Finsler geometry modeling of complex fluids: reduction in viscous resistance

Masahiko Okumura(1), Ippei Homma, Shuta Noro, Hiroshi Koibuchi(2)
National Institute of Technology(KOSEN), Sendai College, Natori Campus, Japan
E-mail: (1)masahikookumura@sendai-nct.ac.jp; (2)koi-hiro@sendai-nct.ac.jp

Abstract. Complex fluids refer to a broad range of materials that contain two different phases, such as solid-liquid and fluid-gas mixtures. Generally, in fluids with dispersed matter, the viscous resistance is not always proportional to the velocity of the fluids, and a fluid exhibiting such behavior is called a non-Newtonian fluid. The viscous resistance is considerably decreased in a certain range of velocities compared to the case without macromolecules. This change in the macroscopic viscous resistance is expected to originate from the interactions between fluids and dispersed matter. In this study, we apply the Finsler geometry (FG) modelling technique to implement this complex interaction, and we numerically show a mechanism for the reduction in viscous resistance. In the FG modeling, a new dynamical variable corresponding to the directional degrees of freedom of dispersed matter is introduced and updated by the Monte Carlo simulation technique during iterations of the Navier-Stokes equation. The tentative results indicate that the distribution of the viscous force and its position dependence are considerably different from those in standard Newtonian fluids. This finding indicates the possibility that the FG modeling technique can properly describe the effects of dispersed matter on the flow behavior.

1. Introduction

In a complex fluid, the fluid is mixed with vapors and solids; an example is a liquid with dispersed matter (or simply particles). Liquids with particles exhibit a unique behavior compared to single-phase fluids, in which the relationship between the shear stress and the velocity gradient is linear; hence, this type of fluid is called a Newtonian fluid. In contrast, in liquids with dispersed particles, the above relationship becomes nonlinear because the particles affect the flow behavior of the fluid in the liquid phase. Examples of such complex fluids are mixtures of cornstarch and water[1] and polymer solutions[2]. Fluids of this type are called non-Newtonian fluids.

In complex fluids with particles, the interactions between the fluid and particles are expected to be relevant. However, the computational fluid dynamics (CFD) technique, which is used to solve the Navier-Stokes (NS) equation, is efficient only for fluids in the liquid phase. This implies that the interactions between fluids and particles are beyond the scope of the CFD technique. Therefore, to study non-Newtonian fluids, the interactions between fluids and particles should be considered during the implementation of the CFD technique.

However, including the interactions between fluids and particles in a complex fluid would make the flow behavior anisotropic. This anisotropic behavior can be studied with a computational
2. Numerical analysis

2.1. Computational domain

We study the velocity field of an incompressible fluid, in which the particles are dispersed, with a boundary condition comprising two moving parallel plates (Figure 1). One of the plates moves in the negative $x$ direction with a constant velocity, while the other moves in the positive direction with the same velocity.

2.2. Navier-Stokes equation described with the stream function and vorticity

The NS equation is described by

$$\frac{\partial \omega}{\partial t} = -\vec{V} \cdot \nabla \omega + \nu \Delta \omega,$$

$$\Delta \psi = -\omega,$$

where $\nu = \mu/\rho$ is the kinematic viscosity and $\psi$ and $\omega$ are the stream function and vorticity, respectively. These variables are connected to the velocity field as follows:

$$\frac{\partial \psi}{\partial x} = -V_y, \quad \frac{\partial \psi}{\partial y} = V_x, \quad \omega = \frac{\partial V_y}{\partial x} - \frac{\partial V_x}{\partial y}.\quad (2)$$

2.3. Calculation technique, internal variable and Finsler metric

The discrete NS equation is obtained simply by employing the finite difference method. In the finite difference method, the computational domain should be covered by grids (Figure 2(a)), and the physical quantities are defined at each grid point (Figure 2(b)).

In addition to the stream function and vorticity, we introduce an internal variable, $\sigma$, at each grid point $i$ to describe the orientation or state of dispersed matter in the fluid (Figure 2(c)):

$$\sigma_i = (\sigma_{i,x}, \sigma_{i,y}) \ (\in S^1: \text{unit circle}).\quad (3)$$

Using this $\sigma$, the Finsler lengths $v_{i,x}$ and $v_{i,y}$ along the $x$ and $y$ directions at $i$ are defined as follows:

$$v_{i,x} = |\sigma_{i,y}| + v_0, \quad v_{i,y} = |\sigma_{i,x}| + v_0,\quad (4)$$

where $v_0$ is a cutoff. These Finsler lengths depend on $\sigma$, reflecting the nontrivial effect of the particle state in the flow field. To implement this effect, we define the Finsler metric $g_{ab}$ as

$$g_{ab} = \begin{pmatrix}
1/v_{i,x}^2 & 0 \\
0 & 1/v_{i,y}^2
\end{pmatrix}.\quad (5)$$
2.4. Hamiltonian for the internal variable

The internal variable $\sigma$ is updated by the Monte Carlo technique with the Hamiltonian $S$ defined by

$$S = \lambda S_0 + \nu S_\nu + S_E,$$

$$S_0 = \frac{1}{2} \sum_{ij} \left[ 1 - 3 (\mathbf{\sigma}_i \cdot \mathbf{\sigma}_j)^2 \right], \quad S_\nu = \sum_{ij} \nu_{ij} (\omega_j - \omega_i)^2, \quad S_E = -E \sum_{ij} (\mathbf{\sigma}_i \cdot \mathbf{e}),$$

where $ij$ in $\sum_{ij}$ denotes the bond connecting two neighboring points $i$ and $j$. The first term $S_0$ is the Lebwohl-Lasher potential [4]. The second term $S_\nu$ is the term corresponding to $\Delta \omega$ in Eq. (1). The symbol $\nu_{ij}$ in $S_\nu$ is given by

$$\nu_{ij} = \frac{1}{4} \left( \frac{v_{ij}}{v_{ik}} + \frac{v_{ji}}{v_{jl}} \right), \quad \nu_{ij} = \nu_{ji}.$$  

The partition function is defined by

$$Z = \sum_\sigma \exp(-S),$$

where $\sum_\sigma$ denotes the sum over all possible configurations. The term $S_E$ represents the energy of the external field $\mathbf{E}$.

2.5. Simulation procedure

The simulation procedure is a hybrid technique consisting of the Monte Carlo (MC) simulation of $\sigma$ and the CFD simulation of $\psi$ and $\omega$. The first step is to update $\sigma$ by the canonical (or metropolis) MC technique. The variable $\sigma$ is randomly updated to new $\sigma'$ with the probability $\text{Min}[1, \exp(-\delta S)]$, and hence, the acceptance rate is not controllable. This MC process is performed for a sufficient number of MC sweeps (MCSs). The next step is to solve the NS equation in Eq. (1) using the finite difference technique. These two steps are repeated for a sufficient number of iterations such that the mean values of $\psi$, $\omega$ and $\sigma$ become accurate.
3. Results and discussion

Without dispersed matter, the NS equation in (1) for the boundary condition in Figure 1 describes the Couette flow problem, the exact solution for which is well known. For this reason, the effectiveness of the proposed technique is discussed by comparing the simulation results with the solution $V_x = V_B(y - d/2)$, $V_y = 0$, where $V_B(=1)$ is the velocity of the two plates and $d(=1)$ is the height of the computational domain. First, $\vec{V}$ and $\vec{\sigma}$ are shown in Figure 3. We find that the flow direction is only along the x axis, and the velocity varies linearly along the y direction. These findings are consistent with those expected by the exact solution. On the other hand, many $\vec{\sigma}$ values are parallel to the x axis, although these values are not always uniformly aligned.

Next, the position dependencies of $dV_x/dy$ and of $d^2V_x/dy^2$ along the y axis are shown in Figure 4. We should note that a decrease in $dV_x/dy$ at the boundary plates corresponds to a reduction in viscous resistance.

These results indicate that the interaction between the fluid and dispersed matter can be implemented in the CFD simulation by FG modeling and hybrid simulation techniques.

Acknowledgments
This work is supported in part by the Collaborative Research Project of the National Institute of Technology (KOSEN), Sendai College.

References
[1] E. Brown and H.M. Jaeger, Rep. Prog. Phys., 77, 046602 (2014).
[2] H. Rehage and H. Hoffmann, J. Phys. Chem., 92, 4712 (1988).
[3] H. Koibuchi and H. Sekino, Phys. A: Stat. Mech. Appl., 393, 37 (2014).
[4] P. Lebwohl and G. Lasher, Phys. Rev. A, 6, 426 (1972).