13th International Symposium on Turbulence and Shear Flow Phenomena (TSFP13)
Montreal, Canada, June 25–28, 2024

DIRECT NUMERICAL STUDY OF TURBULENT PIPE FLOW LADEN WITH THERMAL PARTICLES USING A HYBRID LATTICE BOLTZMANN METHOD

Farshad Gharibi
Laboratory of Fluid Dynamics and Technical Flows,
University of Magdeburg "Otto von Guericke"
D-39106 Magdeburg, Germany.
Farshad.Gharibi@ovgu.de

Amir E. Fard
School of Engineering,
Newcastle University
Newcastle NE1 7RU, UK.
amir.fard@newcastle.ac.uk

Dominique Thévenin
Laboratory of Fluid Dynamics and Technical Flows,
University of Magdeburg "Otto von Guericke"
D-39106 Magdeburg, Germany.
thevenin@ovgu.de

ABSTRACT

This study delves into the turbulent modulation and dynamics of spherical particles in isothermal and non-isothermal pipe flows at a frictional Reynolds number of $\mathrm{Re}_\tau = 180$. The investigation is conducted through three-dimensional direct numerical simulation (DNS), employing a hybrid approach that combines the lattice Boltzmann, immersed boundary, and finite difference methods. This study focuses on a particulate flow with a solid volume fraction of 0.03, examining the relation between turbulence modulation and particle size, as well as the influence of heat generation within particles on the dynamics of particle motion and flow. The analysis of the probability density functions (PDFs) of particle positions sheds light on the spatial distribution of particles within the turbulent flow. The results unveil that the presence of particles alters the velocity profile of turbulent pipe flows, notably reducing the peak of streamwise velocity fluctuations. Furthermore, the heat generation of particles significantly alters the distribution of particles within the pipe.

1 Introduction

Turbulent flows laden with particles are relevant in a variety of natural and industrial processes, e.g. fluidized beds. Studies show that the addition of particles to turbulent flows can notably change turbulence characteristics. However, changes in particle properties and geometry of the process can affect in a very different manner the turbulent characteristics and heat flux. Numerous research groups have conducted experimental studies on non-isothermal particle-laden flows. Nevertheless, with the advancement of supercomputing capabilities in recent decades, the field of direct numerical simulations has also been expanding in parallel. While point-particle simulations have been extensively explored, contributing to our understanding of such two-phase systems, there has been much less focus on the numerical simulation of interface-resolved particles within the context of turbulent flow with heat transfer. This limited attention is primarily attributed to the additional computational cost and time required to resolve the intricacies of particle interfaces.

Zonta *et al.* (2008) showed that small and large micrometer-size particles have different effects on the heat transfer flux in a channel flow. Kuerten *et al.* (2011) performed two-way coupled DNS of non-isothermal particle-laden flows. They observed a rise in the Nusselt number for heavy inertial particles resulting from reduced heat transfer through turbulent velocity fluctuations. Ardekani *et al.* (2018) found that, in turbulent pipe flows with high particle volume fractions, the average Nusselt number decreases over time, dropping below the single-phase values. Chang & Ge (2020) simulated heat transfer in a turbulent channel flow with various particle shapes. Their findings revealed that neutrally buoyant particles decreased the average Nusselt number but increased its local value near the walls. However, further investigations concerning other aspects of thermal flows with particles, such as the effects of particles on heat transfer and turbulence statistics, as well as the distribution of particles in such flows, are obviously necessary to clarify the underlying physical processes.

With respect to the flow field solver, most of the above-mentioned studies relied on classical Navier-Stokes-based solvers, which is obviously fully appropriate. Nevertheless, the locality of all calculations coming with the lattice Boltzmann method (LBM) and leading to an extremely efficient parallelization has made the LBM quite popular in recent years. However, the application of LBM to direct numerical simulations of particle-laden turbulent flows with heat transfer in situ is still quite new. Even though recent studies have shed first light on the matter, a comprehensive investigation of the heat transfer in finite-size particle suspensions is still lacking in the literature. In the present study, the influence of isothermal particles and heat generation at the surface of the particles on turbulent properties in a pipe flow will be investigated using a hybrid lattice Boltzmann/immersed boundary/finite difference model (LB/IB/FDM).

2 Numerical method: lattice Boltzmann solver

The main focus of this section is on introducing a hybrid method that effectively combines the strengths of LBM and classical CFD (computational fluid dynamics). The lattice Boltzmann method has been used to simulate the flow field in this study. The discretization of the Boltzmann equation leads to the stream-collision equation for discrete populations denoted by f_i:

$$f_i(\mathbf{x}+\mathbf{e}_i\delta t, t+\delta t) = f_i(\mathbf{x}, t) + \Omega_i + F_i^{ext}, \quad (1)$$

where $\mathbf{x}$ and t represent the spatial location and time, $\mathbf{e}_i$ is the discrete velocity vector determined according to the D3Q27 model in this study, and F_i^{ext} denotes the external forces. In Equation (1), Ω_i is the discrete collision operator. In this study, a modified Hermite central moments space with a multiple relaxation time collision operator is utilized that provides independent control over the bulk viscosity. Unlike the traditional Hermite polynomial space, this modified formulation allows for the independent relaxation of trace and trace-free contributions to the second-order moments (Hosseini *et al.* (2022)), and is written as:

$$\Omega_i = \mathcal{T}^{-1} S \mathcal{T}(f_i^{eq} - f_i) + E_i, \quad (2)$$

where S represents the diagonal tensor of relaxation rates, while $\mathcal{T}$ and $\mathcal{T}^{-1}$ refer to the transform tensor and its inverse, respectively. In Equation (2), f_i^{eq} is the equilibrium distribution function that is expressed as a finite-order expansion of the Maxwell–Boltzmann distribution using Hermite polynomials, which are orthogonal polynomials computed through Gauss-Hermite quadrature. In Equation (2), E_i is a term that accounts for variations in the diagonal elements of the equilibrium third-order moments. By incorporating this term, we are able to preserve the Galilean invariance of the dissipation rate of shear modes, which is vital for accurately capturing the behavior of fluid flow. This enhancement allows to accurately represent the complex dynamics of flows at the Navier-Stokes level, thus offering improved accuracy and reliability in our simulations. In the context of this new solver E_i is written following (Hosseini *et al.* (2022)):

$$E_i = \left(1 - \frac{\omega_b \omega_s}{\omega_b + \omega_s}\right) \frac{w_i}{2c_s^4} \nabla \mathcal{H}^{(2)} : \left(\rho u_\alpha \left(\frac{3p}{\rho} + u_\alpha^2\right) - \sum_i e_{i,\alpha}^3 f_i^{eq}\right), \quad (3)$$

where α is space coordinate, ω_s is shear relaxation frequency, and ω_b is the relaxation frequency related to bulk viscosity.

The energy equation for incompressible flows, considering variable properties and neglecting viscous heating, can be expressed as:

$$\frac{\partial(\rho C_p T)}{\partial t} + \nabla\cdot(\mathbf{u}\,\rho C_p T) = \nabla\cdot(k\nabla T) + Q, \quad (4)$$

where T is the temperature, C_p is the specific heat capacity, k is the thermal conductivity, and Q is the heat source term. To numerically solve the energy equation, a classical finite-difference (FD) technique is employed. In particular, the advection term is discretized using the third-order Weighted Essentially Non-Oscillatory (WENO) approach. This method ensures stability, even in regions with large gradients, by minimizing numerical oscillations. The diffusion terms in the energy balance equation are discretized using a fourth-order central finite difference approximation in space, and to update the corresponding fields in time, a first-order explicit Euler discretization is employed.

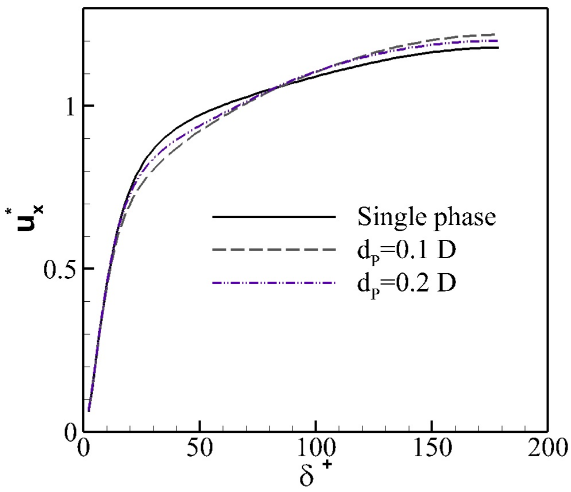

Figure 1. Mean streamwise velocity normalized by bulk flow velocity.

In the present investigation, we incorporate the Direct-Force Immersed Boundary Method to describe the interaction forces between the fluid and particles, including the heat transfer effect. This methodology builds upon the work initially introduced by Uhlmann (2005) and later extended to include thermal considerations by others, e.g. Eshghinejadfard & Thévenin (2016).

By coupling all these numerical techniques and modeling approaches, we aim to accurately capture the behavior of energy transfer and the intricacies of fluid-particle interactions.

3 Setup and results

The DNS is utilized to solve the incompressible flow within a periodic pipe driven by a constant mean pressure gradient and laden with particles. Our in-house code, ALBORZ, is used for this purpose. The length of the pipe is denoted as $2\pi D$, where D represents the pipe diameter. In this study, a turbulent pipe flow with a friction Reynolds number of $Re_\tau = 180$ is considered for the unladen flow case. For the particulate flow, the volume fraction of particles is set at 0.03, while the particle radius was varied at $d_p/D = 0.2$ and 0.1, and the particle-to-fluid density ratio was fixed at 1.2. To ensure a fair comparison, all simulations, whether laden or unladen, were subjected to the same energy input.

Figure 1 illustrates a comparison of mean streamwise normalized velocity profiles between isothermal particulate flow and single-phase flow. The findings suggest that smaller particles have a more pronounced effect on altering the shape of the velocity profile. Figure 2 exhibits rms velocity fluctuations for both unladen and isothermal laden flows. It is evident from this figure that the inclusion of particles in the flow significantly diminishes the peak of streamwise velocity fluctuations. Also, the Q-criterion visualization, showcasing the vortical structures within isothermal particle-seeded pipe flow, is depicted in Figure 3.

To investigate the influence of particles with heat generation in turbulent pipe flow, a concentration of 0.03 with particle radius (d_p) of 0.2 times the pipe diameter (D) is considered. In this part, the gravity and Boussinesq approximation are incorporated into the simulation. The particles exhibit constant heat

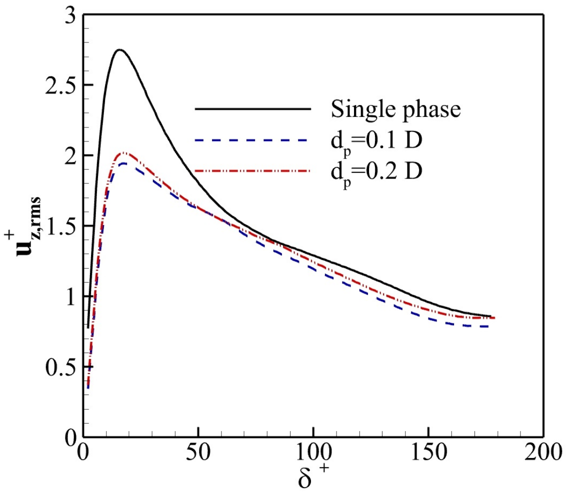

Figure 2. RMS streamwise velocity fluctuations.

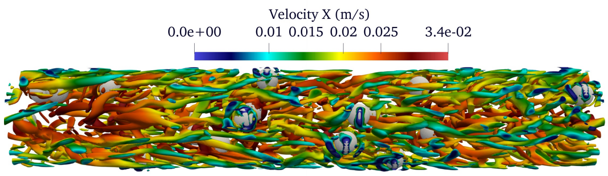

Figure 3. Iso-surface of Q-criterion colored by streamwise velocity for particle-laden flow with $d_p = 0.2D$.

generation on their surfaces. The dimensionless particle's heat source which is defined by Equation (5) is set to $\overline{Q_p^g} = 1.5$.

$$\overline{Q_p^g} = \frac{Q_p^g\, d_p}{\rho_f\, C_{pf}\, U_c\, \Delta T}\,; \tag{5}$$

where Q_p^g is the particle's heat generation, and U_c is the characteristic velocity defined as:

$$U_c = \sqrt{\frac{4 D_p g(\rho_p - \rho_f)}{3\rho_f}}. \tag{6}$$

Here, g represents the gravitational acceleration, and the subscripts f and p denoting fluid and particle, respectively. To analyze the behavior of particles within this flow, Figure 4 illustrates the PDF of particle distribution in relation to normalized distance from the wall (δ^+). It is evident from this figure that in both isothermal flow and thermal flow, particles tend to migrate towards the buffer area, with a peak observed at $\delta^+ = 37$. Moreover, particles with heat generation exhibit a more pronounced tendency to migrate towards this area.

Figure 5 portrays the PDF of azimuthal particle distribution within the pipe. It is observed that in the absence of heat transfer, particles exhibit a relatively uniform distribution across all directions. However, in thermal flows characterized by heat generation on particles, there is a pronounced concentration of particles towards the upper side ($\theta = 0, 360$), while the PDF minimum is located at the lower side ($\theta = 180$).

Figure 7 presents the cumulative center positions of particles over time across the pipe cross-section. This figure also

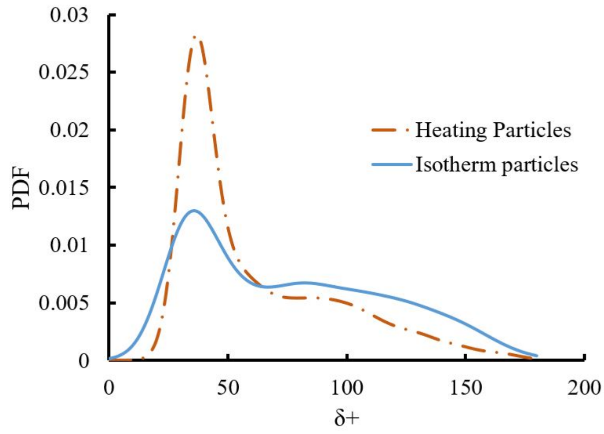

Figure 4. PDF of particle distribution in relation to normalized distance from the wall in turbulent pipe flow

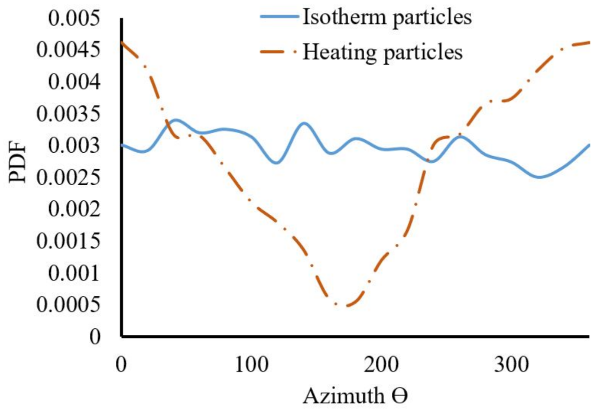

Figure 5. PDF of azimuthal particle distribution within the pipe in turbulent flow, comparing thermal and non-thermal conditions

reveals that while the distribution of isothermal particles remains relatively homogeneous, particles engaged in heat transfer tend to concentrate towards the upper portion of the horizontal pipe. This phenomenon can be attributed to the natural convection around particles, which exerts a drag force surpassing gravity force on particles, thereby transporting them towards the upper side of the pipe.

4 Conclusion

A hybrid approach, combining lattice Boltzmann, immersed boundary, and finite difference methods, is used for a comprehensive exploration of turbulence modulation influenced by particle size and heat generation of particles. DNS simulations reveal significant alterations in turbulent flow characteristics due to the presence of particles. Smaller particles are observed to have a more pronounced effect on velocity profile alterations. The particles substantially diminish peak streamwise velocity fluctuations. Furthermore, analysis of particle motion and distribution within the turbulent flow reveal distinct behaviors between isothermal particles and those with heat generation. Isothermal particles exhibit a more homogeneous distribution, while particles with heat generation tend to concentrate towards the upper portion of the pipe cross-section.