Numerical simulation of the accumulation of heavy particles in a circular bounded vortex flow

Citation for published version (APA):

DOI:
10.1016/j.ijmultiphaseflow.2016.06.008

Document status and date:
Published: 01/01/2016

Document Version:
Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.
Numerical simulation of the accumulation of heavy particles in a circular bounded vortex flow

Xiaoke Ku\textsuperscript{a}, Jianzhong Lin\textsuperscript{a,b,*}, Martin van Sint Annaland\textsuperscript{c}, Rob Hagmeijer\textsuperscript{d}

\textsuperscript{a}Department of Mechanics, State Key Laboratory of Fluid Power Transmission and Control, Zhejiang University, Hangzhou, China
\textsuperscript{b}China Jiliang University, Hangzhou, China
\textsuperscript{c}Department of Chemical Engineering and Chemistry, Eindhoven University of Technology, 5612 AZ Eindhoven, The Netherlands
\textsuperscript{d}Department of Engineering Technology, University of Twente, 7522 NB Enschede, The Netherlands

ABSTRACT

In present work, an Eulerian-Lagrangian CFD model based on the discrete element method (DEM) and immersed boundary method (IBM) has been developed, validated and used to investigate the accumulation of heavy particles in a circular bounded viscous vortex flow. The inter-particle and particle-wall collisions are resolved by a hard-sphere model. Effects of one-way and two-way coupling, Reynolds number, and particle diameter are systematically explored. Results show that, in case of one-way coupling, the majority of particles will spiral into an accumulation point located near the stagnation point of the flow field. The accumulation point represents a stable equilibrium point as the drag created by the flow field balances the destabilizing centrifugal force on the particle. However, in case of two-way coupling, there does not exist a stable accumulation point due to the strong interaction between the particles and fluid dynamics. Instead most particles are expelled from the circular domain and accumulate on the confining wall. The percentage of accumulated particles on the wall increases with increasing Reynolds number and particle diameter. Moreover, influence of three well-known drag models is also studied and they give consistent results on the particle accumulation behavior, although small quantitative differences can still be discerned.

*Corresponding author. E-mail address: mecjzlin@public.zju.edu.cn (J. Lin).
Keywords: Discrete element method, immersed boundary method, drag models, inter-phase coupling, particle accumulation

1. Introduction

The motion of small heavy particles (like droplets, impurities and dust) in gas flows plays an important role in both the cloud formation in the earth’s atmosphere and many industrial applications, such as steam turbines, gas-particle separators and combustion engines. Thus, it has gathered much attention from both the theoretical and numerical point of views in the past two decades (Balachandar and Eaton 2010; Gañán-Calvo and Lasheras 1991; Ijzermans 2007; Ijzermans and Hagmeijer 2006; Provenzale 1999; Sánchez and Moreno-López 2005; Tio et al. 1993a; Tio et al. 1993b). A salient feature of such suspensions is the appearance of strong heterogeneities in the spatial distribution of particles, often dubbed preferential accumulation.

The general conclusion from previous works is that heavier-than-fluid particles are expelled from regions of high vorticity and tend to accumulate in regions of high strain and low vorticity. There is also broad consensus that preferential accumulation is most pronounced when the particle Stokes number $St$, defined as the ratio of a characteristic time of particle motion and the characteristic time scale of the flow, is of order unity. Nevertheless, most work in literature has dealt almost exclusively with systems in which both the effects of the particles on the motion of the flow and the particle-particle collisions are ignored. While these are valid approximations for small size of particles and low particle concentration, for large particles and high particle concentration, both the inter-phase coupling and particle-particle interaction play a significant role and must be taken into account.

From a physical point of view, the coupling between phases currently comprises the effects of (a) volume displacement by the particles, and (b) fluid-solid interaction forces (e.g., drag force, Saffman lift...
force, added mass effect, Basset history force). For the small heavy particles studied in the present paper, the particles have a much higher mass density than the gaseous flow ($\rho_p/\rho_g >> 1$) and thus the viscous drag force will dominate over other interaction forces in determining the particle motion (IJzermans 2007). There are various drag models available in the literature (Ku et al. 2013). The Gidaspow model (Gidaspow 1994) is a combination of the Wen and Yu equation (Wen and Yu 1966) for the dilute granular regime (porosity $\varepsilon_g >= 0.8$) and the Ergun equation (Ergun 1952) for the dense granular regime ($\varepsilon_g < 0.8$). Di Felice (1994), using an empirical fit to a wide range of fixed and suspended-particle systems covering the full practical range of flow regimes based on particle Reynolds number and porosities, proposed a continuous single-function correlation for the drag force. More recently, Hill et al. (2001a, 2001b) proposed a Hill-Koch-Ladd (HKL) correlation applicable to different ranges of Reynolds numbers and porosities based on data from Lattice-Boltzmann simulations. Later Benyahia et al. (2006) blended the HKL correlation with known limiting forms of the gas-solids drag function and constructed an extended HKL drag correlation (EHKL) which is applicable to the full range of solid volume fractions and Reynolds numbers. In our earlier paper (Ku et al. 2015b), we reported some preliminary results of the qualitative influence of various drag models on the motion of heavy particles in vortex flow. In this paper, as an extension, we present in detail an Eulerian-Lagrangian approach with a hard-sphere collision model for the simulation of the accumulation of heavy particles in a circular bounded vortex flow. The objective of the present work is to explore the effects of one-way and two-way coupling, Reynolds number, and particle diameter on the accumulation of heavy particles in vortex flow. In addition, the differences resulting from the above mentioned drag models are also highlighted. The circular bounded vortex flow considered is characterized by an eccentrically placed point vortex in a closed circular domain and directly relevant to industrial gas-particle or gas-condensate separators with the aim of separating small heavy particles and droplets from the gas by a selective swirl. By
studying the effects as described above, we will gain further insight into the understanding of particle
dynamics in vortex flows and the results can also be useful in the design of industrial gas-particle or
gas-condensate separators.

This paper is organized as follows: In section 2, the equations of motion describing the evolution of the
heavy particles and fluid phase are formulated and the three drag models are tabulated. In section 3, the flow
geometry and parameter settings are presented. In section 4, the numerical results of accumulation of heavy
particles in a circular domain containing one vortex are documented. Here, we first verify our approach by
comparing our simulation results with both analytical solutions and numerical predictions reported in the
literature. Subsequently, we investigate the particle accumulation behavior for both one-way and two-way
coupling, where the effects of Reynolds number and particle diameter are explored and the differences
resulting from different drag models are highlighted. A short summary and conclusions are provided in
section 5.

2. Mathematical modeling

2.1. Discrete particle phase

The Discrete Element Model (DEM) used in this work is an Eulerian-Lagrangian model, which
describes the gas-phase as a continuum, whereas each of the individual particles is treated as a discrete entity
(Ku et al. 2014, 2015a). The inter-particle collisions are handled through a hard-sphere collision model. In
this model the particles are assumed to be rigid spheres, and collisions among particles are treated as binary,
instantaneous and impulsive events. For particle-wall collisions the wall is treated as a “special particle” with
infinitely large mass. Particle collision dynamics are described by collision laws, which account for energy
dissipation due to non-ideal particle-particle and particle-wall interactions by means of the empirical
coefficients of normal and tangential restitution, and the coefficient of friction. An advantage of the
hard-sphere model is that there is an analytical solution available for the collision model. Given the
velocities of the particles prior to collision together with the particle properties, the post-collision velocities
can be calculated analytically. For not too dense systems, the hard-sphere model is considerably faster than
the soft-sphere model. For further details on the implementation of DEM and the hard-sphere collision
model the interested reader is referred to the works of Deen et al. (2007), Hoomans (1999), Hoomans et al.
(1996) and Hoomans et al. (2000).

In the DEM, the motion of every individual particle with mass \( m_i \) and volume \( V_i \) is calculated from
Newton’s second law of motion, which reads in a stationary coordinate system and in dimensional form:

\[
m_i \frac{d\mathbf{v}_i}{dt} = -V_i \nabla p + \frac{V_i \beta}{\varepsilon_p} (\mathbf{u}_g - \mathbf{v}_i) + m_i g
\]

where \( \mathbf{v}_i \) is the velocity of the particle \( i \), \( \mathbf{u}_g \) is the instantaneous gas velocity at the particle position and \( g \) the
gravitational acceleration. The term on the left hand side of Eq. (1) represents particle acceleration. The
forces on the right hand side are respectively due to the pressure gradient, drag and gravity. In this paper, we
restrict the mathematical formulation to the case where the direction of gravity is opposite to that of the \( z \)
age and consider a two-dimensional plane (\( x-y \) plane) motion only, so the term related to gravity can be
neglected. The factor \( \beta \) represents the inter-phase momentum transfer coefficient due to drag. Three drag
models for calculating \( \beta \) are summarized in Table 1, where \( \varepsilon_g \) is porosity, \( \varepsilon_p = 1 - \varepsilon_g \), and the subscripts \( g \)
and \( p \) refer to the gas and particle, respectively.

**Table 1**

The angular momentum of the particle is computed with:

\[
I_i \frac{d\omega_i}{dt} = \mathbf{T}_i
\]

where \( \mathbf{T}_i \) is the torque and \( I_i \) is the moment of inertia which for spherical particles with radius \( r_i \) is
\[
I_i = 2l (5m_i r_i^2).
\]
Note that \( T_i = -r_i \mathbf{n} \times \mathbf{J} \) only exists when particle \( i \) has a collision with another particle (e.g., particle \( j \)), where \( \mathbf{n} \) is the normal unit vector pointing in the direction from the center of particle \( j \) to the center of particle \( i \), and \( \mathbf{J} \) is the impulse vector exerted on particle \( i \) during the collision.

### 2.2. Continuous gas phase

In the DEM the gas phase hydrodynamics are calculated from the volume-averaged Navier–Stokes equations which are coupled with those dynamics of the particle phase and the immersed large body through the porosity and the inter-phase momentum exchange:

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

(3)

\[
\frac{\partial}{\partial t} (\varepsilon_g \rho_g \mathbf{u}_g) + \nabla \cdot (\varepsilon_g \rho_g \mathbf{u}_g \mathbf{u}_g) = -\varepsilon_g \nabla p + \nabla \cdot (\varepsilon_g \mathbf{\tau}_g) + \varepsilon_g \rho_g \mathbf{g} - \mathbf{S}_p + \mathbf{S}_{ibm}
\]

(4)

Here, \( \varepsilon_g \) is porosity, and \( \rho_g \), \( \mathbf{u}_g \), \( \mathbf{\tau}_g \) and \( p \) are the density, velocity, viscous stress tensor and pressure of the gas phase, respectively. \( \mathbf{S}_p \) is a source term that describes the momentum exchange of the gas with the small heavy particles and \( \mathbf{S}_{ibm} \) is a source term for the momentum exchange with the immersed large bodies. These two source terms will be discussed in more detail below. \( \rho_g \) is determined using the ideal gas law,

\[
\rho_g = \frac{p M_g}{R_g T_g}
\]

(5)

where \( M_g \) and \( T_g \) are the molecular weight and temperature of the gas, respectively. \( R_g \) is the universal gas constant. \( \mathbf{\tau}_g \) is assumed to obey the general form for a Newtonian fluid (Bird et al. 1960):

\[
\mathbf{\tau}_g = (\lambda_g - \frac{2}{3} \mu_g) (\nabla \cdot \mathbf{u}_g) \mathbf{I} + \mu_g ((\nabla \mathbf{u}_g) + (\nabla \mathbf{u}_g)^T)
\]

(6)

where the bulk viscosity \( \lambda_g \) can be set to zero for gas, and \( \mu_g \) is the dynamic gas viscosity.

The two-way coupling between the gas-phase and the small heavy particles is achieved via the source term \( \mathbf{S}_p \) as shown in Eq. (4), which is a function of the inter-phase momentum transfer coefficient \( \beta \) and is computed from:
\[ S_p = \frac{1}{V_{cell}} \sum_{i \in \text{cell}} \frac{V_i \beta}{\varepsilon_p} (u_i - v_i) D(r - r_i) \] (7)

where \( r_i, v_i \) and \( V_i \) are the position, velocity and volume of the particle \( i \), respectively. \( u_g \) is the local flow velocity of the gas phase and \( V_{cell} \) is the volume of the fluid cell. The distribution function \( D \) is a discrete representation of a Dirac delta function that locally distributes the reaction force acting on the gas-phase to the near Eulerian computational grids via appropriate volume-weighing technique whose details can be found in the works of Hoomans (1999) and Deen et al. (2007).

**Figure 1**

The Immersed Boundary Method (IBM) is adopted to resolve the circular boundary and the large rotating cylinder in our computational domain. In IBM the complex boundary and large immersed body are represented by a set of Lagrangian marker points uniformly distributed over the solid-fluid interface (see Fig. 1). Each marker point exerts a force on the gas phase such that the local velocity of the gas is equal to the velocity of that marker. Specifically, the interaction between the gas phase and the immersed body is then controlled by a source term \( S_{ibm} \) as shown in Eq. (4). \( S_{ibm} \) guarantees the imposition of the no-slip boundary condition over the solid-fluid interface. Implementation details of IBM are described in references (Lima E Silva et al. 2003; Mittal and Iaccarino 2005; Peskin 2002; Taira and Colonius 2007; Uhlmann 2005).

**3. Computation parameters**

**Figure 2**

Figure 2 shows a closed circular domain with a small rotating cylinder placed inside the domain (Ku et al. 2015b). As shown in Fig. 2, the radii of the circular domain and the small rotating cylinder are \( R \) and \( r_s \), respectively. In order to mimic the flow field generated by a point vortex, the small rotating cylinder is assumed to rotate not only around its own axis, but also around the vertical axis through the center of the
circular domain. Then, the small rotating cylinder appears as a point vortex with circulation $\Gamma$. The position of the vortex is determined by the center-to-center distance $r_1$ between the small rotating cylinder and the big circular domain. The orientation of the vortex is denoted by the angle $\theta_1$. The rotation rate of the cylinder around its own axis is $\Gamma/(2\pi r_1^2)$ and the constant angular velocity around the vertical axis through the center of the circular domain $\dot{\theta}_1$ is given by (IJzermans and Hagmeijer 2006):

$$\dot{\theta}_1 = \frac{\Gamma}{2\pi R^2} \frac{1}{R^2 - r_1^2}$$

The no-slip boundary condition on the circular walls and the surface of the small rotating cylinder is imposed via the immersed boundary method.

Our interest focuses on the two-dimensional $(x,y)$ plane particle dynamics in the flow field. The position is denoted by $(x, y)$ in Cartesian coordinates, and by $(\xi, \eta)$ in a co-rotating reference frame which co-rotates with the vortex. Then the following coordinate transform is easily derived:

$$\xi(x, y, t) = x \cos \theta_1(t) + y \sin \theta_1(t)$$

$$\eta(x, y, t) = -x \sin \theta_1(t) + y \cos \theta_1(t)$$

Moreover, let the velocity field in the $(x,y)$-frame be denoted by $(u_x, u_y)$ and in the $(\xi, \eta)$-frame by $(u_\xi, u_\eta)$, then:

$$u_\xi = u_x \cos \theta_1 + u_y \sin \theta_1 + \dot{\theta}_1 \eta$$

$$u_\eta = -u_x \sin \theta_1 + u_y \cos \theta_1 - \dot{\theta}_1 \xi$$

In the following, all variables are made dimensionless by choosing $R$ as the characteristic length, $\Gamma/R$ as the characteristic velocity, and $R^2/\Gamma$ as the characteristic time. The flow Reynolds number is $Re=\rho_R \Gamma/\mu_g$.

Unless stated otherwise, we will only deal with dimensionless quantities in the co-rotating reference frame.

Table 2 summarizes the parameter settings used in the simulation.
4. Results and discussions

4.1. Validation

Figure 3

The accuracy and validity of our model are firstly checked by simulating the steady velocity profile of pipe flow. The implementation of the immersed boundary method to impose the no-slip condition on the circular wall is validated for steady flow in a tube. Figure 3 shows the steady velocity profile at the pipe outlet, where the circular symbols represent the simulation results and the coincident solid line threading the circles represents the analytical solution. The excellent agreement demonstrates the accuracy and validity of the present immersed boundary method implementation.

Table 3

In order to further validate the integrated approach and the three drag models, Table 3 shows the comparison of predicted values of the particle accumulation point in the frame co-rotating with the vortex by the three drag models with the numerical data reported by IJzermans (2007). The simulation condition is based on one-way coupling, $Re = 200$ and $St = 0.25$. It can be observed that good agreement is obtained. The relatively small deviations are induced by the nonlinear drag models adopted in the present work, whereas IJzermans (2007) used the Stokes drag force law. The particle accumulation point is a key index for vortex flows and the successful prediction of this quantity thereby provides an important example to verify the proposed approach.

4.2. Initial flow field

Figure 4
The first step of the simulations is to obtain an initial steady flow field, during which no particles are introduced. This initial flow field is then saved as a starting flow field for the study of particle accumulation.

Figure 4 shows the streamlines of the initial flow field for $Re = 200$. This flow field is induced by the small rotating cylinder in a circular domain (see Fig. 2), plotted in the frame co-rotating with the vortex. The flow field is characterized by the presence of a single stagnation point, which is a point of zero flow velocity in the $(\xi, \eta)$-frame. The location of the stagnation point is $(\xi, \eta) = (-0.263, -0.132)$. As also shown in Fig. 4, the flow around the rotating cylinder is dominated by a counterclockwise motion, whereas the left half plane is dominated by a large rotating motion in clockwise direction around the stagnation point. The rotation of the flow field around this stagnation point is always opposite to the rotation of the frame, which is generally called anti-cyclonic motion. In addition, it is noted that the viscous flow field is not symmetric: the stagnation point is situated well below the negative $\xi$-axis.

### 4.3. One-way coupling

Using the one-vortex flow as the background flow field (see Fig. 4), we now consider the motion of heavy particles in such a flow. Firstly, we turn off the influence of particles on the flow field and start with the relatively simple case of one-way coupling. Figure 5 shows the evolved positions of a group of 482 heavy particles at eight instants in (dimensionless) time: $t = 0, 10.6, 20.3, 30.9, 100.4, 200.9, 500.3, 600.7$ under one-way coupling for different drag models. The particles’ positions are plotted in the frame co-rotating with the vortex and the solid dots denote the positions of those particles. At the start of the simulation ($t = 0$), the particles are uniformly distributed over the circular domain with zero velocity. There is no overlap between particles at $t = 0$ which is required for the hard-sphere collision model. Each particle is traced individually. As can be clearly seen from Fig. 5, depending on the initial positions, some particles
released close to the wall quickly escape to the wall and remain permanently on the wall. However, the majority of particles gradually spiral towards a single point located inside the domain and remain there permanently. This accumulation feature is robust and does not appear to depend on the specific drag model.

Note that the particle-particle collisions are also taken into account for the one-way coupling case. In contrast to the point-particle results obtained by IJzermans and Hagmeijer (2006), the present results are obtained for finite size particles that may collide with others frequently due to the high particle concentration in the vicinity of the accumulation point. Because the particles cannot overlap, it is better to refer to an accumulation ‘island’ instead of an accumulation point. The bigger the particle size is, the larger the accumulation island is. The physical mechanism leading to particle accumulation is that the Coriolis force drives inertial particles to the center of anti-cyclonic region of the flow field, where the particles are trapped by a balance between the viscous drag and the centrifugal force (IJzermans and Hagmeijer 2006). Note that the particle Stokes number $St$, defined as the ratio between the characteristic time of particle motion and the characteristic time of the flow, can characterize the particles’ inertia and is 1.0 in the present paper.

4.4. Two-way coupling

From the analysis of the one-way coupling case (see Fig. 5), we observe that the heavy particles will spiral to an accumulation point within the domain. What will happen when two-way coupling is considered? A major distinction between one-way coupling and two-way coupling is that, for two-way coupling the fluid flow is significantly modified due to the influence of the particles on the carrier fluid and the flow evolves dynamically with spatial and temporal structures.

Figure 6

Figure 6 presents the evolved positions of a group of 482 initially uniformly distributed heavy particles into the initial flow field at eight dimensionless time instants: $t = 0, 10.6, 20.3, 30.9, 200.9, 600.7, 700.2,$
750.4 under two-way coupling for different drag models. The conditions are the same as in Fig. 5 except for
the influence of the particles on the flow field being taken into account. It can be firstly found that the stable
accumulation point as observed for the one-way coupling case ceases to exist in the case of two-way
coupling. Secondly, as we go from $t = 0$ to $t = 30.9$, it is found that many particles are still spiraling inside
the domain, while the rest of them have been attracted by the wall. However, for larger times ($t > 30.9$), the
particles no longer accumulate in one single position, instead they periodically orbit along a non-regular
trajectory inside the domain. Thirdly, as we progress from $t = 0$ to $t = 750.4$, it is observed that the number of
particles inside the domain decreases and most particles are collected at the wall. As also shown in Fig. 6,
although the qualitative features described above are the same for the three different drag models, some
differences can still be discerned at larger times ($t > 200.9$).

**Figure 7**

In order to explore the physical reason for these phenomena, Figure 7 shows the corresponding
streamlines of the carrier flow at eight dimensionless time instants for the three drag models. It is clearly
seen that the flow field varies with time due to the influence of particles. Specifically, at the initial stage ($t <
10.6$), the streamlines change slightly because fairly weak inter-phase interaction is introduced into the flow
field due to the dilute loading and the initial spread of the particles. Therefore, the evolution of particles is
similar to the one-way coupling case and many particles accumulate somewhere. However, at later stages
($t > 20.3$), the flow modification is large and the streamlines are much different from the initial flow because
a region of concentrated particles is locally formed inside the domain and the inter-phase interaction is
strong. The big changes of the local spatial structures of the flow will in turn significantly affect the particle
motion and push particles to the wall. At further later stages ($t > 200.9$), the streamlines become similar to
those of the initial flow since most particles are collected at the wall and there are only few particles moving
inside the domain.

Clearly shown in Figs. 5, 6 and 7, the one-way coupling approximation may give some misleading estimates of the accumulation process. The results here imply that the local dynamical interaction of particles with flow structures are strong and that the interaction plays a significant role on the particle motion in vortex flow. So when the particle accumulation is considered, a proper account of two-way coupling is essential.

In order to quantify the particle accumulation, the percentage of accumulated particles on the wall, \( P \), is defined as follows

\[
P = \frac{\text{number of particles collected on the wall}}{\text{total number of initially uniformly distributed particles}} \times 100\%\tag{13}
\]

4.4.1. Effect of Reynolds number

Figure 8

In our earlier works (IJzermans and Hagmeijer, 2006, 2007), the accumulation of heavy particles in potential vortex flows were numerically studied. The results showed that the heavy particles tended to accumulate in elliptic islands of the flow, which was consistent with the findings of the one-way coupling case in the present paper. However, the previous numerical models were constructed based on the assumptions of one-way coupling and low Reynolds number where the Stokes drag force law holds. Considering the flow solver and the drag models in this paper are applicable to cases of both two-way coupling and high Reynolds number which are not yet examined, their influences are explored in this subsection. Figure 8 shows the effect of Reynolds number on the percentage of accumulated particles on the wall for the three investigated drag models. It is clear that \( P \) increases monotonically with increasing \( Re \).
This result is consistent with physical intuition. Since a higher Reynolds number corresponds to a larger centrifugal force acting on the particle which tends to push more particles to the wall. It is also found that the differences among the different drag models decrease with an increase in $Re$.

### 4.4.2. Effect of particle diameter

Figure 9

Figure 9 presents the effect of particle diameter ($d_p$) on the $P$ for the three drag models. It is easily seen that $P$ increases monotonically with an increase in $d_p$. The reason is that a larger $d_p$ means a particle with a greater inertia and thus a greater chance of avoiding the captivity inside the flow field and escaping to the wall. It is also observed that $P$ is the largest for the Di Felice model and almost the same for the Gidaspow and EHKL models within the $d_p$ range studied. These small differences are attributed to the different non-linear drag exerted on the particles by various drag models as shown in Table 1.

### 4.5. Discussion

From the above analyses, it is found that the heavy particles will spiral to an accumulation point within the domain for the one-way coupling case (see Fig. 5) and this accumulation point is located near the stagnation point of the flow field in the left half plane. The specific location of the accumulation point depends on the particle Stokes number $St$ and the distance between the accumulation point and the stagnation point becomes large as $St$ increases. Note that in the limit case of $St = 0$, the accumulation point will coincide with the stagnation point of the flow field. However, in case of two-way coupling, there does not exist a stable accumulation point due to the strong inter-phase interaction. Instead, most particles are expelled from the circular domain and accumulate on the confining wall. Qualitatively, one-way coupling holds when the particle concentration is low, while the two-way coupling must be taken into account when the particle
concentration is high. In order to check the transition concentration at which the inter-phase interaction becomes important, we initially put more and more particles at the accumulation point to see at what concentration they will leave that location. The results show that the particles can still locate near the accumulation point when the local particle concentration is below 3%, above which they start to deviate from the accumulation point. Furthermore, the deviation distance increases with increasing concentration and some particles can collide with the wall and be trapped at the wall when the initial local particle concentration is above 6%.

5. Conclusions

The accumulation of heavy particles in a circular bounded vortex flow has been numerically investigated by an Eulerian-Lagrangian CFD model based on the discrete element method (DEM) and immersed boundary method (IBM). The inter-particle and particle-wall collisions are resolved by a hard-sphere model. Effects of one-way and two-way coupling, Reynolds number, particle diameter, and three drag models (Gidaspow 1994, Di Felice 1994 and EHKL 2006) are systematically explored. Results show that, in case of one-way coupling, the majority of particles will spiral into an accumulation point located near the stagnation point of the flow field for all the drag models. The accumulation point represents a stable equilibrium point as the drag created by the flow field balances the destabilizing centrifugal force on the particle. However, in case of two-way coupling which is a more general and practical regime, there does not exist a stable accumulation point due to the strong interaction between the particles and fluid dynamics. Instead most particles are expelled from the circular domain and accumulate on the confining wall. The three drag models employed give consistent results on the percentage of accumulated particles on the wall, although small quantitative differences can still be discerned. Moreover, the percentage of accumulated particles on the wall increases with an increase in Reynolds number and particle diameter. In addition, the
transition particle concentration at which the inter-phase interaction must be taken into account is also numerically studied.

The computational model in this study corresponds to a flow induced by one slender helical vortex filament in a pipe where a three-dimensional swirl flow can be approximated by a two-dimensional rotating flow. The results obtained give a global idea of the motion of heavy particles in such system and may have useful implications for the design of industrial gas-particle separators and also have significant practical application in estimating particle accumulation in strongly rotational flows such as centrifuges.

Acknowledgements

The authors would like to thank the National Natural Science Foundation of China with Grant No 11132008, the startup fund from the “hundred talents program” of Zhejiang University, and the University of Twente for financial support.

References


Fig. 1. Lagrangian marker points of immersed boundary method for a circular boundary.
Fig. 2. A closed circular domain with a small rotating cylinder placed inside the domain.
Fig. 3. Steady radial profile of the axial velocity at the outlet for pipe flow.
Fig. 4. Initial steady flow filed in the frame co-rotating with the vortex, $Re = 200$. 
Fig. 5. Instantaneous positions of heavy particles at eight instants in the frame co-rotating with the vortex for one-way coupling. \( t \) is the dimensionless time, \( Re = 200, St = 1.0, d_0 = 0.00125 \). (a) Gidaspow model; (b) Di Felice model; (c) EHKL model.
Fig. 6. Instantaneous positions of heavy particles at eight instants in the frame co-rotating with the vortex for two-way coupling. $t$ is the dimensionless time, $Re = 200$, $St = 1.0$, $d_p = 0.00125$. (a) Gidaspow model; (b) Di Felice model; (c) EHKL model.
Fig. 7. Streamlines of the flow field at eight instants in the frame co-rotating with the vortex for two-way coupling. $t$ is the dimensionless time, $Re = 200$, $St = 1.0$, $d_p = 0.00125$. (a) Gidaspow model; (b) Di Felice model; (c) EHKL model.
Fig. 8. Effect of Reynolds number on the percentage of accumulated particles on the wall for different drag models. $d_p=0.00125$. 
Fig. 9. Effect of particle diameter on the percentage of accumulated particles on the wall for different drag models. $Re = 200$. 
Table 1 Three drag models commonly used for particulate flows.


\[
\beta = \begin{cases} 
150 \frac{\varepsilon_p^2 \mu_g}{\varepsilon_g d_p^2} + 1.75 \frac{\varepsilon_p \rho_g}{d_p} |u_g - v_p| \varepsilon_g < 0.8 \\
\frac{3}{4} C_d \frac{\varepsilon_p \rho_g}{d_p} |u_g - v_p| \varepsilon_g^{-1.65} \varepsilon_g \geq 0.8 
\end{cases}
\]

\[
C_d = \begin{cases} 
\frac{24}{Re_p} (1 + 0.15 Re_p^{0.687}) & Re_p < 1000 \\
0.44 & Re_p \geq 1000 
\end{cases}
\]

\[
Re_p = \varepsilon_g \rho_g d_p |u_g - v_p| / \mu_g
\]

2. Di Felice model (1994)

\[
\beta = \frac{3}{4} C_d \frac{\varepsilon_p^2 \rho_g}{d_p} |u_g - v_p| \varepsilon_g^{-\chi}
\]

\[
C_d = (0.63 + \frac{4.8}{\sqrt{Re_p}})^2, \quad \chi = 3.7 - 0.65 \exp \left[ -\frac{(1.5 - \log_{10}(Re_p))^2}{2} \right]
\]

\[
Re_p = \varepsilon_g \rho_g d_p |u_g - v_p| / \mu_g
\]

3. EHKL model (2006)

\[
\beta = \frac{18 \mu_g \varepsilon_p^2 \varepsilon_p}{d_p^2} F
\]

\[
F = \begin{cases} 
1 + 3/8 Re_p, & \varepsilon_p \leq 0.01 \text{ and } Re_p \leq (F_2 - 1)/(3/8 - F_2) \\
F_0 + F_1 Re_p^2, & \varepsilon_p > 0.01 \text{ and } Re_p \leq (F_3 + \sqrt{F_3^2 - 4F_1(F_0 - F_2)}/(2F_1)) \\
F = F_2 + F_3 Re_p, & \text{otherwise} 
\end{cases}
\]

\[
F_0 = \begin{cases} 
(1 - w) \frac{1 + 3(\varepsilon_p^3/5) + 135/64 \varepsilon_p \ln(\varepsilon_p) + 17.14 \varepsilon_p}{1 + 0.681 \varepsilon_p - 8.48 \varepsilon_p^2 + 8.16 \varepsilon_p^3} + w \frac{10 \varepsilon_p}{\varepsilon_g^3} & 0.01 < \varepsilon_p < 0.4 \\
10 \varepsilon_p / \varepsilon_g^3 & \varepsilon_p \geq 0.4
\end{cases}
\]

\[
F_1 = \begin{cases} 
\sqrt{2/\varepsilon_p} / 40 & 0.01 < \varepsilon_p \leq 0.1 \\
0.11 + 0.00051 e^{11.6 \varepsilon_p} & \varepsilon_p > 0.1
\end{cases}
\]
\[ F_2 = \begin{cases} 
(1 - w) \frac{1+3\sqrt{\varepsilon_p}}{2} + (135/64)\varepsilon_p \ln(\varepsilon_p) + 17.89\varepsilon_p + w \frac{10\varepsilon_p}{\varepsilon_g^3} & \varepsilon_p < 0.4 \\
10\varepsilon_p / \varepsilon_g^3 & \varepsilon_p \geq 0.4 
\end{cases} \]

\[ F_3 = \begin{cases} 
0.9351 \varepsilon_p + 0.03667 & \varepsilon_p < 0.0953 \\
0.0673 + 0.212 \varepsilon_p + 0.0232 / \varepsilon_g^5 & \varepsilon_p \geq 0.0953 
\end{cases} \]

\[ Re_p = \varepsilon_i \rho d_p \left| u_p - v_p \right| / (2 \mu) \quad w = e^{-10(0.4 - \varepsilon_p) / \varepsilon_p} \]
Table 2 Parameter settings for the simulations.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value (non-dimensional)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain radius ((R))</td>
<td>1</td>
</tr>
<tr>
<td>Small rotating cylinder radius ((r_s))</td>
<td>0.1</td>
</tr>
<tr>
<td>Distance from the vortex to the domain center ((r_1))</td>
<td>0.5</td>
</tr>
<tr>
<td>Particle diameter ((d_p))</td>
<td>0.00075, 0.00125, 0.001875</td>
</tr>
<tr>
<td>Reynolds number ((Re))</td>
<td>200, 400, 800</td>
</tr>
<tr>
<td>Normal restitution coefficient between particles</td>
<td>0.97</td>
</tr>
<tr>
<td>Tangential restitution coefficient between particles</td>
<td>0.33</td>
</tr>
<tr>
<td>Friction coefficient between particles</td>
<td>0.1</td>
</tr>
</tbody>
</table>
Table 3 Comparison of predicted locations of the accumulation point in the frame co-rotating with the vortex by the three drag models with the data reported by IJzermans (2007). One-way coupling, Re =200 and St =0.25.

<table>
<thead>
<tr>
<th>Drag model used</th>
<th>Location of accumulation point, (ζ, η)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>This work</td>
</tr>
<tr>
<td>Gidaspow</td>
<td>(-0.271, -0.112)</td>
</tr>
<tr>
<td>Di Felice</td>
<td>(-0.271, -0.111)</td>
</tr>
<tr>
<td>EHKL</td>
<td>(-0.271, -0.112)</td>
</tr>
</tbody>
</table>

1 The drag model used by IJzermans (2007) is Stokes drag law.