DEM/CFD Simulations of a Pseudo-2D Fluidized Bed: Comparison with Experiments

Ziad Hamidouche 1, Yann Dufresne 2, Jean-Lou Pierson 1,*, Rim Brahem 1, Ghislain Lartigue 2 and Vincent Moureau 2

1 IFP Energie Nouvelles, Rond-Point de l’échangeur de Solaize, BP 3, 69360 Solaize, France; hamidouche_ziad@yahoo.com (Z.H.); rim.brahem@ifpen.fr (R.B.); 2 CORIA, CNRS, INSA & Université de Rouen, 76801 Saint-Etienne-du-Rouvray, France; yann.dufresne@insa-rouen.fr (Y.D.); ghislain.lartigue@coria.fr (G.L.); vincent.moureau@coria.fr (V.M.)

* Correspondence: jean-lou.pierson@ifpen.fr

Received: 12 February 2019; Accepted: 4 March 2019; Published: 15 March 2019

Abstract: The present work investigates the performance of a mesoscopic LaGrangian approach for the prediction of gas–particle flows under the influence of different physical and numerical parameters. To this end, Geldart D particles with 1 mm diameter and density of 2500 kg/m³ are simulated in a pseudo-2D fluidized bed using a Discrete Element Method (DEM)/Large-Eddy Simulation (LES) solver called YALES2. Time-averaged quantities are computed and compared with experimental results reported in the literature. A mesh sensitivity analysis showed that better predictions regarding the particulate phase are achieved when the mesh is finer. This is due to a better description of the local and instantaneous gas–particle interactions, leading to an accurate prediction of the particle dynamics. Slip and no-slip wall conditions regarding the gas phase were tested and their effect was found negligible for the simulated regimes. Additional simulations showed that increasing either the particle–particle or the particle–wall friction coefficients tends to reduce bed expansion and to initiate bubble formation. A set of friction coefficients was retained for which the predictions were in good agreement with the experiments. Simulations for other Reynolds number and bed weight conditions are then carried out and satisfactory results were obtained.

Keywords: DEM/CFD simulations; Euler/Lagrange approach; fluidized beds; frictional effects

1. Introduction

Based on their effectiveness regarding gas–solid heat and mass transfers, fluidized beds are among the best options for developing economically and environmentally viable techniques for fossil-fuel-based energy generation. Such systems involve complex physical mechanisms such as momentum, heat, and mass exchanges between the gas and the particulate phases. In addition, fluidized beds exhibit an unsteady and inhomogeneous behavior leading to wide characteristic length and time scales. Particle–particle and gas–particle interactions at the micro-scale (1 to 5dp, where dp is the characteristic size of a particle) result in meso-scale structures, such as bubbles (10 to 100dp), which can affect the macro-scale gas–particle flow [1]. Furthermore, different local behaviors can be observed depending on the local particle–particle, gas–particle and particle–wall interactions, and may profoundly modify the bed hydrodynamics. Therefore, various numerical approaches have been developed over recent decades to simulate those flows at microscopic, mesoscopic and macroscopic scales with the aim of elucidating the mechanisms underlying the origin and the evolution of the heterogeneous gas–particle flow pattern. The accurate prediction of the underlying physics makes possible to improve existing processes and to design more efficient new facilities. In this context, the development of reliable numerical approaches is an essential prerequisite.
Discrete Element Method (DEM) is among the most appropriate meso-scale approaches to simulate small scale fluidized beds, with $O(10^6)$ particles [2]. In this technique the particle motion is given by the Newtonian equations. Particles interact with each other through collisions that can be described using models from molecular dynamics [3], the so-called sphere models [4–6]. In this work the particles are considered as soft spheres that can overlap slightly and exert both normal and tangential forces on each other [4]. This model, known as the soft-sphere model, requires a contact force model to account for the inter-particle collision dynamics. The gas flow is solved on a Eulerian grid using a continuum approach based on a volume averaging of the Navier-Stokes equations [7]. Interphase momentum transfer terms are included in the modeling to account for the fluid-particle interactions [8]. In this work, the ability of a DEM/Large-Eddy Simulation (LES) approach to reproduce the gas–particle flow behavior in a 2D-fluidized bed is assessed. A specific attention will be paid to the influence of numerical (grid cell size) and physical parameters (friction coefficient).

In Section 2, the modeling strategy is given. Detailed models, based on averaged Navier-Stokes equations for the gas phase, DEM technique for the particles and coupling procedure between the phases are provided in Sections 2.1–2.3, respectively. Detailed information about the numerical schemes can be found in Section 2.4. Finally, an overview of the simulation cases is presented in Section 3 and results discussed in Section 4.

2. Modeling Strategy

All the numerical simulations presented in this work are performed using the finite-volume code YALES2 [9], a LES and Direct Numerical Simulation (DNS) solver based on unstructured meshes. This code solves the low-Mach number Navier-Stokes equations for turbulent reactive flows using a time-staggered projection method for constant [10] or variable density flows [11]. YALES2 is specifically tailored for solving these low-Mach number equations on massively parallel machines with billion-cell meshes thanks to a highly optimized linear solvers [12].

Recently, a meso-scale four-way coupling approach for the modeling of solid particles has been implemented in the YALES2 solver. This approach combines DEM approach to represent the solid phase with LES equations solved on a Eulerian unstructured grid for the fluid phase. The CFD/DEM solver has been thoroughly optimized for massively parallel computing. It features a dynamic collision detection grid for unstructured meshes and packing/unpacking of the halo data for non-blocking Message Passing Interface (MPI) exchanges.

2.1. Gas Phase Modeling

The governing equations for the averaged fluids flow are obtained from the filtering of the unsteady, low-Mach number Navier-Stokes equations, taking the local fluid and solid fractions into account. If $G$ is a filtering kernel (see for instance [7]), the local fluid fraction $\varepsilon$ is defined as:

$$\varepsilon(x, t) = \int_{V_f} G(|x - y|) dy,$$

where $V_f$ is the volume occupied by the fluid. Defining $\Phi(x, t)$ as a function of position and time, the volume filtered field $< \Phi > (x, t)$ refers to the regular spatial average and is computed by taking the convolution product with the filtering kernel $G$, giving:

$$\varepsilon < \Phi > (x, t) = \int_{V_f} \Phi(y, t) G(|x - y|) dy.$$

Further details concerning the volume filtering operations can be found in [8]. The governing equations finally read:

$$\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = 0,$$

where $\mathbf{u}$ is the fluid velocity.
\[
\frac{\partial}{\partial t} (\rho \langle \mathbf{u} \rangle) + \rho \mathbf{u} \cdot \nabla (\rho \langle \mathbf{u} \rangle) = -\nabla \langle P \rangle + \nabla \cdot (\rho \langle \mathbf{u} \rangle \otimes \langle \mathbf{u} \rangle) + \epsilon \rho \mathbf{g} + \frac{1}{\Delta V} \langle \mathbf{F} \rangle_{p\rightarrow f}, \tag{4}
\]

where \(\mathbf{u}, \rho, P, \) and \(\mathbf{\tau}\) are the gas velocity, density, dynamic pressure respectively, and viscous stress tensor. The closure used to calculate the turbulent viscosity \(\mu_t\) is the localized dynamic Smagorinsky model [13] proposed by [14,15]. The term \(\tilde{\mathbf{F}}_{p\rightarrow f}\) is the momentum source term due to particle displacement. \(\Delta V\) is the local control volume. Details concerning the computation of this term can be found in Section 2.3. For the sake of clarity, the notation \(<>\) will be dropped for the averaged quantities in the following.

2.2. Discrete Particle Modeling

Generally speaking, the forces acting on a particle in motion can be divided into two categories, volume and surface forces. The volume forces are due to gravity and buoyancy acting against the weight, whereas, the surface forces consist of hydrodynamic and contact forces. While the hydrodynamic forces arise from fluid-particle interactions, such as drag, pressure gradient, and lift forces, the contact forces are due to particle–particle and particle–wall interactions. Such forces can be further classified in collision and adhesive forces. In the present work, due to the high solid/gas density ratio, as reported in Table 1, the buoyancy force is neglected. The hydrodynamic forces being also omitted are the lift, the added and the history (Basset) forces. For the lift forces, two contributions are proposed in the literature, Magnus and Saffman forces, which are extensively investigated in dilute flows. However, a consensus on their use in dense regimes is still missing and their relevance in fluidized-bed configurations in the absence of a mean shear is questionable. For high density ratios, the added and the Basset forces may have only very weak contributions and thus can be neglected in our case. Concerning the contact forces, only collision forces are accounted for. Adhesive effects that may originate from electrostatic or Van Der Vaals forces can safely be here neglected since, as it can be seen in Table 1, the simulated particles are relatively large (Geldart D particles).

DPMs, in which each particle is tracked in a LaGrangian fashion, have clearly shown their ability to simulate the behavior of granular flows, and originate from molecular dynamics methods initiated in the 1950s [3]. A soft-sphere model [16] is employed to compute contact between each particle. They may overlap other particles or walls in a controlled manner. A resulting contact force accounting for the added and the Basset forces may have only very weak contributions and thus can be neglected in our case. Concerning the contact forces, only collision forces are accounted for. Adhesive effects that may originate from electrostatic or Van Der Vaals forces can safely be here neglected since, as it can be seen in Table 1, the simulated particles are relatively large (Geldart D particles).

Particle movement is then given by Newton’s second law for translation assuming high solid/gas density ratio:

\[
m_p \frac{d\mathbf{u}_p}{dt} = \mathbf{F}_D + \mathbf{F}_C + \mathbf{F}_p + \mathbf{F}_C \quad \text{with} \quad \frac{d\mathbf{x}_p}{dt} = \mathbf{u}_p, \tag{5}
\]

where \(m_p, \mathbf{u}_p,\) and \(\mathbf{x}_p\) are the particle mass, velocity, and position, respectively. \(\mathbf{F}_D\) is the drag force, \(\mathbf{F}_P\) is the pressure gradient force and \(\mathbf{F}_C\) is the collision force. The relation between \(\mathbf{F}_D, \mathbf{F}_C\) in Equation (5) and \(\mathbf{F}_{p\rightarrow f}\) in Equation (4) is detailed in Section 2.3. Preliminary simulations including particle rotation provided similar results as those without rotation. Consequently, for computational reason, the particle rotation is not accounted for in this paper.

2.2.1. Modeling of Collisions

The total collision force \(\mathbf{F}_C\) acting on particle \(a\) is computed as the sum of all forces \(f^{\text{col}}_{b\rightarrow a}\) exerted by the \(N_p\) particles and \(N_w\) walls in contact. As particles and walls are treated similarly during collisions, the \(b\) index refers to both:

\[
\mathbf{F}_C = \sum_{b=1}^{N_p+N_w} f^{\text{col}}_{b\rightarrow a}, \tag{6}
\]

\[
f^{\text{col}}_{b\rightarrow a} = f^{\text{col}}_{n,b\rightarrow a} + f^{\text{col}}_{t,b\rightarrow a}. \tag{7}
\]

Depending on the desired compromise between accuracy and numerical cost, a lot of soft-sphere models can be found in the literature. Here, a linear-spring/dashpot [16] is used along with a simple Coulomb sliding model accounting for the normal \(f^{\text{col}}_{n,b\rightarrow a}\) and tangential \(f^{\text{col}}_{t,b\rightarrow a}\) components of
the contact force, respectively, as in the work of Capecelatro [8]. For one particle (or wall) \( b \) acting on a particle \( a \):

\[
\mathbf{f}_{\text{col}}^{n,b \rightarrow a} = \begin{cases} 
-k_n \delta_{ab} \mathbf{n}_{ab} - 2\gamma_n M_{ab} \mathbf{u}_{ab,n} & \text{if } \delta_{ab} > 0, \\
0 & \text{else,}
\end{cases}
\]

\[
\mathbf{f}_{\text{col}}^{t,b \rightarrow a} = -\mu \tan \theta \mathbf{f}_{\text{col}}^{n,b \rightarrow a} || t_{ab} .
\]

The term \( \delta_{ab} \) is defined as the overlap between the \( a \) and \( b \) entities expressed using each particle radius \( r_p \) and center coordinates \( x_p \) such as:

\[
\delta_{ab} = r_a + r_b - (x_b - x_a) \cdot \mathbf{n}_{ab}.
\]

The system effective mass \( M_{ab} \) is expressed using each particle mass \( m_p \) such as:

\[
M_{ab} = \frac{1}{1/m_a + 1/m_b}.
\]

In case of a particle–wall collision, the wall is considered to be a particle with infinite mass and null radius. This model requires three user-defined parameters; \( k_n \), \( \gamma_n \), and \( \mu \tan \theta \) respectively accounting for the spring stiffness, normal damping, and friction coefficient of the \( a - b \) binary system. \( \mathbf{n}_{ab} \) and \( \mathbf{t}_{ab} \) respectively account for the unit normal vector from particle \( a \) towards entity \( b \) and the unit tangential vector. \( \mathbf{n}_{ab} \) is calculated as follows:

\[
\mathbf{n}_{ab} = \frac{x_b - x_a}{||x_b - x_a||}.
\]

with \( \mathbf{u}_{ab} \) being the relative velocity of the colliding system, its normal and tangential components are then given by:

\[
\mathbf{u}_{ab,n} = ((\mathbf{u}_a - \mathbf{u}_b) \cdot \mathbf{n}_{ab}) \mathbf{n}_{ab},
\]

\[
\mathbf{u}_{ab,t} = \mathbf{u}_{ab} - \mathbf{u}_{ab,n}.
\]

Finally, \( \mathbf{t}_{ab} \) is given by:

\[
\mathbf{t}_{ab} = \frac{\mathbf{u}_{ab,t}}{||\mathbf{u}_{ab,t}||}.
\]

Using Newton’s third law and a projection on \( \mathbf{n}_{ab} \) yields the following ordinary differential equation for the overlap evolution of an \( a - b \) binary system undergoing collision without taking any other force into account:

\[
\frac{d^2 \delta_{ab}}{dt^2} + 2\gamma_n \frac{d\delta_{ab}}{dt} + \omega_0^2 \delta_{ab} = 0,
\]

where \( \omega_0 \) stands for the system’s natural frequency and is defined as:

\[
\omega_0^2 = \frac{k_n}{M_{ab}}.
\]

The damping parameter \( \gamma_n \) accounting for the energy dissipation occurring during contact is calculated by the mean of a normal restitution coefficient \( e_n \) verifying \( 0 < e_n \leq 1 \) such as:

\[
\gamma_n = -\frac{\omega_0 \ln e_n}{\sqrt{\pi^2 + (\ln e_n)^2}}.
\]
A contact time $T_C$ can also be analytically extracted from Equation (16) corresponding to the time during which the particle $a$ and the entity $b$ are overlapping:

$$T_C = \frac{\pi}{\sqrt{\omega_0^2 - \gamma_n^2}}. \quad (19)$$

### 2.2.2. Closure for Drag

The drag force $F_D$ acting on a particle $p$ is written:

$$F_D = \frac{m_p}{\tau_p} (\mathbf{u}_@p - \mathbf{u}_p), \quad (20)$$

where $\mathbf{u}_@p$ is the local gas velocity interpolated at the center of the particle $p$ and $\tau_p$ is the drag relaxation time expressed as follows:

$$\tau_p = \frac{4\rho_p d_p^2}{3\mu C_D Re_p} \quad \text{with} \quad Re_p = \frac{\varepsilon_p \rho |\mathbf{u}_@p - \mathbf{u}_p| d_p}{\mu}, \quad (21)$$

where $\rho_p$ is the particle density, $d_p$ its diameter, $\mu$ the dynamic viscosity, $\varepsilon_p$ is the local fluid fraction interpolated at the center of the particle, and $C_D$ is the drag coefficient. To compute $C_D$, the classical closures proposed by Ergun ($C_{D,ER}$) [17] for high $\varepsilon_p$ values and Wen & Yu ($C_{D,WY}$) [18] for low $\varepsilon_p$ values are used along with the smoothing function $\phi_{gs}$ introduced by Huilin & Gidaspow [19] to avoid discontinuities when switching models:

$$\phi_{gs} = \frac{1}{\pi} \arctan(150 \times 1.75(0.2 - (1 - \varepsilon_p))) + 0.5, \quad (22)$$

in such a way that:

$$C_D = \phi_{gs} C_{D,WY} + (1 - \phi_{gs}) C_{D,ER}, \quad (23)$$

with

$$C_{D,WY} = \begin{cases} \frac{24}{Re_p} \left( 1 + 0.15 Re_p^{0.687} \right) \varepsilon_p^{-2.7} & \text{for } Re_p < 10^3, \\ 0.44 \varepsilon_p^{-2.7} & \text{for } Re_p \geq 10^3, \end{cases} \quad (24)$$

and

$$C_{D,ER} = \left( \frac{200}{3 Re_p} + \frac{7}{3} \right) \varepsilon_p^{-1}. \quad (25)$$

### 2.2.3. Other Forces

The gravity force $F_G$ acting on a particle $p$ is written:

$$F_G = m_p g. \quad (26)$$

The pressure gradient force $F_P$ in Equation (5) reads as:

$$F_P = -V_p \nabla P_{@p}, \quad (27)$$

where $V_p$ is the particle’s volume and $\nabla P_{@p}$ is the local pressure gradient interpolated at the center of the particle.

### 2.3. Phase Coupling

The coupling between the particle and fluid phases is a key point for the modeling of particle-laden flows, especially when the particle size approaches the Eulerian cell size. Many Eulerian fields must
be interpolated at the center of the particles for the numerous closures, as shown in Section 2.2. In the YALES2 solver, particles are in a unique mesh cell (C) using the position of their center. For any Eulerian scalar or vector field $\Phi(x,t)$, its value taken at the particle $p$ center $\Phi_{p}(t)$ obeys:

$$\Phi_{p}(t) = \sum_{i \in C} \omega_{p,i} \Phi(x,t) \quad \text{with} \quad \sum_{i \in C} \omega_{p,i} = 1. \quad (28)$$

Here $i$ is a node index so that ‘$i \in C$’ means ‘all nodes composing the mesh cell C in which the particle $p$ is located’. $\omega_{p,i}$ is the interpolation weight of the particle $p$ on cell node $i$ and is calculated using a trilinear interpolation on tetrahedra and on hexahedra. The same interpolation weights are used for data transfer from grid to particles (interpolation step) and from particle onto the grid (projection step).

The conservative projection operator needed to compute $F_{p \rightarrow f}$ (see Equation (4)) is thus written on each node $i$ as:

$$F_{p \rightarrow f,i} = - \sum_{p \in SC_{i}} \omega_{p,i}(F_{D} + F_{P}), \quad (29)$$

and the fluid fraction at node $i$ is written:

$$\varepsilon_{i} = \frac{1}{\Delta V_{i}} \sum_{p \in SC_{i}} \omega_{p,i}V_{p}. \quad (30)$$

Here $\Delta V_{i}$ denotes the control volume of node $i$ and ‘$p \in SC_{i}$’ means ‘all particles belonging to any surrounding cell (SC) of node $i$’. Referring to Figure 1, it means that any particle belonging to one of the cells will be accounted for when computing $\varepsilon$ (as well as $F_{p \rightarrow f}$) at node 3.

![Figure 1](image-url) 2D representation of a particle $p$ located in a triangular cell (■) moving towards a neighboring cell (□). $\Phi$: mesh nodes. ---: contour of node 3 control volume. $\omega_{p,i}$: interpolation weight of particle $p$ on node $i$.

This method, consisting of distributing particle quantities only in the cell where its center resides, referred to as Particle Centroid Method (PCM), can lead to large calculation errors in particular regarding the fluid fraction, as pointed out in [20]. This is partly due to the fact that many CFD/DEM codes feature a staggered grid where the fluid fraction is defined at cell centers, causing dramatic discontinuities in time and space derivatives when a particle enters or leaves a cell. On the contrary, in the YALES2 code the fluid fraction, as all the Eulerian fields, is computed at the grid nodes. As depicted in Figure 1, it is then straightforward that the particle crossing from the green cell to the red one will not cause any discontinuity on the computation of neither $\omega_{p,1}$ nor $\omega_{p,3}$ involved in their interface. Moreover, $\omega_{p,2}$ will not be much affected neither during the crossing because as the particle approaches the cell’s interface, $\omega_{p,2}$ tends towards 0. This is still true in 3D cases and on Cartesian meshes.

Nevertheless, it is well known that the PCM method can induce inaccuracies and lead to numerical instabilities because it cannot prevent the fluid fraction from reaching unrealizable values, in particular when dealing with close to unity particle diameter/mesh cell size ratios. In such cases, the fluid fraction value can locally decrease below the theoretical packing limit. To cope with this limitation, a
filtering operator well suited for distributed memory machines is used. Taking a 2D case as shown on
the left in Figure 2, this filtering operator is built for any Eulerian scalar or vector field noted \( \Phi_i \), its
filtered value being \( \hat{\Phi}_i \). At node \( i_1 \), \( \hat{\Phi}_i \) reads:

\[
\hat{\Phi}_{i_1} = \frac{1}{3} \Phi_{i_1} + \frac{1}{3\Delta V_{i_1}} \left[ (S_{23} + S_{34})\Phi_{i_3} + (S_{34} + S_{45})\Phi_{i_4} + \cdots + (S_{72} + S_{23})\Phi_{i_2} \right],
\]

where \( \Delta V_{i_1} \) is the control volume associated with node \( i_1 \) and \( S_{mn} \) is the part of \( \Delta V_{i_1} \) contained in the
face delimited by nodes \( i_1, i_m \), and \( i_n \), as shown on the left in Figure 2. If all the control volumes are
equal, on a structured mesh for instance, Equation (31) becomes:

\[
\hat{\Phi}_{i_1} = \frac{1}{3} \Phi_{i_1} + \frac{1}{9} \sum_{m \in [2,7]} \Phi_{i_m}.
\]

\[\text{Figure 2. On the left: 2D representation of an unstructured mesh. \( \bullet \): mesh nodes. The control volumes of nodes } i_1(\#), i_2(\#), i_3(\#), i_4(\#), i_5(\#), i_6(\#), i_7(\#) \text{ are shown. The control volume of node } i_1 \text{ is composed of surfaces } S_{23}, S_{34}, S_{45}, S_{56}, S_{67} \text{ and } S_{72}. \quad \text{On the right: 3D representation of a regular Cartesian mesh part. } \bullet \text{: node } i_1. \quad \bullet \text{: nodes at faces’ center. } \bullet \text{: nodes at edges’ center. } \bullet \text{: nodes at corners.} \]

The same type of filter can be derived in 3D. The following equation gives the value of \( \hat{\Phi}_{i_1} \) in a
3D structured case with all equal control volumes as shown on the right in Figure 2, where RN is the
abbreviation of Red Nodes, BN of Blue Nodes and GN of Green Nodes:

\[
\hat{\Phi}_{i_1} = \frac{1}{8} \Phi_{i_1} + \frac{1}{64} \left[ 4 \sum_{m \in RN} \Phi_{i_m} + 2 \sum_{m \in BN} \Phi_{i_m} + \sum_{m \in GN} \Phi_{i_m} \right].
\]

This fully conservative operation being performed on all volumes at the same instant provides a
fully filtered field, and can be repeated several times to increase the filter width. In all the simulations
presented in this paper, only one filtering step was used. It should be underlined that for the
computation of \( \hat{\varepsilon} \), the filtering step is applied before dividing by the local control volume to conserve
total solid mass over the whole computational domain volume \( V \):

\[
\text{Total solid mass } = \rho_p \int_V (1 - \varepsilon) dV = \rho_p \int_V (1 - \hat{\varepsilon}) dV.
\]

The properties of such a filtering operator, i.e., its moments, are not straightforward to determine
on unstructured meshes but it can be noticed that it is based on direct neighbors and thus does not
need distant nodes, hence its attractiveness regarding parallelism. The main drawback is that the
filter width cannot be directly obtained because it depends on the local mesh size. Thus, when using
this filtering operator, the user cannot prescribe the filter width. Other projection methods have been
developed to circumvent this drawback while ensuring interesting mathematical properties as moment
conservation for instance. One can cite the work of Capecelatro [8], who used a Gaussian kernel filter
in a method called mollification for the same kind of application as presented here, and the work of Mendez [21] in which a projection operator based on high-order moments conservation was built for deformable red cell membrane modeling purposes.

2.4. Numerical Schemes

2.4.1. Fluid Advancement Procedure

This section presents some numerical features of the YALES2 code. This code solves the filtered incompressible Navier-Stokes equations presented in Section 2.1 with an explicit time advancement. Among the various implemented numerical schemes, a fourth-order central scheme was used for the spatial integration, and a fourth-order scheme called TFV4A [22] combining Runge–Kutta and Lax–Wendroff methods was used for the explicit time integration.

The time advancement uses a time-staggered projection method for variable density flows [11] described below in which the \( n \) superscript refers to discrete times:

1. LaGrangian phase advancement

First, the particles are advanced. The full description of this step is available in Section 2.4.2. After being relocated on the grid, \( \epsilon^{n+3/2} \) can be computed using Equation (30).

2. Density prediction for scalar advancement

The density predictor \( (\epsilon \rho)^* \) is then determined by the mean of the mass conservation equation (Equation (3)):

\[
\frac{(\epsilon \rho)^* - (\epsilon \rho)^{n+1/2}}{\Delta t} = -\nabla \cdot (\epsilon \rho u)^n. \tag{35}
\]

3. Velocity prediction

Once \( (\epsilon \rho)^{n+1} \) is known, the velocity can be predicted reusing the dynamic pressure gradient of the previous time step:

\[
\frac{(\epsilon \rho u)^* - (\epsilon \rho u)^n}{\Delta t} = -\nabla \cdot ((\epsilon \rho u)^n u^n) - \nabla \cdot \tau^n. \tag{36}
\]

4. Velocity correction

Velocity correction is performed by updating the pressure gradient:

\[
\frac{(\epsilon \rho u)^{n+1} - (\epsilon \rho u)^*}{\Delta t} = -\nabla (p_2^{n+1/2} - p_2^{n-1/2}). \tag{37}
\]

The Poisson equation aiming at calculating \( p_2^{n+1/2} \) is obtained by taking the divergence of Equation (37) and inserting the condition imposed by the following equation of mass conservation written for \( u^{n+1} \):

\[
\nabla \cdot (\epsilon \rho u)^{n+1} = -\frac{(\epsilon \rho)^{n+3/2} - (\epsilon \rho)^{n+1/2}}{\Delta t}. \tag{38}
\]

The Poisson equation finally reads:

\[
\nabla \cdot \left( \nabla \left( p_2^{n+1/2} - p_2^{n-1/2} \right) \right) = \frac{(\epsilon \rho)^{n+3/2} - (\epsilon \rho)^{n+1/2}}{\Delta t^2} + \frac{\nabla \cdot (\epsilon \rho)^*}{\Delta t}. \tag{39}
\]

This linear system requires an efficient and accurate iterative solver. For all our simulations, a Deflated Preconditioned Conjugate Gradient (DPCG) algorithm [23] is used.

The resulting time advancement is fully mass and momentum conserving. The time step for the fluid phase \( \Delta t \) is calculated at each solver iteration by enforcing a maximum value of 0.2 for the Courant-Friedrichs-Lewy number (CFL) criterion and 0.15 for the Fourier number criterion for all the simulations.
2.4.2. Particle Advancement Procedure

A second-order explicit Runge–Kutta (RK2) algorithm is used to advance the position $x_p$, the velocity $u_p$ (see Equation (5)) of the particles in time:

\[
\begin{align*}
\text{RK2 - 1st step:} & \quad \begin{cases} 
    x_{p,n+1/2} = x_{p,n} + \frac{\Delta t}{2} u_{p,n}, \\
    u_{p,n+1/2} = u_{p,n} + \frac{\Delta t}{2} \sum F^p_{m,n},
\end{cases} \\
\text{RK2 - 2nd step:} & \quad \begin{cases} 
    x_{p,n+1} = x_{p,n} + \Delta t u_{p,n+1/2}, \\
    u_{p,n+1} = u_{p,n} + \Delta t \frac{\sum F^p_{m,n+1/2}}{m_p},
\end{cases}
\end{align*}
\] (40, 41)

where $\sum F$ refers to the summation of all forces acting on particle $p$ (see Section 2.2).

At the particle scale, several phenomena need to be integrated properly by the mean of an associated characteristic time, among these: drag, gravity, etc. Thus, several stability criteria must be computed on each particle to determine the smallest time step needed for the most constraining characteristic time. In dense fluidized beds simulations, the collision time step is generally the limiting one, so it will be noted $\Delta t_p$ from now on.

$\Delta t_p$ must be inferior to the contact time $T_C$ described in Equation (19) to be able to solve collisions properly. Furthermore, it must be small enough to ensure numerical stability depending on the selected numerical scheme, without compromising the performance of the code. It can be noted that there is a unique value of $T_C$ because all the particles are identical. In our simulations, the following criterion was used:

\[\Delta t_p = T_C / 6.\] (42)

This criterion has been tested against the following non-dimensional analytical solution of Equation (16) that can be found in [24] for a normal collision of a single particle on a wall with given collision parameters as shown in Figure 3:

\[
\begin{align*}
\delta^{*}_{ab}(t) &= \frac{\delta_{ab}(t)}{\delta_{ab}^{\text{max}}} = \frac{\omega_0}{\Omega} \exp \left( \gamma_n \left[ \frac{1}{\Omega} \arcsin \left( \frac{\Omega}{\omega_0} \right) - t \right] \right) \sin (\Omega t), \\
u^{*}_{ab,n}(t) &= \frac{u_{ab,n}(t)}{u_0} = \frac{1}{\Omega} e^{-\gamma_n t} (-\gamma_n \sin(\Omega t) + \Omega \cos(\Omega t)),
\end{align*}
\] (43, 44)

with

\[\Omega = \sqrt{\omega_0^2 - \gamma_n^2}.\] (45)

**Figure 3.** The simple test consists of a single particle colliding a wall in the normal direction. Only the collision force is accounted for and the parameters used for the resolution are visible on the right side. The contact starts at $t = 0$.

The time evolution of both overlap and particle velocity during contact has been plotted in Figures 4 and 5, respectively.
Non-dimensional overlap 

\[ \eta_{ab} = \frac{a \cap b}{\max(a \cap b)} \]

Figure 4. Non-dimensional overlap as a function of the non-dimensional time during the contact presented in Figure 3. Comparison between analytical method (---), Euler method with \( t_p = T_C/80 \) (▲) and RK2 method with \( t_p = T_C/6 \) (●).

Non-dimensional velocity 

\[ \eta_{ab,n} = \frac{u_{ab,n}}{u_{\text{max}ab,n}} \]

Figure 5. Non-dimensional velocity as a function of the non-dimensional time during the contact presented in Figure 3. Comparison between analytical method (---), Euler method with \( t_p = T_C/80 \) (▲) and RK2 method with \( t_p = T_C/6 \) (●).

The agreement with the analytical solution is not perfect; however, the scheme is stable and in this case the absolute error on the restitution velocity was found to lie below 5%.

2.5. Performance and Parallelism

A critical point in the coupling between the gas and solid phase is the time advancement, as they can have very different time scales. The choice was made to sub-step the solid phase, of which time advancement is generally limited by the collision time step in dense fluidized beds, during a gas phase time step, which is itself governed by convective and diffusive time scales. The mean number of particle sub-steps observed during our simulations was approximately 3 for the refined mesh, 7 for the intermediate mesh and 16 for the coarse mesh (mesh details are described in the following section).

As the computation of the collision force requires a distance estimation for each particle pair \textit{a priori}, solving Equations (40) and (41) can become critically time consuming. The work of Lubachevsky [25] provides an analysis of the linked-cell method to tackle this problem, which is the most commonly used method. The solution consists in the definition of a Cartesian grid superimposed on the grid mesh allowing each particle to search for its potential collision partners, as shown in Figure 6. First, each particle is in one of the Cartesian grid cells. Then looping over the 8 SCs plus the one where the particle stands (26 + 1 cells in 3D) provides a list of the closest particles that are stored as potential collision partners. Eventually, during the computation of the collision force, only these particles are checked.
The linked-cell method is used even if the grid is Cartesian. This choice is justified by the fact that the cells of the detection grid should probably be (i) at least as big as the particle diameter (ii) as small as possible, in order to prevent too many neighboring particles from being detected. These requirements may be hard to guarantee if the Cartesian mesh is too coarse, or locally refined (which is one of the future aims of the code). For these reasons, a Cartesian grid is superimposed in any case.

Parallelism also requires special treatment for particles, as collision might occur between some of them although they do not belong to the same processor domain. To cope with this requirement, a ghost particle method is used, which is also classical in CFD/DEM. Ghost particles are identified using a cell halo surrounding each processor domain as shown in Figure 7 for processor ranked #1 in a cylindrical geometry discretized with an unstructured mesh. The particles belonging to the cell halo are exchanged between involved processors with the necessary data to compute collision force only by the mean of non-blocking MPI exchanges. The size of the cell halo is locally determined by the cell mesh size and particle diameter to avoid unwanted distant particles from being identified as ghost particles, which is not straightforward on unstructured meshes. On each processor, ghost particles are treated to be located as well on the afore mentioned Cartesian detection grid (see Figure 6).
3. Simulation Cases

3.1. Configuration and Meshes

In this work, CFD/DEM simulations of a fluidized bed, similar to that used in the experiments reported by Patil et al. [26], are performed. A sketch of the simulated configuration of the fluidized bed is shown in Figure 8 (left) including its dimensions. The bed contains inert glass particles fluidized by fresh nitrogen injected through the bottom of the bed at different gas flowrates. Physical properties of both gas and particles are summarized in Table 1. In the experiments, the bottom area of the bed was equipped with a small circular nozzle of 1.2 cm diameter, through which no gas was supplied during the experiments. To reproduce the effect of the switched nozzle as faithfully as possible, a zero gas flow was set through an area located at the center of the lower horizontal section of the bed. This area was considered as a wall for both gas and particles and is referred to as the bottom wall in the simulations. In the simulations, three different grid refinements were used, and their characteristics are reported in Table 2, including that of the bottom wall retained for each mesh. As an example, a sketch of the intermediate mesh is illustrated in Figure 8 (right).

![Figure 8. Sketches of the simulated configuration (left) and of the intermediate mesh (right).](image)

<table>
<thead>
<tr>
<th>Table 1. Gas and particle properties [26].</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Gas Properties (at 20 °C)</strong></td>
</tr>
<tr>
<td>Density, $\rho$</td>
</tr>
<tr>
<td>Viscosity, $\mu$</td>
</tr>
<tr>
<td><strong>Particle Properties</strong></td>
</tr>
<tr>
<td>Mean diameter, $d_p$</td>
</tr>
<tr>
<td>Density, $\rho_p$</td>
</tr>
<tr>
<td>Norm. coef. of rest., $e_n$</td>
</tr>
<tr>
<td>Solid/gas density ratio, $\rho_p/\rho$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2. Mesh characteristics.</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Mesh</strong></td>
</tr>
<tr>
<td>$N_x$</td>
</tr>
<tr>
<td>Coarse</td>
</tr>
<tr>
<td>Intermediate</td>
</tr>
<tr>
<td>Fine</td>
</tr>
</tbody>
</table>
3.2. Description of the Simulation Cases

Several simulations based on different grid refinements, combined with different physical parameters, are investigated. In our modeling strategy, a friction coefficient, previously denoted as $\mu_{\text{tan}}$ in Section 2.2.1, is required to account for the tangential contact force during binary particle and particle–wall collisions. Since such a coefficient is not provided in the work of Patil et al. [26], we examined available values from the literature. As reported by Lorenz et al. [27], its value may depend on the state and history of the particle surfaces. The authors experimentally evaluated the friction coefficient of glass particles having experienced a considerable time in granular flows to 0.177 and 0.141 for inter-particle and particle-spent aluminum plate collisions, respectively. In the work of Goldschmidt et al. [28], values of 0.10 and 0.09, as measured by Gorham et al. [29], were respectively used in the modeling of the particle–particle and the particle–wall frictional collisions in a dense gas-fluidized-bed configuration. In Gorham et al., both the particles and the pseudo-2D bed were made of glass. In the following sections, we denote the particle–particle and the particle–wall friction coefficients as $\mu_p$ and $\mu_w$, respectively. All the simulations are summarized in Table 3. A mesh sensitivity analysis, drawn from simulations C1, C2, and C3, for which $\mu_p = \mu_w = 0.2$ is assumed, is first presented. In a second step, various values of the friction coefficients, $\mu_p$ and $\mu_w$, are investigated. This study refers to the sets of simulations C4 and C5, as shown in Table 3. The C4 simulations are performed using a constant value of $\mu_p$, with different values of $\mu_w$, whereas in the C5 simulations, the effect of $\mu_p$ is investigated for a constant value of $\mu_w$. From these investigations, values for the friction coefficients are chosen by comparing numerical results of the volume fraction and the axial flux of the particulate phase with the available experimental measurements. Finally, additional Reynolds number and bed weight conditions are used as variable parameters for the simulations C6 and C7. It must be mentioned that the inlet gas velocity, $U_f$, given in Table 3 is the superficial velocity through the horizontal cross-sectional area of the bed (so, this area also includes the bottom wall). For each tested gas velocity, the corresponding particle Reynolds number, given as $Re_p = \rho U_f d_p / \mu$, is reported in the table, together with the Stokes number $St = \rho d_p U_f / (18 \mu)$. It must be noted that the bulk Reynolds number based on the bed width is relatively high ($5600 \leq Re \leq 8000$). Therefore, for all the performed simulations, a turbulent viscosity based on Smagorinsky model is used. The additional parameter of interest is the restitution coefficient during particle–wall collisions, $e_w$, which is considered equal to that of the particle–particle collisions, $e_n$.

<table>
<thead>
<tr>
<th>Simulation Cases</th>
<th>Bed Mass (g) at 20 °C</th>
<th>$U_f$ (m/s)</th>
<th>$Re_p$</th>
<th>$St$</th>
<th>Friction Coefficients</th>
<th>Mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>$\mu_p = \mu_w = 0.2$</td>
<td>Coarse</td>
</tr>
<tr>
<td>C2</td>
<td>75</td>
<td>1.20</td>
<td>70</td>
<td>$\sim 4$</td>
<td>0.0</td>
<td>Intermediate</td>
</tr>
<tr>
<td>C3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.2</td>
<td>Fine</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>C4</td>
<td>75</td>
<td>1.20</td>
<td>70</td>
<td>$\sim 4$</td>
<td>0.1</td>
<td>Intermediate</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>C5</td>
<td>75</td>
<td>1.20</td>
<td>70</td>
<td>$\sim 4$</td>
<td>$\mu_p = \mu_w = 0.1$</td>
<td>Intermediate</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.8</td>
<td></td>
</tr>
<tr>
<td>C6</td>
<td>75</td>
<td>1.71</td>
<td>100</td>
<td>$\sim 5.5$</td>
<td>$\mu_p = \mu_w = 0.1$</td>
<td>Intermediate</td>
</tr>
<tr>
<td>C7</td>
<td>125</td>
<td>1.54</td>
<td>90</td>
<td>$\sim 5$</td>
<td>$\mu_p = \mu_w = 0.1$</td>
<td>Intermediate</td>
</tr>
</tbody>
</table>
4. Results

Non-dimensional results are presented in this section. The time scale is defined as \( t/(d_p/U_f) \). x-, y- and z-distances are divided by the bed width \( (L = 0.08 \text{ m}) \), the bed depth \( (D = 0.015 \text{ m}) \) and the bed height \( (H = 0.25 \text{ m}) \), respectively. The velocities are divided by the inlet gas velocity \( (U_f) \) and the mass flow rates by the inlet gas mass flow rate \( (\rho U_f) \). To characterize the bed hydrodynamics in an Eulerian fashion, the instantaneous fields of any Lagrangian variable, \( \psi_p \), are obtained by performing a spatial average over the grid cell. In addition, a 2D-behavior is considered for this fluidized-bed configuration since the hydrodynamic variations in the depth direction are negligible. This can be observed in Figure 9, which displays the instantaneous particle volume fraction, \( \alpha_p \), on different slices chosen at \( y/D = 0.2 \) (close to the back side), \( y/D = 0.5 \) (in the middle of the bed) and at \( y/D = 0.8 \) (close to the front side). As an example, in Figure 10, profiles along the depth direction at the height \( z/H = 0.16 \) are also given for the time-averaged vertical velocity of the solid phase normalized by the inlet gas velocity \( (\overline{U_p z}/U_f) \). It can be observed that variations along the depth direction of such a quantity are not strong. Thus, two-dimensional variables are computed by averaging over the bed thickness (i.e., in the \( y \)-direction). In the present work, the 2D-variables corresponding to the computed field, \( < \psi_p > \), are denoted as \( < \psi_p >_{xz} \). All the simulations are run for 20 s to reach a steady regime. In such a regime, particles and fluid hydrodynamic fields are statistically stationary and time-averaged quantities, \( < \psi_k > \), for which \( k \) denotes the gas (\( g \)) or the particulate (\( p \)) phase, may be computed for comparison with the experimental measurements.

![Figure 9. Instantaneous particle volume fraction on different slices of the bed. Simulation C2, dimensionless time \( t/(d_p/U_f) = 12,000 \), \( \mu_w = \mu_p = 0.2 \), \( Re_p = 70 \) and bed weight 75 g.](image)

![Figure 10. Profiles of the time-averaged normalized vertical velocity of the particulate phase \( (\overline{U_p z}/U_f) \) at different locations \( x/L \). Simulation C2, \( \mu_w = \mu_p = 0.2 \), \( Re_p = 70 \) and bed weight 75 g.](image)

4.1. Effect of the Grid Refinement

In this section, the simulations C1, C2, and C3 are presented and discussed. The three simulations differ from each other by their mesh size, as shown in Tables 2 and 3, and consequently by their filtering kernel length \( (\Delta x) \). However, because the kernel length is always larger than the particle
diameter, the volume-averaged equations are well posed in the sense of Jackson [7]. Snapshots of the time-averaged fields concerning the volume fraction, $\langle \alpha_p \rangle_{xz}$, and the mass flux, $\langle \alpha_p \rho_p U_p \rangle_{xz}$, of the particulate phase are depicted in Figure 11 for the different grid refinements. Experimental results are also included in the right column.

The simulations match quite well the experiments. The particle volume fraction exhibits high values (dense regions) close to the side and bottom walls and small values (dilute regions) in the center of the bed. However, the average height of the fluidized bed seems to be slightly overestimated by the simulations. The solid flux exhibits a macroscopic double mixing loop, and, on average, the particles move upward at the center of the bed and downward close to the side walls. Comparison to experiments reveals that the center of the mixing loops, as predicted by the simulations, is located slightly higher than that experimentally observed. The three simulations indicate clearly that the mesh size affects the bed hydrodynamics prediction. The reason is that particles continuously interact with the gas flow through the drag and the pressure gradient forces, which then contributes to the particle dynamics evolution. Since the mesh resolution affects the hydrodynamics prediction of the gas phase, this consequently modifies the particle dynamics predictions. The upper panel of Figure 11 shows that the finest grid leads to the best predictions, because of an accurate resolution of the gas hydrodynamics. The high solid concentrations close to the side walls, as reported by the experiments, are reproduced with a relatively good accordance by the fine-grid simulation. Nevertheless, refining the mesh does not seem to have a significant effect on the height of these dense regions from the bottom of the bed, nor on the mean bed height. The analysis of the time-averaged fields of the solid flux reveals that apart from the progressive intensification of these fields due to the increasing mesh resolution, this latter did not influence the position of the observed double mixing loop.

Figure 11. Numerical predictions of the time-averaged fields of the volume fraction, $\langle \alpha_p \rangle_{xz}$ (upper panel) and the mass flux, $\langle \alpha_p \rho_p U_p \rangle_{xz}$ (bottom panel) of the solid phase. From left to right, increasing grid refinement with $\mu_w = \mu_p = 0.2$, and comparison with the experiments (right column). $Re_p = 70$ and bed weight 75 g.

Figure 12 shows profiles of the time-averaged vertical solid mass flux normalized by the gas mass flow rate, $\langle \alpha_p \rho_p U_p \rangle_{xz} / \rho U_f$. The profiles are taken at the height $z/H = 0.092 (z = 2.3 \text{ cm})$ above the bottom of the bed, for which experimental data are available. We notice that the predictions are
improved when the finest mesh is used, especially close to the side walls. However, at the center of the bed, the effect of the bottom wall on the axial solid flux is still poorly reproduced even when a finer mesh is used.

Figure 12. Time-averaged normalized vertical mass flux of the solid phase, $\frac{\alpha_p \rho_p U_p}{\rho U_f}$, at the height $z/H = 0.092$ above the bottom bed, for different grid refinements. $\mu_w = \mu_p = 0.2$, $Re_p = 70$ and bed weight 75 g.

With the attempt to improve the numerical results, the wall boundary condition type on the gas phase was also investigated. Slip and no-slip fine-grid simulation cases were run and compared to each other. The time-averaged results (here not shown) relative to the particulate phase were very slightly affected by the selected wall condition of the gas phase. Such a behavior is inherent to inertial particles, as those simulated in the present work, for which the Stokes number is much greater than unity, as shown in Table 3.

4.2. Investigation of the Frictional Effects

Effects of the inter-particle and the particle–wall friction coefficients on the bed hydrodynamics are here discussed. In the previous study, the computational time required by the fine-grid simulation was nearly twice higher than that of the intermediate-grid simulation. Therefore, the present study is carried out using the intermediate mesh.

The time-averaged fields concerning the particle volume fraction, as obtained from the sets of simulations C4 and C5 are shown in Figure 13. As previously mentioned, a dense region is observed close to the side and bottom walls, whereas, the center of the bed is characterized by a dilute solid concentration. In addition, the average height of the fluidized bed seems to decrease as the frictional effects become stronger. However, more subtle differences between the situations can be observed. The upper panel as obtained by the simulations C4, reveals that for the same value of the particle friction coefficient, when the wall frictional effects become stronger, the dense regions become denser and larger near the side walls, while they are slightly narrowing above the bottom wall. Results also show that such regions move slightly upwards as $\mu_w$ increases. It must be remembered that the influence of the wall friction coefficient is also exerted at the front and the back walls, but should be stronger in the confined parts of the bed, namely at the side walls, where the denser regions of the bed are observed. The bottom panel, obtained by the simulations C5, shows that when $\mu_p$ increases up to 0.2, the bed hydrodynamics exhibit similar behavior as that observed when $\mu_w$ varies and $\mu_p$ is kept constant (simulations C4). Such results are in agreement with the work of Yang et al. [30], who showed the same trend of particle volume fraction distribution when $\mu_p$ increased from 0.05 to 0.15, in their comparison study between two fluid model (TFM) and discrete particle model (DPM) simulations. However, unlike simulations C4, continuing to increase the particle–particle friction coefficient above the value of 0.2 in the simulations C5 resulted in decreasing the solid concentration near the side walls and increasing it above the bottom wall. The inter-particle friction acts on the whole bed volume...
and, consequently, high values of the inter-particle friction coefficient may considerably modify the bulk-bed hydrodynamic behavior. Finally, when the particle friction is not considered ($\mu_p = 0$ for $\mu_w = 0.1$), a homogeneous fluidized bed is obtained, which is not the case for $\mu_p \neq 0$ and $\mu_w = 0$ (simulations C4). These various bed behaviors inform about the effect of the particle friction on the bubble formation. As an example, snapshots of the instantaneous bed porosity (gas volume fraction), obtained at different instants by the simulations C5 for two values of $\mu_p$, are depicted in Figure 14. In the case of frictionless particles ($\mu_p = 0$), no bubble formation occurs, whereas in the case of relatively frictional particles ($\mu_p = 0.2$), a realistic bubble growth is observed. This is consistent with the work of Hoomans et al. [31], who showed the same bubble patterns in 2D-fluidized-bed DPM simulations.

Figure 13. Time-averaged particle volume fraction, $<\alpha_p>_{zz}$, for different values of the particle–wall $\mu_w$ and the particle–particle $\mu_p$ friction coefficients. Upper panel: constant $\mu_p$ and different $\mu_w$. Bottom panel: constant $\mu_w$ and different $\mu_p$. $Re_p = 70$ and bed weight 75 g.
Figure 14. Instantaneous gas volume fraction, with $\mu_w = 0.1$. Upper panel: $\mu_p = 0$. Bottom panel: $\mu_p = 0.2$. $Re_p = 70$ and bed weight 75 g.

Time-averaged particle agitation energy, $q_{p}^{2}$, has been computed as $q_{p}^{2} = \frac{u'_{p,i}u'_{p,i}}{U_{p,i}}$, where $u'_{p,i} = u_{p,i} - U_{p,i}$. In these expressions, $u'_{p,i}$ and $U_{p,i}$ are the $i^{th}$ components of the fluctuating and the mean particle velocities, respectively, with the mean particle velocity computed as $U_{p,i} = \langle u_{p,i} \rangle$. The normalized results are shown in Figure 15 for the two sets of simulations. The first panel obtained from the set C4 shows that most of the particle agitation is produced at the approximate height $z/h = 0.25$. This height corresponds to the average height of the dense bed ($\alpha_p > 0.2$). Furthermore, it increases when friction at wall becomes stronger. Results of the simulations C5, as depicted in the bottom panel, show a decline, on average, of the particle agitation energy when the particle friction increases. In addition, some particle agitation is produced at the height $z/h = 0.25$, similarly to what is observed in the C4 simulations. On the contrary, fewer amount of particle agitation is produced as the particle frictional effects become more significant. These results show that for low friction between particles, the fluidized bed may be considerably heterogeneous when friction at wall is high and become homogeneous when friction between particles is strong. It must be noted that in all these simulations, the computed particle agitation energy in the upper part of the bed ($z/H \geq 0.375$), has a very low significance since, in this part, the particle volume fraction is nearly zero.
Figure 15. Time-averaged dimensionless particle agitation energy, $q_p^2/U_f^2$ for different values of the particle–wall $\mu_w$ and the particle–particle $\mu_p$ friction coefficients. Upper panel: constant $\mu_p$ and different $\mu_w$. Bottom panel: constant $\mu_w$ and different $\mu_p$. Re$_p$ = 70 and bed weight 75 g.

Figure 16 shows the time-averaged field of the solid mass flux for the two sets of simulations. From the visualizations, it appears that friction significantly affects the magnitude of the solid flux and the extent of the mixing loops. In some cases, it also affects their position from the bottom of the bed. As previously observed for the particle volume fraction distribution, the position of the center of the double mixing loop moves upwards when $\mu_w$ increases (upper panel). This point is not straightforward when $\mu_p$ is increased (bottom panel). However, qualitatively speaking, the magnitude of the solid flux exhibits a maximum for $\mu_p = 0.2$ in both sets of simulations.
A quantitative analysis of the solid flux may be performed via its vertical profiles at a given height of the bed. Profiles of the time-averaged normalized vertical solid mass flux are depicted in Figure 17a,b for the two sets of simulations. The profiles are taken at the height $z/H = 0.092$ ($z = 2.3$ cm) above the bottom of the bed, for which experimental data are available. In all the simulations, two distinct parts for each profile can be observed. The first part corresponds to a nearly flat profile located in the region spanning the center of the bed and the second one is a sloped profile starting from the side walls. The expansion of one part to the detriment of the other depends on the values of the friction coefficients $\mu_w$ and $\mu_p$. These coefficients also determine the maximum values of the upward and the downward mass fluxes in the flat and the sloped parts, respectively. The figures show that when $\mu_w$ or $\mu_p$ increases up to 0.2, the downward flux increases, and the upward flux, close to the center of the bed, decreases. This leads, at the same time, to a steeper slope of the solid mass flux profile in the sloped part and to a less broad flat part at the center of the bed. For a stronger friction, either at wall or between particles, larger sloped parts are still observed, but with less steep slopes, leading to the decrease of both the downward and the upward solid fluxes. Therefore, despite the differences observed concerning the volume fraction and the agitation energy of the particulate phase, simulations C4 and C5 predict similar characteristics (field and magnitude) of the solid mass flux. It is conjectured that in this confined configuration, when the wall friction coefficient increases, the front and back walls may considerably contribute together with the side walls to propagate stronger wall frictional effects towards the bulk bed. This has already been shown in a previous CFD-DEM study of Li et al. [32] for bed thicknesses of 10 and 20 particle diameters in a bubbling fluidized bed. Consequently, stronger wall friction may produce almost the same effect on the solid flux as that induced by the friction between particles.

Finally, comparison with the experimental measurements shows that the case with $\mu_p = \mu_w = 0.1$, is particularly interesting as it allows to reproduce the effect of the bottom wall on the axial solid flux (Figure 17) and to accurately predict both the location from the bottom of the bed and the shape of the solid mixing loops (Figure 16 vs. experimental data from Figure 11). However, the solid concentration close to the side walls seems to be slightly overestimated (Figure 13 vs. experimental data from Figure 11).
Figure 17. Time-averaged vertical solid mass flux normalized by the gas mass flow rate, \( \frac{\alpha_p \rho U_p}{\rho U_f} \), for different values of the particle–wall and particle–particle friction coefficients, and comparison with experiments. \( Re_p = 70 \) and bed weight 75 g.

4.3. Application of the Results to Other Physical Configurations

To generalize the previous results to different physical configurations, other Reynolds number and bed weight conditions are investigated using the intermediate mesh. The same value of \( \mu_w \) and \( \mu_p \) equal to 0.1 is used in the simulations since, as shown in Section 4.2, this leads to minor deviations from the experiments conducted for \( Re_p = 70 \). A gas no-slip condition at the wall is also retained. In the simulation C6, a superficial gas velocity of 1.71 m/s corresponding to \( Re_p = 100 \), as reported in Table 3, is imposed at the bottom of the bed. Predictions obtained by this simulation concerning the time-averaged fields of volume fraction and mass flux of the solid phase are reported in Figure 18. Experimental measurements are also included for comparison purpose. One can see that the simulation reproduces very well the experimental solid distribution and the solid mass flux field. However, slight mismatches can be observed concerning the mean bed height and the position of the solid mixing loops. At the height \( z/H = 0.092 \), numerical and experimental vertical mass fluxes are displayed on Figure 19. Globally, results are conveniently reproduced by the simulation and confirm the accuracy of the selection \( \mu_w = \mu_p = 0.1 \).

(a) \( \alpha_p \) vs (b) \( \alpha_p \rho U_p \) vs z

Figure 18. Numerical (a) against experimental (b) time-averaged fields of the volume fraction and the mass flux of the solid phase. \( Re_p = 100 \) and bed weight 75 g. In the simulation, \( \mu_w = \mu_p = 0.1 \).
Fluids 2019, 4, 51

Figure 19. Numerical against experimental time-averaged vertical mass flux, $\langle \alpha_p \rho_p U_p \rangle_{xz}/\rho U_f$, of the solid at the height $z/H = 0.092$ above the bottom of the bed. $Re_p = 100$ and bed weight 75 g. In the simulation, $\mu_w = \mu_p = 0.1$.

Results of a fluidization experiment of 125 g of particles by nitrogen injected at a velocity of 1.54 m/s, corresponding to $Re_p = 90$, have also been reported in Patil et al. [26]. This operating point is simulated in the present work (simulation C7) and results displayed in Figure 20a,b, together with the experimental data. In general, good agreements can be observed concerning the time-averaged distributions of the volume fraction and the mass flux of the particulate phase. Nevertheless, the deviation from the experimental average height of the bed is still exhibited, although the friction coefficient seems to be accurately estimated.

The observed difference between our simulations and experiments for the average bed height is related to the drag law chosen to model the hydrodynamic force.

Figure 20. Numerical (a) against experimental (b) time-averaged fields of the volume fraction and the mass flux of the solid phase. $Re_p = 90$ and bed weight 125g. In the simulation, $\mu_w = \mu_p = 0.1$.

5. Conclusions

In this work, a DEM/LES approach is used to simulate a confined fluidized-bed configuration for several physical and numerical conditions. Results are compared with experimental data of [26]. This study shows that compared with the experimental measurements, predictions regarding the volume fraction and the mass flux of the particulate phase, obtained from the fine grid, are slightly better than that of the coarse and the intermediate grids. This slight improvement is attributed to a better prediction of the gas–particle interactions through the drag and the pressure gradient forces. Another simulation, using the no-slip wall condition of the gas phase, is then performed using the fine grid and
results compared with the slip wall fine-grid simulation. This investigation shows that in the case of inertial particles, the gas velocity boundary condition at the walls is of minor importance in predicting the particulate phase behavior, since the volume fraction and the solid circulation patterns from the slip and the no-slip wall simulations are very similar. The present work is also dedicated to frictional effects between particles and between particles and walls on the bed hydrodynamics behavior. It is shown that after increasing either the coefficient of the inter-particle friction or that of the particle–wall friction, the average bed height decreases, and the bubble formation is enhanced. At low friction between particles, increasing the wall friction coefficient leads to similar solid circulation patterns (fields and magnitudes) as in the simulations with increasing friction between particles. Thus, it is conjectured that in such confined 2D-configuration, when the wall friction coefficient increases, the friction of the front and back walls may have a significant impact on the inner-flow hydrodynamics of the fluidized bed. This finally may lead to the same bed hydrodynamics behavior as that observed when the friction coefficient between particles increases. Nevertheless, this point must be further investigated. Indeed, to clearly understand the contribution of the friction at the front and back walls, simulations with various bed thicknesses combined with different values of the friction coefficients must be performed. Additional simulations, performed at higher Reynolds numbers and/or other bed weights, showed globally good agreement with the experiments, reproducing almost similar particle volume fraction and solid circulation patterns as the experiments. However, deviations from the experiments concerning the average height of the dense bed is still exhibited in all the simulations. This could be further enhanced by studying other modeling features, such as the drag law.

Author Contributions: Investigation, Z.H.; Methodology, Y.D.; Supervision, J.-L.P., R.B., G.L. and V.M.

Funding: Computations were performed on the IFPEN supercomputer ENER110 and on the supercomputer CURIE under the project GENCI A0032B07345. The ENER110 and GENCI supercomputers teams are gratefully acknowledged. This work is done in the frame of the MORE4LESS project funded by the French National Research Agency.

Acknowledgments: Olivier Simonin at Institut National Polytechnique de Toulouse (France) is gratefully acknowledged for fruitful discussions throughout this work.

Conflicts of Interest: The authors declare no conflict of interest. The funders had no role in the design of the study; in the collection, analyses, or interpretation of data; in the writing of the manuscript, or in the decision to publish the results.

Abbreviations

The following abbreviations are used in this manuscript:

- CFL: Courant-Friedrichs-Lewy
- DEM: Discrete Element Method
- DPCG: Deflated Preconditioned Conjugate Gradient
- DPM: Discrete Particle Model
- DNS: Direct Numerical Simulation
- LES: Large-Eddy Simulation
- MPI: Message Passing Interface
- PCM: Particle Centroid Method
- RK: Runge-Kutta
- SC: Surrounding Cell
- TFM: Two Fluid Model
- BN: Blue Nodes
- GN: Green Nodes
- RN: Red Nodes
References


© 2019 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (http://creativecommons.org/licenses/by/4.0/).