FLOW AROUND AN OBSTACLE BETWEEN PARALLEL WALLS

Wenchao Yu, Lionel Le Penven, Ivana Vinkovic & Marc Buffat
Laboratoire de Mécanique des Fluides et d'Acoustique, Université de Lyon, Université Claude Bernard Lyon 1, Ecole Centrale de Lyon, INSA de Lyon, CNRS UMR 5509, 36 Avenue Guy de Collongue, F-69134 Ecully, France

Summary In order to study the interaction between finite size particles and wall turbulence, we analyze here the effect of the walls on the flow around an obstacle. The flow is simulated by an efficient Fourier Chebyshev method. The obstacle is accounted for by an implicit penalty method. The case of a cylinder at different distances to the wall and the case of a sphere in the middle of the channel are computed. For both test cases, our results are in good agreement with others.

INTRODUCTION

The interaction between finite-size particles and a carrier flow is a fundamental hydrodynamic problem for many natural and industrial applications, such as the transport of sediment in river flows [1]. This interaction depends strongly on the ratio between the particle diameter D_p and the Kolmogorov length scale η. When D_p/η is very small, the particles can be treated as points [2]. But when D_p/η is nearly one, the finite-size effect must be taken into account [3]. Direct numerical simulation has been used to solve this problem [3,4,5]. However, several difficulties still exist. Accurate simulations are time consuming and can not treat a large number of particles [4]. Most methods based on finite differences [6] do not have enough grid points on the surface of the particles to accurately resolve the flow in its vicinity. In view of this discussion, the aim here is to develop an accurate and efficient method for simulating particle transport by wall turbulence. Therefore, as a first step, an efficient spectral code is coupled with a penalty method for simulating the effect of the walls on a fixed 2D and 3D obstacle.

METHODOLOGY

We consider an incompressible, Newtonian fluid moving between two fixed, parallel plates distant by $2h$. After scaling the velocities by a constant speed U_0 and the length by h, the Navier-Stokes equations are

$$\frac{\partial \vec{u}}{\partial t} = -\vec{u}\cdot\nabla\vec{u} - \nabla p + \frac{1}{\mathrm{Re}_0}\Delta\vec{u} + \vec{f}_{Fringe} + \vec{f}_{IBM} \qquad (1)$$

$$\nabla\cdot\vec{u} = 0 \qquad for\ \vec{x}\in\Omega\setminus\Omega_p \qquad (2)$$

where $\vec{u}(x,y,z,t)$ is the velocity vector, p the pressure divided by density, $\mathrm{Re}_0 = U_0 h/\nu$ the Reynolds number and ν the kinematic viscosity of the fluid. The studied domain is presented in figure 1. Here x, y and z are the streamwise, wall-normal and spanwise directions, respectively. Equations (1) and (2) are solved by an efficient Fourier Chebyshev spectral method [7]. The force $\vec{f}_{Fringe}$ is introduced in order to simulate flows that are not naturally periodic in the x direction, while keeping the benefits of Fourier expansions.

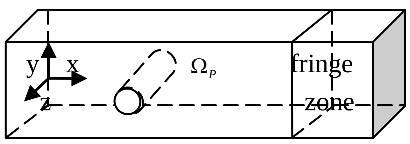

Figure 1 Computational domain

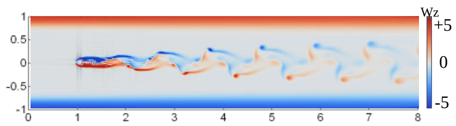

Figure 3 Vorticity contour for $\mathrm{Re}_p = 400$

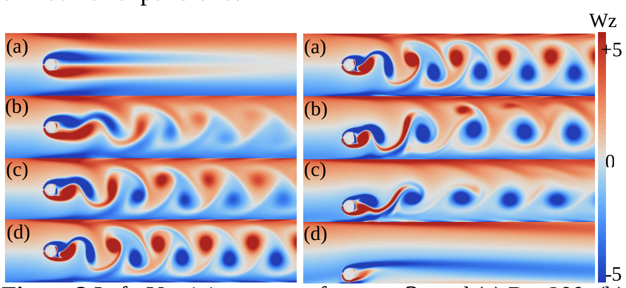

Figure 2 Left: Vorticity contour for $\gamma = 2$ and (a) Re=200; (b) Re=300; (c) Re=500; (d) Re=1000. Right: Vorticity contour for Re=1000 and (a) $\gamma = 2$; (b) $\gamma = 1.25$; (c) $\gamma = 0.75$; (d) $\gamma = 0.25$.

The boundary conditions on the obstacle are imposed by means of a Darcy term, $\vec{f}_{IBM} = -\vec{u}/\eta_p$ for $\vec{x}\in\Omega_p$ where η_p is the porosity of the obstacle. This force is introduced implicitly in the Navier-Stokes equations to avoid the constraint of stability $dt < \eta_p$.

2D CASE - CYLINDER

First, the case of a cylinder placed at different distances from one wall in a Poiseuille flow is studied. The results of our simulations are compared to those of Zovatto & Pedrizzetti [8] for validation. The channel contains a circular cylinder of diameter D_c, which position is defined by the gap Δ, the minimal distance from the cylinder surface to the nearest wall. The channel Reynolds number is $\mathrm{Re} = 2Uh/\nu$, where U is the steady average velocity inside the channel. The blockage ratio is given by $d = D_c/2h$, and the gap parameter by $\gamma = \Delta/D_c$. Details of the numerical simulations are given in table 1.

Re	Mesh	Domain	Porosity
1000	1280x1024x4	26hx2hx0.4h	0.00001

Table 1 Parameters of the simulations for the cylinder

Rep	Mesh	Domain	Porosity
400	1280x512x128	12.8hx2hx4h	0.00001

Table 2 Parameters of the simulations for the sphere

Figure 2 (left) shows the spanwise vorticity for $\gamma = 2$ with different Re. The transition to the unsteady regime happens between $\mathrm{Re} = 200$ and 300. This is in good agreement with experimental observations [9]. On the right of Figure 2, as the cylinder approaches one wall, the wake vorticity is significantly reduced. The transition to the unsteady regime is delayed. The forces acting on the cylinder are then calculated by integration of the wall pressure and wall shear stress on the cylinder surface. These forces can be computed from the penalty force $\vec{f}_{IBM}$ since,

$$\int_{\partial\Omega_p} \mathrm{T}\cdot d\vec{S} = \int_{\Omega_p} \nabla\cdot \mathrm{T} d^3x = \int_{\Omega_p} \vec{f}_{IBM} d^3x, \text{ where } \mathrm{T} = -pI_3 + \frac{1}{\mathrm{Re}_0}(\nabla\vec{u} + \nabla\vec{u}^T).$$

The drag coefficient $C_D = F_x / 0.5\rho U^2 D_c L_z$, where F_x is the steady or time-average (for periodic flow) longitudinal component of the total force and L_z the spanwise length of the cylinder, is given in Figure 4 for different γ and Re. Good agreement with the DNS of [8] is achieved except for $\gamma = 0.25$. In this case, the cylinder is close to the wall. The DNS of [8] has only 10 elements to resolve the flow between the lower edge of the cylinder and the wall, so that the results may be questioned.

3D CASE - SPHERE

The flow around a sphere is then computed. The results of our simulations are compared to the DNS of Homann et al. [10] for validation. The sphere is placed in the middle of the channel far from the boundary layers that develop at each wall. The Reynolds number is given here by $\mathrm{Re}_p = U_0 D_s / \nu$, where D_s is the diameter of the sphere. Details of the numerical simulations are given in table 2.

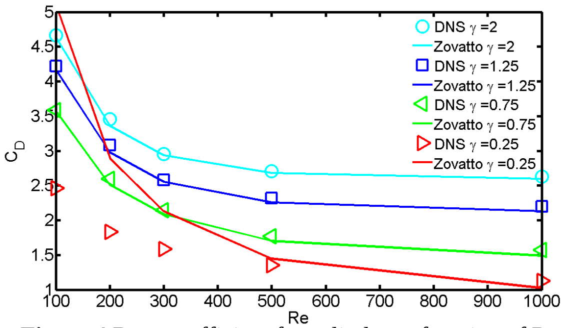

Figure 4 Drag coefficient for cylinder as function of Re

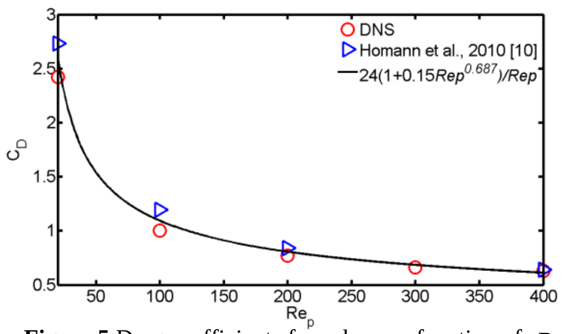

Figure 5 Drag coefficients for sphere as function of Re_p

Figure 3 illustrates the spanwise vorticity behind the sphere. The von-Karman vortex street is well observed. For the sphere, the drag coefficient is given by $C_{DS} = 8F_x / \pi D_s^2 \rho U_0^2$. Figure 5 shows the drag coefficient as function of Re_p. Our DNS results are in good agreement with the empirical formula and the DNS of Homann et al. [10].

CONCLUSION

In conclusion, there is good agreement between our DNS and the two test cases chosen. The penalty method used here is fully implicit and no stability constraint has been imposed on the time step. However, the grids used in both cases are extremely refined and the simulations are time consuming. Therefore, in future work a spectral fictitious domain method will be introduced to satisfy exactly the no-slip boundary condition on the surface of the obstacle [11].

References

[1] Kaftori D., Hestroni G. & Banerjee S.: Particle behavior in the turbulent boundary layer. I. Motion, deposition and entrainment. *Phys. Fluids* 7:1095–1106, 1995

[2] Lavezzo V., Soldati A., Gerashchenko S., Warhaft Z. & Collins L.R.: On the role of gravity and shear on inertial particle accelerations in near wall turbulence. *J. Fluid Mech.* 658:229-246, 2010.

[3] Homann H. & Bec J.: Finite-size effects in the dynamics of neutrally buoyant particles in turbulent flow. *J. Fluid Mech.* 651:81–91, 2010.

[4] Naso A. & Prosperetti A.: The interaction between a solid particle and a turbulent flow. *New J. of Physics* 12:33–40, 2010.

[5] Zeng L., Balachandar S., & Fischer P.: Wall-induced forces on a rigid sphere at finite Reynolds numbers. *J. Fluid Mech.* 536:1-25, 2005.

[6] Uhlmann M.: Interface-resolved direct numerical simulation of vertical particulate channel flow in the turbulent regime. *Phys. Fluids* 20:053305, 2008.

[7] Buffat M., Le Penven L. & Cadiou A.: An efficient spectral method based on an orthogonal decomposition of the velocity for transition analysis in wall bounded flow. *Computers & Fluids* 42:62-72, 2011.

[8] Zovatto L. & Pedrizzetti G.: Flow about a circular cylinder between parallel walls. *J. Fluid Mech.* 440:1–25, 2001.

[9] Chen J.-H., Pritchard W.G. & Tavener S. J.: Bifurcation for flow past a cylinder between parallel planes. *J. Fluid Mech.* 284:23-41, 1995.

[10] Homann H., Bec J., & Grauer R.: Immersed boundaries with Fourier-spectral method. *Physics.Flu-dyn*, 2010.

[11] Buffat M. & Le Penven L.: A spectral fictitious domain method with internal forcing for solving elliptic PDEs. *J. Comput. phys.*230:2433–2450, 2011.