V International Conference on Particle-based Methods – Fundamentals and Applications
PARTICLES 2017
P. Wriggers, M. Bischoff, E. Oñate, D.R.J. Owen, & T. Zohdi (Eds)

THE IMPACT OF COUPLING AND PARTICLE VOLUME FRACTION ON FLUID-PARTICLE INTERACTIONS IN A TURBULENT CHANNEL FLOW

DAVID A. RUPP, DERRICK O. NJOBUENWU AND MICHAEL FAIRWEATHER

University of Leeds
School of Chemical and Process Engineering, University of Leeds, Leeds, LS2 9JT, UK
pmdr@leeds.ac.uk

Key words: Direct numerical simulation, Particles, Concentration, Lagrangian particle tracking, Stochastic, Spectral element method

Abstract. Direct numerical simulation, facilitated by a spectral element method, is used to predict a multi-phase fluid flow through a channel at a shear Reynolds number of 300. Following validation of single- and multi-phase flow results against other DNS predictions available in the literature, a channel flow is simulated utilising a Lagrangian particle tracker to model 300,000 particles with a diameter of 100 μm, having a density ratio equivalent to that of water to glass, and a particle volume fraction of approximately 0.01%. This flow is calculated using multiple levels of coupling between the particles and the flow; one-way, two-way and four-way. The mean streamwise velocity of the fluid and the particles, along with the shear and normal stresses, are compared for the different coupling methods, with the differences between them analysed and, although small, they are found to be consistent across the channel. A second set of runs is performed using in excess of 2 million particles in order to facilitate a tenfold increase in the particle volume fraction, to 0.1%, with the particles expected in this case to have a greater impact upon the properties of the fluid. The statistics of the fluid and particles in these simulations are then compared with those from the simulation with a lower concentration of particles in order to determine the magnitude of the effect the particles have on the fluid in this flow. The effects of the different couplings on the flow are much greater in this case due to the increased number of particles affecting the flow. Also, the presence of the particles is seen to increase the turbulence levels of the fluid, especially in the streamwise direction. The accuracy of the simulations clearly increases with the level of coupling. However, the speed of the simulations decreases. One way of achieving decreased run times, for both volume fraction cases, is to use a faster stochastic version of the particle tracking code for four-way coupling. This is tested, replacing the Lagrangian collision mechanics in the four-way coupled simulations with a probabilistically-determined mechanistic model. For the lower volume fraction, the normal stresses of the particles are exaggerated somewhat using the stochastic method. The simulation time is decreased compared to the Lagrangian approach, although the results presented suggest that the stochastic method requires further refinement.

1 INTRODUCTION

Accurately simulating particle-laden flows is of fundamental importance in many industries that, as part of their operations, are required to transport multi-phase flows, including waste. This includes industries in the chemical, nuclear, agriculture, pharmaceutical and minerals processing sectors. Without proper optimisation, these processes can be inefficient and the time between maintenance and/or replacement of equipment can be increased, with all of the associated costs.

Under complex turbulent conditions, it is laborious, if not impossible, to exactly predict how the dynamic interactions between the fluid and particles affect the flow as a whole using conventional mathematical methods, and experimentation often produces inconsistent results due to the difficulty in repeating initial conditions. Hence this work turns to computer modelling methods. Direct numerical simulation is a method which resolves the entire flow, over all length and time scales, and all of the turbulence effects within it and, when coupled to a particle tracking technique, offers the potential to provide basic understanding of particle-laden flows. However, it is also far more computationally expensive than the less inclusive, and accurate, large eddy simulation and Reynolds-averaged Navier-Stokes approaches.

By assessing the behaviour of particles within a multi-phase flow it is possible to set a useful benchmark which can be used to predict the behaviour of such flows. Depending on the scale of the change in the behaviour of the fluid when increasing its level of simulated coupling with the particles, it can be determined if the particle phase has a negligible effect or otherwise.

A number of previous works have simulated these flows in recent years [1-3]. Important findings include the fact that turbulence intensities grow more anisotropic as the particle mass loading is increased. Furthermore, qualitative descriptions have been offered [4] concerning the way particles affect the continuous phase at high concentrations. However, there is insufficient literature on the effect of various physical and flow properties of the fluid and particles in simulations of particle transport and interactions in turbulent flow. The present work, therefore, considers particle-laden channel flows using direct numerical simulation coupled to a Lagrangian particle tracking technique, with simulations performed using one-, two- and four-way coupling between the particles and the fluid flow. Two particle loadings are considered, equivalent to particle volume fractions of approximately 0.01% and 0.1%, to elucidate the effect of particle concentration on the continuous phase properties. Lastly, a stochastic method, replacing the Lagrangian collision mechanics in the four-way coupled simulations, is assessed in terms of its accuracy and potential to decrease simulation times.

2 METHODOLOGY

2.1 Fluid flow modelling

The fluid flow is simulated using a spectral element method, a hybrid of the finite-element method and the spectral method, which was developed by Patera [5]. The fluid domain is divided into elements with a resolution finer than 15 times the Kolmogorov scale, which Moser and Moin [6] have shown to be the minimum requirement to resolve the smallest turbulent structures.

$$\frac{\delta \boldsymbol{u}^*}{\delta t^*} + (\boldsymbol{u}^* \cdot \nabla)\boldsymbol{u}^* = -\nabla p^* + \frac{1}{Re}\nabla \cdot \tau^* + f_i \quad (1)$$

$$\nabla . \boldsymbol{u}^* = 0 \quad (2)$$

Equations (1) and (2) are the non-dimensional, incompressible Navier-Stokes and continuity equations solved, where $\boldsymbol{u}^*$is the fluid velocity vector, which has been non-dimensionalised using the bulk velocity U_B, and t^* is the non-dimensionalised time tU_B/δ, where δ is half the height of the channel. p^* is a non-dimensionalised pressure term equal to $p/\rho U_B^2$, where ρ is the density of the continuous phase. Re is the bulk Reynolds number given by $\delta U_B/\nu$, and τ^* is the non-dimensionalised deviatoric stress tensor given by $\tau^* = (\nabla \boldsymbol{u}^* + \nabla \boldsymbol{u}^{*\tau})$. f_i is an arbitrary forcing term that is only applied to flows with multiple phases.

The code used to simulate the fluid flow was Nek5000, an efficient DNS code developed by Fischer et al. [7].

2.2 Channel geometry

The fluid flow equations were solved for a turbulent flow in a channel with dimensions 12δ × 2δ × 6δ, with δ being the half-height of the channel, at a bulk Reynolds number of 4900, which translates to a shear Reynolds number of 300. The channel had periodic boundary conditions in the streamwise and spanwise directions, making its size arbitrary in those directions, with the channel wall having a significant effect on the flow. The computational domain was divided into 32 × 32 × 32 elements consisting of 8 × 8 × 8 nodes each, giving a total of ~16.8M nodes.

2.3 Multi-phase simulations

The first simulation considered below concerned validation of predictions of a single-phase channel flow against previous runs by Marchioli and Soldati [8]. Subsequent runs were for multi-phase flows with varying number of particles that were simulated using a highly versatile Lagrangian particle tracking code which synchronises with the Eulerian fluid flow model, time-step for time-step.

$$\frac{\partial x_P^*}{\partial t^*} = \boldsymbol{u}_P^* \tag{3}$$

$$\frac{\partial \boldsymbol{u}_P^*}{\partial t^*} = \underbrace{\frac{3C_D|\mathbf{u}_s^*|}{4d_p^*\rho_P^*}\mathbf{u}_s^*}_{Drag} + \underbrace{\mathbf{g}^*(1-\rho^*)}_{Gravity} + \underbrace{\frac{3}{4}\frac{C_L}{\rho_P^*}(\mathbf{u}_s^* \times \boldsymbol{\omega}_F^*)}_{Lift} + \underbrace{\frac{1}{2\rho_P^*}\frac{D'\mathbf{u}_F^*}{Dt^*}}_{Virtual\ Mass} + \underbrace{\frac{1}{\rho_P^*}\frac{D\mathbf{u}_F^*}{Dt^*}}_{Pressure\ Gradient} \tag{4}$$

The tracking code solves Equations (3) and (4) for $\partial x_P^*/\partial t^*$, the change in non-dimensional position over non-dimensional time, of each particle using the fourth order Runge-Kutta method [9], although in this work gravitational effects were ignored. d_p^*, ρ_P^* and $\rho^*$are the non-dimensional particle diameter, density and fluid density, $\boldsymbol{u}_s^*$, $\boldsymbol{u}_F^*$, $\boldsymbol{\omega}_F^*$ are the slip velocity, fluid velocity and vorticity, and C_D, C_L and $\mathbf{g}^*$are coefficients of drag, lift and gravity, respectively.

The first set of particle runs used 300k 10.2 μm particles with a high density compared with that of the fluid, corresponding to fly ash with a density of 1500 kg m^{-3} in air of density 1.3 kg m^{-3}. These particles had a Stokes number of 1 and predictions were validated against the results of Marchioli and Soldati [8]. All subsequent results were collected from runs containing 100 μm particles with a density corresponding to glass, 2500 kg m^{-3}, in water, 1000 kg m^{-3}. These particles had a Stokes number of 0.3125.

The particles used for the validation noted were one-way coupled, such that the fluid applied a force on the particles but not vice-versa. In four-way coupling, the fluid is affected by the particles and the particles are allowed to collide. The two-way coupling term in Equation (5) is added to the initial Navier-Stokes equations to account for the force the particles have on the fluid:

$$f_C = \frac{1}{V}\Sigma(P = 1)^N F_P \tag{5}$$

Here, V is the volume of a computational cell, N is the number of particles it contains, and F_P is the force exerted on any particle P. In order to gauge the effect of neglecting higher levels of coupling, one-way coupled and four-way coupled 100 μm glass particles were run at volume fractions of 0.01% and 0.1%.

Inter-particle collisions were simulated by checking the radius of a particle and comparing it to each potential collision partner in the region, extrapolating backwards to find the time of collision and changing the particle velocity and position accordingly. In order to limit computer run times, the flow was divided into small segments and only those particles in the same segment of the flow were considered for collision. Despite this, the computational expense of this process rises drastically with the number of particles.

2.4 Stochastic method

Another way of decreasing the computational expense present in monitoring particle-particle interactions is to use the stochastic method as described by Sommerfeld [10], and forgo monitoring the colliding of particles altogether, instead treating them as probability distributions for the purpose of collision. The probability of a collision according to this method is given by first calculating the velocity $u'_{fict,i}$ of a fictional collider using Equation (6), which accounts for the Stokes number *St* in the form of a correlation function *R(St)* which determines whether the fictional particle has a velocity closer to the real particle's velocity $u'_{real,i}$ or a random value calculated from the root mean squared velocity $\sigma_{p,i}$ and a Gaussian random number ξ.

$$u'_{fict,i} = R(St)u'_{real,i} + \sigma_{p,i}\sqrt{1 - R(St)^2\xi} \tag{6}$$

$$R(St) = \exp(-0.55 \times St^{0.4}) \tag{7}$$

$$P_{coll} = f_c\Delta t = \frac{\pi}{4}\left(2D_{\mathrm{p}}\right)^2\left|\overrightarrow{u_{\mathrm{p},i}} - \overrightarrow{u_{\mathrm{p},j}}\right|n_{\mathrm{p}}\Delta t \tag{8}$$

Kinetic theory is then used to calculate the probability of the collision from Equation (8) based on the instantaneous relative velocity $|\overrightarrow{u_{p,i}} - \overrightarrow{u_{p,j}}|$ between the fictional and real particles, the number density n_{p} and the particle diameter D_{p}.

3 RESULTS AND DISCUSSION

3.1 Validation

Figure 1 shows the results of the single-phase validation runs against the predictions obtained by Marcholi and Soldati [8], and Morinishi and Tamano [11]. Few runs have been performed at a shear Reynolds of 300, and Morinishi and Tamano [11] only gave results for y^* values close to the wall. Nevertheless, there is good agreement between the present results and those of these previous works. There is a slight discrepancy between the height of the peak of the stream-wise fluid normal stress, but this is at an acceptable level, and is most likely due to minor differences in the numerical methods employed.

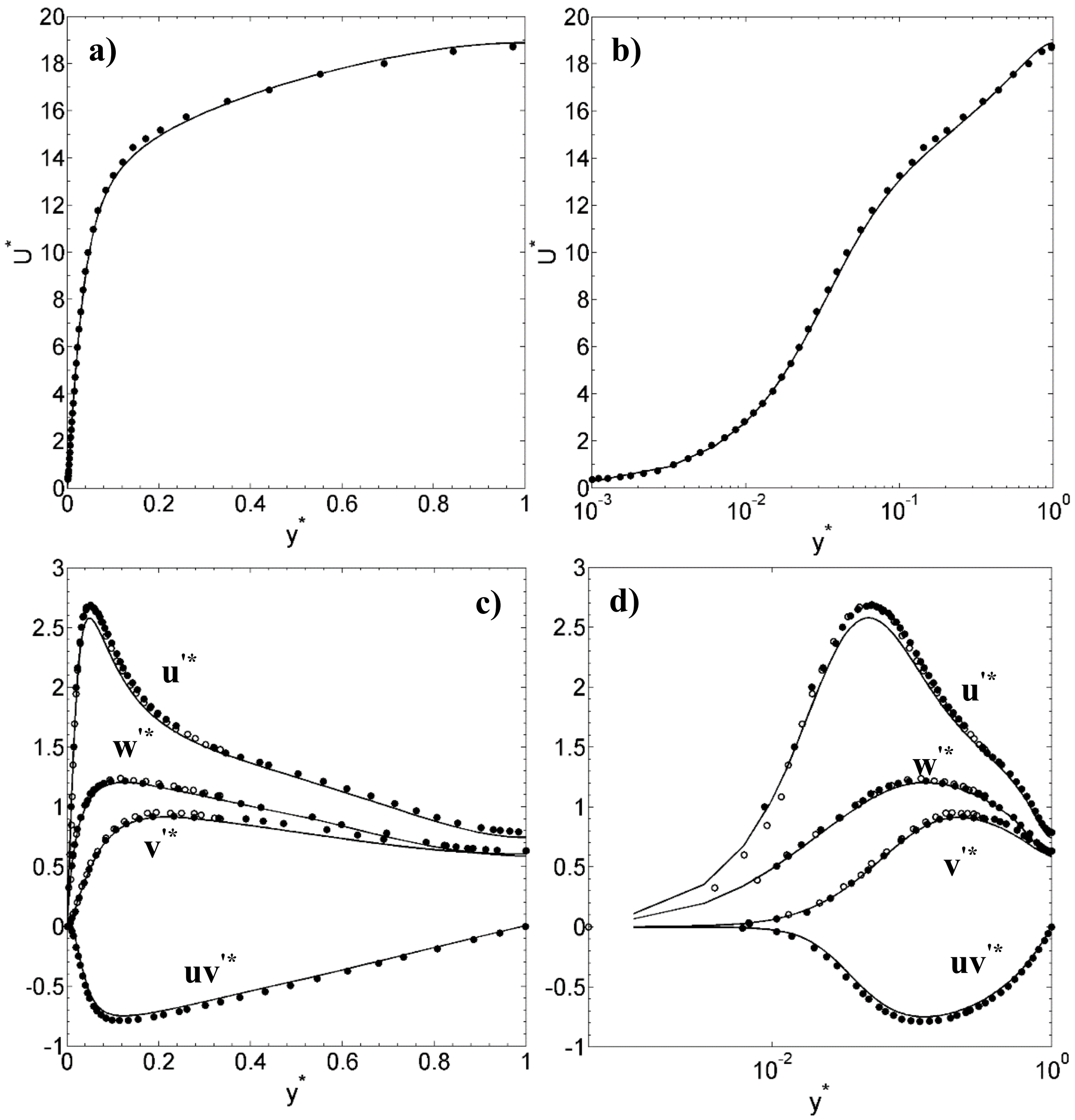

Figure 1: Validation of the mean streamwise fluid velocity U^* for a) linear scale and b) log scale, and the normal and shear stresses, u'^*, v'^*, w'^* and uv'^*, for c) linear scale and d) log scale of a single-phase flow at 300 shear Reynolds number. Present results as line, Marchioli and Soldati (2007) and Morinishi and Tamano (2005) as black and white points, respectively.

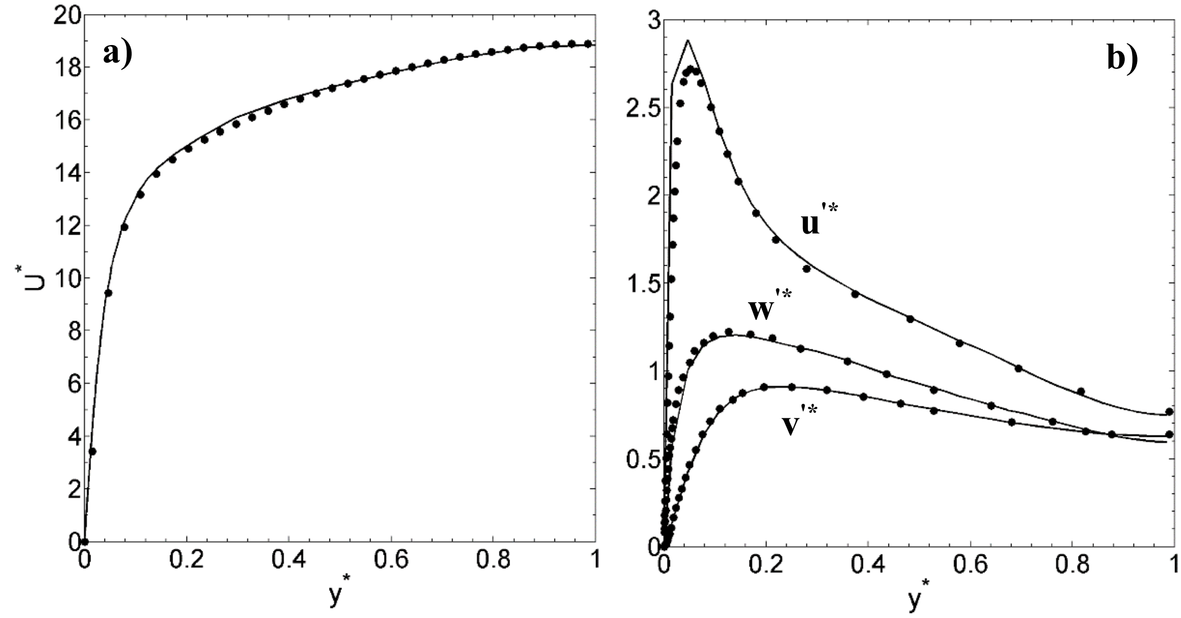

Figure 2: Validation of a) the mean streamwise particle velocity U^* and b) the normal stresses u'^*, v'^* and w'^* of a one-way coupled multiphase flow at 300 shear Reynolds numer. Present results as line, Marchioli and Soldati (2007) as points.

Figure 2 gives the results of a multi-phase validation run, plotting the mean streamwise velocity and the normal stresses for the particles. Since the flow was one-way coupled the fluid phase for this run is as in Figure 1. The particles again show mean streamwise velocity and normal stress values which are a good match to the predictions of Marchioli and Soldati [8].

3.2 Low concentration comparisons

The glass spheres were then run in a water flow at a volume fraction of 0.01% to compare the effects of one-way and four-way coupling between the fluid and particle phases at a Reynolds number of 300. The results are given in Figure 3. At first glance the difference in the fluid flow seems negligible, with the mean streamwise velocity unchanged between the two sets of results. There is a not imperceptible increase in the fluid and particle velocities towards the centre of the flow, however, and between $\pm 0.2\delta$ and $\pm 0.6\delta$, it is clear that the four-way coupled fluid flow has a slightly greater streamwise velocity fluctuation than for the one-way case.

In the region between $\pm 0.7\delta$ and $\pm\delta$, towards the centre of the flow, the streamwise velocity fluctuation of the four-way coupled run dips below the one-way case, but the spanwise and wall normal fluctuations increase slightly. In the region between $\pm 0.2\delta$ the peak shows a similar trend and this would appear to accord with energy conservation laws, namely, the kinetic energy from the streamwise direction is transferred to or from the other directions. The particle velocities and stresses show markedly similar responses .

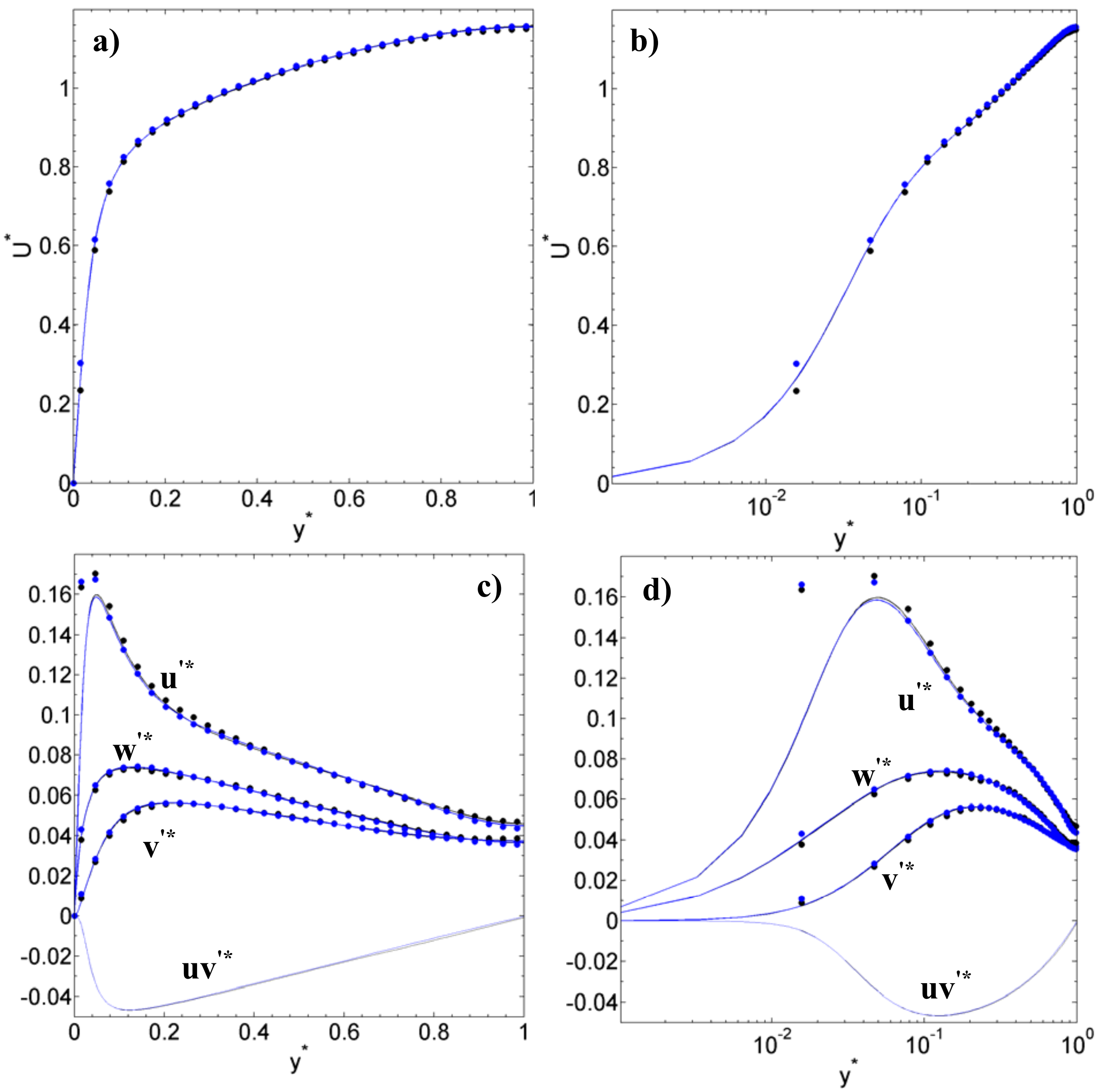

Figure 3: Mean streamwise velocities U^* for a) linear scale and b) log scale, and the normal and shear stresses u'^*, v'^*, w'^* and uv'^*, for c) linear scale and d) log scale of a multiphase flow with 100μm particles at a volume fraction of 0.01% and 300 shear Reynolds number. Fluid and particles as solid line and points, respectively, with one-way coupled in black and four-way coupled in blue.

3.3 High concentration comparisons

The particles at a volume fraction of 0.1% show a magnified difference between the effects of one-way and four-way coupling (Figure 4). The mean streamwise velocity and all of the normal stresses are greater in the latter case. The turbulence level in the fluid flow has thus seen a noticeable increase. These effects are more pronounced away from the centre of the channel, as for the low concentration results.

Although increasing the number of particles does change the behaviour of the fluid, and increases the impact of the particles on the flow, the particles themselves show a strong tendency to follow the flow that does not change with concentration, despite the fact that they are affecting the medium in which they are carried. The low Stokes number of the particles makes this unsurprising.

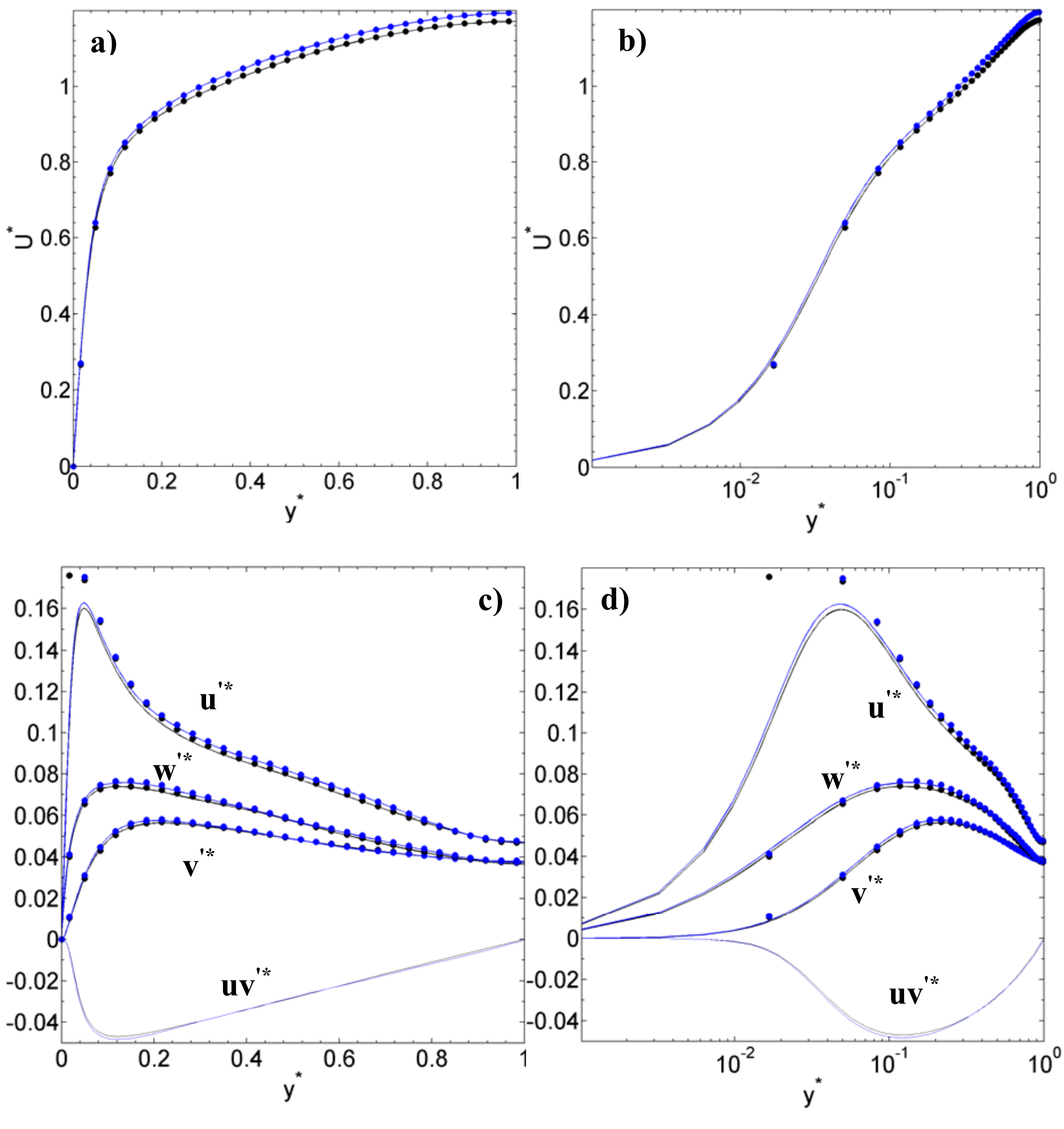

Figure 4: Mean streamwise velocities U^* for a) linear scale and b) log scale, and the normal and shear stresses u'^*, v'^*, w'^* and uv'^*, for c) linear scale and d) log scale of a multiphase flow with 100µm particles at a volume fraction of 0.1% and 300 shear Reynolds number. Fluid and particles as solid line and points, respectively, with one-way coupled in black and four-way coupled in blue.

3.4 Stochastic model

Under the same conditions as the previous low concentration runs, the results of which are given in Figure 3, the stochastic method was used with the aim of providing a suitably accurate set of predictions without the relatively slow computational speed of the Lagrangian collision mechanics that the regular four-way coupled approach used. The mean streamwise velocity, as before, varied only slightly due to the change in methodology, as shown in Figure 5, but the particle normal stresses now differ considerably from the fluid normal stresses in a way that suggests a methodical overprediction of these values, when compared to the results of Figure 3, especially considering the fact that the divergence in predictions increases considerably towards the centre of the flow. For the low volume fraction, therefore, the normal stresses of the particles are exaggerated somewhat using the stochastic method, although this was found to be less the case for the higher volume fraction (not shown). In both cases the simulation time was decreased compared to the Lagrangian approach, although the results suggest that the stochastic method requires further refinement.

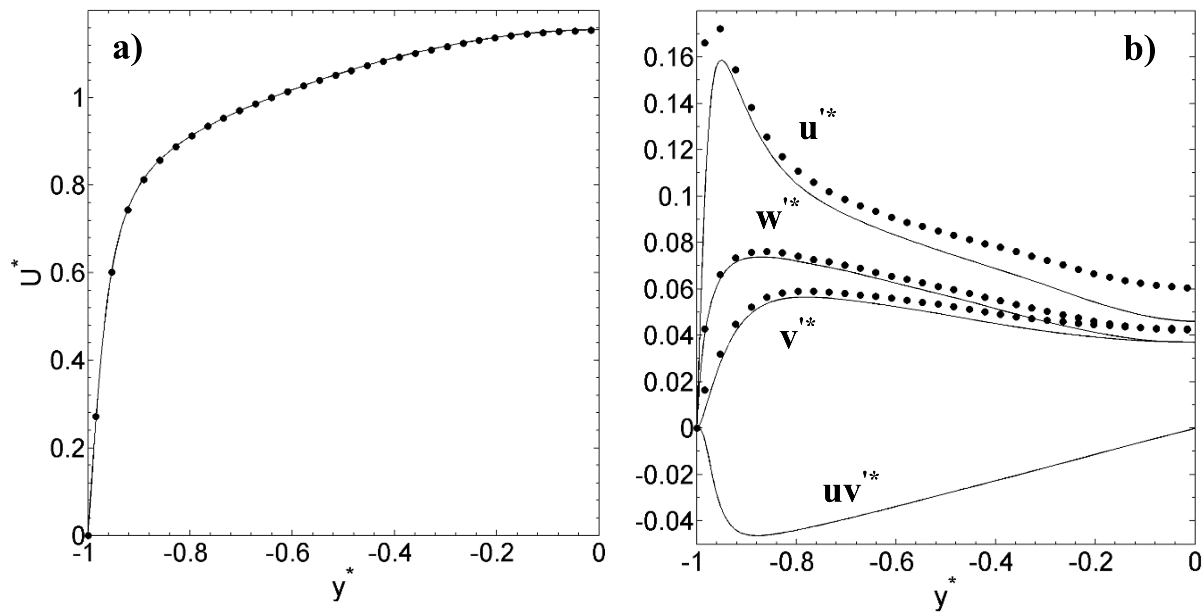

Figure 5: a) Mean streamwise velocity U^* and b) the normal and shear stresses u'^*, v'^*, w'^* and uv'^* for 100μm particles at a volume fraction of 0.01% and 300 shear Reynolds number using a stochastic collision metric. Fluid flow results as line, particles as points.

4 CONCLUSIONS

The study reported considered the effects of different particle-fluid coupling regimes on the turbulence statistics of a simulated multi-phase channel flow at a shear Reynolds number of 300. By performing direct numerical simulations, coupled to particle tracking, at different particle concentrations and differing levels of coupling, results have been established that demonstrate how particles affect the turbulent flow at this Reynolds number. Further work should consider investigating the impact of lower and higher volume fractions of particles than considered herein, as well as different sizes of particles, to establish precisely those flows in which it is necessary to use higher order coupling and for which flows such coupling is unnecessary. For higher concentrations of particles, the implementation of a stochastic collision metric, rather than a full four-way coupling approach, becomes advantageous in terms of model run times, although the results presented suggest that the stochastic method used requires further refinement prior to its use in computing such flows.

REFERENCES

[1] Squires, K.D. and Eaton, J.K. Particle response and turbulence modification in isotropic turbulence. *Phys. Fluids* (1990) **A2**:1191-1203.

[2] Sundaram, S. and Collins, L.R. A numerical study of the modulation of isotropic turbulence by suspended particles. *J. Fluid Mech.* (1999) **379**:105-143.

[3] Li, Y., McLaughlin, J.B., Kontomaris, K. and Portela, L. Numerical simulation of particle-laden turbulent channel flow. *Phys. Fluids* (2001) **13**:2957-2967.

[4] Vreman, B., Geurts, B.J., Deen, N., Kuipers, J.A. and Kuerten, J.G. Two-and four-way coupled Euler-Lagrangian large-eddy simulation of turbulent particle-laden channel flow. *Flow Turbul. Combust.* (2009) **82**:47-71.

[5] Patera, A.T. A spectral element method for fluid dynamics: Laminar flow in a channel expansion. *J. Comput. Phys.* (1984) **54**:468-488.

[6] Moser, R.D. and Moin, P. Direct numerical simulation of curved turbulent channel flow. NASA TM 85974 (1984).

[7] Fischer, P.F., Lottes J.W. and Kerkemeier S.G. Nek5000 Web Page. http://nek5000.mcs.anl.gov (2008).

[8] Marchioli, C. and Soldati, A. Reynolds number scaling of particle preferential concentration in turbulent channel flow. In: Palma J., Lopes A.S. (Eds) Advances in Turbulence XI, Springer Proceedings Physics, 117, Springer, Berlin (2007).

[9] Süli, E. and Mayers, D. *An introduction to numerical analysis*. Cambridge University Press, 2003.

[10] Sommerfeld, M. Validation of a stochastic Lagrangian modelling approach for inter-particle collisions in homogeneous isotropic turbulence. *Int. J. Multiph. Flow* (2001) **27**: 1829-1858.

[11] Morinishi, Y. and Tamano, S. Study on differences in turbulence statistics between compressible and incompressible low-Reynolds number turbulent channel flows using semi-local scaling. *JSME Int. J. Ser. B* (2005) **48**:743-749.