Direct Numerical Simulation of Particle-laden Flow Around an Obstacle at Different Reynolds Numbers

Shengxiang Lin¹, Huanxiong Xia¹*, Zhenyu Zhang¹, Jianhua Liu¹ and Honglei Wang²
¹School of Mechanical Engineering, Beijing Institute of Technology, Beijing 100081, China
²China Coal Research Institute, Beijing 100013, China

*E-mail: hxia@bit.edu.cn

Abstract. Inspired by the practical operation of the fluid machineries, direct numerical simulation of fluid with a lot of finite-size particles flowing around a large-size obstacle at three different Reynolds numbers is implemented by using a two-way coupled finite-volume, discrete-element and immersed-boundary method. The results show that, for a low Reynolds number Re=20, the flow is dominated by viscosity, and under the circumstances of a small Stokes number, the particles follow fluid streamlines closely. The flow suggests regular movement characteristics of laminar flow, although the vortices behind the obstacle tend to collapse under the perturbation of particles. For a moderate Reynolds number Re=100, the phenomenon of vortex shedding is also observed. Due to the centrifugal force induced by the vortices, particles are distributed around the main vortices behind the obstacle, forming particle-free zones in these vortices. For a high Reynolds number Re=300, the flow is chaotic. The vortices of many sizes appear irregularly in the domain and the distribution of particles tends to be uniform.

1. Introduction
The phenomenon of fluid flowing around bluff bodies is widely encountered in natural and industrial processes. Wind blowing past high-rise buildings and bridges, water flowing over submarines, flows passing spoilers in industrial equipment (e.g., flue gas in industrial boilers blowing around heat exchange tubes) are some of the common examples. It is worth noting that these fluids often contain solid particles, such as sand in the wind, suspended solids in the water and dust in the flue gas. The presence of particles has a great impact on the mass, momentum and energy exchange, making the flow more complicated.

Due to the complexity of the mechanism and the wide range of industrial applications, this type of flow has been the subject of numerous numerical [1-5] and experimental [6,7] researches. Limited by factors such as control, observation, measurement, etc., it is extremely difficult to conduct a relevant experimental study. Therefore, simulation on this issue is the most popular research method used by scholars. Apostolou and Hrymak [8] simulated the flow of suspensions of particles in viscous liquids using the discrete element method (DEM). They studied the effect of the accounted forces on the agglomeration process, but when modeling the interaction between fluid and particles, they assumed that the fluid is unaffected by the presence of the particles. After that, Zhao’s group [9-11] performed two-way coupled investigations of turbulent channel flow laden with spherical particles.
research results revealed the influence of the particle characteristics on the particle–turbulence interactions.

The above-mentioned two research teams [8-11] studied the flow in a simple domain, that is, there are no obstacles in the domain. Obviously, the existence of obstacle can make the dynamics of the flow much different. Haugen and Kragset [12] examined particle impaction on a cylinder in a crossflow and found that there are three modes of impaction on the front side of the cylinder, and the impaction on the back side is strongly dependent on the flow Reynolds number. Haddadi et al. [13] investigated the flow of a suspension of different particle solid fractions over a circular cylindrical post in a channel experimentally and numerically. The results indicated that the recirculating wake behind the obstacle at moderate Reynolds numbers is depleted or devoid by particles, and with an increase of the number of particles in the wake alone, particles can escape the wake due to velocity fluctuations. Tao et al. [14] conducted a numerical investigation of dilute aerosol particle transport and deposition in oscillating multi-cylinder obstructions. They found that the oscillating motion of fibers has significant influence on the filtration performance and the collection efficiency is linearly related to the oscillation amplitude or frequency. Kleinhans er al. [15] proposed a detailed ash deposition model and used the model to predict early deposit formation in a suspension fired boiler. The results suggested that boundary effects and particle cooling are crucial factors during early stages of deposit formation.

Literature analysis shows that scholars have different focuses on particle-laden fluid flowing around an obstacle. According to the actual operating conditions of the fluid machinery, fluids tend to flow at different Reynolds numbers. So in the present numerical study of fluid with many finite-size particles flowing around an obstacle, we have chosen to focus on the interactions between the fluid and the inertial particles at three different Reynolds numbers. It is hoped that the results can provide certain theoretical guidance for the safety and economic operation of the fluid machinery.

2. Problem formulation and numerical method

The physical problem studied here is sketched in Figure 1. Fluid and finite-sized particles flow into uniformly from the inlet at the same velocity \( U_x \), and then flow out of the domain after bypassing a stationary obstacle. Here we set \( H = 5D \), \( L_n = 5D \) and \( L_d = 25D \) to ensure that the domain is sufficiently large and the flow will not be affected by the domain size. Full-slip boundary condition is imposed on the top and bottom walls, and inflow (\( u = U_x \), \( v = 0 \)) and outflow (\( \partial u / \partial x = 0 \), \( \partial v / \partial x = 0 \)) boundary conditions are applied to the inlet and outlet, respectively.

![Figure 1](image)

**Figure 1.** A sketch of the structure of fluid with finite-sized particles flowing around an obstacle.

For the fluid-structure interaction problem investigated in this paper, the fluid flow is evaluated under the Eulerian formalism, while the stationary obstacle and moving particles are described by the Lagrangian formalism. Specifically, a fully-resolved direct numerical simulation of the fluid-obstacle interactions is achieved by using the immersed boundary method (IBM), where no-slip boundary condition is directly imposed on the immersed boundary. The finite-sized particles suspended in the fluid are represented by DEM. Thus, the evolution of the particle motion is governed by the Newton’s second law:
\[\rho V_i \frac{dU_i}{dt} = \sum_{j=1}^{N} F_{P,i,j} + F_{W,i} + F_{O,i} + F_{F,i} \]  
(1)

\[ I_i \frac{d\omega_i}{dt} = \sum_{j=1}^{N} M_{P,i,j} + M_{W,i} + M_{O,i} \]  
(2)

Here, subscripts \(i\) and \(j\) denote particle \(i\) and particle \(j\), respectively, \(I\) stands for the moment of inertia, \(V\) is the particle volume, \(\rho\) is the density, \(U\) and \(\omega\) are the translational velocity and angular velocity of particles, respectively, \(F_{P,i,j}\), \(F_{W,i}\), \(F_{O,i}\) and \(F_{F,i}\) are the particle-particle, particle-wall, particle-obstacle and particle-fluid interaction forces, respectively, and \(M_{P,i,j}\), \(M_{W,i}\) and \(M_{O,i}\) are the torques corresponding to \(F_{P,i,j}\), \(F_{W,i}\) and \(F_{O,i}\).

In this paper, we focus on inertial impaction and assume that the flow is not subject to external force fields including gravity field. Therefore, \(F_{F,i}\) mainly refers to drag force (i.e., \(F_{F,i} = F_d\)). Given radius \(r\) of the spherical particles, the Stokes drag force can be written as:

\[ F_d = 6\pi \mu r (u - U) \]  
(3)

Here, subscript \(f\) denotes fluid, \(\mu\) is the dynamic viscosity and \(u\) is the fluid velocity vector. The Stokes drag acts on the particle’s center of mass, so it does not produce torque on the particle.

The Newtonian carrier fluid is treated as incompressible and viscous, then the continuity and momentum equations are:

\[ \nabla \cdot (\varepsilon \rho_j u_j) = \frac{1}{V_c} \frac{dV_p}{dt} \]  
(4)

\[ \frac{\partial(\varepsilon \rho_j u_j)}{\partial t} + \nabla \cdot (\varepsilon \rho_j u_j u_j) = -\varepsilon \nabla p + \nabla \cdot (\varepsilon \tau_j) + S_m \]  
(5)

Where \(\varepsilon\) and \(\tau\) are the volume fraction and viscous stress tensor, respectively, \(V_c\) stands for the volume of each grid cell, and \(S_m\) is the momentum source term that accounts for the effect of particles on the behaviour of the fluid. Equation (5) is solved by employing the second-order projection method. We first find the temporary velocity without accounting the pressure gradient \(\nabla p\)

\[ \varepsilon^{n+1}_j \rho^{n+1}_j u'_j = (\varepsilon_j \rho_j u_j)_n + \Delta t \left[ -\nabla \cdot (\varepsilon_j \rho_j u_j u_j) + \nabla \cdot (\varepsilon_j \tau_j) + S_m \right] \]  
(6)

then, the temporary velocity \(u'_j\) is corrected by

\[ (\varepsilon_j \rho_j u_j)^{n+1} = (\varepsilon^{n+1}_j \rho^{n+1}_j u'_j) - \Delta t \varepsilon^n_j \nabla p \]  
(7)

where the pressure gradient \(\nabla p\) is obtained from the non-separable elliptic equation derived by combining equations (4) and (7). Moreover, the QUICK scheme for the advection terms and a second-order predictor-corrector method for time integration are used. For more details on the utilized numerical methods, please refer to papers [16-18].

\section{3. Results and discussions}

The flow is determined by a Reynolds number based on the free-stream velocity \(U_\infty\) and the side length \(D\) of the obstacle. The particle motion is characterized by the particle void fraction and the ratio of the particle response time to fluid relaxation time, given by \(\varepsilon_p\) and the Stokes number \(St\), respectively. The governing non-dimensional numbers are therefore:

\[ Re = \frac{\rho_j U_\infty D}{\mu_j}; \quad St = \frac{\rho_j U_\infty R^2}{18 \mu_j D}; \quad \varepsilon_p \]
In addition, nondimensionalized time $\bar{t} = t U_\infty / D$ and normalized velocity $\bar{u} = u / U_\infty$ are adopted.

**Table 1.** The parameters used in the simulations.

| Case                  | A     | B     | C     |
|-----------------------|-------|-------|-------|
| Side length of the obstacle (m) | $1.0 \times 10^{-3}$ | $1.0 \times 10^{-3}$ | $1.0 \times 10^{-3}$ |
| Free-stream velocity (m/s) | 0.1   | 0.1   | 0.1   |
| Particle diameter (m)    | $1.0 \times 10^{-5}$ | $1.0 \times 10^{-5}$ | $1.0 \times 10^{-5}$ |
| Particle density (kg/m$^3$) | 6500  | 6500  | 6500  |
| Fluid density (kg/m$^3$) | 200   | 1000  | 3000  |
| Fluid viscosity (kg/(m s)) | $1.0 \times 10^{-3}$ | $1.0 \times 10^{-3}$ | $1.0 \times 10^{-3}$ |
| Reynolds number       | 20    | 100   | 300   |
| Stokes number         | 0.0036 | 0.0036 | 0.0036 |
| Particle void fraction | 1%    | 1%    | 1%    |

In engineering practices, the Reynolds number of fluid flow will change according to different mechanical operating conditions. Under various Reynolds numbers, the dynamics of particle-laden flow are different. We therefore study three cases with Reynolds number of 20, 100 and 300, while keeping the other parameters as constant values (see Table 1). For the flow around a square cylinder, when Reynolds number is 20, the flow is symmetric with stationary recirculating vortices behind the cylinder. As the Reynolds number is increased to 100, the vortices shed up and down alternately, and turbulence may appear in the wake. Elevating the Reynolds number further to 300, turbulence in the wake flow is stronger.

**Figure 2.** The instantaneous particle distribution and contours of streamwise velocity (right column) and vorticity (left column) at $\bar{t} = 150$. Here, $Re = 20$ (top row), $Re = 100$ (middle row) and $Re = 300$ (bottom row).

The particle motion is driven via the drag force exerted by the fluid, and the flow field is also modulated by the inertia particles at the same time. The dynamics of the fluid and the particles determine the performance of fluid machinery. To examine the influence of the interactions between fluid and particles on their dynamics, we start by plotting the instantaneous contours of vorticity and streamwise velocity and particle distribution in Figure 2. Notice that although the particles flow into the domain through the entire cross section, here we only show the distribution of particles near the
centerline of the channel along the streamwise direction. As shown in the top row of Figure 2, the flow is symmetrical and laminar at Re=20. Since the St number is very small, particles follow fluid streamlines closely. At the same time, the vortices behind the obstacle tend to collapse under the action of particles. For a moderate Reynolds number Re=100, shown in the middle row, the vortex shedding behavior is fully developed. In addition to the main vortex shedding behind the obstacle, there are many small vortices scattered in the flow field. Under the action of vortex centrifugal force, particles are distributed around the main vortices, forming particle-free zones in the vortices. For a high Reynolds number Re=300, shown in the bottom row, the flow presents a large pulsation. The velocity tends to be chaotic and the vortices of many sizes appear irregularly in the domain. Similarly, the distribution of particles also demonstrates a strong irregularity.

Figure 3 illustrates the $\bar{u}$ velocity distribution on the centerline of the channel along the streamwise direction. When Re=20, viscosity dominates the flow and because the St number is small, the particles mainly follow the fluid in motion. This means that the impact of particle motion on the overall trend of flow is limited. The red dash-dot line indicates the regular movement characteristics of laminar flow. Corresponding to Re=100, the green solid line suggests that there is a certain fluctuation of the streamwise velocity, which is mainly caused by the alternating shedding behavior of vortex. Compared with the green line, the pulsation degree of the blue dotted line is greater. This tells that the fluid and particles are constantly exchanging kinetic energy in the spanwise direction under Re=300.

In order to further observe the movement of particles, in Figure 4 we have also plotted the instantaneous local void fraction profile across the channel. The local void fraction is found by dividing the width of the channel into 25 equal-sized bins and computing the void fraction in each bin. In correspondence with different Reynolds numbers, the three curves in Figure 4 show that there are more particles near the walls, but less particles in the center of the channel. In these three cases, the order of the number of particles in the center region from low to high is case A, case B, and case C. Due to the turbulent fluctuation, the distribution of particles in case C tends to be more uniform throughout the domain. As what we can see from the blue dotted line, the difference of the void fraction between the center region and the walls is relatively small. In addition, these three curves all appear to be approximately symmetrical respect to the centerline of the channel, which is probably due to the width limitation of the domain.

4. Conclusion
Two-way coupled direct numerical simulation of fluid with many solid particles flowing around a stationary obstacle at three different Reynolds numbers is performed by a FVM-IBM-DEM method. Examination of the interactions between the fluid and particles on their dynamics shows that the flow
is laminar with regular movement characteristics at $Re=20$. Due to the action of particles, the vortices behind the obstacle tend to be collapsed. When $Re=100$, the vortices shed up and down alternately behind the obstacle. In addition to the main vortices, there are many randomly distributed small vortices in the flow field. When $Re=300$, the flow is turbulent, presenting a large pulsation. And there are vortices of various sizes in the channel. Under these three Reynolds numbers, there are more particles near the walls, while less in the center of the channel. But beyond that, at a low Reynolds number, the particles mainly follow the fluid in motion. At a moderate Reynolds number, the particles form particle-free zones in the main vortices because of the centrifugal force of vortex. At a high Reynolds number, the distribution of particles tends to be more uniform throughout the domain for the turbulent fluctuation.

5. References

[1] Ramesh G K, Gireesha B J and Gorla R S R 2015 *Heat Mass Transf.* 51 1061-1066
[2] Gireesha B J, Mahanthes B, Gorla R S R and Manjunatha P T 2016 *Heat Mass Transf.* 52 897-911
[3] Wang J and Levy E K 2006 *Exp. Therm. Fluid Sci.* 30 473-483
[4] Wang Y, Zhao Y and Yao J 2019 *Adv. Powder Technol.* 30 656-671
[5] Zhao P, Xu J, Ge W and Wang J 2020 *Chem. Eng. J.* 389 124343
[6] Fei J, Jie Y, Sun X and Chen X 2020 *Powder Technol.* 359 36-46
[7] Ouchegou O A, Pointeau V, Ricciardi G, Guazzelli E and Bergougnoux L 2020 *Mech. Ind.* 21 517
[8] Apostolou K and Hrymak A N 2008 *Comput. Chem. Eng.* 32 841-856
[9] Zhao L, Marchioli C and Andersson H I 2012 *Phys. Fluids* 24 021705
[10] Zhao L, Andersson H I and Gillissen J J J 2013 *J. Fluid Mech.* 715 32-59
[11] Zhao L, Challabotla N R, Andersson H I and Variano E A 2015 *J. Fluid Mech.* 115 244501
[12] Haugen N E L and Kragset S 2010 *J. Fluid Mech.* 661 239-261
[13] Haddadi H, Shojaei-Zadeh S, Connington K and Morris J F 2014 *J. Fluid Mech.* 760 R2-1
[14] Tao S, Zhang H, Guo Z and Wang L 2018 *Adv. Powder Technol.* 29 2003-2018
[15] Kleinhans U, Wieland C, Babat S and Spliethoff H 2019 *Fuel* 241 585-606
[16] Tryggvason G, Scardovelli R and Zaleski S 2011 *Direct numerical simulations of gas-liquid multiphase flows* (Cambridge: Cambridge University Press) p 50–69
[17] Xia H, Lu J and Tryggvason G 2019 *Comput. Meth. Appl. Mech. Eng.* 346 242-259
[18] Lin S, Lu J, Tryggvason G and Zhang Y 2018 *Phys. Fluids* 30 127105

Acknowledgments

This work was supported by the National Key Research and Development Program of China (No. 2018YFB1105304), the National Natural Science Foundation of China (No. 51905038), CCTEG Science and Technology Innovation Fund (No. 2020-2-TD-ZD002), and the Youth Academic Launch Program of Beijing Institute of Technology.