A Numerical Study of Fluid-Particle Interaction with Slip Boundary Condition

Xing Zhang¹, Li Luo² and Xiaoping Wang¹,*

¹ Department of Mathematics, The Hong Kong University of Science and Technology, Hong Kong
² Shenzhen Institutes of Advanced Technology, Chinese Academy of Sciences, Shenzhen, 518055, P. R. China

Received 30 August 2017; Accepted (in revised version) 19 December 2017

Dedicated to Professor Xiaoqing Jin on the occasion of his 60th birthday

Abstract. In this paper, we present a numerical study of the effect of slip in the fluid-particle interaction. The motion of the particle is described by the Newton’s second law and the flows are simulated by solving the incompressible Navier-Stokes equations with the Navier slip boundary condition. Numerical schemes are designed using the extended finite element method (XFEM) combined with the temporary arbitrary Lagrangian-Eulerian (tALE) technique. In this method, both the fluid dynamics and the motion of particle are efficiently computed on a fixed Cartesian mesh. With the XFEM, the discontinuities at the particle boundary are naturally captured by the Heaviside-enriched finite element basis functions. With the tALE technique, field variables at the previous time level are mapped onto the computational mesh at the current time level, hence regeneration or deformation of meshes can be avoided. To study the effect of the slip, we simulate the rotation of an ellipsoidal particle in a simple shear flow and compare with the analytic results from the theory of Jeffery orbit.

AMS subject classifications: 65M10, 78A48

Key words: Fluid-particle interaction, direct numerical simulation, extended finite element method, temporary arbitrary Lagrangian-Eulerian, Jeffery orbit.

1. Introduction

Considerable attentions have been paid for both experimental and numerical investigations of fluid-particle interaction problems. Numerical studies of fluid-particle systems concentrate on the description of interface, field variable discontinuity, and hydrodynamic interaction between fluid and solid objects. In the past decade, two categories of numerical simulations have been developed, including the continuum approach and the direct numerical simulation (DNS) approach. In the continuum approach, solid particles and
fluids are viewed as interpenetrating mixtures with different viscosities that governed by conservation laws [21, 27, 29, 30]. Despite its efficiency and flexibility, the continuum approach suffers from false response from the inside viscous material which is used to mimic the rigid objects, causing inaccurate hydrodynamical effects. On the other hand, the DNS approach, where the fluid is described by Navier-Stokes equations and the motion of rigid-body is governed by the Newton’s second law, provides a fundamental understanding of the mechanism and more details of fluid-particle interactions.

A classical method in DNS for fluid-particle interaction problems is to use the boundary-fitted mesh by aligning the solid boundary with element edges [18–20]. In this approach, the governing equations are solved only in the fluid domain and the discontinuous characteristics are naturally captured on the boundaries of particles. To handle moving particles, this approach incorporates the ALE technique where mesh nodes near particles follow the motion of particles in a Lagrangian manner, while mesh nodes far away from particles are remained stationary in an Eulerian manner. The creation of new unstructured meshes is often needed for the region occupied by the fluid. In general, it is very time consuming to generate boundary-fitted meshes if complex geometries are involved.

An alternative approach is the fictitious domain method developed by Glowinski et al. [11–13]. The basic idea of this approach is to extend the problem in a geometrically complex domain to a simple and regular domain by generalizing the weak formulation of flows from the fluid domain to the fictitious domain that represents the particles. This approach enforces the constraint of rigid-body motion with a distributed Lagrange multiplier to introduce an additional body force to the interior of the region of particles. Field variables at the interface are required to be interpolated between the values of the physical fluid and the fictitious fluid.

Recently, a novel numerical method, the extended finite element method, or XFEM, was developed to capture arbitrary discontinuous at the interface on a fixed Cartesian mesh. The XFEM generalizes the standard Galerkin finite element method with additional degrees of freedom to handle discontinuities. The methodology was first developed by Moës et al. [25] to solve crack problems. It has been extended to several other problems including elastic problems with holes [26], fluid-structure interaction [10] and particulate flow problems [8]. Gerstenberger and Wall [10] utilized the XFEM for problems of fluid-structure interaction (FSI) using a Heaviside function to define the discontinuities. Choi et al. [8] presented an application of XFEM for the simulation of moving particles in a viscoelastic fluid, with a no-slip boundary condition imposed on the particle boundary through Lagrange multipliers. In their work, a temporary arbitrary Lagrangian-Eulerian (tALE) scheme was proposed to handle the motion of particles by mapping the time dependent variables along with the ALE meshes between two time levels.

In [30], Zhang et al. investigated the effects of boundary slip on the orientational motion of an anisotropic particle in a simple shear flow. A fluid-particle system is numerically solved using the fluid particle dynamics (FPD) method [27]. The dependence of the cross-coupling coefficient on the slip length at particle surface is measured, showing that the boundary slip can enhance the effective anisotropy of the particle and hence the cross coupling between the rotational torque and the shear stress.
In this paper, we generalize the approach in [8] to system with slip boundary condition and study the effect of slip in the fluid-particle interaction. In our model, the fluid is governed by the Navier-Stokes equations with Navier slip boundary condition and the motion of particle is governed by the Newton’s second law. To handle the movement of both the fluid and rigid body, we incorporate the XFEM with the tALE technique to the DNS approach such that the whole calculation is performed on a fixed Cartesian mesh. Specifically, the discontinuous characteristics at the particle boundary are captured by Heaviside-enriched finite element basis functions, and the boundary conditions at the interfaces are included in the weak form by using a penalty method. To cope with the time dependent domain caused by the moving particle, we map the field variables at the previous time level to the computational mesh at the current time level so that only a fixed mesh is needed. Combining the above techniques, we have an efficient algorithm to fully decouple the calculations of particle motion, mapping of variables, and fluid dynamics. We then simulate the rotation of an ellipsoidal particle in a simple shear flow and compare with the analytic results from the theory of Jeffery orbit and the results obtained in [30].

The paper is organized as follows. In Section 2, the governing equations for the motion of incompressible fluid and solid particle are described. In Section 3, we present the overall numerical algorithm, including the temporal and spacial discretization and the implementation of XFEM and tALE technique. In Section 4, we show numerical tests for simulating an elliptic particle interacting with a shear flow. The paper is concluded in Section 5.

2. Governing equations

2.1. The fluid-particle system

Let $\Omega$ be the entire computational domain, including the fluid and the particle, and $P(t)$ be the time-dependent particle domain. Boundaries are denoted by $\partial \Omega$ and $\partial P(t)$, as shown in Fig. 1. The motion of an incompressible fluid is given by the incompressible Navier-Stokes equations, in the dimensionless form, as follows,

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot \mathbf{\sigma} + \mathbf{g} \quad \text{in} \quad \Omega \setminus P(t), \quad (2.1a)$$

$$\nabla \cdot \mathbf{u} = 0 \quad \text{in} \quad \Omega \setminus P(t), \quad (2.1b)$$
where \( \mathbf{u} = (u^x, u^y) \) is the velocity vector, \( \mathbf{\sigma} := -p \mathbf{I} + \frac{1}{Re} \mathbf{e} \), \( p \) is the pressure, \( \mathbf{I} \) is the unit matrix, \( Re \) is the Reynolds number, \( \mathbf{e} := \nabla \mathbf{u} + (\nabla \mathbf{u})^T \), and \( \mathbf{g} \) is a driving external force. The motion of particle is determined by external forces that balanced with the net hydrodynamic force \( \mathbf{F} \) and torque \( \mathbf{T} \), which are defined by

\[
\mathbf{F} := - \int_{\partial \mathcal{P}(t)} \mathbf{\sigma} \cdot \mathbf{n} ds, \tag{2.2a}
\]

\[
\mathbf{T} := - \int_{\partial \mathcal{P}(t)} (\mathbf{\sigma} \cdot \mathbf{n}) \times (\mathbf{x} - \mathbf{x}_s) ds, \tag{2.2b}
\]

where \( \mathbf{n} \) is the unit outward normal vector on the solid boundary for fluid, and \( \mathbf{x}_s \) is the particle position (mass center). In our model, the motion of particle is governed by the Newton’s second law, i.e.

\[
M_s \frac{d\mathbf{U}_s}{dt} = \mathbf{F} + M_s \mathbf{g}, \tag{2.3a}
\]

\[
I_s \frac{d\mathbf{\omega}_s}{dt} = \mathbf{T}. \tag{2.3b}
\]

Here \( M_s, \mathbf{U}_s, I_s, \mathbf{\omega}_s \) are the mass, the translational velocity, the moment of inertia, and the angular velocity of the particle, respectively. The particle position \( \mathbf{x}_s \) and the angular orientation \( \Theta_s \) are obtained from the following kinematic equations

\[
\frac{d\mathbf{x}_s}{dt} = \mathbf{U}_s, \quad \mathbf{x}_s|_{t=0} = \mathbf{x}_0, \tag{2.4a}
\]

\[
\frac{d\Theta_s}{dt} = \mathbf{\omega}_s, \quad \Theta_s|_{t=0} = \Theta_0. \tag{2.4b}
\]

On the particle boundary, we apply an impermeability condition along the normal direction

\[
(\mathbf{u} - \mathbf{u}_s) \cdot \mathbf{n} = 0 \quad \text{on} \quad \partial \mathcal{P}(t), \tag{2.5}
\]

and a Navier slip boundary condition along the tangent direction

\[
\left( \mathbf{u} - \mathbf{u}_s \right) + \frac{\mathbf{L}_s}{Re} (\mathbf{e} \cdot \mathbf{n}) \cdot \mathbf{\tau} = 0 \quad \text{on} \quad \partial \mathcal{P}(t), \tag{2.6}
\]

where \( \mathbf{L}_s \) is the slip length, and \( \mathbf{u}_s = \mathbf{U}_s + \mathbf{\omega}_s \times (\mathbf{x} - \mathbf{x}_s) \) is the speed of the particle boundary.

The boundary conditions on \( \partial \Omega \) can be divided into two parts, \( \partial \Omega = \Gamma_D \cup \Gamma_N \), including a Dirichlet boundary condition on \( \Gamma_D \) (top and bottom) and a traction-free boundary condition on \( \Gamma_N \) (left and right).

\[
\mathbf{u} = \mathbf{u}_w \quad \text{on} \quad \Gamma_D, \tag{2.7a}
\]

\[
\mathbf{\sigma} \cdot \mathbf{n} = 0 \quad \text{on} \quad \Gamma_N. \tag{2.7b}
\]
where $\mathbf{u}_w$ is the wall speed on $\Gamma_D$.

To impose the impermeability condition (2.5), we consider a penalty method to replace it by

$$
\left( (\mathbf{u} - \mathbf{u}_w) + \varepsilon_p \mathbf{a} \cdot \mathbf{n} \right) \cdot \mathbf{n} = 0 \quad \text{on} \quad \partial P(t),
$$

where $\varepsilon_p$ is a small parameter, such that a Robin type boundary condition is prescribed on $\partial P(t)$ which can be directly introduced to the weak formulation. This method does not require to solve extra degrees of freedom introduced by other methods such as constraint with Lagrange multipliers [2]. The penalty method for finite element approximation was first proposed by Babuška [3] and studied in [5]. A mathematical analysis of selecting effective penalty parameters was given in Arnold et al. [1].

3. Numerical methods

3.1. The extended finite element method

In this section, we describe the XFEM for handling the jump at the interface between physical quantities in fluid domain $\Omega \setminus P(t)$ and those in rigid body domain $P(t)$. The XFEM extends the classical FEM by enriching the solution space of differential equations with discontinuous functions such that the non-smooth solutions can be obtained independent of the mesh. Let $\Omega_h$ be a conforming mesh of $\Omega$ with $h$ being the mesh size. For our fluid-particle system in two dimensions, the velocity and pressure can be discretized as

$$
\begin{align*}
\mathbf{u}_h^x(x) &= \sum_i \phi_i(x) \mathcal{H}(s) u_i^x, \\
\mathbf{u}_h^y(x) &= \sum_i \psi_i(x) \mathcal{H}(s) u_i^y, \\
p_h(x) &= \sum_i N_i(x) \mathcal{H}(s) p_i,
\end{align*}
$$

where $\phi_i(x) \in H^1(\Omega_h)$, $\psi_i(x) \in H^1_0(\Omega_h)$, and

$$
N_i(x) \in H^1_c(\Omega_h) := \left\{ p : p \in H^1(\Omega_h), \int_{\Omega_h} p \, dx = 0 \right\}
$$

are standard element shape functions, $u_i^x$, $u_i^y$, and $p_i$ are degrees of freedom and $\mathcal{H}(s)$ is a Heaviside function defined by a scalar level set function $s$,

$$
\mathcal{H}(s) = \begin{cases} 
1, & s \geq 0, \\
0, & s < 0,
\end{cases}
$$

where $s(x)$

$$
\begin{align*}
s(x) &= \begin{cases} 
> 0, & x \in \Omega \setminus P(t), \\
< 0, & x \in P(t), \\
= 0, & x \in \partial P(t).
\end{cases}
\end{align*}
$$

(3.1)
In this paper, a signed distance function is used as the level set function $s(x)$. For example, the interface of an elliptic particle with major radius $a$ and minor radius $b$ located at $X_s = (x_s, y_s)$ is defined by the zero level of function

$$s(x) = d(x, y) = \sqrt{\frac{(x - x_s)^2}{a^2} + \frac{(y - y_s)^2}{b^2}} - 1. \quad (3.2)$$

A simple interpretation of the XFEM enrichment procedure for the fluid-particle system is shown in Fig. 2. Elements entirely inside the fluid domain $\Omega \setminus P(t)$ have standard degrees of freedom (○) and are assembled to the system matrix one by one. Elements entirely inside the rigid particle body $P(t)$ have no contribution to the system, therefore, they are not assembled into the system matrix thus the associated degrees of freedom (×) can be removed. Elements intersected by the interface $\partial P(t)$ have both standard degrees of freedom and virtual degrees of freedom (■) for the jump enrichment. The virtual degrees of freedom have no variables defined on the node but help interpolating values at the fluid domain part of an intersected element. Due to the discontinuities of field variables, numerical quadratures on intersected elements (colored in blue in Fig. 2) are only performed on the fluid domain part. It is well known that the accuracy of numerical quadrature on
part of an intersected element is crucial for the robustness of XFEM. To achieve better approximation, we perform local mesh subdivisions near the interface. First, the intersected elements are divided successively into smaller quads using a quadtree technique. Then, the quads near the interface are further divided into triangles aligned with the interface \( \partial P(t) \). The subdivision near the interface is shown in Fig. 3.

### 3.2. The temporary arbitrary Lagrangian-Eulerian technique

As the particle moves, field variables at the previous time level may become undefined near the boundary of the particle since there may be no fluid flow at time \( t^n \), as shown by the dashed blue region in Fig. 4. To resolve this issue, we adopt an ALE description for the fluid dynamics by introducing a mesh velocity \( u_m \) so that the equations can be solved on a referential domain. In the classical ALE method, the computational mesh should be updated in order to track the motion of the particle. For instance, starting from an initial mesh \( x_{AL E}^0 \), a simple update scheme of ALE mesh reads

\[
x_{AL E}^{n+1} = \Phi\left(x_{AL E}^n\right) = x_{AL E}^n + u_m \left(x_{AL E}^n, t^n\right) dt. \tag{3.3}
\]

After a few steps, the ALE mesh may become too distorted, hence mesh regeneration is often needed, which is still a challenging task in view of computational costs.

In this paper, we aim to solve all equations on a fixed Eulerian mesh based on the tALE technique [8]. The idea is to extract each pair of time steps \((t^n, t^{n+1})\) separately from the whole process of time iterations. At each time step, a tALE mesh is constructed from the inverse mapping of \( \Phi \),

\[
x_{tALE} = \Phi^{-1}(x), \tag{3.4}
\]

then the field variables at the previous time step \( t^n \) are mapped along with the tALE mesh, i.e., the previous velocity can be obtained with

\[
u^n_t = u(x_{tALE}^{n}, t^n) = u(\Phi^{-1}(x), t^n). \tag{3.5}
\]

Here \( \Phi^{-1}(x) \) is regarded as a fixed point of the mapping equation (3.3) that can be calculated using a simple iterative procedure. The field variables at \( \Phi^{-1}(x) \) are computed via a bi-linear interpolation on the element that encloses the fixed point. Fig. 5 shows the computational mesh at \( t^{n+1} \) and the constructed tALE mesh at \( t^n \) for the case of an
ellipse rotating in clockwise direction. Fig. 6 shows the inverse mapping $\Phi^{-1}$ between the computational mesh and the tALE mesh.

We define $u_m$ as the solution to the following elliptic equation

\begin{align*}
-\nabla^2 u_m &= 0 \quad \text{in } \Omega, \quad (3.6a) \\
u_m &= 0 \quad \text{on } \partial\Omega, \quad (3.6b) \\
u_m &= u_s \quad \text{on } \partial P(t), \quad (3.6c)
\end{align*}

such that mesh nodes near the particle follow the motion of particle, while mesh nodes far away from the particle remain stationary. The boundary condition (3.6c) is realized by using a penalty method as the following

$$u_m - u_s + \epsilon_p \frac{\partial u_m}{\partial n} = 0 \quad \text{on } \partial P(t). \quad (3.7)$$
3.3. A direct numerical approach for fluid-particle interaction

For the fluid dynamics governed by the Navier-Stokes equations (2.1a)-(2.1b), the coupling of velocity and pressure through the incompressibility constraint results in a saddle point system, which is generally difficult to solve. Over the past decades, a class of efficient methods, so-called the projection methods, have been developed to overcome this difficulty (cf. [9, 14, 16, 23] and the review [15]). The core of these methods is to decouple the Navier-Stokes equations into an elliptic system for velocity and a Poisson system for pressure. In this paper, we extend the classical Chorin scheme [9] to a Q1-Q1 finite element approximation with second order accuracy in space and a first order semi-implicit tALE integration in time.

We denote by $\langle \cdot , \cdot \rangle$ the $L^2(\Omega_h)$ inner product and by $\langle \cdot , \cdot \rangle$ the $L^2(\partial \Omega_h(t))$ inner product. At $t = 0$ we specify initial values for $u^0$, $p^0$, $X^0_s$, $U^0_s$, $\Theta^0_s$, and $\omega^0_s$. Then, we apply the following update procedure:

**Step 1.** Define the mapping

$$\Phi(x) = x + u_m(x,t^n) dt,$$  \hspace{1cm} (3.8)

and use the inverse mapping $\Phi^{-1}$ to obtain the velocity at the previous time step

$$u^n_h = u(\Phi^{-1}(x),t^n).$$  \hspace{1cm} (3.9)

**Step 2.** Update the particle position and orientation using an explicit scheme,

$$X^{n+1}_s = X^n_s + U^n_s dt,$$  \hspace{1cm} (3.10a)

$$\Theta^{n+1}_s = \Theta^n_s + \omega^n_s dt.$$  \hspace{1cm} (3.10b)

**Step 3.** Update the particle translational and angular velocity by

$$U^{n+1}_s = U^n_s - \frac{dt}{M_s} \int_{\partial \Omega_h(t^n)} \sigma^n_h \cdot n ds + dt g,$$  \hspace{1cm} (3.11a)

$$\omega^{n+1}_s = \omega^n_s - \frac{dt}{I_h} \int_{\partial \Omega_h(t^n)} \left( \sigma^n_h \cdot n \right) \times \left( x - X^n_s \right) ds.$$  \hspace{1cm} (3.11b)

**Step 4.** Solve the velocity system of Navier-Stokes equations: find $u^{n+1}_h$ in the following weak form such that for $\forall v_h \in H^1(\Omega_h) \times H^1_0(\Omega_h)$,

$$\frac{1}{dt} \left( u^{n+1}_h , v_h \right) + \left( \left( u^n_h - u_m \left( \Phi^{-1}(x),t^n \right) \right) \cdot \nabla \right) u^{n+1}_h , v_h + \frac{1}{Re} \left( \varepsilon^{n+1}_h , \nabla v_h \right) + \frac{1}{L_s} \left( u^{n+1}_h \cdot \tau , v_h \cdot \tau \right) + \frac{1}{\varepsilon_p} \left( u^{n+1}_h \cdot n , v_h \cdot n \right)$$

$$= \frac{1}{dt} \left( u^n_h , v_h \right) - \left( \nabla P^n_h , v_h \right) + \left( g , v_h \right)$$

$$+ \frac{1}{L_s} \left( u^{n+1}_s \cdot \tau , v_h \cdot \tau \right) + \frac{1}{\varepsilon_p} \left( u^{n+1}_s \cdot n , v_h \cdot n \right),$$  \hspace{1cm} (3.12)
where \( u_i^{n+1} = U_i^{n+1} + \omega_i^{n+1} \times (x - X_i^{n+1}) \).

**Step 5.** Solve the pressure system of Navier-Stokes equations: find \( p_h^{n+1} \) in the following weak form such that for \( \forall q_h \in H^1_c(\Omega_h) \),

\[
\left( \nabla p_h^{n+1}, \nabla q_h \right) = -\frac{1}{dt} \left( \nabla \cdot u_h^{n+1}, q_h \right).
\] (3.13)

**Step 6.** Solve the elliptic equation using a standard finite element scheme: find \( u_m \) in the following weak form such that for \( \forall v_h \in H^1_0(\Omega_h) \times H^1_0(\Omega_h) \),

\[
\left( \nabla u_m, \nabla v_h \right) + \frac{1}{\epsilon_p} \left( u_m, v_h \right) = \frac{1}{\epsilon_p} \left( u_h^{n+1}, v_h \right).
\] (3.14)

**Remark 3.1.** The proposed decoupled algorithm leads to several linear elliptic systems at each time step and hence is computationally efficient and easy-to-implement.

**Remark 3.2.** In contrast to the continuum approach where the conservative properties are common issues, the XFEM with the TALE technique fundamentally guarantee the rigid motion of the particle. The fluid-particle interface is defined by the level-set function \( s(x) \) that strictly preserves the volume and length of the solid region. The solid region is rebuilt once the position and angular orientation of the particle are updated from the kinematic equations, and the updated boundary velocity of the particle in turn has an impact on the surrounding fluid.

### 4. Results and discussions

#### 4.1. Implementation

Preconditioned GMRES methods are used to obtain the solutions in each time step. Specifically, systems arising from implicit discretization of the velocity equation (3.12) are solved using a restricted additive Schwarz (RAS [7]) preconditioner, and the pressure Poisson system (3.13) and the elliptic system (3.14) are solved using an algebraic multigrid (AMG) preconditioner. The algorithm is implemented using libMesh [24] for generating the system matrices, and PETSc [4] for the preconditioned solvers.

#### 4.2. Rotation of an ellipse in a shear flow

The phenomena of suspension and dynamics of a particle in a viscous flow has attracted many interests of scientific community. Experiments [28] showed that an ellipse undergoes a periodic revolution. The particle motion consists of a spin about the axis of symmetry and a precession of this axis about the vorticity of the undisturbed flow. The rate of spin is equal to the component of the vorticity in the direction of the axis of symmetry. The precession of the axis is described, in terms of polar angles, by the well-known solution of Jeffery [22],
which is called the Jeffery orbit. Bretherton [6] demonstrated the general validity of the Jeffery orbit for particles with rotational symmetry. Recently, Zhang et al. [30] studied the slip effect on Jeffery orbit using a continuum approach, in which the solid particle was treated as a fluid phase with fictitious high density and viscosity. In this section, we perform a direct simulation for the concerned problem by coupling the fluid dynamics and the motion of particle through hydrodynamic forces and torques on the particle boundary.

As shown in Fig. 7, the domain of interest \( \Omega \) is a rectangular region with the top and bottom boundaries \( \Gamma_D \) assumed to be solid walls with distance \( H \). A viscous fluid is confined between the walls and a homogeneous solid ellipse \( P(t) \) is located at the center. The aspect ratio of the ellipse \( e = b/a \leq 1 \) where \( a \) is the major radius and \( b \) the minor radius. The solid walls move with speed \( W \) on the top boundary and \( -W \) on the bottom boundary. Under the generated shear flow, the ellipse rotates with an angular velocity \( \omega_s \).

The left and right boundaries \( \Gamma_N \) are prescribed with a traction-free condition.

In the absence of gravity, the ellipse undergoes a periodic tumbling which is determined by the Jeffery orbit equation,

\[
\omega_s = \frac{\dot{\gamma}}{2} (-1 + k \cos 2\Theta_s),
\]

where \( \dot{\gamma} = 2W/H \) is the shear rate and \( k = \frac{1-e^2}{1+e^2} \) is a dimensionless parameter measuring the anisotropy of the ellipse.

In our experiments, the computational domain is \([-1,1] \times [-1,1]\) and the center of the ellipse is \((0,0)\). The mass and inertia of the ellipse are computed with

\[
M_s = \rho_s \pi ab \quad \text{and} \quad I_s = \rho_s \pi ab (a^2 + b^2)/4,
\]

respectively, where \( \rho_s \) is the density ratio of the ellipse versus the fluid. A fixed Cartesian mesh of size \( 201 \times 201 \) is used and the time step size is \( dt = 0.001 \). Other parameters are taken as follows,

\[
Re = 1, \quad \mathcal{L} = 0.0025, \quad \rho_s = 1, \quad W = 2, \quad \epsilon_p = 10^{-10}.
\]
We start with a very small slip length that closed to no-slip boundary condition. Fig. 8 shows the fitted curves to Eq. (4.1) with various elliptic radiiues. It is observed that the numerical results are in good agreement with the Jeffery orbits. For the case of \( a = 0.25 \) and \( b = 0.125 \), a snapshot of the motion with the velocity field at \( t = 4.8 \) is shown in Fig. 9. Fig. 10 shows the fitted curves obtained with different mesh resolution: \( 51 \times 51 \), \( 101 \times 101 \), and \( 201 \times 201 \), the corresponding fitted anisotropy parameters \( k_f \) and shear rates \( \dot{\gamma}_f \) are listed in Table 1. Here \( k_f \) and \( \dot{\gamma}_f \) are obtained from the direct simulation of the Jeffery orbit by fitting the angular velocity \( \omega_s(\Theta_s) \) according to Eq. (4.1). As shown in the figure, there are a few sparks on the fitted curve obtained with mesh \( 51 \times 51 \) which is due to the lack of mesh resolution to accurately capture the fluid-particle interface. As the mesh is refined, the sparks are effectively eliminated and a better fitted Jeffery orbit is achieved.

Next, we study the effect of slip length on the fluid-particle interaction by varying \( L_s \).
Figure 10: Fitted curves obtained with different mesh resolution for the case \( a = 0.25, b = 0.125 \).

Table 1: The fitted anisotropy parameter \( k_f \) and shear rate \( \dot{\gamma}_f \) obtained with different mesh resolution.

<table>
<thead>
<tr>
<th>Fitted results</th>
<th>( a = 0.25, b = 0.125 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>51x51</td>
<td>0.598 1.970</td>
</tr>
<tr>
<td>101x101</td>
<td>0.603 1.973</td>
</tr>
<tr>
<td>201x201</td>
<td>0.601 1.974</td>
</tr>
<tr>
<td>Jeffery orbit</td>
<td>( k ) ( \dot{\gamma} )</td>
</tr>
<tr>
<td></td>
<td>0.6 2</td>
</tr>
</tbody>
</table>

Figure 11: The fitted anisotropy parameter \( k_f \) versus different slip lengths \( L_s \) for the case \( a = 0.25, b = 0.125 \).

Figure 12: Fitted curves obtained with different slip lengths for the case \( a = 0.25, b = 0.125 \).

from 0.0025 to 0.03. It is observed in Fig. 11 that the anisotropy parameter \( k_f \) increases as the slip length grows. This means that the effective anisotropy of the particle can be enhanced by the boundary slip, i.e., the effective anisotropy is jointly determined by the geometric shape and the degree of boundary slip. Fig. 12 shows the fitted curves obtained with different slip lengths \( L_s = 0.0025, 0.005, 0.01, 0.02, \) and 0.03. As \( L_s \) increases, the
cosine curves shift with an enhanced amplitude, which are in accordance with the results in Zhang et al. [30].

5. Conclusions

In this work we present a direct simulation of the particle motion in a shear flow. We use the XFEM and the tALE technique to solve all equations on a fixed Cartesian mesh. The decoupled algorithm is very efficient and easy-to-implement. For the rotation of an ellipse in a shear flow, we study the effect of the boundary slip on the dynamics of the particle. It is shown that the effective anisotropy of the particle can be enhanced by the boundary slip, which is in good agreement with those in Zhang et al. [30]. Further extensions of the proposed numerical method to more general problems can be considered in two folds: first, the XFEM and tALE technique can be generalized to cases with multiple particles as long as the interaction between particles are explicitly defined [8, 13]; second, extension of the implementation to three dimensions is practicable, where the complexity lies in the numerical integration on part of an element with the division of intersected elements into sub-elements, specifically, the classification of further triangulation of the sub-elements [17].

Acknowledgments The research was supported in part by the Special Project on High-performance Computing under the National Key R&D Program (No. 2016YFB0200601), the NSFC under 11701547, 61531166003, and the Hong Kong RGC-GRF grants 16324416, 16302715, RGC-GRF grant C6004-14G, NSFC-REGC joint research scheme N-HKUST620/15.

References


