Direct simulation of granular flow with fluid applied to sedimentation

Vincent KOMIWES\textsuperscript{1}, Philippe MEGE\textsuperscript{1}, Yves Meimon\textsuperscript{1} and Hans HERRMANN\textsuperscript{2}

Abstract A numerical model applied to simulation of granular flow with fluid are presented. The physical model selected to describe particles flow is a discrete approach. Particle trajectories are calculated by Newton law and collision is describe by a soft-sphere approach. The fluid flow is modelled by Navier-Stokes equation. The modelling of the momentum transfer is directly calculated by stress tensor computation around particles. This model is validated by the calculation of the coefficient of drag, making it possible to leave the limitations on the Reynolds number according to the size of the mesh and complexity of the computational time. This model is then used for the sedimentation of 2 particles to reproduce Drift, Kiss and Tumbled. This shows the capacity of the model to reproduce hydrodynamic interactions acting on the scale of the particle. The terminals velocity of particles are in good agreement with the experimental laws. Simulations of the sedimentation of a system of particles makes it possible to find the law of Richardson and Zaki, to validate its local character with an acceptable time CPU. The impact of the size of the mesh shows that the dense mediums impose a size of mesh smaller than for the diluted mediums.

1 Introduction

Several industrial processes which use mobile beds, pneumatic transport, fluidized catalyst beds. Their correct operation is related to the comprehension of the behavior of the granular bed in interaction with a fluid. However, the internal hydrodynamics such processes is still badly known, bringing into play physical phenomena acting on the scale of the particle. The numerical simulation is a tool which must make it possible to understand these phenomena in order to model them. Several models can be used to simulate the diphasic flows of fluid-solid type:

1. a continuous model Eulerian-Eulerian for the phases liquid and solid [1], [2], [3], [4], [5] more popular in the engineering sciences.

In this model, there are remaining open questions in the determination of the proper constitutive equation for the solid and the modelling of the momentum transfer between the solid and fluid phases. In general, the modelling of the momentum transfer is made by a local drag force depending on the local relative velocity between the solid and the fluid, the Reynolds number and the local volume fraction of the solid and the fluid. This drag law is partly based on the Ergun’s law [6] given the pressure drop in a granular media.

2. a Eulerian-Lagrangian model for the fluid and solid phases respectively [7], [8], [9] that implement with finite element method no-slip boundary conditions on the surface of each particles have been employed for very few particles.

3. a Eulerian model for the fluid phase coupled to a discrete approach for the solid phase. [10], [11], [12], [13], [14], [15], [16], [17] use a drag law, [18] use an idea of Fogelson and Peskin [19] and [20], [21] use the integration of stress tensor around the surface of particles.

In this paper, a model of type 3, is presented: The physical model selected to describe particles flow is a discrete approach. Particle trajectories are calculated by Newton law and collision is describe by a soft-sphere approach. The fluid flow is modelled by Navier-Stokes equation and is solved on a staggered grid. The modelling of the momentum transfer is directly calculated by stress tensor computation around each particles.

2 Model

2.1 Particle flow

The physical model selected to describe particles flow is a discrete approach [22], [23], [24]. Particle trajectories are calculated by Newton law and collision is describe by a soft-sphere approach as follow:

\[ f_g + f_n + f_f + f_h = m_{ip} \frac{d^2 \mathbf{x}_{ip}}{dt^2} \quad (i_p = 1, \ldots, n_p) \quad (1) \]

The gravity force \( f_g \) is given by:

\[ f_g = -m_{ip} g \mathbf{a}_{\text{grav}} \quad (2) \]

with:

\[ m_{ip} = 4 \pi r_{ip}^3 \rho_p \quad (3) \]

The collision force in the normal direction is given by:

\[ f_n = f_d + f_{en} \quad (4) \]

and in the tangential direction by:

\[ f_t = -\min(\mu_c ||f_n||, ||f_{ct} + f_{et}||) t \quad (5) \]

with \( \mu_c \) is the Coulomb coefficient of friction

\[ f_{et} = -k_n [(r_{ip} + r_{jp}) - \delta r] \mathbf{n} \quad (6) \]

Vincent KOMIWES, Philippe MEGE, Yves Meimon and Hans HERRMANN

Institut Français du Pétrole
69360 SOLAIZe - FRANCE,

Institut für Computerphysik, Pfaffenwaldring 27
D-70569 Stuttgart
with:

\[ n = \frac{\mathbf{x}_{i_p} - \mathbf{x}_{j_p}}{||\mathbf{x}_{i_p} - \mathbf{x}_{j_p}||} \]  
(7)

\[ k_n, r_{p} \text{ et } r_{p}' \text{ are respectively the elastic coefficient, radius of particle } i_p \text{ and } j_p \]  
(8)

\[ f_d = -k_n[(r_{p} + r_{p}')] - \delta r]n \]  
(9)

\[ f_{fr} = -2\gamma_n \mathbf{m}_n \mathbf{v}_n \]  
(10)

\[ \gamma_n \text{ is the normal coefficient of dynamic friction with:} \]  
(11)

\[ \mathbf{v}_n = (\mathbf{v}_{i_p} - \mathbf{v}_{j_p}) \cdot n \]  
(12)

\[ \delta r = (\mathbf{x}_{i_p} - \mathbf{x}_{j_p}) \cdot n \]  
(13)

\[ m_{re} = \frac{m_{i_p} m_{j_p}}{m_{i_p} + m_{j_p}} \]  
(14)

\[ f_{fr} = -2\gamma m_{re} \mathbf{v}_n \]  
(15)

\[ \mathbf{v}_t = \mathbf{v}_{i_p} - \mathbf{v}_n \]  
(16)

\[ f_s = -k_s \int_{0}^{t_c} \mathbf{v}_t \, dt \]  
with:

\[ \mathbf{x}_{i_p}, m_{i_p}, n_{p}, k_s, \text{ and } t_c \text{ are respectively the position, the mass of the particle } i_p \text{, the total number of particles, the coefficient of static friction and the time of the choc where } t_c \text{ is the time of the choc.} \]

### 2.1.1 Adimensional equations of particle flow motion

The variables without dimension are defined from the following characteristic sizes: the average radius \( \bar{r} \) for the length, Stokes velocity \( V_s \) for the velocity, the fluid density \( \rho_f \):

\[ \mathbf{x}_{i_p} = \bar{r}\mathbf{x}_{i_p} \]  
(17)

\[ t = \frac{\bar{r}}{V_s \cdot t} \]  
(18)

\[ \mathbf{v}_{i_p} = V_s \mathbf{v}_{i_p} \]  
(19)

By replacing these quantities in the equation of the movement, the adimensional equation of the movement is written:

\[ f_g^* + f_s^* + f_{fr}^* + f_{frn}^* + f_n^* = m_{i_p}^* \frac{d^2 \mathbf{x}_{i_p}^*}{dt^2} \quad (i_p = 1, \ldots, n_{p}) \]  
(20)

\[ f_g^* = -m_{i_p}^* \bar{r} \mathbf{u}_{k_{eq}} \]  
(21)

\[ f_{fr}^* = -2\gamma \mathbf{m}_n \mathbf{v}_n \]  
(22)

\[ f_{frn}^* = -k_n \int_{0}^{t_c} \mathbf{v}_t^* \, dt \]  
with:

\[ r_{p}^* = \frac{r_{p}}{\bar{r}} \]  
(23)

\[ m_{i_p}^* = \frac{m_{i_p}}{\bar{r}^2} \]  
(24)

\[ \gamma_n = \frac{\gamma_n}{\bar{r}} \]  
(25)

The velocity at the time step \( (n + \frac{1}{2}) \Delta t_p \) and the position at the time step \( (n + 1) \Delta t_p \) are obtained by soustraining the equations (25) and (26):

\[ \mathbf{v}_{i_p}^{n + \frac{1}{2}} = \mathbf{v}_{i_p}^{n - \frac{1}{2}} + \Delta t_p m_{i_p} a_{i_p}^{n} + O(\Delta t_p^3) \]  
(26)

\[ \mathbf{v}_{i_p}^{n + 1} = \mathbf{v}_{i_p}^{n} + \Delta t_p m_{i_p} a_{i_p}^{n} + O(\Delta t_p^3) \]  
(27)

\[ a_{i_p}^{n} = \frac{f_{frn}^*}{m_{i_p}} \]  
(28)

\[ \mathbf{x}_{i_p}^{n + 1} = \mathbf{x}_{i_p}^{n} + \Delta t_p m_{i_p} v_{i_p}^{n + \frac{1}{2}} + O(\Delta t_p^3) \]  
(29)

### 2.1.2 Numerical scheme

The integration of the equations of the movement is carried out by using the explicit Leap-Frog scheme:

\[ \mathbf{x}_{i_p}^{n} = \mathbf{x}_{i_p}(n \Delta t), \quad \mathbf{v}_{i_p}^{n} = \mathbf{v}_{i_p}(n \Delta t_p), \quad f_{frn}^{n} = f_{frn}(n \Delta t_p) \]

\[ a_{i_p}^{n} = a_{i_p}(n \Delta t_p) \quad \text{on } n = 0, 1/2, 1, \ldots \]

Leap-Frog is obtained by writing the following developments of Taylor:

\[ \mathbf{v}_{i_p}^{n - \frac{1}{2}} = \mathbf{v}_{i_p} + \frac{\Delta t_p}{2} \mathbf{a}_{i_p}^{n} + \frac{\Delta t_p^2}{4} \mathbf{v}_{i_p} + O(\Delta t_p^3) \]  
(30)

\[ \mathbf{v}_{i_p}^{n + \frac{1}{2}} = \mathbf{v}_{i_p} - \frac{\Delta t_p}{2} \mathbf{a}_{i_p}^{n} + \frac{\Delta t_p^2}{4} \mathbf{v}_{i_p} + O(\Delta t_p^3) \]  
(31)

\[ \mathbf{a}_{i_p}^{n} = \mathbf{a}_{i_p}^{n - \frac{1}{2}} + \Delta t_p \mathbf{v}_{i_p}^{n + \frac{1}{2}} + O(\Delta t_p^3) \]  
(32)

\[ \mathbf{x}_{i_p}^{n} = \mathbf{x}_{i_p} + \Delta t_p \mathbf{v}_{i_p}^{n} + \frac{\Delta t_p^2}{2} \mathbf{a}_{i_p}^{n} + O(\Delta t_p^3) \]  
(33)

\[ \mathbf{x}_{i_p}^{n + 1} = \mathbf{x}_{i_p} + \Delta t_p \mathbf{v}_{i_p}^{n + \frac{1}{2}} + O(\Delta t_p^3) \]  
(34)

### 2.1.3 Linked-Cell algorithm

The integration of the equations of the movement requires for a given particle the calculation of the forces of interaction between the \( (n_p - 1) \) other particles. The resulting cost is \( O(n_p^2) \).
The use of an algorithm of research of the possible neighbors of collisions of linked-cell type is used to reduce complexity by dividing space in cells containing of the particles. The particles of a cell are considered not to be able to interact with the other particles of the cell itself or with the particles of the cells neighbor. Once the space is divided in cells and the particles indexed by cell, the calculation of the interaction forces carries out itself while traversing the every cell particles and of its cells neighbor. This algorithm is in $O(n_p)$ and has the advantage of well to lend itself to the parallelisation [23, 25].

The first step of algorithm consist in to calculate the size of the cells $Cdim[i]$ and the number of cells present in every direction: $Cdim[1]$, $Cdim[2]$ and $Cdim[3]$. The size of the cells is superior to 2 times the maximum radius of the particles.

The second step consist in to index the particles in the cells as follows:

- a cell $cell[i][j][k]$ of coordinated ones $i,j$ and $k$ is an entire type containing the number of the first particle found in the cell or containing the entire one any in the case where the cell is empty. This particle found contains an entire type containing the number of a particle neighbors of this cell or containing the entire one any. This is simplified for a case bidimensional in Fig. 2. below.

While designating by $p[i].next$ the used variable to index the particles present in the cell of the Fig. 2., algorithm would give the following values:

- $cell[2][3] = 2, p[2].next = 0$
- $cell[4][3] = 4, p[4].next = 11, p[4][3].next = 0$

The other cells are indexed by the entire one any.

While designating by $ip$ the variable representing the number of the particles, the algorithm of remissage of the cells is given hereafter where $lb[i][j][k]$ is an intermediary array:

$$\text{for } i = 0 \text{ to } n_p$$
$$p[i].next = 0$$

The calculation of the interaction forces carries out itself cell by cell, while traversing the cells neighbor. For a three-dimensional case, every cell contains in average the following number of particles:


As every cell possesses 26 cells neighbor, the number of interactions to calculate is of $26n_p n_c$.

While using the principle of the action and reaction, alone the cells neighbor on the inferior part are traversed (see Fig. 3. below). The cost of the algorithm for the calculation of the forces of interactions is divided by two, be on the order of $13n_p n_c$.

### 2.2 Fluid flow

The fluid is considered incompressible and Newtonian. The variables without dimension are defined from the following characteristic values: the average radius $r$ for the length, Stokes velocity $V_s$ for the velocity, the fluid density $\rho_f$. The adimensional Navier-Stokes equations describe the motion of interstitial fluid flow with resolution scale $h$ smaller than the particle diameter $d_p$ as shown by Fig. 4:

$$\frac{\partial v_i}{\partial t} = -\frac{\partial (r v_j)}{\partial x_j} - \frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 v_i}{\partial x_i^2} + f_i$$

with:

$$f = f_i u_{x_i} = -Fr u_{x_i}$$
and the conservation of the fluid mass give with incompressibility consideration:
\[
\frac{\partial \psi}{\partial x_i} = 0
\]
where the Reynolds number is given by:
\[
Re = \frac{\rho f V_y}{V_x}
\]
and the Froude number is given by:
\[
Fr = \frac{\frac{g}{V_x}}{\frac{v_y}{V_x}}
\]

2.2.1 Numerical scheme
An explicit operator splitting, fractional-time-step method, first order in time proposed by Chorin [26] is used as the following to solve Navier-Stokes equation:
\[
\frac{v_i^{n+1} - v_i^n}{\Delta t_f} = -\frac{\partial p^n_{i,j} + 1}{\partial x_j} + \frac{1}{Re} \Delta \psi_i^n
\]
The introduction of the quantity \(v_i^{n+\frac{1}{2}}\)
\[
\frac{v_i^{n+\frac{1}{2}} - v_i^n}{\Delta t_f} = \frac{(v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}})}{\Delta t_f} - \frac{\partial (v_i^{n+\frac{1}{2}})}{\partial x_j} + \frac{1}{Re} \Delta \psi_i^n
\]
and
\[
\frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{\Delta t_f} = \frac{\partial p^{n+\frac{1}{2}}}{\partial x_i}
\]
by taking the divergence from equation (49) give a Poisson Equation for pressure:
\[
\frac{\partial^2 p^{n+\frac{1}{2}}}{\partial x_j x_j} = \frac{1}{\Delta t_f} \frac{\partial v_i^{n+\frac{1}{2}}}{\partial x_i}
\]
A staggered marker and cell (MAC) mesh as the base for a second order spatial finite difference centered discretization is used for the each partial differential terms [27] (see Fig. 5.)
The discrete Poisson equation for pressure is given by:
\[
\frac{1}{h^2} (p_{i,j+1,m}^{n+1} + p_{i,j-1,m+1}^{n+1} + p_{i,j-1,m}^{n+1} + p_{i,j+1,m}^{n+1}) = \frac{1}{h} \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h} + \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h} + \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h}
\]
(51)
The centered discretization produce exterior node, for example:
\[
v_1^{n+\frac{1}{2},-1,i-\frac{1}{2},m} = v_1^{n+\frac{1}{2},i-\frac{1}{2},m} + \frac{1}{h} \psi_i^{n+\frac{1}{2}}
\]
The velocity value at this node is determined by a linear extrapolation. For example at the node \((-1, l + \frac{1}{2}, m)\), we have:
\[
v_2^{n+\frac{1}{2},-1,i+\frac{1}{2},m} = 2v_2 - v_2^{n+\frac{1}{2}}
\]
(52)
For other boundary node, to fix the value of \(v_1^{n+\frac{1}{2},i-\frac{1}{2},m}\) for example, we write:
\[
v_1^{n+\frac{1}{2},-1,i-\frac{1}{2},m} = v_1^{n+\frac{1}{2},-1,i-\frac{1}{2},m}
\]
(53)
The boundary condition for pressure is determined by calculated the scalar product of discretized equation (49) with the normal unit vector of the domain boundary:
\[
\frac{p_{i,n}^{n+1} - p_{i-1,n}^{n+1}}{h} = \frac{1}{h} (v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}})
\]
Replaces (54) in (51), the quantity \(v_i^{n+\frac{1}{2}}\) is cancelled and give for the boundary node:
\[
v_1^{n+\frac{1}{2},-1,i-\frac{1}{2},m} = v_1^{n+\frac{1}{2},i-\frac{1}{2},m}
\]
(55)
\[
p_{i,n}^{n+1} = p_{i-1,n}^{n+1}
\]
(56)
The poisson equation (51) for pressure with the boundary condition (56) is solved by the conjugate gradient method [28].

2.3 Momentum transfer : the particle-fluid coupling
The momentum transfer between the fluid and the particle is directly calculated by integration of stress constraint on the surface of the particle given the hydrodynamic force:
\[
\int_{S_{p}} \sigma_{ij,nj'd} S_{ip} =
\int_{S_{p}} \int_{p_{i,n} S_{ip}} p_{i,n} dS_{ip} + \frac{2}{Re} \frac{\partial v_i^{n+1}}{\partial x_j} n_j
\]
(57)
The discrete expression of the hydrodynamic force is given by:
\[
(f_{h_{ip}})_{i} = \sum_{node(k,l,m) of the particle surface} \sum_{node(k,l,m) of the particle surface} (f_{h_{ip}})_{i}
\]
(58)
with:
\[
(f_{h_{ip}})_{i} = -\frac{1}{Re} \sigma_{k,l,m} n_{i,k,l,m} h^2
\]
(59)
\(n_{i,k,l,m}\) is the unit normal vector of the surface of the particle \(i\) at node \((k,l,m)\)
\(\sigma_{i,k,l,m}\) is given by:
\[
\frac{1}{h^2} (p_{i,j+1,m}^{n+1} + p_{i,j-1,m+1}^{n+1} + p_{i,j-1,m}^{n+1} + p_{i,j+1,m}^{n+1}) = \frac{1}{h} \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h} + \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h} + \frac{v_i^{n+\frac{1}{2}} - v_i^{n+\frac{1}{2}}}{h}
\]
Fig. 5. Staggered marker and cell (MAC) mesh.

\[ m_{\text{max}} \] is the necessary number of time step for the particle collision.

**Drag law**

### 3.1 Empirical expression of the drag law

The drag coefficient \( C_d \) on a spherical particle of radius \( r \) fixed in a fluid flow is defined by:

\[
C_d = \frac{f_h}{\rho_f S_F U_s^2} \tag{67}
\]

\( f_h \) is the norm of the hydrodynamic force acting on the spherical particle, \( U_s \) is the superficial mean velocity and \( S_F = \pi r^2 \) is the frontal surface of the particle.

With the Stokes approximation \( Re << 1 \), the hydrodynamic force is calculated by the analytic formula:

\[
f_h = 6\pi \mu U_s \tag{68}
\]

and give the following drag coefficient:

\[
C_d = \frac{24}{Re^\prime} \tag{69}
\]

with: \( Re^\prime = \frac{2 r U_s \rho_f}{\mu_f} \)

Schiller and Naumann validated with experiment at another Reynolds number an empirical drag coefficient given by:

\[
C_d = \frac{24}{Re^\prime(1 + 0.15 Re^\prime)^{0.87}} \tag{70}
\]

### 3.2 Parameters and boundary conditions

The dimension of the simulation box are: \( L_x = 20r \), \( L_y = 40r \) et \( L_z = 20r \).

We are not seen change of results with a bigger box (relative error less than 1%).

The particle is placed and fixed on the position (see Fig. 6): \( x_p = (\frac{L_x}{2}, \frac{L_y}{4}, \frac{L_z}{2}) \).
The mesh size take the value : $h_1 = 0.4r$, $h_2 = 0.2r$ at fixed Reynolds number. The time step is calculated by the stability condition (64).

Reynolds number is a variable parameter of the study at fixed mesh size.

A uniform adimensional velocity $U_s = 1.0$ is used as boundary condition on $x_2 = 0$. The pressure is set at 0 on $x_2 = Lx_2$.

### 3.3 Results

The Fig. 7, show the drag coefficient calculated by the simulations as a function of Reynolds Number $Re'$. Expression (70) of drag coefficient for $Re' \leq 20$ is calculated with a maximal error of 14% with a mesh size of $h_1 = 0.4r$ and a maximal error of 10% with a mesh size of $h_2 = 0.2r$. The simulation is in good agreement with the drag empirical law coefficient of Schiller and Nauman for $Re' \leq 20$.

The more important Reynolds numbers requires a small mesh size given by the relation (71):

$$\frac{h}{r} = O(Re^{-2})$$

and consume too much computational time because the complexity of the computational time is $O(k^5)$, where $k$ is the reduction ratio of the mesh size ($O(k^2)$ for the stability condition (64) and $O(k^3)$ for the number of nodes). For example, when a smaller mesh size $h_2 = 0.2r$ is used instead of $h_1 = 0.4r$, the reduction ratio is $k = 2$ and a CPU time is multiplied by $2^5 = 32$.

### 4 Draft, Kiss and Tumbled

The model is used to simulate the sedimentation of two particles and to reproduce the Draft, Kiss and Tumbled phenomena.

#### 4.1 Description of phenomena

The Draft, Kiss and Tumbled phenomena may be observed when two particles fall under the gravity force action in a viscous Newtonian fluid. This phenomena is described by the following steps named Draft, Kiss and Tumbled: the particle 1 is first placed above the hydrodynamic drag of the particle 2. As the hydrodynamic drag of the particle 2 is a depression zone, the particle 1 is attracted inside: the Draft. One by one, the particle 1 increase his vertical velocity until touch the particle 2: the Kiss. The particle 2 is turned: his horizontal velocity increase and his vertical velocity decrease more than the particle 1. The particle 1 with a same horizontal velocity, with higher vertical velocity than particle 2 overtake the particle 2: the Tumbled.

### 4.2 Parameters and boundary conditions

We used the following physical parameters: $\rho_f = 1000kg.m^{-3}$, $\rho_p = 2000kg.m^{-3}$, $\mu_f = 10^{-3}kg.s.m^{-1}$, $d_p = 200\mu m$ and give the 3 adimensional parameters:

$Re = 1.2, \; St = 0.97 \; et \; Fr = 1.2$.

The mesh size is $h = 0.4r$ and the stability condition gives the following time step $\Delta t_f = 5.8 \times 10^{-2}$. The dimensions
of the simulation box are: $L_{x_1} = 12\bar{r}$, $L_{x_2} = 24\bar{r}$ et $L_{x_3} = 12\bar{r}$.
We use periodic boundary conditions on velocity and pressure of the fluid on each space direction.

4.3 Results

The simulation describe the Draft, Kiss and Tumbled phenomena of the sedimentation of 2 particles under gravity and Fig. 8 and Fig. 9 show the velocity and pressure fields of the fluid $x_3 = \frac{L_{x_3}}{2}$ at respective time $t = 14, 77, 88, 126$ corresponding to the draft, kiss and tumbled.

The Fig. 10. show the evolution in time of the horizontal and vertical velocities of the 2 particles.

The horizontal and vertical velocities of the 2 particles are in agreement with the description of the Draft, Kiss and Tumbled phenomena: One by one, the particle 1 increase his vertical velocity more than the particle 2 until touch the particle 2: The Kiss arrive at time $t = 70.0$. After this time, the vertical velocity of particle 2 decrease and his vertical velocity increase. The terminals velocities of the 2 particles are in good agreement with the terminal velocity of a single particle: found by independant simulation and calculated by the empirical Schiller and Nauman formula (70).
5

Sedimentation of a system of particles

5.1 The empirical law of Richardson and Zaki

The mean velocity $\bar{v}$ of a system of particle sedimenting in a fluid is correlated with the volume concentration of solid $\phi$ by the Richardson and Zaki [30] law:

$$\bar{v}(\phi) = v_s (1 - \phi)^n$$  \hspace{1cm} (72)

where $v_s$ is the Stokes velocity of an isolated particle.

The exponent $n$ is a function of the Reynolds number and is close to 5 for Reynolds number $Re \leq 1$. This law is validated by many experiments and in particular by Nicolai [31].

5.2 Parameters

We use the following physical parameters: $\rho_f = 1000 \text{kg.m}^{-3}$, $\rho_p = 2500 \text{kg.m}^{-3}$, $\mu_f = 10^{-3} \text{kg.s.m}^{-1}$, $d_p = 100 \mu m$

and we calculate: $Re = 0.41$, $Fr = 7.4$ et $St = 0.23$ and a adimensional time step of $\Delta t_f = 10^{-2}$.

The dimensions of the simulation box are:

$L_x_1 = 40r$, $L_x_2 = 40r$ et $L_z = 40r$.

Periodic boundary conditions are used on each space direction.

5.3 Results

The simulations carried out for sizes of meshes $h_1 = 0.4r$ and $h_2 = 0.2r$ make it possible to find the law (72) for voluminal concentrations of increasingly important solid $\phi$ (see Fig. 12). These simulations make it possible to leave the limitations on the level of the size of the mesh: direct simulations in diluted medium can be carried out with a grid of coarse size $h_1 = 0.4r$ and in dense medium ($\phi < 0.2$) must be necessarily realized with a size of fine mesh of $h_2 = 0.2r$, to see even finer to be able to simulate mediums even denser. The reason of these limitations same as that is evoked for the preceding application and comes owing to the fact that the denser the medium is, the more the interstices where the fluid can pass are small what requires a finer resolution.

These results validate by the same occasion the local character of the empirical law of Richardson-Zaki.

6

CPU time

The CPU time is about 6h3s per node and per time step on a Silicon Graphics Workstation Octane with a processor R10 000.

With the same physical parameters and the same dimensions of the simulation box $L_{x1} = 40r$, $L_{x2} = 40r$ et $L_z = 40r$ used in 5.2, a number of particles $n_p = 500$, a mesh size $h_1 = 0.4r$ and a physical time of simulation equal to $t = 0.001s$ gives an adimensional time of $t^* = \frac{V_s}{r_p}$

$t^* = 16,35$ what nécessite 1635 steps of time. The CPU
time of this simulation is about $t_1 = 3h$.

A mesh size $h_2 = 0.2r = \frac{h_1}{2}$ gives a CPU time about $t_2 = 2^5t_1 = 96h$ according to the complexity in $O(k^5)$ with $k = 2$.

7 Conclusions

We presented a model to simulate granular flow with fluid. A discrete method is used for the granular medium, where the trajectories of the particles are calculated by the law of Newton with a soft sphere approach. The flow of the fluid is determined by the direct resolution of the Navier-Stokes equations on a fixed grid with a size of small mesh in front of the size of the radius of the particles. The transfer of momentum between the fluid and solid phases is carried out directly by the integration of the tensor of the constraints on the surface of each particle.

The use of this model gives a good agreement with a relative error of less than 10 particle insulated for lower Reynolds numbers or equal to 20. Larger Reynolds numbers are not possible to simulate with this method and lead to prohibitory computing times because of complexity.

The simulation of the sedimentation of 2 particles makes it possible to reproduce the phenomenon of Draft, Kiss and Tumbled with a good agreement with terminals velocity of particles in comparison with the experimental laws. This shows the capacity of the model to reproduce hydrodynamic interactions acting on the scale of the particle.

Simulations of the sedimentation of a system of particles makes it possible to design the law of Richardson and Zaki, to validate its local character with an acceptable time CPU. The impact of the size of the mesh shows that the dense mediums impose a size of mesh smaller than for the diluted mediums. This method gives an acceptable computing time and makes it possible to treat on WokStation a number of particles of the order of the thousand in three dimensions.

In prospect, this model can be used to model and validate...
more finely the terms of local transfers between the solid and fluid phases used in the other approaches.

8
Acknowledgments
We thank Bernd Wachmann and Stephan Schwarzer of Institut fur Computerphysik for several motivating and enlightening discussions and for contributions to the simulation code.

References
2. A. Boëlle, Validation d’un modèle à deux fluides appliqué à la simulation des lit fluidisés Thèse de Doctorat (1997), Université de Paris 6
4. O. Simonin, Eulerian formulation for particle dispersion in turbulent two-phase flows 5th Workshop on two phase flow predictions, 19-22-03/90, Erlangen RFA
6. S. Ergun, Fluid Flow through packed columns Chemical Engineering Progress Vol.48, No.2, (1952)
20. B. Wachmann, Microsimulation of Two-Phase System in 3D Doctoral Thesis (1999), University of Stuttgart
22. P.D. Cundall, D.J. Strack Geotechnique, 29, 47, (1979)