoffbeladenen Gasströmungen (Westdeutscher Verlag, Köln), 1960, 32.