

Modeling and Simulation of Finite-Size Particles in Turbulence

Holger Homann

Abstract This course gives an introduction to the dynamics of finite-size particles moving in laminar and especially turbulent flows. Finite-size means that their diameter is larger than the smallest active scale of the carrier flow and that their slipping velocity is not negligible. An emphasis will be on the differences in the dynamics of these particles compared with those that are much smaller than any scale of flow variation. We will see that the differences are substantial: The dynamics of finite-size particles is hard to model, numerical simulations are challenging, large particles have a complicated imprint on the carrier flow and their mutual hydrodynamic interactions show rich properties.

1 Introduction

Many environmental and industrial flows are in a turbulent state. This means that inertial effects of the gas or liquid are important which might be for example due to high streaming velocities or low viscosities. There are various different types of turbulent flows that are distinguished by the properties of the flowing medium (water, air, plasmas, ...) or by the geometry and type of the boundaries (open, closed, moving, ...). However, turbulence has universal properties that motivates a systematic study of selected flow prototypes. Results might than be applied to similar systems.

In this course we are not only dealing with turbulence but with turbulent transport that is the motion of particles in a turbulent environment. It is not hard to imagine that this problem appears very often in environmental and industrial settings. Examples are the motion of pollutants in the atmosphere, the transport of organic material in the oceans, mixing devices, pumping of particle suspension through pipes or the motion of dust grains in protoplanetary discs.

The importance of turbulent transport for many open problems led to a intensive study of its statistical features. It has for example been found that turbulence leads to a

H. Homann (✉)

Laboratoire J.-L. Lagrange, Observatoire de la Côte d'Azur, CNRS,
Université Côte d'Azur, 06304 Nice, France
e-mail: holger@oca.eu

fast separation of neighboring particles and hence to very efficient mixing. Another feature is that particles that have inertia decouple from the dynamics of the fluid thereby creating spatial inhomogeneities called clusters. Recent studies were related to turbulent fluctuations that can throw particles with high speeds against each other. This phenomenon called caustics and has to be considered for the problem particle collisions.

Most of the collected data and theory is on the dynamics of very small particles, so called *point-particles*. This is because in applications transported particles are indeed often much smaller than any scale of the fluid motion and because the smallness assumption simplifies significantly their study: if a particle is much smaller than any scales on which the flow varies and if in addition it is moving only slowly with respect to the fluid the particle will be surrounded by a Stokes flow (see Fig. 1 (left)). The corresponding flow structure originates only from viscous and pressure stresses and has been calculated analytically. The Stokes flow is symmetric and the inflow does (by definition) not vary on scales comparable to the diameter of the particle; the inflow is said to be uniform. Any inertial effect or inhomogeneity of the fluid is negligible in this limit and the drag on the particle is given by

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\tau_p}(\mathbf{v} - \mathbf{u}), \quad (1)$$

where $\mathbf{u}$ is the fluid velocity, $\mathbf{v}$ the particle velocity and $\tau_p = 2\rho_p a^2/(9\rho_f \nu)$ its response time (ρ_p is the particle density, ρ_f the fluid density, a the radius of the particle and ν the kinematic viscosity of the fluid). τ_p is the time that a particle needs to relax to the velocity of the fluid and is a measure of inertia of the particle. In the particle equation (1) the fluid velocity $\mathbf{u}$ means more precisely the velocity at the position of the particle. Taken literally, this definition has of course no sense (the fluid is sticking to the surface of the particle) but for a Stokes flow a particle is

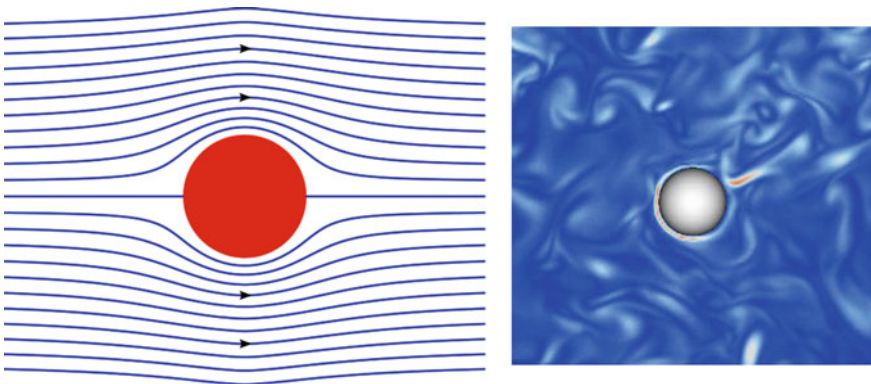


Fig. 1 *Left* Stokes flow around a point-particle. *Right* Vorticity in a meridian plane around a finite-size particle

facing a flow which is uniform at distances large compared to its diameter. In Fig. 1 (left) we could take any value for $\mathbf{u}$ at the boundary of the shown box. The nice thing about Eq. (1) is that it is just an ordinary differential equation for a given $\mathbf{u}$. This makes both analytic and numerical studies much simpler than studies of the transport of finite-size particles as we will see soon. Direct numerical simulations of turbulent point-particle suspensions can nowadays faithfully integrate the trajectories of billions of particles. Resulting data sets provide thus the basis for high precision statistical studies.

If one tries to model the force on bigger particles one encounters not only serious mathematical but also conceptual problems. The computation of the force in the point-particle approach relies (as we have seen) on the definition of a relative velocity of the particle with respect to the fluid. This very definition is not obvious if possible at all for a finite-size particle in turbulent flows. Particles that are larger than the Kolmogorov scale can be surrounded by a complicated flow structure (see Fig. 1 (right)). The fluid velocity in the vicinity of the particle can vary on scales that are smaller than the particle diameter. It is completely unclear which single velocity one should take from Fig. 1 (right) to define a meaningful velocity seen by the particle.

Such conceptual problems are one reason why the understanding of turbulence seeded with big particles is still in its infancy. Most of the works rely on the generation and exploitation of experimental or numerical data sets. But even the production of precise data is more complicated than for suspensions of very small particles. Numerically, one has to account for the interaction of the particle with the fluid which requires a precise modeling of the no-slip boundary condition at the surface of the particle and an adequate resolution of its boundary layer. Experimentally, one has to find ways to precisely track both the motion of the large and small particles. All this difficulties are responsible for an unpleasant situation: Even fundamental questions such as if turbulence is enhanced or attenuated by the presence of big particles is still under discussion. We will later come back to those open questions.

This course shall serve as an introduction to the topic of finite-size particle dynamics. Finite-size here means more precisely that either the diameter is of the order or larger than a characteristic scale of the flow (i.e. the Kolmogorov scale in turbulence) or that it has an important relative velocity with respect to the fluid so that it creates a wake. We can of course not discuss all possible flow types, open questions, numerical, experimental and analytic approaches. This is not the aim of this course so that we will only discuss selected problems and results. The main focus will be on the motion of finite-size particles in homogeneous isotropic turbulence. From time to time we will also consider the dynamics of sedimenting particles in a still fluid. This course includes a section that presents different approaches how the motion of finite-size particles and their fluid-particle interaction can be solved numerically. For experimental techniques the reader is asked to look at the course of Mickaël Bourgoïn. The hope is that this introduction provides sufficiently many ideas, reflections and open questions that motivate the reader to get deeper into the story. A good starting point would be to read the cited publications as in each case only one or two of their results can be mentioned here.

This course is organized as follows: We will start with a very brief introduction to turbulence. Then, some selected numerical methods are discussed that are used for direct numerical simulations of the transport of finite-size particles. After that, an important place is reserved for the study of differences in the dynamics of small and large particle. Hereby, the modification of the carrier flow by finite-size particles will be presented. The last section deals with collective effects of many particles that explicitly arise from a large particle diameter.

2 Basics of Turbulence

In this section we will introduce some basic properties of turbulent flows that are important for the subsequent sections. We will focus on incompressible flows obeying to the Navier–Stokes equations

$$\partial_t \mathbf{u} = -\mathbf{u} \cdot \nabla \mathbf{u} - \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f}_e, \quad \nabla \cdot \mathbf{u} = 0, \quad (2)$$

$\mathbf{u}$ denoting the velocity field, p the pressure, ν the kinematic viscosity, and $\mathbf{f}_e$ a possible external force maintaining the flow. The degree of turbulence can be characterized by the Reynolds number $Re = u_{rms} L / \nu$, L being the forcing scale and u_{rms} the root-mean-square value of the velocity. Often also the Taylor Reynolds number $R_\lambda = \sqrt{15} Re$ is used.

One of the properties of high Reynolds number turbulence is the creation and interaction of an enormous range of spatial and temporal scales. Let us take the example of a cloud and assume that it has a total size of one hundred meters. Its turbulent motion is driven at comparable scales by convection and shear. In three dimensional turbulence the energy contained in these large scales cascades to smaller and smaller scales. At very small scales of the order of one millimeter the viscosity of air transforms the kinetic energy into heat. In our cloud example we have thus a scale separation from 100 m to 1 mm, thus 10^5 different interacting scales involved. The largest scale is called *integral scale* L and the smallest scales is called *Kolmogorov scale* η . The mentioned energy cascade from large to small scales is a basic property of 3D turbulence. In between the two limiting scales (100 m and 1 mm) a purely inertial transfer of kinetic energy is taking place. In this interval of scales, called *inertial range of scales*, neither the particular mechanism of driving at large scales nor the dissipation mechanism at small scales is important so that the flow is more or less homogeneous, isotropic and approximately scale invariant.

Many scientific activities are devoted to the study of the idealized flow type of homogeneous isotropic turbulence (HIT). In this case statistical quantities such as velocity gradients are invariant under translation and rotations. Practically, one can think for example of a small parcel of a cloud in a co-moving (with the average velocity) frame of reference. In order to get a rough idea of the flow structure of HIT a snapshot of vorticity is shown in Fig. 2. This picture is taken from a direct numerical simulation (see subsequent section) of three-dimensional incompressible

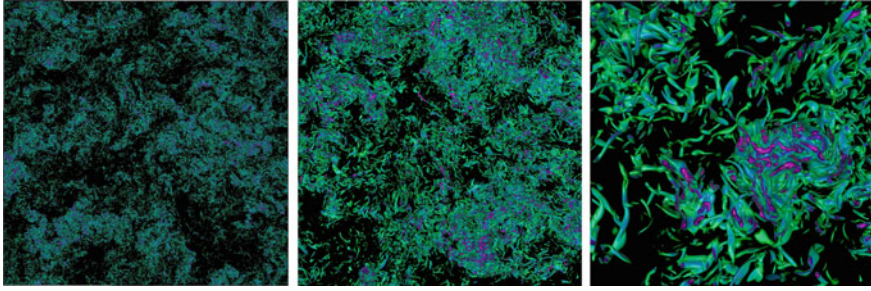


Fig. 2 Rendering of vorticity in a small slice of a direct numerical simulation of homogeneous isotropic Navier–Stokes turbulence with 4096^3 grid points ($R_\lambda = 730$). The *middle figure* applies a zoom of 4 while the *right figure* applies a zoom of 16 compared to the *left figure*

turbulence with periodic boundary conditions. One can see that HIT is composed of regions of strong vorticity (and strong dissipation in its vicinity). Looking in more detail into these regions (Fig. 2 (middle and right)) reveals that they consist of entangled vorticity filaments obviously interacting with each other. The diameter of such filaments is of the order of 10η and they belong to the smallest structures of HIT. These filaments are also present in other types of turbulent flows.

The fact that there is no characteristic scale within the inertial range of scales implies that scale (l) dependent quantities such as velocity differences along a separation l

$$\delta_l u = (\mathbf{u}(\mathbf{x} + \mathbf{l}) - \mathbf{u}(\mathbf{x})) \cdot \mathbf{l} / |\mathbf{l}| \quad (3)$$

behave as power laws. Indeed, the (second-order) structure function $S_2(l) = \langle (\delta_l u)^2 \rangle$ scales as $S_2(l) \sim l^{\zeta_2}$ within the inertial range and the extend of this spatial scaling range can in turn be estimated from S_2 . Two examples of S_2 are plotted in Fig. 3

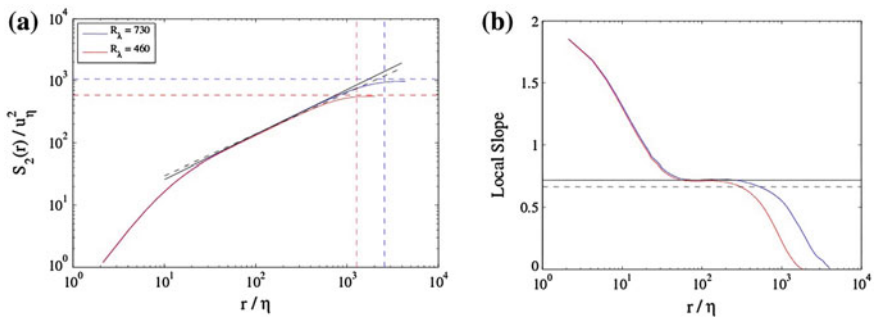


Fig. 3 *Left* Second order structure function S_2 in HIT for two different Reynolds numbers $R_\lambda = 730$ ($Re = 35000$) and $R_\lambda = 460$ ($Re = 14000$). *Right* Local slope of S_2 that is its logarithmic derivative $d \log(S_2) / d \log(r)$. The inertial range of scales can be defined as the interval where the local slope of S_2 is zero. Its scaling exponent ζ_2 corresponds to the value of this plateau (Bitane et al. 2013)

(left) for the simulation shown in Fig. 2 and for a simulation with a smaller Reynolds number. The main message is that at sufficiently high Reynolds number an inertial range of scales clearly exists but that Re has to be sufficiently high ($Re \approx 10000$, corresponding to a grid resolution of 2048^3 grid points).

Another important fact is that the scaling exponent ζ_2 is much smaller in the inertial range (see Fig. 3 (right)) than in the dissipation range where the flow is smooth ($\zeta_2 = 2$). The size of the vorticity filaments can roughly be seen as a landmark between the small and inertial range scales. Turbulence is thus not smooth within the inertial range of scales. However, these are the scales we are talking about if particles are meant to be of finite-size!

In this course we will consider the transport of particles having a size equal to or larger than the vorticity filaments. The basic equations of motion of a particle of any size read

$$m_p \ddot{\mathbf{X}} = \int_S \nabla \cdot \mathbf{T} dV = \int_{\partial S} \mathbf{T} dS, \quad (4)$$

$$\frac{2}{5} m_p a^2 \dot{\boldsymbol{\Omega}} = \int_{\partial S} \mathbf{n} \times (\mathbf{T} \cdot dS), \quad (5)$$

$\mathbf{T}$ being the stress tensor

$$\mathbf{T} = -p \mathbf{I}_3 + \nu (\nabla \mathbf{u} + \nabla \mathbf{u}^T). \quad (6)$$

The first equation (4) determines its translational motion and the second equation (5) its rotation. The particle Reynolds number $Re_p = D_p V_p / \nu$ usually involves the particle diameter D_p , its relative velocity V_p and the kinematic viscosity. The velocity $\mathbf{u}$ appearing in (4) is a solution to the Navier–Stokes equations (2) with a no-slip boundary condition at the particle surface. This is a coupled non-linear system of partial differential equation that shows the complexity of the problem.

Let us conclude this section by stressing that homogeneous isotropic turbulence is one type of turbulent flow on which physicists often concentrate. However, one should not forget that in real flows often a multitude of physical processes is contributing to its evolution. Concerning a cloud, for example radiation, latent heat and electricity play for example a role. Also there are of course other archetypal and important flows such as convecting, rotating, pipe-, wall- and channel flows.

3 Numerical Methods for Finite-Size Particles

Finite-size particles in turbulence are surrounded by a complicated flow structure (recall Fig. 1 (right)) for which no general analytic solution can be found. Indeed, finding a simple equation modeling the force on such a particle in the spirit of (1) is part of ongoing research activities. In order to get insight into the dynamics of finite-size particles one can perform direct numerical simulations (DNS). DNS means that

the basic equations (2), (4), and (5) are solved directly at all scales and no modeling or simplifications are used. This is especially adequate for systems that are not well understood. In the case of large particle dynamics this kind of simulations has become a valuable tool because the computing power of supercomputers and the performance of algorithm increased to such an extend that DNS of suspension of big particles are nowadays possible.

Several different strategies have recently been proposed in order to perform DNS of spherical particles. Most of them make use of a homogeneous grid that covers the complete physical domain and resolves all turbulent scales. The principle idea is to use a solver of the Navier–Stokes equations and to impose in some way the no-slip boundary condition at the surface of the particle. Let us now mention some of those strategies:

“Physalis”, developed by Prosperetti and Oguz (2001) and extended by Naso and Prosperetti (2010), uses the analytic solutions to the Stokes equation for the viscous boundary layer in the vicinity of the particle. This Stokes solution is used to prescribe the velocity at the nearest surface points of the grid. Iteratively, the analytic stokes solution and the outer solution on the homogeneous grid are matched. That way the no-slip condition is imposed in an exact manner if the grid spacing is much smaller than the viscous boundary layer.

The “Overset Grid” technique proposed by Burton and Eaton (2005b) consists in using and matching two different grids. A spherical one encloses the particle surface and is designed to resolve the boundary layer very accurately and a second homogeneous grid that covers the entire domain. A third-order interpolation is used to transfer values from one grid to the other in order to couple the two solutions.

The “force coupling” method of Maxey and Patel (2001) solves the Navier–Stokes equations including a body force that originates from the transported particles. This body force is computed from a low-order multipole expansion of the force exerted by a particle on the fluid. From this expansion they retain the monopole term including buoyancy and forces from the displaced fluid and the dipole term including straining and rotation effects. Due to numerical manageability they replace the Dirac delta function by a Gaussian envelop of a width which is connected to the size of the particle. The no-slip boundary condition is therefore only approximately fulfilled.

The “immersed boundary” approach solves the Navier–Stokes equations in the entire domain (usually on a uniform grid) including the particles. The no-slip boundary condition is imposed via additional forces acting on the fluid in the volume occupied by the particles. This technique has been used together with finite-difference schemes (Uhlmann 2005) and pseudo-spectral schemes (Homann et al. 2013). Let us now look at the latter in more detail.

The main idea consists in combining a Fourier pseudo-spectral method for the fluid with an immersed-boundary technique to impose the no-slip boundary condition on the surface of the particle. Pseudo-spectral means that spatial derivatives arising in (2) are computed in Fourier-space while convolutions from the non-linear term are computed in real space. A Fast Fourier Transformation (FFT) is used to switch between the two spaces. Spectral schemes are a standard tool of DNS of HIT because of their accuracy.

The no-slip boundary condition at the particle surface is treated via an immersed boundary technique. For a particle at rest this condition reads

$$\mathbf{u}(\mathbf{x}, t) = 0 \quad \text{for } \mathbf{x} \in \partial\Omega_p, \quad (7)$$

for a particle at rest and center $\mathbf{x}_0$. Here, $\Omega_p = \{\mathbf{x} : |\mathbf{x} - \mathbf{x}_0| \leq D_p/2\}$ denotes the volume occupied by the particle. The immersed boundary technique consists in introducing in the right-hand side of the Navier–Stokes equation (2) a force $\mathbf{f}_b(\mathbf{x}, t)$ associated to the constraint defined by the boundary condition (7). The full problem (2)–(7) can then be rewritten as

$$\partial_t \mathbf{u} = L(\mathbf{u}) + \mathbf{f}_b, \quad \nabla \cdot \mathbf{u} = 0, \quad \text{for } \mathbf{x} \in \Omega, \quad (8)$$

where $L(\mathbf{u})$ denotes the right hand side of (2). There are now different ways to compute the force $\mathbf{f}_b$. One of them is called direct forcing method (see Fadlun et al. 2000) and directly imposes the particle velocity to the grid points in the volume occupied by the particle. As a uniform grid does not coincide with the particle surface some sort of interpolation of the surface velocity to the nearest grid points has to be used. This results in an effective smoothing of the particle surface making its surface slightly porous.

Here might be the right place to stress today’s limits of direct numerical simulations. Modern efficient numerical codes which include optimized algorithms and parallelization run on thousands of computing cores. They use the fastest supercomputers in the world and consume millions of CPU hours per simulation. Nevertheless, the accessible parameter space is still limited. The Reynolds number of the flow is usually of the order of one thousand and also the number of particles does not exceed a few thousand. This is one reason why the amount of available data on this topic is also still limited. One should mention that experiments have the advantage to reach higher Reynolds and particle numbers but often suffer from other difficulties.

4 Finite-Size Effects of Individual Particles

In the introduction we saw that the flow structure around finite-size particles in turbulent flows is in general different from that of point particles. However, we do not know yet if their dynamics differ. But in fact they do. In this section we are going to present finite-size features of particle transport that definitely distinguish large from small particle dynamics.

4.1 Slip Velocity

Let us start by discussing the slip velocity of large particles. Slip velocity means the relative velocity of a particle with respect to the fluid. In the introduction we saw that this velocity difference had a simple meaning for point particles. They are so small that the surrounding flow is nearly uniform at large distances from the particle. In this region, every fluid parcel is moving with the same velocity relative to the particle. This velocity is obviously the slip velocity.

A finite-size particle in turbulence is (by definition) facing a flow that varies on scales smaller than the particle diameter (recall Fig. 1 (right)). There is no such region of uniform fluid velocity. However, also a big particle is somehow moving through the fluid and creating eventually even a clearly visible wake. There might be therefore at least some hope that a meaningful definition of the slip velocity exists. Because of the turbulent fluctuations spatial averages might be a good candidate. Lucci et al. (2010) and Kidanemariam et al. (2013) define the fluid velocity seen by the particle to be the average velocity on a concentric shell around the particle. They take a shell with a radius of the order of the particle diameter. The idea is that at this distance the flow is only weakly influenced by the boundary layer of the particle. The slip velocity is then the difference of this averaged velocity and the particle velocity.

Cisse et al. (2013) computed the shell averaged velocity as a function of the shell radius. They introduced a two dimensional coordinate system consisting of the slip direction and a normal direction. The slip direction $\vec{e}_r(t)$ is computed from the fluid flux through each shell as

$$\vec{e}_r(t) = \vec{\Phi}_r(t)/|\vec{\Phi}_r(t)|, \quad \text{where } \vec{\Phi}_r(t) = \int_{S_r} \left(\vec{u}(\vec{x}, t) - \vec{V}_p(t) \right) \cdot \vec{n} \, d\vec{S}, \quad (9)$$

where $\vec{u}$ and $\vec{V}_p$ are the fluid and the particle translational velocity, respectively, $\vec{n}$ is the unit vector normal to the shell. In other words, on each shell an average of the direction weighted by the fluid mass flux is performed, so that $\vec{e}_r$ points in the direction of the flux on the shell at distance r . This choice is physically motivated as the fluid enters such a shell upstream and exits it in the wake. For a Stokes flow, $\vec{e}_r$ would be independent of r and exactly parallel to the slip direction.

Having the direction $\vec{e}_r$ defined, one can project the velocity difference $\vec{u} - \vec{V}_p$ onto it and perform a time average to construct the mean velocity profile of the flow relative to the particle

$$U_{\text{rel}}(\rho, z) = \left\langle \left(\vec{u}(\vec{x}, t) - \vec{V}_p(t) \right) \cdot \vec{e}_r \right\rangle, \quad (10)$$

with $z = (\vec{x} - \vec{X}_p(t)) \cdot \vec{e}_r$ and $\rho = [|\vec{x} - \vec{X}_p(t)|^2 - z^2]^{1/2}$. The coordinates z and ρ , which are defined at each instant of time, are in the direction of $\vec{e}_r$ and perpendicular to it, respectively. By rotational symmetry around the axis defined by $\vec{e}_r$, the mean profile U_{rel} depends on z and ρ only and not on the angle.

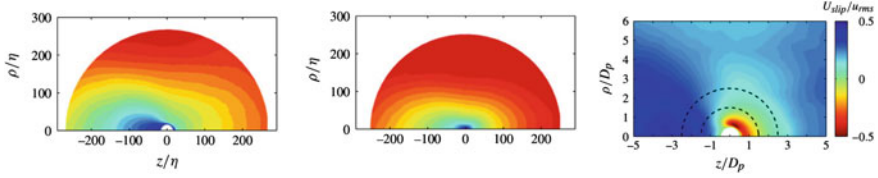


Fig. 4 *Left* Temporal- and angle-averaged fluid relative velocity projected on the direction of motion for $D_p = 34\eta$ (*left*) and for a tracer (*middle*). *Right* slip velocity U_{slip} for $D_p = 34\eta$ defined as the difference between the mean relative velocity profile U_{rel} around the particle and that around a tracer; the two *black dashed circles* represent distance equal to D_p and $2D_p$ from the particle surface (Cisse et al. 2013)

The relative velocity profiles (see Fig. 4 (left)) have a marked asymmetry resembling the inflow and wake structure of a particle facing a uniform flow. However, performing the same computations for a tracer particle moving in a similar flow produces a similar velocity profile (see Fig. 4 (middle)). The problem is that a tracer has by definition no slip velocity or wake. Indeed, the measured velocity profile is a consequence of the intrinsic asymmetry of the longitudinal velocity increments (3) in HIT and the specific choice of the coordinate system. In HIT, velocity differences in the direction of the separation increase with the separation and are skewed. Cisse et al. (2013) therefore proposed to study the difference of the mean velocity profiles of a finite size particle and that of a tracer (see Fig. 4 (right)). This difference shows strong variations close to the particle (up to twice its diameter) while it is approximately constant at further distances. This constant is then defined to be the average slip velocity.

A completely different strategy to define a relative velocity is pursued by Bellani and Variano (2012). They define a *stochastic* slip velocity using the variance of the fluid and particle velocity

$$U_{\text{rel}} = (\langle \mathbf{u}'^2 \rangle - \langle \mathbf{v}'^2 \rangle)^{1/2}. \quad (11)$$

Here, the primes denote the fluctuations with respect to the mean $\mathbf{u}' = \mathbf{u} - \langle \mathbf{u} \rangle$. An advantage of this definition is its simplicity and direct applicability to experiments. However, from its very definition it is clear that this only gives an average and no local definition.

We see from this that already the definition of slip velocity is a subtle problem for finite-size particles. Next we will look at the modification of turbulence in the vicinity of an individual particle.

4.2 Modification of Turbulence

It is well known that finite-size particles in a uniform flow create a boundary layer and wake. One may ask how this modifies the properties of the turbulent flow. How do for example particles change the energy dissipation rate in its vicinity?

Burton and Eaton (2005a) performed 3D DNS of freely decaying HIT in which they put one fixed particle with a diameter comparable to the Kolmogorov length scale. By means of ensemble averages they find that the local energy dissipation is increased close to the particle compared to the outer turbulence. The particle displaces the fluid, creates a boundary layer and increases thereby velocity gradients. Further away, there is some indication that at intermediate distances the dissipation rate might be reduced. This has also been found by Naso and Prosperetti (2010) who studied the same situation as Burton and Eaton but in stationary (forced) HIT (see Fig. 5 (left)). A reduced energy dissipation rate means that the particle attenuates turbulent velocity fluctuations at small scales (the energy dissipation is a small scale quantity). The particle influences obviously scales smaller than its diameter.

Cisse et al. (2013) studied the same question but for a moving particle. They performed numerical measurements around a neutrally buoyant (having the same density as the fluid) sphere. Using their definition of the slip direction (see previous section) they distinguished the energy dissipation rate in the upstream and downstream flow (see Fig. 5 (right)). In the upstream flow the dissipation is strongly increased. Downstream, at distances from $0.2D_p$ to $1D_p$, the dissipation is reduced compared to the bulk turbulence. It is thus the wake of the particle that attenuates turbulence. However, very close to the particle (in the viscosity dominated boundary layer) the dissipation is even downstream increased.

These results concern particles moving with moderate slip velocity through the fluid. Rough estimates of their particle Reynolds numbers range from 20 to 200. At these speeds the wake of the particle is still laminar. But higher speeds (for bigger

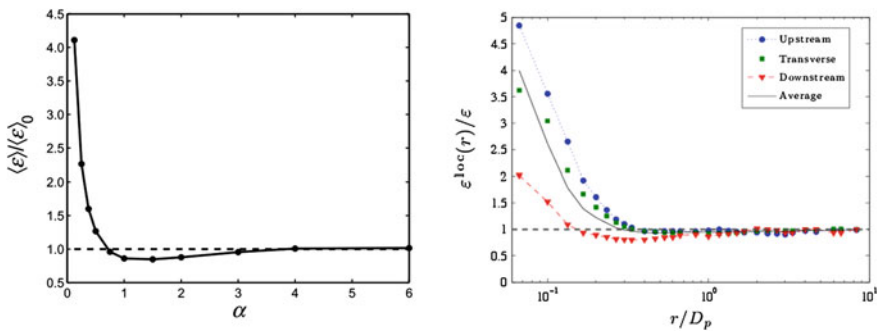


Fig. 5 Energy dissipation around a finite-size particle in HIT as a function of the distance from the particle surface. *Left* Data for a fixed particle ($\alpha = (r - R)/R$, (Naso and Prosperetti 2010)). *Right* Data for a moving neutrally buoyant particle distinguishing contributions from up- and downstream (Cisse et al. 2013)

or heavier particles) would imply the creation of turbulent fluctuations in the wake and those might lead to an increase of the energy dissipation rate.

Let us now turn to the drag force on a large particle facing turbulent fluctuations. It is clear that this quantity is crucial for the possible modeling of large particle dynamics via a simple equation such as (1).

4.3 Drag Force in Turbulent Flows

The drag force on a particle moving at a constant speed U_c (see Fig. 6) through a fluid at rest has been studied for a long time. The corresponding drag coefficient $C_D = 8 F / (\rho U_c^2 \pi D_p^2)$ is a non-trivial function of the particle Reynolds number Re_p (see Fig. 7 (left)). Analytic results have only been obtained for small Re_p . Other formulas such as the one of Schiller and Naumann (1933)

$$C_D(Re_p) = \frac{24}{Re_p} (1 + 0.15 Re_p^{0.687}) \quad (12)$$

are only reasonable fitting functions to experimental and numerical data up to a certain Re_p .

At small Re_p the drag is increasing linearly (C_D is decreasing) according to the Stokes drag. From $Re_p \approx 10^3$ to $Re_p \approx 10^5$ it increases quadratically so that C_D becomes flat. Associated to these different drag regimes are specific flow patterns around the particle. For small Re_p one finds a steady, approximately symmetric flow resembling the Stokes flow. At intermediate Re_p the flow is still steady but a recirculation region appears in the wake. In the quadratic drag regime the wake is turbulent (see also Fig. 6). At a even higher Re_p the boundary layer is said to become turbulent (a regime that we will not consider).

We see that the drag force and the according flow patterns are already complicated in the case of an uniform inflow. The question arises what happens if the inflow is turbulent (see Fig. 8). Clearly, turbulent velocity fluctuations will interact with the boundary layer and wake of the particle. The question is whether one can also apply the empirical formula (12) to the case of a turbulent flow interacting with a particle or will the drag force be modified?

Bagchi and Balachandar (2003) analyzed this question by means of different numerical simulations. They either placed a finite-size particle or point particle into

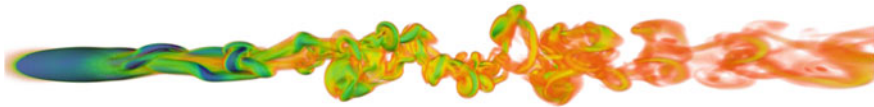


Fig. 6 Volume rendering of vorticity of a uniform flow passing a spherical particle with $Re_p = 1000$

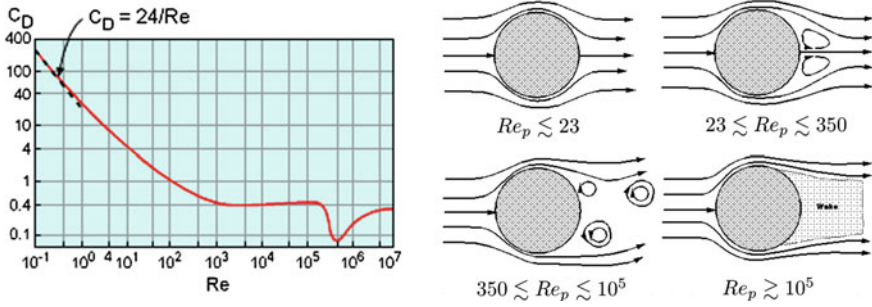


Fig. 7 *Left* Drag coefficient of a spherical particle facing a uniform inflow. C_D . *Right* Flow structure depending on the particle Reynolds number



Fig. 8 Volume rendering of vorticity of a turbulent flow with average speed U_c passing a spherical particle with $Re_p = 400$ and $I = u_{rms}/U_c = 0.12$

the same turbulent inflow. They found that already for a particle size of $d/\eta = 1.5$ only the large scale variations are correctly reproduced while the small scales are clearly not captured (see top panel of Fig. 9). This can be understood by considering the large scale variations as a slowly varying inflow. If the inflow velocity changes on time-scales much larger than the time it takes the fluid to pass the particle, the latter faces a quasi-stationary headwind. The small scale fluctuations are more complicated to understand and model. They are not included in standard drag correlations for uniform inflows.

The agreement of the finite-size DNS and the point-particle simulations is getting even worse when increasing the particle size (see middle and bottom panel of Fig. 9). Bagchi and Balachandar (2003) checked whether this can be improved by taking additional force terms into account. Indeed, Maxey and Riley (1983) and Gatignol (1983) derived the equation

$$\frac{d\mathbf{v}}{dt} = -\frac{1}{\tau_p}(\mathbf{v} - \mathbf{u}) + \beta \frac{D\mathbf{u}}{Dt} - \sqrt{\frac{3\beta}{\pi \tau_p}} \frac{d}{dt} \int_0^t \frac{\mathbf{v} - \mathbf{u}}{t - \tau} d\tau \quad (13)$$

with $\beta = 3/(2\rho_p/\rho_f + 1)$ (ρ_p and ρ_f being the mass density of the particle and fluid, respectively). The second and third term on the right hand side are called added mass term and history term, respectively. However, Bagchi and Balachandar (2003) showed that the inclusion of the added mass and even the history term did not improve the agreement of the point model with the finite-size simulations. This is a bit surprising, as Daitche (2015) analyzed in detail the role of the history force

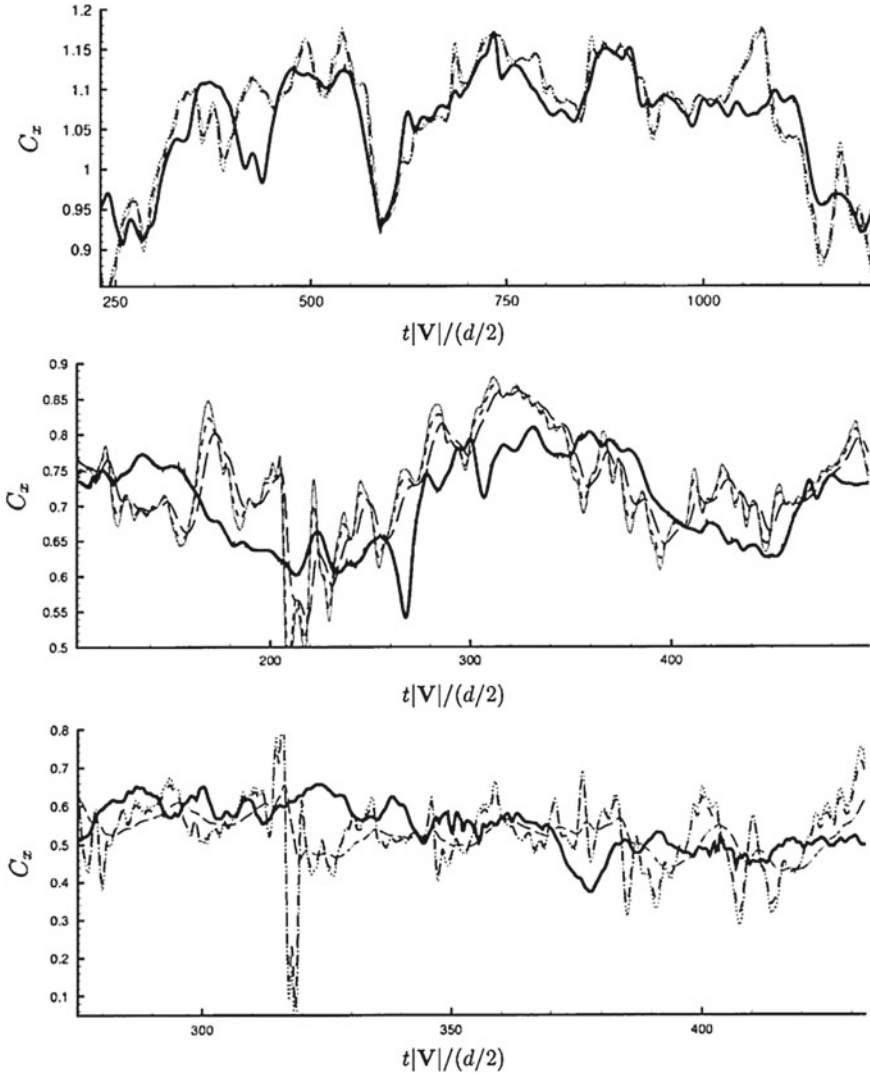


Fig. 9 Time series of the stream-wise force component of a spherical particle facing a turbulent inflow. *Solid line* DNS of a finite-size particle (*top* $d/\eta = 1.5$, $Re_p = 107$; *middle* $d/\eta = 3.8$, $Re_p = 261$; *bottom* $d/\eta = 9.6$, $Re_p = 609$) facing a turbulent inflow of $I = 0.1$. *Long dashed line* Schiller and Naumann law (12). *Short dashed line* plus the inertial force. *Dotted line* plus the history force (Bagchi and Balachandar 2003)

in a turbulent flow and found that its relative contribution compared to the Stokes drag scales as a/η . One could have thus expected the importance of this force term to grow with the particle size.

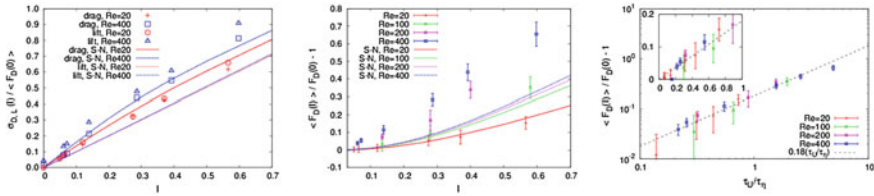


Fig. 10 *Left* Standard deviation of the drag force in free-stream turbulence for different turbulent intensities I . The *lines* correspond to a simple model assuming a quasi-static inflow speed. *Middle* Mean drag force. *Right* Mean drag force this time as a function of τ_U / τ_η = particle passing time/turbulent dissipation time scale (Homann et al. 2013)

Kim and Balachandar (2012) found indications that the drag force might be increased in turbulent compared to uniform flow. And here, increased means beyond that what one would expect from quasi-static changes of the inflow speed. Indeed, a slowly varying inflow speed already leads to an increase of the average drag force due to its non-linear dependence on the inflow speed (see (12)). The observed additional increase, however, was within the error bars.

Let us note here that this fluid dynamical problem has three independent parameters: the fluid Reynolds number $Re = u_{rms} L / \nu$, the particle Reynolds number $Re_p = U_c D_p / \nu$ and the turbulent intensity $I = u_{rms} / U_c$. Other non-dimensional quantities such as D_p / η are consequences of them.

Later, Homann et al. (2013) redid DNS of a similar situation (fixed sphere in turbulent headwind) and found that the standard deviation of the fluctuations of the drag force follow approximately the prediction of quasi-static changes of the inflow speed (see Fig. 10 (left)). The large scale fluctuations of the inflow translate into a shaking of the particle that is captured by the quasi-static assumption. However, the mean drag force was significantly higher than what one would expect (see Fig. 10 (middle)). From the right panel in Fig. 10 Homann et al. (2013) claimed that this increase might be due to small scale fluctuations of the inflow: Once the drag force is plotted as a function of the ratio of two small-scale time-scales all data collapse to one single straight line. The two time-scales are the time it takes the flow to pass the particle and the turbulent dissipation time scale. The mean drag force might be increased by small scale fluctuations that modify the boundary layer in such a way to increase the velocity gradients. Steeper gradients in turn lead to larger forces.

There are of course other situations than turbulent flows in which things change for finite-size particles. One of them is the motion of a particle sedimenting under the action of gravity in a still fluid. Let us look at this now. This will serve as a preparation for the complicated case of the interaction on many sedimenting finite-size particles discussed later.

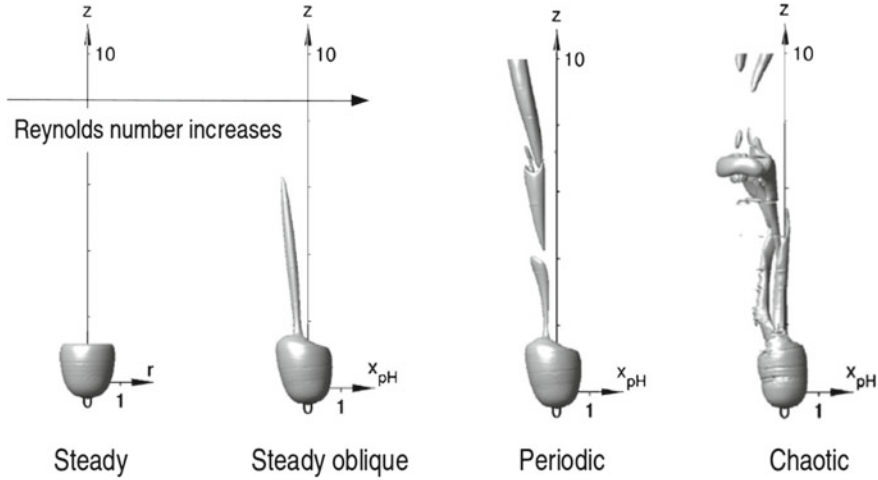


Fig. 11 Flow structure (vorticity) of a spherical particle sedimenting in a quiescent fluid. The particle Reynolds number increases from left to right. Remark that the wake structure is not vertical (Uhlmann and Dušek 2014)

4.4 Sedimenting Particles

We will consider the case of a single finite-size particle sedimenting in quiescent fluid. Despite the fact that the wake structure depends on the particle Reynolds number one finds that depending on Re_p particles are not just falling along vertical lines. Indeed, for intermediate Re_p particles move on straight but oblique trajectories (see Fig. 11). Even if the wake is steady it can give rise to a horizontal velocity. One would of course not expect this feature for point-particle as they are (by definition) moving in a symmetric (Stokes) flow.

5 Collective Effects of Finite-Size Particles

This section is devoted to collective effects in the particle dynamics, i.e. effects arising from the interaction of many (at least two) particles. Let us already distinguish two different classes of collective effects: Those simply arising from a superposition of the imprints of individual particles onto the flow in very diluted systems and those originating from the hydrodynamical interactions of particles.

Collective effects are worth studying because the admixture of impurities to a turbulent flow can have a significant impact onto the flow. It is for example well known that even a very small polymer concentration can lead to a significant drag reduction in oil pipes. A feature used in the petroleum business. Oil with polymers is a visco-elastic fluid for which models exists that allow for an exchange of the kinetic

energy of the fluid and the elastic energy of the polymers. Polymers suppress large velocity gradients and lead in this way to a drag reduction.

Finite-size particles have a more complicated imprint on the flow. In turbulent settings its radius of influence is of the order of its diameter. This is true for neutrally buoyant particles while other particles with higher Reynolds numbers might produce wakes that are visible at even longer distances. It is not surprising that in suspensions of many particles these flow modifications can have a macroscopic effect on the flow structure. In very dilute suspensions (the simplest case) the individual particle contributions might simply add up so that modifications can be understood from the one-particle dynamics. But in denser suspensions particles and their respective wakes interact with each other thereby creating new effects. The aim of this section is to present a selection of collective effects of finite-size particles. Data and results on this topic are still quite rare as already the understanding of the dynamics of a single particle is lacking. So let us go step by step. Before considering a large number of particles we will study what happens if two particles come close to each other and interact hydrodynamically.

If two particles approach each other they squeeze out the fluid between them. This creates a repulsive force that is called lubrication force. Particles with smooth surfaces cannot touch *a priori* because this force goes to infinity at contact. Reality, however, might be different. The surface might be rough, van der Waals forces might act or the fluid assumption (inter-particle distance much larger than the mean free path) might break down. The lubrication force is also at play if close particles separate. Now, fluid has to be squeezed in so that particles slow down by transferring their kinetic energy to the fluid.

In the next paragraph we will see how the drag and lift forces on a particle are affected by the presence of a second particle in its vicinity.

5.1 *Two Interacting Particles*

The main question of this section is what happens to the drag force on a fixed particle in a uniform flow if a second particle comes close to it? There are two very different possible arrangements of the particle positions. They can be on a line in stream-wise direction or on a line perpendicular to it. In the first case we expect that the drag on the following particle is reduced in the slipstream of the leading particle (an experience made by cyclists and motorists). Let us now consider this configuration in more detail.

Zhu et al. (1994) experimentally measured the drag force on two particles that are aligned in stream-wise direction (see Fig. 12 (left)). The particles build a tandem with a fixed separation. They varied their distance in a uniform flow and measured the associated drag forces. They found that the trailing particle experiences a significantly reduced drag force that can be as small as one fifth of the drag force of an isolated particle (see Fig. 12 (right)). The closer the particles are the smaller is the drag. The reason for this is the wake of the leading particle in which the difference between

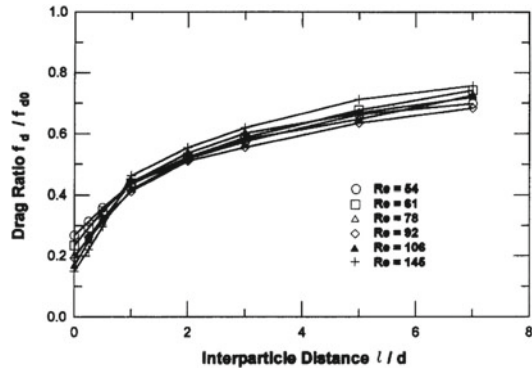
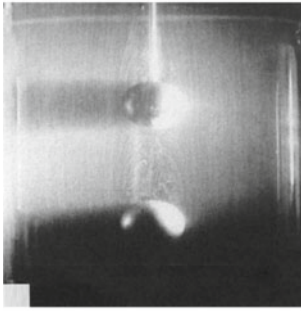


Fig. 12 *Left* Two particles aligned in stream-wise direction. *Right* Drag force on the trailing particle (Zhu et al. 1994)

the particle velocity and that of the fluid is reduced. This so called velocity deficit is zero at the particle surface and recovering only asymptotically the inflow value. The leading particle is entraining the fluid. The trailing particle will thus face a reduced headwind speed that in turn reduces its drag. This is quite obvious. However, as can also be seen in the right panel of Fig. 12, there is some non-trivial Reynolds number dependence of this drag reduction. At small distances from the particle the drag reduction is the larger the smaller is the particle Reynolds number. This tendency is reversed at larger distances. The cross-over happens at approximately at a distance of one diameter.

Before continuing let us stress the importance of this effect of drag reduction, called *wake attraction effect*. It is clear that this effect has the potential to bring particles together so that for example sedimenting particles will tend to accumulate. And it is also clear that a bunch of particles will have other dynamics than an individual particle. We will later come back to this point.

The work of Zhu et al. (1994) documents another interesting observation: Also the leading particle experiences a drag reduction when a trailing particle catches up. This reduction is quite small and reaches only ten percent at a distance of one diameter before it is slightly reincreasing. For the leading particle it is thus beneficial to have a particle following closely. The reason is that a part of the drag on a particle originates from its wake generation. The trailing particle obviously modifies the boundary layer and wake of the leading particle in such a way to reduce stresses.

Now let us come to the second configuration: We will put the particles side by side instead of one after another. Kim et al. (2006) fixed two of them as sketched in Fig. 13 (left) in a uniform flow and varied their Reynolds number and distance. They report two important findings: First, their individual drag force increases significantly for distances smaller than one diameter. The increase is larger the closer they are. The second particle has a sort of blocking effect on the fluid passing the first. The second important finding is that the lift force i.e. the force between the particles varies non-trivially as a function of the inter-particle distance. Close particles (closer than 1.5

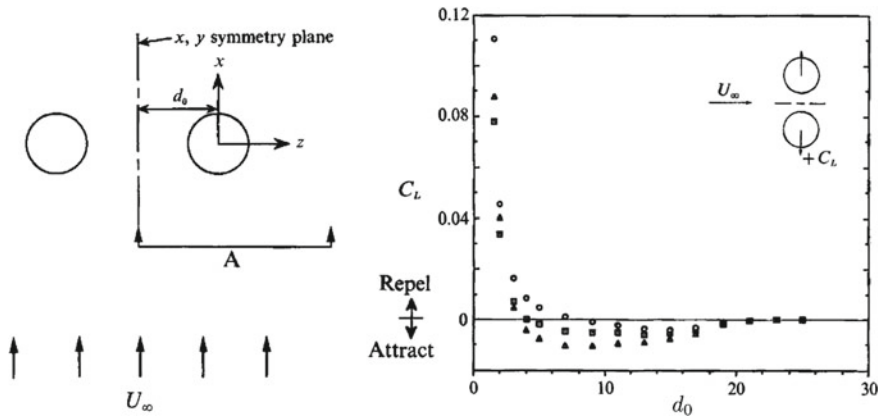


Fig. 13 Left Two particles aligned side by side. Right Lift force on the particles (Kim et al. 2006)

diameter) are repelled. The flow between them pushes them strongly apart. However, particles at distances of two to twenty diameter are slightly pulled together (see Fig. 13 (right)). This attraction effect increases with the particle Reynolds number. Roughly speaking this sounds like the Venturi effect where the static pressure decreases when the dynamics pressure increases. However, it is not obvious why this effect inverses at particle separations of the order of one diameter.

So far the two particles were fixed. Let us now see what happens to two sedimenting particles in still fluid (see Fig. 14). If they are vertically aligned the wake attraction effect will accelerate the trailing particle to catch up the leading one. They will collide which gives horizontal momentum to the particles and make them separate again. Their boundary layers and wake interact in a complicated manner leading to a tumbling motion. This dynamics is called *drafting, kissing, tumbling*.

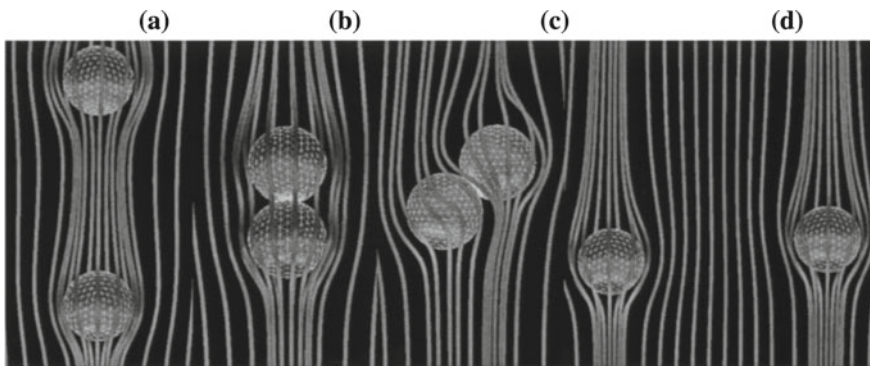


Fig. 14 Interaction of a sedimenting pair of particles - drafting (a), kissing (b), tumbling (c-d) (Prosperetti and Tryggvason 2007)

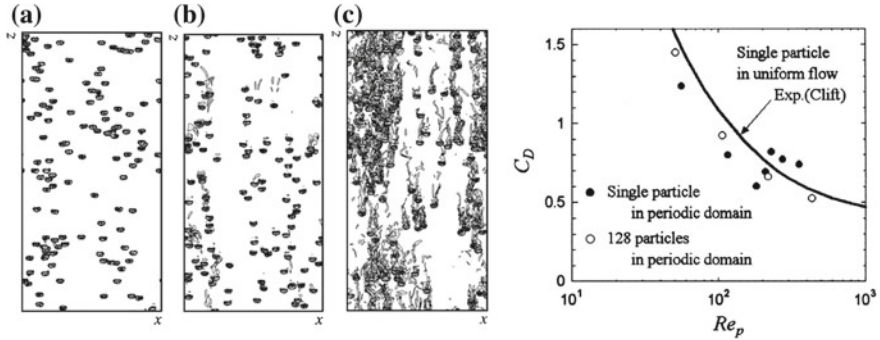


Fig. 15 *Left* Instantaneous flow field including 128 particles for particle Reynolds numbers 100, 200, and 400 (from *left to right*). *Right* Drag coefficient of an individual particle and a sedimenting cloud of 128 particles (Kajishima and Takiguchi 2002)

5.2 Sedimentation

The last paragraph introduced the important effect of wake attraction. We can expect that it will also play an important role when many particles sediment under gravity. This situation has been studied by Kajishima and Takiguchi (2002) who performed DNS of 128 identical particles (same size and density) sedimenting under the action of gravity. They found two remarkable finite-size effects: First, wake attraction leads to the formation of clusters (see Fig. 15 (left)). Trailing particles are slip-streaming and thus approaching the leading particles. In principle, this works for many particles as for two particles. In consequence particles agglomerate in vertically elongated clouds. The stability and evolution of such clusters with respect to the intrinsically generated velocity fluctuations is a more complicated and still an open question. The second finite-size effect is that, the average drag force experienced by sedimenting particles is reduced if many particles are present (see Fig. 15 (right)). The wake attraction effect gives thus rise to a collective effect in reducing the force on the individual particles.

This drag reduction is also reflected in the sedimentation speed. Uhlmann (2005) considered the evolution of the average vertical velocity of particles initially at rest. He performed several DNS and varied the number of particles. A single particle accelerates monotonously and reaches asymptotically its terminal velocity. Uhlmann (2005) showed that a set of particles with initial positions on a regular grid passes through different stages. Once released they accelerate as expected (see Fig. 16 (right)). Soon they reach (on average) a higher velocity than a single particle. During this phase, the wake attraction effect reduces the drag and allows for higher vertical velocities. However, once wake induced fluctuations make the particle deviate from their initially regular spatial distribution their settling velocity reduces. It reduces to values smaller than that of a single particle. The important point here is that the particle Reynolds number ($Re_p \approx 400$) is sufficiently high that the particle wakes

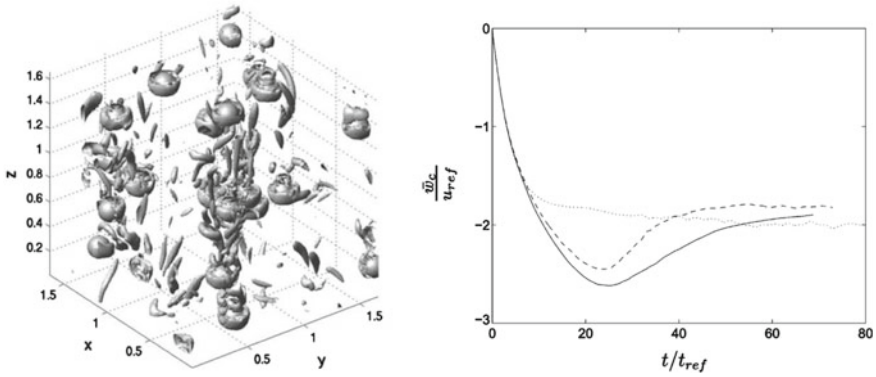


Fig. 16 *Left* Instantaneous flow field including $Re_p \approx 400$ particles. *Right* Average settling speed as a function of time (Uhlmann 2005)

are turbulent (see Fig. 16 (left)). The particle-wake interactions lead to an increased drag and decreased settling velocity. Uhlmann (2005) shows that a higher particle number leads to a lower average settling speed. This is in agreement with the observation (from Sect. 4.3) that the drag force increases with increasing turbulent intensity because a higher number of particles leads to more fluctuations (see Fig. 16 (left)) and in turn to a higher turbulent intensity.

In a later numerical work Uhlmann and Doychev (2014) focused on possible clusters of many sedimenting particles. They considered two different systems having the same solid to fluid density ratio of 1.5 and the same solid volume fraction of

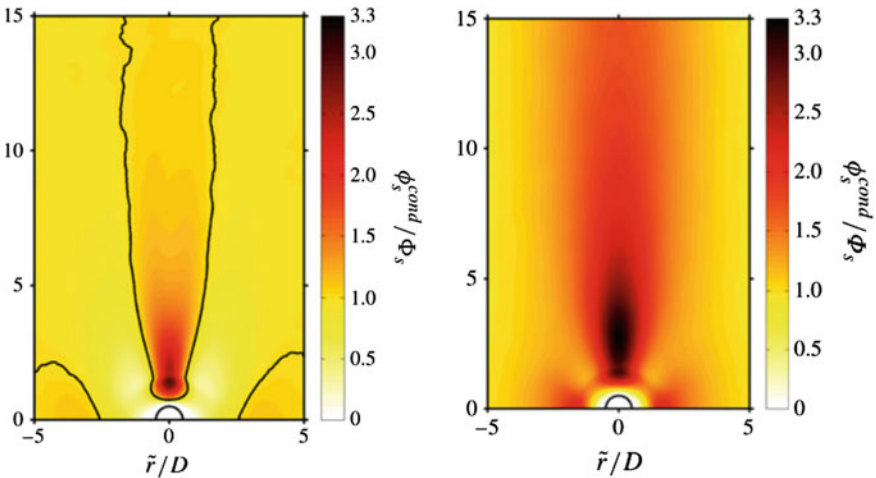


Fig. 17 Averaged solid volume fraction conditioned on particle positions, normalized by the global solid volume fraction. *Left* $Ga = 121$. *Right* $Ga = 178$ (Uhlmann and Doychev 2014)

0.005 but different Galileo numbers $Ga = u_g D / \nu$, $u_g = \sqrt{|\rho_p / \rho_f - 1| D g}$. Starting from initially random particle positions, they found that the particles do not always agglomerate. Only the system with the higher Galileo number ($Ga = 178$) showed significant particle concentrations while the simulation with $G = 121$ did not. This is visualized in Fig. 17 where the normalized average solid volume fraction conditioned on the particle position is shown. This quantity is the lateral-angle averaged volume fraction of particles surrounding a given particle in the cylindrical coordinate system $((\tilde{r}, \tilde{z}) = (\text{lateral}, \text{vertical}))$. It is normalized to the mean solid volume fraction. This representation reveals important features about the average particle distribution seen by individual particles. The leading particle often pulls a trailing particle very close to it manifested by the dark disc at a distance of approximately one diameter. In the lower Ga case (Fig. 17, left) this disc has a short shadow indicating the presence of second trailing particle. In the higher Galileo number case (Fig. 17, right) this shadow is not only much longer but also broader showing the coordinated movement of many particles and thus a cluster. The existence of the latter can also be concluded from the lateral density profile. For $Ga = 121$ one does not observe an increased particle density at the sides of the reference particle while this is the case for $Ga = 178$.

Their explication of these Galileo number based differences is based on the single particle dynamics: Above a critical value of $Ga = 155$ the trajectory of a sedimenting particle undergoes an instability and becomes oblique (see Sect. 4.4). They claim that this enhanced span-wise mobility of the particles compared to that of vertically sedimenting particles facilitates the occurrence of clusters. The idea is thus that many particles moving on random oriented oblique paths have better chances the encounter the wakes of other particles than particles falling simply vertically. The end of the story is here again the wake attraction effect which make the particles catch up each other.

The existence of these clusters has important effects on the dynamics of the particles and also on that of the fluid. For instance, the particle motion is more agitated in the clustering case. The standard deviation of the particle velocity is higher which is a consequence of the wake interactions and creation of turbulence within the clusters.

We will now turn to the interaction of many particles with a turbulent flow.

5.3 Turbulence Modulation

We have seen in Sect. 4.2 that an individual particle increases the energy dissipation rate close to its surface while it calms turbulence in its wake. Imagine now, that we seed our turbulent flow with many finite-size particles. How do the particles change the turbulent flow? Do they increase or decrease the kinetic energy dissipation rate?

Cate et al. (2004) studied these questions by seeding a forced turbulent flow with a few thousand finite-size particles that were slightly heavier than the fluid. The volume fraction (the volume occupied by the particles compared to the total volume of the fluid) varied from 2 to 10%. They found that the spatial distribution of energy

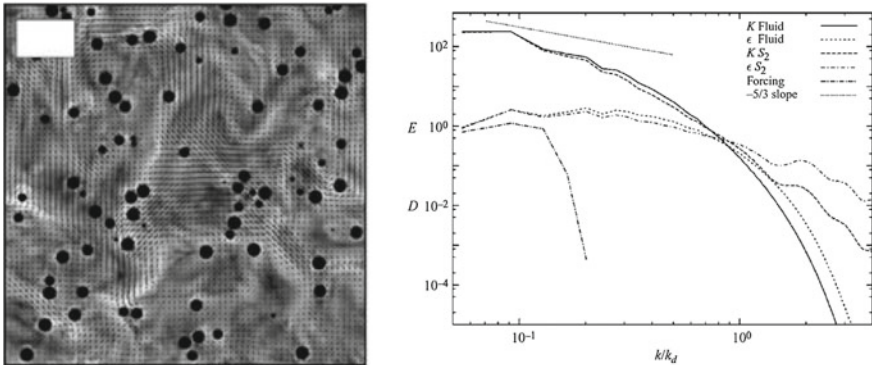


Fig. 18 *Left* Instantaneous local energy dissipation. *Right* Spectrum of energy dissipation (Cate et al. 2004)

dissipation is significantly influenced by the particles (see Fig. 18 (left)). Beside their boundary layer they also create inter-particle structures of increased dissipation. This could be a real collective effect in the sense that it is might be different from a simple superposition of the imprints of individual particles.

Changes in the energy dissipation are also clearly reflected in the global dissipation spectrum (see Fig. 18 (right)). Compared to a single phase flow (without particles) the dissipation rate is reduced at scales larger than the particle diameter while it is increased at smaller scales. Particles thus reduce shear at large scales while they increase it at small scales. The authors find the analog for the scale dependent kinetic energy. Particles transfer energy from large scales to scales of the order of the particle diameter or smaller.

There is another interesting result in the work of Cate et al. (2004). They computed the characteristic collision time scale of binary collisions and compared it to a prediction for small particles in a turbulent flow. Their data agrees with the prediction for small volume fraction of 2% and start to deviate at 5%. In fact, their collision times are longer than what one expects which in turn means that collective effects might reduce the probability of collisions.

Lucci et al. (2010) studied a case similar to the one discussed before. The main difference is that their flow is not forced so that they investigate the influence of many finite-size particles on decaying turbulence. They also find that the particles increase the energy dissipation rate and that they transfer energy from large to smaller scales. Furthermore, they find that the two-way coupling rate $\Psi_p(t) = \langle u_i f_i \rangle$ where $\mathbf{f}$ is the force exerted by the particle on the fluid is always positive. In decaying turbulence finite-size particles act as a source of kinetic energy while $\Psi_p(t)$ can be positive or negative for small particles.

Yeo et al. (2010) compared the dynamics of many finite-size particles in forced turbulence for particles that were heavier or lighter than the fluid. They found that heavy particles are expelled from vortex filaments while light particles are entrapped in these structures, an observation also made for point-particles. Concerning the

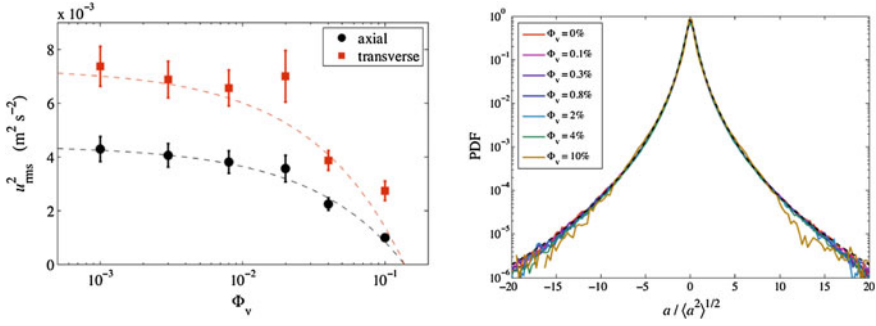


Fig. 19 *Left* Variance of the turbulent velocity fluctuations u_{rms}^2 as a function of the particle volume fraction; The *dashed lines* are $u_{rms}^2(\Phi_v) = u_{rms}^2(0)(13.75 \Phi_v)^{-2/3}$. *Right* Normalized probability density functions of acceleration components for various values of the volume fraction Φ_v as labeled (Cisse et al. 2015)

energy dissipation spectrum they confirm the previously discussed results. Interestingly, they found that the modulation of turbulence only weakly depends on the type of particles (light or heavy) but is dominated by the finite-size contributions to the flow disturbances.

So far, there are very few experimental investigations on turbulence modulation by finite-size particles. The reason are mostly technical difficulties. In a water experiment Bellani et al. (2012) studied a turbulent suspension of hydro-gel particles by means of a particle-image velocimetry technique. Hydro-gel is nearly neutrally buoyant and has a refraction index close to that of water. They found that for a volume fraction of $1.3 \cdot 10^{-3}$ the turbulent kinetic energy is reduced by 14% compared to an unladen flow.

Recently, Cisse et al. (2015) optically tracked the movement of tracer-like particles transported by a von Kármán flow. For their experiments they seeded water with a varying number of finite-size particles. For the latter they choose super-absorbent polymer spheres whose optical index and mass density match those of water. From the tracer dynamics they deduced that the big particles globally attenuate the fluid turbulence. When increasing the particle occupied volume Φ_v the turbulent velocity fluctuations reduces as $\Phi_v^{2/3}$ (see Fig. 19 (left)). It is not surprising that the level of turbulence decreases because this has already been observed for an individual particle (see Sect. 4.2). What is surprising is the exponent $2/3$. The authors interpret the latter by the possibility that not all added particles (their total volume) contribute to the turbulence reduction but only a fraction as if particles agglomerate on a surface rather than distribute in the full volume of the flow. Unfortunately, the authors did not track the large particles to conclude on this hypothesis. The evoked clustering is a possible collective effect and needs further investigation especially because Fiabane et al. (2012) have found that neutrally buoyant finite-size particles do not cluster in homogeneous isotropic flows even if their Stokes number is large. But neutrally buoyant particle clustering can occur in non-homogeneous flows: Machicoane et al.

(2014) found clear evidence of preferential concentrations in a turbulent von Kármán flow. Particles tend to stay in the vicinity of the rotating impellers and avoid the central region of the device.

Not all statistics of the flow are affected by the presence of the finite-size particles. Following Cisse et al. (2015), it seems that higher-order small-scale quantities are insensitive to the addition of big particles. The normalized acceleration probability density functions fall on top of each other irrespective of the volume fraction Φ_V (see Fig. 19 (right)).

6 Concluding Remarks

Let us summarize the most important findings that distinguish finite-size from point particles.

We first begin with results for individual particles:

- *Point particles* have (by definition) a negligible Reynolds number and are much smaller diameter than any characteristic scale of the flow. Their surrounding flow profile (Stokes flow) and drag force (Stokes drag) can be computed analytically.
- *Finite-size particles* are surrounded by a more complicated boundary layer and wake structure. In general, no analytic expression is known for the force on such a particle.
 - Their numerical treatment is demanding and various different strategies are proposed to integrate the Navier–Stokes equations together with the no-slip boundary condition at the particles surface.
 - The very definition of a slip velocity with respect to the fluid and consequential particle Reynolds number is difficult.
 - The energy dissipation rate is increased very close the particle while it can be reduced in the wake.
 - Large and small scale turbulent fluctuations lead to an increase of the drag force on the particle.
 - Particles with even steady wakes can sediment under gravity on non-vertical paths.

Let us now list collective effects arising from the mentioned finite-size effects:

- A particle entrains fluid in its wake that can reduce the drag force on a trailing particle. This is called the wake attraction effect.
- Particles separated in cross-stream direction are repulsed from each other at very small distances and are attracted at larger distances.
- The wake attraction effect leads to the formation of particle clusters.
- Clusters sediment on average at higher speeds than individual particles.
- Finite-size particles transfer turbulent kinetic energy from scales of the order of the particle diameter to smaller scales.
- Collective hydrodynamic interactions might reduce the collision rate.

In summary one can conclude that a better understanding of finite-size particle suspension will need much more analytic, numerical and experimental efforts.

Finally, we should mention that the Reynolds number of the presented works are quite limited and will not allow for a clear extension of the inertial range of scales. The results will probably include finite-Reynolds number effects and might thus change for higher Reynolds numbers. This kind of question has also to be answered in the future.

References

- P. Bagchi and S. Balachandar. Effect of turbulence on the drag and lift of a particle. *Phys. Fluids*, 15(11):3496, 2003. doi:[10.1063/1.1616031](https://doi.org/10.1063/1.1616031).
- G. Bellani and E.A. Variano. Slip velocity of large neutrally buoyant particles in turbulent flows. *New J. Phys.*, 14:125009, 2012.
- G. Bellani, M.L. Byron, A.G. Collignon, C.R. Meyer, and E.A. Variano. Shape effects on turbulent modulation by large nearly neutrally buoyant particles. *J. Fluid Mech.*, 712:41–60, 2012.
- R. Bitane, H. Homann, and J. Bec. Geometry and violent events in turbulent pair dispersion. *Journal of Turbulence*, 14:23–45, 2013.
- T. Burton and J. Eaton. Fully resolved simulations of particle-turbulence interaction. *J. Fluid Mech.*, 545:67–111, 2005a.
- T. Burton and J. K. Eaton. Fully resolved simulations of particle-turbulence interaction. *J. Fluid Mech.*, 545:67, 2005b.
- M. Cisse, H. Homann, and J. Bec. Slipping motion of large neutrally buoyant particles in turbulence. *Journal of Fluid Mechanics*, 735:R1, 2013.
- M. Cisse, E.-W. Saw, M. Gibert, E. Bodenschatz, and J. Bec. Turbulence attenuation by large neutrally buoyant particles. *Physics of Fluids*, 27:061702, 2015.
- A. Daitche. On the role of the history force for inertial particles in turbulence. [arXiv:1501.04770](https://arxiv.org/abs/1501.04770), 2015.
- E. A Fadlun, R. Verzicco, P. Orlandi, and J. Mohd-Yusof. Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations. *J. Comp. Phys.*, 60:35–60, 2000.
- L. Fiabane, R. Zimmermann, R. Volk, J.-F. Pinton, and M. Bourgoïn. Clustering of finite-size particles in turbulence. *Phys. Rev. E*, 86:035301, 2012.
- R. Gatignol. The Faxén formulae for a rigid sphere in an unsteady non-uniform Stokes flow. *J. Méc. Théor. Appl.*, 1:143–160, 1983.
- H. Homann, J. Bec, and R. Grauer. Effect of turbulent fluctuations on the drag and lift forces on a towed sphere and its boundary layer. *J. Fluid Mech.*, 721:155–179, 2013.
- T. Kajishima and S. Takiguchi. Interaction between particle clusters and particle-induced turbulence. *International Journal of Heat and Fluid Flow*, 23:639–646, 2002.
- A. Kidanemariam, C. Chan-Braun, T. Doychev, and M. Uhlmann. Direct numerical simulation of horizontal open channel flow with finite-size, heavy particles at low solid volume fraction. *New J. Phys.*, 15:025031, 2013.
- J. Kim and S. Balachandar. Mean and fluctuating components of drag and lift forces on an isolated finite-sized particle in turbulence. *Theoretical and Computational Fluid Dynamics*, 26:185–204, 2012.
- I. Kim, S. Elghobashi, and W. A. Sirignano. Three-dimensional flow over two spheres placed side by side. *Journal of Fluid Mechanics*, 246:465, 2006.
- F. Lucci, A. Ferrante, and S. Elghobashi. Modulation of isotropic turbulence by particles of Taylor length-scale size. *J. Fluid Mech.*, 650:55, 2010.

- N. Machicoane, R. Zimmermann, L. Fiabane, M. Bourgoïn, J.-F. Pinton, and R. Volk. Large sphere motion in a nonhomogeneous turbulent flow. *New Journ. Phys.*, 16:013053, 2014.
- M. R. Maxey and B. K. Patel. Localized force representations for particles sedimenting in stokes flow. *International Journal of Multiphase Flow*, 27:1603–1626, 2001.
- M. R. Maxey and J. Riley. Equation of motion for a small rigid sphere in a nonuniform flow. *Phys. Fluids*, 26(4):883, 1983.
- A. Naso and A. Prosperetti. The interaction between a solid particle and a turbulent flow. *New Journ. Phys.*, 12(3):033040, 2010. ISSN 1367-2630.
- A. Prosperetti and H.N. Oguz. Physalis: A New $O(N)$ Method for the Numerical Simulation of Disperse Systems: Potential Flow of Spheres. *J. Comp. Phys.*, 167:196–216, 2001.
- A. Prosperetti and G. Tryggvason. *Computational methods for multiphase flow*. Cambridge University Press, 2007.
- L. Schiller and A. Naumann. über die grundlegenden berechnungen bei der schwerkraftauflbereitung. *Verein Deutscher Ingenieure*, 77:318, 1933.
- A. Ten Cate, J. Derksen, L. M. Portela, and H. E. Van Den Akker. Fully resolved simulations of colliding monodisperse spheres in forced isotropic turbulence. *Journal of Fluid Mechanics*, 519:233–271, 2004.
- M. Uhlmann. An immersed boundary method with direct forcing for the simulation of particulate flows. *J. Comp. Phys.*, 209(2):448–476, 2005.
- M. Uhlmann and T. Doychev. Sedimentation of a dilute suspension of rigid spheres at intermediate galileo numbers: the effect of clustering upon the particle motion. *Journal of Fluid Mechanics*, 752:310–348, 2014.
- M. Uhlmann and J. Dušek. The motion of a single heavy sphere in ambient fluid: A benchmark for interface-resolved particulate flow simulations with significant relative velocities. *International Journal of Multiphase Flow*, 59:221–243, 2014.
- K. Yeo, S. Dong, E. Climent, and M.R. Maxey. Modulation of homogeneous turbulence seeded with finite size bubbles or particles. *International Journal of Multiphase Flow*, 36:221–233, 2010.
- C. Zhu, S. C. Liang, and L. S. Fan. Particle wake effects on the drag force of an interactive particle. *International journal of multiphase flow*, 20(1), 1994.

Collective Dynamics of Particles

From Viscous to Turbulent Flows

Marchioli, C. (Ed.)

2017, VII, 128 p. 67 illus., Hardcover

ISBN: 978-3-319-51224-2