Wake asymmetry weakening in viscoelastic fluids: A numerical discovery and mechanism understanding

Sai Peng\textsuperscript{1,2}, Tao Huang\textsuperscript{1,2}, Taiba Kouser\textsuperscript{3}, Yong-liang Xiong\textsuperscript{4}, and Peng Yu \textsuperscript{1,2,5}\textsuperscript{*}

\textsuperscript{1}Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, Shenzhen, China
\textsuperscript{2}Guangdong Provincial Key Laboratory of Turbulence Research and Applications, Southern University of Science and Technology, Shenzhen, China
\textsuperscript{3}Department of Mathematics, Government College University Faisalabad, 38000, Pakistan
\textsuperscript{4}Department of Mechanics, Huazhong University of Science and Technology, Wuhan 430074, China
\textsuperscript{5}Center for Complex Flows and Soft Matter Research, Southern University of Science and Technology, Shenzhen, 518055, China

Abstract

Viscoelasticity weakens the asymmetry of laminar shedding flow behind a blunt body in a free domain. In the present study, this finding is confirmed by four unsteady viscoelastic flows with asymmetric configuration against each blunt body, i.e., flow over an inclined flat plate with various angles of incidence, flow over a rotating circular cylinder, flow over a circular cylinder with asymmetrical slip boundary distribution, and flow over eight equal side-by-side circular cylinders placed inclined closely through direct numerical simulation (DNS) combined with the Peterlin approximation of the finitely extensible nonlinear elastic (FENE-P) model. At high Weissenberg number ($\textit{We}$), an arc shape region with high elastic stress, which is similar to shock wave, forms in the frontal area of each blunt body. This region acts as a stationary shield to divide the flow into different regimes. Thus, the free stream resembles to pass this shield instead of the original blunt body. As this shield has symmetric feature, the wake flow restores symmetry.

**Keywords:** Viscoelastic flow, shedding flow asymmetry, inclined flat plate, rotating circular cylinder, asymmetrical slip distribution, side-by-side cylinders.

I. Introduction

When a long cylindrical structure is immersed in a cross-flow, Kármán vortices are periodically shed downstream of the structure at sufficiently large Reynolds number ($\textit{Re}$).\textsuperscript{1} These vortex shedding phenomena from blunt cylinders have received great attention since they are associated with numerous cases of flow-induced structural and acoustic vibrations.\textsuperscript{2,3} For the simplest case of unconfined viscous flow past a circular cylinder, depending upon $\textit{Re}$, the flow undergoes several transitions from one flow regime to another. At very low $\textit{Re}$, since fluid inertia is negligible, fluid parcels are able to adjust the shape of the submerged blunt body and thus closely follow its contours, i.e., the flow remains attached to the surface. The flow behaves symmetrically at front and back, up and down area of rigid body. As $\textit{Re}$ is increased gradually ($\textit{Re} > \sim5$), fluid inertia increases and the adverse pressure gradient along the surface of the blunt

*Corresponding Author: yup6@sustech.edu.cn
body leads to the appearance of a separation bubble. The flow remains symmetric at up and down areas of cylinder. But the flow in front and rear of the cylinder is asymmetric. With a further increment in $Re$, the separation bubble grows in size until the wake becomes asymmetric (about the mid-plane) and unsteady at $Re \approx 48$. Beyond this $Re$, the vortices are shed downstream of the cylinder and the wake becomes periodic in time, albeit the flow is still two-dimensional and laminar. The transient flow shows the up-down asymmetry. However, previous numerical simulation and experimental results indicated that the time average flow field around a cylinder still maintains the spatial symmetry.\textsuperscript{4,5} Statistically, the time average lift acted on cylinder is about zero.

Flow over blunt bodies is also affected by the shape, orientation, number and arrangement, and motion (such as rotation)\textsuperscript{6} of the blunt bodies, boundary condition (such as wall slip),\textsuperscript{10} as well as the blockage ratio of flow domain,\textsuperscript{11} etc. In Newtonian fluid, when the flow configuration (i.e., geometry or boundary condition) is asymmetric, the pressure distribution on the blunt body and the vortex shedding in the wake show up-down asymmetry. For example, shear layer attachment on an inclined flat plate could lead to an effective change of the flow pattern based on the inclination of the inclined flat plate,\textsuperscript{7} which may lead to asymmetric flow beyond a certain angle of attack ($\alpha$). With an increase in $\alpha$, the lift coefficient of inclined flat plate gradually increases. This results from the asymmetric pressure distribution on the upper and lower walls of the inclined flat plate, while the vortex shedding trajectory in the wake deviates from the center line of the flow field. Over a rotating circular cylinder, the cross flow would also generate lateral lift due to asymmetry. As the cylinder rotation drives additional motion of the surrounding fluid, the fluid velocity increases at one side of the cylinder and decreases at the other side. This difference causes the pressure on the upper and lower sides of the cylinder to be inconsistent, resulting in the force perpendicular to the incoming flow direction, which is the so-called Magnus effect.\textsuperscript{12} Correspondingly, the wake flow at $Re = 100$ could be divided into four categories, including two types of vortex shedding modes and two types of steady states.\textsuperscript{7} Similar to flow over a rotating circular cylinder, for flow over a circular cylinder with asymmetric slip distributions, no-zero average torque and lift may occurs for asymmetrical flow distribution on cylinder’s wall.\textsuperscript{10} Meanwhile, an asymmetric vortex shedding trajectory appears downstream of the cylinder. Flow over two side-by-side circular cylinders is a simple representation of side-by-side multi-body in a free domain. Special to $Re = 100$ (based on cylinder’s diameter $D$), the flow behaviors can be classified into four different modes, depending on $L/D$ (the minimum spacing between two cylinders is $L_{0}$), i.e., the single vortex street mode when $L/D < 0.4$, the flip-flopping mode when $0.4 \leq L/D < 1.5$, the in-phase-synchronized flow mode when $1.5 \leq L/D < 2.0$, and the antiphase-synchronized flow mode when $L/D \geq 2.0$.\textsuperscript{3} If the distance between cylinders is small ($L/D < 0.4$), the whole blunt body can be regarded as a whole large blunt body. The main mode of this multi-bluff body flow is single bluff body flow. If the cylinders distributes asymmetrically up and down, an asymmetric vortex shedding trajectory appears downstream for the main flow mode. Moreover, a strong repulsive force exists between the cylinders if they are placed closely.\textsuperscript{8} The flow behaviors at the upper and lower sides of each cylinder show different features. For example, when the fluid flows past the upper side of the top cylinder, the velocity gradually recovers the incoming flow velocity along the streamwise direction. However, the flow velocity past the lower side of the top cylinder is much smaller, due to the blockage effect of the narrow gap between the two cylinders. Thus, the near wall flow is asymmetric with respect to the
horizontal center line of each cylinder, which results in the nonzero time average lift force acted on each cylinder.

Adding dilute concentration of polymers in a fluid could significantly change the flow characteristics. For the internal and external flows in the turbulent flow regime, when a small amount of polymer is added into water, the friction factor and the drag coefficient are known to dramatically decrease, which implies many applications ranging from fluid transportation to flow control.\textsuperscript{13,14} For example, Xiong et al.\textsuperscript{15} proposed a strategy to suppress vortex-induced vibration by introducing a small amount of soluble long-chain polymer in water. Adding soluble polymer into water could also suppress cavitation.\textsuperscript{16-19} Dissolving polymer into a working fluid may either inhibit heat dissipation\textsuperscript{20-22} or enhance heat transfer.\textsuperscript{23-25} Solutions containing polymer additives may exhibit more complex rheological behavior than Newtonian fluids, owing to the non-Newtonian effects such as shear-thinning, anisotropy, and viscoelasticity. Therefore, the underlying mechanisms behind these applications are very complicated. When the geometry and the boundary conditions are symmetric, flow asymmetry is more likely to occur in viscoelastic fluid, compared with pure shear-thinning fluid or Newtonian fluid flow.\textsuperscript{26-28} Haward et al.\textsuperscript{29} experimentally studied the flow of a dilute polymer solution (low polydispersity sample of tactic polystyrene dissolved in a viscous organic solvent dioctyl phthalate) over a confined cylinder. This fluid is essentially non-shear-thinning over 3 decades in shear rate. They found that at high Weissenberg number ($We$), a flow asymmetry appeared upstream of the cylinder, due to a high local tensile Weissenberg number. Later, Haward et al.\textsuperscript{30} investigated a series of shear-banding viscoelastic wormlike micellar solutions (hydrolyzed polyacrylamide dissolved at different concentrations in deionized water) and found that strong flow asymmetry appears not only in front of the cylinder but also at both sides of the cylinder. The asymmetry was also found to develop from an initially random sideways fluctuation of the highly-stressed downstream birefringent wake when $We$ is beyond a critical value $We_c$. Besides, the asymmetry also widely appears in other flows, such as cross-flow\textsuperscript{30,31}, planar expansion,\textsuperscript{31} etc.

Xiong et al.\textsuperscript{32} simulated viscoelastic flow over a hydrofoil with a large $\alpha$ and found that the wake field gradually retains symmetry (associated with a decreasing lift) when increasing $We$. For viscoelastic flow over two side-by-side circular cylinders, the lift force between the two cylinders with a low $L_D$ become lower when increasing $We$.\textsuperscript{33} In these two numerical simulations, it is observed that the lift force acted on every blunt body is weakened in viscoelastic fluid, comparing with that in Newtonian fluid. This raises questions as to whether and why the flow asymmetry against each blunt body caused by the asymmetric geometry or boundary condition could be weakened in viscoelastic fluid.

To shed light on this matter, we propose four numerical examples with asymmetric flow configuration against each blunt body, i.e., flow over an inclined flat plate with various angles $\alpha$, flow over a rotating circular cylinder, flow over a circular cylinder with asymmetrical slip boundary distribution, and flow over eight equal side-by-side circular cylinders placed inclined closely. Flow over an inclined flat plate is the representative of geometric asymmetry. Other geometric asymmetry examples include a NACA0012 with angle of attack,\textsuperscript{34,35} an inclined square cylinder,\textsuperscript{36} etc. Flow over a rotating circular cylinder and asymmetric slip distribution on the cylinder surface are the representative of asymmetry in boundary condition. Flow over eight equal side-by-side circular cylinders placed inclined closely, which could be regarded as one blunt body,
is the representative of multi-body flow. The flow asymmetry comes from the asymmetry behavior of multiple blunt body distribution. Other blunt body flows, such as flows over two unequal side-by-side circular cylinders, \textsuperscript{37} and two staggered circular cylinders, \textsuperscript{38} can be classified in this category. For these three types of flows, due to geometric asymmetry, boundary asymmetry or asymmetry behavior of surrounding flow, the lift force acted on each blunt body is not zero, while the wake field becomes asymmetry.

In each example, we carefully explore the influence of viscoelasticity on the laminar unsteady wake behavior, particularly the flow asymmetry. This paper is organized as follows. The governing equations and the numerical methods are presented in Sec. II. In Sec. III, the results of lift force and flow characteristics of the four flow cases are presented and the corresponding underlying mechanisms on flow asymmetry weakening are discussed. Finally, the main conclusions and the future outlook are provided in Sec. IV.

II. Mathematical formulation

While a small amount of polymers are added into water, the incompressible Navier-Stokes (N-S) equations are slightly modified and have the following form on a differential fluid element:\textsuperscript{39-41}

\[ \frac{\partial u_j}{\partial x_j} = 0, \]

\[ \rho \frac{\partial u_j}{\partial t} + \rho u_j \frac{\partial u_j}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \mu_s \frac{\partial^2 u_j}{\partial x_i^2} + \frac{\mu_p}{\lambda} \frac{\partial \tau_{ij}^p}{\partial x_j}, \]

where the summation convention is used, with \( i \) and \( j \) being the summation indices, \( x \) is the coordinate, \( t \) is time, \( u \) is the velocity, \( p \) is the pressure, \( \rho \) is the density of fluid, \( \mu_s \) and \( \mu_p \) are the viscosity contribution from the solvent and the polymer, respectively, \( \lambda \) is the relaxation time of polymer, \( \tau_{ij}^p \) is the additional polymer stress. The last term in Eq. (2) represents the influence of \( \tau_{ij}^p \) due to the elasticity of polymers in the flow. The total viscosity of the solution is defined as \( \mu = \mu_p + \mu_s \). The polymer viscosity ratio at vanishing shear rate is defined as \( \beta = \mu_p/\mu_s \), which is a measurement of polymer concentration and molecular characteristics. Eq. (2) restores to the original N-S equations when \( \beta = 0 \). The polymer stress can be modeled by a molecular-based Peterlin approximation of the finitely extensible nonlinear elastic (FENE-P) model, which describes an individual member of polymers in a dilute concentration as a dumbbell connected with a finitely extensible nonlinear elastic spring by the way of a balance of forces acting on each beads. In this model, the polymeric stress \( \tau_{ij}^p \) can be determined using kinetic theory:\textsuperscript{42,43}

\[ \tau_{ij}^p = \frac{c_y}{1 - \frac{c_{ik}}{L^2}} \delta_{ij} - \delta_{ij}, \]

where \( c_y \) represents the polymer conformation tensor, which is defined as the pre-averaged dyadic product of the polymer end-to-end vector, \( \delta_{ij} \) is the Kroenecker delta function, and \( L \) is the maximum polymer extensibility, which is normalized by the equilibrium length of a linear spring \((\kappa T/H)^{1/2}\) with \( T \) the absolute temperature, \( \kappa \) the Boltzmann constant, and \( H \) the Hookean spring
constant for an entropic spring. The polymer conformation tensor is governed by the following hyperbolic transport equation:

$$\frac{\partial c_{ij}}{\partial t} + u_k \frac{\partial c_{ij}}{\partial x_k} - c_{ik} \frac{\partial u_j}{\partial x_k} - c_{ij} \frac{\partial u_k}{\partial x_k} = - \frac{\tau_{ij}^p}{\lambda}.$$  \hspace{1cm} (4)

It is noted that the FENE-P model is employed in this work based on its ability to properly represent the finite extensibility of the polymer. In order to obtain bound solutions for problems with high $We$ and large strain rate, the finite extensibility is necessary. Those linear spring models such as the Oldroyd-B model cannot be faithfully used in real engineering problems. Furthermore, the FENE-P model has been widely used in previous studies involving viscoelastic flows at high $Re$ and can successfully produce accurate physical results in these problems.

In this paper, $\beta$ is set to be very small ($\beta = 0.1$) to minimize the shear-thinning effect and $L$ is set to 100. Under these parameters, the FENE-P model could describe the rheological behavior of a dilute polymer solution more explicitly. Other values of $L$ are also investigated to provide a basic evaluation of the influence of the maximum polymer extensibility. As $L$ tends to infinity, the present FENE-P fluid approaches to the Oldroyd-B fluid.

To solve the above equations numerically, a finite volume commercial solver ANSYS FLUENT associated with a self-developed user defined function (UDF) for the elastic stress transport equation, i.e., Eq. (4), is applied. The QUICK scheme and the second order implicit scheme are adopted to discretize the space and temporal domains, respectively. An artificial term $\kappa \Delta \epsilon$ is added to the right hand side of Eq. (4). This implementation has been proved to be effective to ensure numerical stability. In this study, $\kappa = 0.1 \mu$ ($\kappa = 0.5 \mu$ for asymmetrical slip boundary cases) is adopted by considering both the numerical stability and accuracy of numerical approximation. Furthermore, an under-relaxation scheme is utilized to avoid the rapid increase of the numerical error. For numerical stabilization, the method is performed at the simulation of flow over an inclined flat plate with various angles of incidence and flow over a circular cylinder with asymmetrical slip boundary distribution.

III. Validation for viscoelastic flow simulation

In order to evaluate the accuracy of the present numerical method for the FENE-P fluid, a comparison is made with the analytical solution for the slit flow. The streamwise velocity profile and the first component of non-dimensional polymeric stress profile at $We = 1$ are both plotted in Fig. 1, which shows excellent agreement between the analytical solutions and the present numerical results.

Furthermore, we extracted some details of the flow field to compare with those in Richter et al. Fig. 2 shows the distribution of the elastic stress and $x$-velocity in front of the cylinder wall at $Re = 100$, $We = 10$ and $L = 100$. For higher maximum polymer extensibility and higher Weissenberg number, the flow field shows a solid like trend in front of the bluff body, characterized by high elastic stress, high pressure, and decreasing flow velocity. It is noted that the present Weissenberg number and maximum polymer extensibility $L = 100$ is quite high to be a numerical challenge. The profile of trace of conformation tensor and $x$-velocity alone the center
line demonstrates that our numerical results are consistent with the literature as shown in Fig. 2. The error for the maximum trace is about 2 percent suggesting that present results suffer from the numerical diffusion slightly. In addition, it is worth mentioning that we get the time average drag coefficient of 2.64, while Richter et al.\textsuperscript{39} gives it as 2.7. In all, these two comparative cases confirm that the present code could simulate the main characteristics of viscoelastic fluid flow.

Fig. 1. A comparison between the analytical solution and the present results of the slit flow for the FENE-P fluid at $We = 1$. (a) Streamwise velocity profile and (b) the first component of non-dimensional polymeric stress.

Fig. 2. The distribution of (a) the trace of elastic stress tensor ($c_{kk}/L^2$) and (b) x-velocity ($u/U_\infty$) in front of the cylinder along the stagnation streamline. X-coordinate origin is taken at the forefront of the cylinder.

In viscoelastic fluid at high $We$, viscoelastic stress plays a key role on those flow dynamics. In order to understand the relationship between elastic stress and velocity (gradient) distribution, the field distribution of $c_{kk}/L^2$ and time-averaged flow velocity magnitude ($\sqrt{\bar{u}^2 + \bar{v}^2}/U_\infty$) are shown in Fig. 3. The concentrated area of elasticity extends from the upstream to the downstream of the cylinder. The velocity distribution in this concentrated area becomes smaller. The existence of elastic stress slows down the flow velocity near the cylindrical wall.
IV. Numerical results and discussion

A. Viscoelastic flow over an inclined flat plate with various angles of incidence

In our previous work, we studied viscoelastic flow over a Naca0012 hydrofoil in a free domain. In this study, the Naca0012 hydrofoil is replaced with an inclined flat plate with various angles of incidence, as shown in Fig. 4(a). The width of the flat plate is set as $a$ and the height is set as $0.02a$. The attack angle between the flat plate and the incoming flow is recorded as $\alpha$ ($10^0$ and $20^0$). The entrance of the calculation domain is rectangle. The center of gravity of the inclined flat plate is set at $(x, y) = (0, 0)$. The distance between the inlet and the origin of coordinate is set as $L_u = 10a$. The distance between the outlet and the origin of coordinate is set as $L_v = 20a$. The distance between the upper and lower boundaries is set as $H = 20a$. A uniform streamwise velocity $U_\infty$ is applied at the inlet boundary. The transverse boundaries of the simulation domain are assumed to be the symmetric boundary condition, which acts as a slip wall in order to eliminate displacement thickness of no-slip wall. The present computation domain has been validated to be able to give an accurate result compared to an almost doubled domain. A zero gauge pressure is applied at the outflow boundary with the far field elastic stress as $c = 1$. The computational grid is generated by the commercial software ICEM. The mesh is generated based on two-dimensional structured O-grid topology in the $x$-$y$ direction giving rise to quadrilateral cells of 271, 800. The
mesh is local refined near the inclined flat plate wall. Fig. 4(b) exhibits the adopted mesh surrounding the flat plate. The near-wall mesh consists of 600 grid points in wide direction and 60 grid points in height direction uniformly distributed along the inclined flat plate wall and 71 grid points stretched over an exponential progression along the radial direction to ensure a fine mesh near the inclined flat plate surface. In this study, the height of the first cell adjacent to the surface of the foil is set to 0.000625. In the x-direction, 501 grid points (for \( L_x \)) are unevenly arranged in the downstream region, and 51 grid points (for \( L_u \)) are set in the upstream region. A grid independence study was performed to verify that the numerical results keep consistent to those by refinement. For this numerical example, the Reynolds number is defined as \( Re = U_∞ \rho / \mu \) and fixed at 500 and the Weissenberg number is defined as \( We = \lambda U_∞ / a \), which ranges from 0 to 1.5. For most simulation cases, the time step is set to 0.00625\( a / U_∞ \). For the certain cases that are more difficult to converge (mainly those at high \( We \)), the calculation time step is reduced to 0.003125\( a / U_∞ \) or even 0.0015625\( a / U_∞ \). Our previous studies\(^{15,32,33}\) indicated that the time step is sufficiently small to obtain reliable and accurate simulation results.

![Diagram](image1.png)

**Fig. 4.** (a) The schematic of the unconfined flow around an inclined flat plate. (b) The near-wall view of the mesh for \( α = 20^0 \).

**Table 1.** Results comparison at \( Re = 500 \) and \( α = 20^0 \) for Newtonian fluid.

| Source          | \( \overline{C}_d \) | \( \overline{C}_l \) | \( St \) | \( C_{l,ms} \) | \( C_{d,ms} \) |
|-----------------|----------------------|----------------------|---------|----------------|----------------|
| Yang et al.\(^{48}\) | 0.4472               | 0.9823               | 0.4959  | 0.1003         | 0.0176         |
| Present Results | 0.4716               | 0.9814               | 0.4923  | 0.1262         | 0.0144         |

![Diagram](image2.png)

**Fig. 5.** The time average lift coefficients.
The drag coefficient \((C_d)\) and lift coefficient \((C_l)\) are calculated by the following equations:

\[
C_d = \frac{2F_x}{\rho U_*^2 a} \quad \text{and} \quad C_l = \frac{2F_y}{\rho U_*^2 a},
\]

where \(F_x\) and \(F_y\) are force acted on the inclined flat plate in \(x\) and \(y\) directions, respectively. The calculation method of force could refer to our previous publications.\(^{15,32,33}\)

**Fig. 6.** The time average streamlines.

**Fig. 7.** The instantaneous vorticity contours. \(\omega_x\) is normalized by \(U_*/a\).
Our simulation results for Newtonian fluid are compared with Yang et al.\textsuperscript{48} for \( \alpha = 20^\circ \), list in Tab. 1. On the whole, our results are close to those of Yang et al.\textsuperscript{48} For example, the time average lift coefficient calculated by us is 0.9814, while the result calculated by Yang et al. is 0.9823. The time average lift coefficient (\( \overline{C_l} \)) of the hydrofoil at various \( \alpha \) and \( We \) in viscoelastic fluid is plotted in figure 5. In viscoelastic fluid, for \( \alpha = 10^\circ \) and \( 20^\circ \), \( \overline{C_l} \) gradually decreases with an increase in \( We \). The magnitude of lift could reflect the degree of asymmetric distribution of reaction force of the reflect flat plate upper and lower surfaces. The decreasing \( \overline{C_l} \) indicates that the flow asymmetry is weakened near the inclined flat plate wall. Following, the downstream wake flow symmetry behaviour in viscoelastic fluid is discussed.

The time average streamlines for \( \alpha = 10^\circ \) and \( 20^\circ \) are shown in Fig. 6. The recirculation region behind the hydrofoil gradually elongates with increasing \( We \). The instantaneous vorticity contours are shown in Fig. 7. In Newtonian fluid, at high \( \alpha \) (e.g. \( \alpha = 20^\circ \)), the trail of wake vortex leans to one side of the hydrofoil. However, as \( We \) increases, the vortex trajectory gradually approaches to the center line of the flow field. The typical instantaneous positions of vortex cores for Newtonian and viscoelastic fluids at \( \alpha = 10^\circ \) and \( 20^\circ \) are extracted and plotted in Fig. 8. Obviously, with an increase in \( We \), the vortices on both sides of the hydrofoil tend to be symmetric against the center line of the flow field.

![Fig. 8](image8.png)

**Fig. 8.** The typical instantaneous positions of vortex cores for Newtonian and viscoelastic fluids at (a) \( \alpha = 10^\circ \) and (b) \( \alpha = 20^\circ \). The moments correspond to Fig. 7.

![Fig. 9](image9.png)

**Fig. 9.** The instantaneous distributions of (a) \( c_{xx} \) and (b) \( c_{yy} \) for the viscoelastic flow over an inclined flat plate at \( \alpha = 20^\circ \) and \( We = 1.5 \).

For the viscoelastic flow over an inclined flat plate with various angles of incidence, a region with high elastic stress appears around the leading edge and then extends to the wake region (combine the regions of high conformation tensor components \( c_{xx} \) and \( c_{yy} \) in Fig. 9(a) and (b)). \( \alpha \) does not fundamentally affect the distribution of the elastic stress (data not shown). The
corresponding time-averaged flow velocity magnitude \((\sqrt{u'^2 + v'^2}/U_\infty)\) distribution is shown in Fig. 26(b), which is compared with that of Newtonian fluid, as shown in Fig. 10(a). The isogram of \(\sqrt{u'^2 + v'^2}/U_\infty = 0.1\) with pink line is marked in the figure. The flow velocity in its surrounding area is lower than this value (0.1), which could be regarded as a whole blunt body. The whole blunt body is symmetry with the centerline of the flow field. The centerline is marked in the figure. Because of the symmetrical distribution of the near-wall flow, the vortex shedding trajectory of the wake also tends to be symmetrical in the flow field.

\[
\text{(a) Newtonian} \quad \text{(b) } We = 1.5
\]

**Fig. 10.** The distribution of time-averaged flow velocity magnitude \((\sqrt{u'^2 + v'^2}/U_\infty)\) field for (a) Newtonian fluid and (b) viscoelastic fluid with \(We = 1.5\) and \(L = 100\), which is dimensionless with incoming flow velocity \((U_\infty)\). Isogram \(\sqrt{u'^2 + v'^2}/U_\infty = 0.1\) is marked with pink line in panel (b).

**B. Viscoelastic flow over a rotating circular cylinder**

Second, we consider viscoelastic flow over a rotating circular cylinder in a free domain, as shown in figure 11(a). The whole computational domain has a rectangular shape, with the length \(L_x+L_d\) and the width \(H\). The center of the cylinder is set at \((x, y) = (0, 0)\). The diameter of the cylinder is \(D\). The distance between the inlet and the cylinder center is set as \(L_w = 25D\). The distance between the outlet and the cylinder center is set as \(L_d = 75D\). \(H\) is set as \(50D\). The free stream boundary condition with the uniform velocity \(u = (U_\infty, 0)\) is imposed at the inlet. The no-slip boundary condition is adopted on the cylinder surface. The angular velocity of rotating cylinder is \(\omega\) and the dimensionless rotation velocity rate is defined as \(\omega_r = \omega D/(2U_\infty)\). In this study, \(\omega_r\) ranges from 0 to 6. The symmetry boundary condition is imposed at the upper and lower boundaries of the computational domain. The pressure at the outlet is set as \(p = 0\). For the conformation tensor, the no-flux condition is approximated at all boundaries. The Reynolds number is defined as \(Re = U_\infty D \rho/\mu\) and fixed at \(Re = 100\) and the Weissenberg number is defined as \(We = \lambda U_\infty/D\), which ranges from 0 to 10.

As shown in Fig. 9(b), the surrounding region of the circular cylinder is discretized by the O-type mesh. The rest of the computational domain is discretized by several blocks of rectangular meshes, with the dense mesh near the cylinder and the coarse mesh near the domain boundaries. The O-type mesh consists of 280 grid points uniformly distributed along the cylinder perimeter and 71 grid points stretched over an exponential progression along the radial direction to ensure a fine mesh near the cylinder surface. In this study, the size of the first cell adjacent to the cylinder surface in the radial direction is set to 0.0025D. In the \(x\)-direction, 501 grid points (for \(L_x\) are...
unevenly arranged in the downstream region, and 37 grid points (for $L_u$) are set in the upstream region. The total number of meshes for the computational domain is approximately 85,000. The time step is set to be small enough in our simulation to ensure the stability of numerical calculations. For most simulation cases, the time step is set to $0.005D/U_\infty$. For the certain cases that are more difficult to converge (mainly those at high $We$), the calculation time step is reduced to $0.0025D/U_\infty$ or even $0.00125D/U_\infty$.

**Fig. 11.** (a) The schematic of flow over a rotating circular cylinder. (b) Near wall mesh distribution.

**Fig. 12.** The absolute time average lift coefficients of the rotating cylinder in (a) Newtonian fluid and (b) viscoelastic fluid.

The lift coefficient ($C_l$) is calculated by the following equations:

$$C_l = \frac{2F_y}{\rho U_\infty^2 D},$$

where $F_y$ is force acted on the cylinder in $y$ direction.

The absolute time average lift coefficients ($|C_l|$) of the cylinder is calculated and plotted in figure 7. In Newtonian fluid, $|C_l|$ increases with $\alpha$, as shown in Fig. 12(a). The comparison between the present results and those of Bourguet & Jacono\(^9\) and Stojković et al.\(^9\) shows excellent agreement, which is with a little difference with the potential flow solution (potted with
dotted line) for fluid viscosity and flow separation and vortex shedding. In viscoelastic fluid, $|\vec{C}_i|$ gradually decreases with an increase in $We$ for all $\alpha_r$ as shown in Fig. 12(b). As mentioned above, the magnitude of lift could reflect the strength of flow asymmetry. The decreasing $|\vec{C}_i|$ indicates that the flow asymmetry is weakened near the cylindrical wall.

![Fig. 13](image)

Fig. 13. (a) The instantaneous vorticity contours and (b) the time average streamlines for $\alpha_r = 1.8$. $\omega_z$ is normalized by $U_\infty/D$.

![Fig. 14](image)

Fig. 14. The typical instantaneous positions of vortex cores for Newtonian and viscoelastic fluids for $\alpha_r = 1.8$. The moments correspond to Fig. 13(a).

The instantaneous vorticity contours (left column) and the time average streamlines (right column) at $(Re, \alpha_r, L) = (100, 1.8, 100)$ for different $We$ are shown in Fig. 13. The vortex core centers are extracted and drawn in Fig. 14. At this $\alpha_r$, in Newtonian fluid, the wake vortex shedding belongs to the mode I shedding (same to $\alpha = 0$), however, the trajectory of vortex core
center deviates from the centerline (dotted line in the figure) of the flow field obviously. In viscoelastic fluid, the vortex core center gradually concentrates on the center of the flow field as $We$ increase. At the same time, the time-averaged wake bubbles are elongated and tend to the center line of the flow field, while an annulus streamline boundary layer formed between the original cylinder surface and the dummy stationary wall.

**Fig. 15.** (a) The instantaneous vorticity contours and (b) the time average streamlines for $\alpha_c = 6.0$. $\omega_z$ is normalized by $U_\infty/D$.

**Fig. 16.** The instantaneous vorticity contours (left column) and the time average streamlines (right column) at $(Re, We, L) = (100, 10, 100)$. $\omega_z$ is normalized by $U_\infty/D$. 
The instantaneous vorticity contours (left column) and the time average streamlines (right column) at \((Re, \alpha, L) = (100, 6.0, 100)\) for different \(We\) are shown in Fig. 15. At this \(\alpha\), in Newtonian fluid, the flow state is time-steady with no wake vortex shedding. For time-average flow, a streamline distribution similar to an egg surrounds the cylinder. In viscoelastic fluid at \(We = 1\), the flow is still time-steady, however the vortex extends downstream. For time-average flow streamline, the ‘egg’ becomes similar to an annulus surrounds the cylinder, while another bubble appears downstream. In viscoelastic fluid at \(We = 2\), the flow becomes time-unsteady. The vortex sheds downstream, which is deviated from the center line (dotted line in the figure) of flow field. For time-average flow streamline, the ‘annulus’ becomes larger. The bubble downstream becomes also larger, while another bubble appears downstream. In viscoelastic fluid at \(We = 4\). The vortex core centers tend to the centerline (dotted line in the figure) of the flow field. For time-average flow streamline, the ‘annulus’ becomes larger while the bubbles downstream become also larger. The bubbles downstream tends to symmetrically with the centerline (dotted line in the figure) of the flow field.

For our selected \(\alpha = 1.8 \text{ and } 6.0\), although the flow state in Newtonian fluid are different, as \(We\) increase in viscoelastic fluid, the wake flow field eventually evolve similarly. We show the instantaneous vorticity contours (left column) and the time average streamlines (right column) at high \(Wi (Wi = 10)\) for different \(\alpha\) in Fig. 16. For different \(\alpha\), the flow fields behave quite similarly. We mainly concern the flow behaviors in two regions, i.e., a thick rotating boundary layer flow around the rotating circular cylinder and vortex shedding in the wake. A simple model, as shown in Fig. 18, could be used to describe the time-average flow behavior of the viscoelastic fluid past a rotating circular cylinder at a high \(We\) (such as \(We = 10\)). \(D_a\) denotes the thickness of the rotating boundary layer \((D_a > D)\), which increases with \(\alpha\). We notice that the profile of the circumferential velocity \(u_\theta\) along the radial direction \(r\) in the rotating boundary layer is almost completely consistent, as shown in Fig. 17. \(u_\theta\) gradually decays with \(r\) and tends to zero at the outer edge of the rotating boundary layer \(r = D_a/2\). Thus, this outer edge \(r = D_a/2\) acts as a dummy stationary solid wall and divides the flow into two regimes, as shown in Fig. 18. The flow induced by the rotating cylinder is shielded by this dummy wall and does not communicate with the free stream outside. On the other hand, the free stream outside resembles to pass a large stationary cylinder with \(r = D_a/2\) and the wake becomes symmetric as shown in Fig. 18. Because of the symmetrical distribution of the near-wall flow, the vortex shedding trajectory of the wake also tends to be symmetrical in the flow field.
**Fig. 17.** (a) The instantaneous distributions of the circumferential velocity $u_\theta$. (b) Circumferential velocity $(u_\theta)$ profiles along the front, back, upper and lower lines near the cylinder for $(Re, We, L, \alpha_r)=(100, 10, 100, 6)$. $r=0.5$ denotes the cylinder surface.

**Fig. 18.** A simple model to illustrate the feature of the viscoelastic flow over a circular cylinder rotating at a high $We$.

In order to understand the above-mentioned change of wall velocity distribution, we consider the distribution of elastic stress. The Conformation tensor components $(c_{xx}$ and $c_{yy})$ distribution for the viscoelastic flow over a rotating circular cylinder are shown in Fig. 19. Two elastic stress boundary layers can be identified by combining the regions of high conformation tensor components $c_{xx}$ and $c_{yy}$ in figure 19(a) and (b). One is a thick inner elastic stress boundary layer surrounds the rotating cylindrical wall, which is caused by the velocity gradient of the rotation of the cylinder. Elastic stress tends to appear in the place where the velocity gradient is larger. Conversely, the existence of elastic stress gradient plays a dissipative role, which in turn weakens the gradient of velocity. The inner elastic stress boundary layer makes the internal flow velocity uniform in the circumferential direction, resulting in an annulus velocity distribution. Another is an outer inner elastic stress boundary layer surrounds the dummy stationary solid cylinder $(D_\alpha)$, shown in Fig. 19. The elastic stress distribution leads to a velocity distribution as shown in Fig. 18, and thus a symmetrical wake vortex shedding trajectory.

**Fig. 19.** The instantaneous distributions of (a) $c_{xx}$ and (b) $c_{yy}$ for the viscoelastic flow over a
rotating circular cylinder at \( \alpha_r = 6 \) and \( We = 10 \). The dash lines denote the position of the dummy stationary solid wall.

**C. Viscoelastic flow over a circular cylinder with asymmetrical slip boundary distribution**

Third, we consider viscoelastic flow over a circular cylinder with asymmetrical slip boundary distribution in a free domain, as shown in figure 20(a). The whole computational domain has a rectangular shape, with the length \( L_u + L_d \) and the width \( H \). The center of the cylinder is set at \((x, y) = (0, 0)\). The diameter of the cylinder is \( D \). The distance between the inlet and the cylinder center is set as \( L_u = 25D \). The distance between the outlet and the cylinder center is set as \( L_d = 75D \). \( H \) is set as \( 50D \). The free stream boundary condition with the uniform velocity \( u = (U_\infty, 0) \) is imposed at the inlet. A partial slip boundary condition is adopted on the upper half of the cylinder surface. The linear slip length (\( L_s \)) method is introduced to describe the degree of cylinder’s upper wall slippage. The slip length is defined as following equation: \(^{50}\)

\[
\mathbf{n} \times \mathbf{u} = L_s \mathbf{n} \times \left[ (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \cdot \mathbf{n} \right],
\]

where \( \mathbf{n} \) is the unit normal to the cylinder surface. A dimensionless number, Knudsen number (\( Kn \)) is used to evaluate the relative relationship between slip length and cylinder diameter, as follows,

\[
Kn = \frac{L_s}{D}.
\]

In our this simulation, \( Kn \) is set as 0.5 for upper half of the cylinder surface. The slip condition (Eqn. 7) calculated refers to Legendre et al. \(^{51}\). The no-slip boundary condition is adopted on the lower half of the cylinder surface. The pressure at the outlet is set as \( p = 0 \). For the conformation tensor, the no-flux condition is approximated at all boundaries. The Reynolds number is defined as \( Re = U_\infty D \rho / \mu \) and fixed at \( Re = 500 \) and the Weissenberg number is defined as \( We = 2U_\infty / D \), which ranges from 0 to 1.0.

![Fig. 20.](image)

**Fig. 20.** (a) The schematic of the unconfined flow around a circular cylinder with asymmetrical slip boundary distribution. (b) The near-wall view of the mesh.

As shown in Fig. 20(b), the surrounding region of the circular cylinder is discretized by the O-type mesh. The rest of the computational domain is discretized by several blocks of rectangular meshes, with the dense mesh near the cylinder and the coarse mesh near the domain boundaries. The O-type mesh consists of 300 grid points uniformly distributed along the cylinder perimeter and 151 grid points stretched over an exponential progression along the radial direction to ensure a
fine mesh near the cylinder surface. In this study, the size of the first cell adjacent to the cylinder surface in the radial direction is set to 0.00125D. In the x-direction, 501 grid points (for L_x) are unevenly arranged in the downstream region, and 71 grid points (for L_u) are set in the upstream region. The total number of meshes for the computational domain is approximately 156,200. The time step is set to be small enough in our simulation to ensure the stability of numerical calculations. For all simulation cases, the time step is set to 0.0125D/\infty.

The drag coefficient \(C_d\) and lift coefficient \(C_l\) are calculated by the following equations:

\[
C_d = \frac{2F_x}{\rho U_x^2 D} \quad \text{and} \quad C_l = \frac{2F_y}{\rho U_x^2 D},
\]

where \(F_x\) and \(F_y\) are force acted on the cylinder in x and y directions, respectively.

**Fig. 21.** Drag coefficients \(C_d\) for \(Re\) from 0.1 to 1 in Newtonian fluid, compared with analytical solution and numerical results of Li et al.\(^{10}\)

**Fig. 22.** The time average lift coefficients.

Li et al.\(^{10}\) gives an approximate analytical solution laminar flow over a circular cylinder with all partial slip boundary. The drag reads as:

\[
C_d = \left[ \frac{1}{1 - \frac{1}{2(1+2Kn)} - \gamma - \ln \left( \frac{Re}{8} \right)} \right] \frac{8\pi}{Re},
\]

(10)
The solution is more accurate when Reynolds number is much less than 1. We simulated Newtonian flow over a circular cylinder with all partial slip boundary at $Re$ from 0.1 to 1. $Kn$ is set as 0.1. Our simulated space area is a large circular area, while the blocking rate ($BR$) of the cylinder is 1%. At this $Re$ range, the calculation area is enough to be unaffected by the blocking rate. Our simulated $C_l$ are compared with the analytical solution and the simulated results of Li et al. ($BR = 1\%$), a shown in Fig. 21. For $Re$ is less than 0.2, our results compares well with the analytical solution. For $Re$ is higher than 0.3, our results compares well with the simulated results of Li et al. This validation confirm that the present code could simulate the wall partial slip.

![Fig. 23.](image)

The time average lift coefficient ($\overline{C_l}$) of the cylinder at various $\alpha$ and $We$ in Newtonian and viscoelastic fluid are plotted in figure 22. In Newtonian fluid, due to the asymmetric slip of cylindrical wall, $\overline{C_l}$ is as high as 2.665. In viscoelastic fluid, $\overline{C_l}$ gradually decreases with an increase in $We$ when $We$ is higher than 0.2. The magnitude of lift could reflect the degree of asymmetric distribution of reaction force of the reflect flat plate upper and lower surfaces. The decreasing $\overline{C_l}$ indicates that the flow asymmetry is weakened near the cylinder’s wall. Following, the downstream wake flow symmetry behaviour in viscoelastic fluid is discussed.

The instantaneous vorticity contours (left column) and the time average streamlines (right column) at ($Re$, $Kn$, $L$) = (100, 0.5, 100) for different $We$ are shown in Fig. 23. The vortex core centers are extracted and drawn in Fig. 24. In Newtonian fluid, the trajectory of vortex core center

![Fig. 23. (a) The instantaneous vorticity contours and (b) the time average streamlines. $\omega_z$ is normalized by $U_{\infty}/D$.](image)
deviates from the centerline (dotted line in the figure) of the flow field obviously. In viscoelastic fluid, the vortex core center gradually concentrates on the center of the flow field as $We$ increase when $We$ is over 0.2. At the same time, the time-averaged wake bubbles are elongated and tend to the center line of the flow field.

Fig. 24. The typical instantaneous positions of vortex cores for Newtonian and viscoelastic fluids. The moments correspond to Fig. 23(a).

Fig. 25. The instantaneous distributions of (a) $c_{xx}$ and (b) $c_{yy}$.

Fig. 26. The distribution of time-averaged flow velocity magnitude ($\sqrt{u^2 + v^2}/U_\infty$) field for (a) Newtonian fluid and (b) viscoelastic fluid with $We = 1.0$ and $L = 100$, which is dimensionless with incoming flow velocity ($U_\infty$). Isogram of $\sqrt{u^2 + v^2}/U_\infty = 0.1$ is marked with pink line in panel (b).

For the viscoelastic flow over a circular cylinder with asymmetrical slip boundary distribution, a region with high elastic stress appears around the leading edge and then extends to the wake region (combine the regions of high conformation tensor components $c_{xx}$ and $c_{yy}$ in Fig. 25(a) and (b)). The corresponding time-averaged flow velocity magnitude ($\sqrt{u^2 + v^2}/U_\infty$) distribution is shown in Fig. 26(b), which is compared with that of Newtonian fluid, as shown in
Fig. 26(a). The isogam of $\sqrt{u^2 + v^2}/U_\infty = 0.1$ with pink line is marked in the figure. The flow velocity in its surrounding area is lower than this value (0.1), which could be regarded as a whole blunt body. The velocity distributions near the upper and lower surfaces of the circular cylinder tend to be symmetric.

D. Viscoelastic flow over eight equal side-by-side circular cylinders placed inclined closely

Fourth, we consider viscoelastic flow over eight equal side-by-side circular cylinders placed inclined closely in a free domain, as shown in Fig. 27. The diameter of each cylinder is $D$ and the nearest distance between each nearby two cylinders is $L_D = 0.1D$. In our previous study, viscoelastic flow over two side-by-side circular cylinders at high $We$, the flow behaves like flow over two lower distance side-by-side circular cylinders for lower $We$. For $L_D = 0.1D$, Newtonian flow over two side-by-side circular cylinders, the wake vortex shedding behaves like one cylinders wake with $O$. In viscoelastic flow, the wake flow also like flow over one large cylinder (the cross-sectional area facing the incoming flow is about $2D + L_D D$). However, if $L_D$ is increased, such as $L_D = 3D$, the two cylinders cannot be considered as one body. Note that the present study only concerns the wake asymmetry weakening of one blunt body or several closely placed bodies which can be regarded as one body. Thus, the results for $L_D = 0.1D$ for eight equal side-by-side circular cylinders placed inclined is presented here. The center points of eight cylinders are on the same straight line. The angle between this line and the incoming flow direction is set at $20^\circ$. The whole computational domain has a rectangular shape, with the length $L_u + L_v$, and the width $L_w$. The distance between the inlet and the cylinder center is set as $L_u = 25D$. The distance between the outlet and the cylinder center is set as $L_v = 75D$. $H$ is set as $50D$. The middle point of the eight cylinders is set at $(x, y) = (0, 0)$. The free stream boundary condition with the uniform velocity $\mathbf{u} = (U_\infty, 0)$ is imposed at the inlet. The no-slip boundary condition is adopted on the eight cylinder surfaces. The symmetry boundary condition is imposed at the two lateral boundaries of the computational domain. The pressure at the outlet is set as $p = 0$. For the conformation tensor, the no-flux condition is approximated at all boundaries. The Reynolds number is defined as $Re = U_\infty D p / \mu$ and fixed at $Re = 100$. The Weissenberg number is defined as $We = \lambda U_\infty D / \mu$, which ranges from 0 to 10. The computational grid is generated by the commercial software ICEM. As shown in Fig. 27, the surrounding region of each cylinder is discretized by the O-type mesh. The rest of the computational domain is discretized by several blocks of rectangular meshes, with the dense mesh near the cylinder and the course mesh near the domain boundaries. When $L_D = 0.1D$, one block of the O-type mesh consists of 51 grid points uniformly distributed along the cylinder perimeter and 250 grid points stretched over an exponential progression along the radial direction to ensure a fine mesh near the cylinder surface. In this study, the size of the first cell adjacent to the cylinder surface in the radial direction is set to $0.005D$. In the $x$ direction, 501 grid points (for $L_x$) are unevenly arranged in the downstream region, and 71 grid points (for $L_u$) are set in the upstream region. The total number of meshes for the computational domain is approximately 244, 300.

The total lift coefficient ($C_l$) for the eight equal side-by-side circular cylinders is calculated by the following equations:

$$C_l = \frac{F_y}{4\rho U_\infty^2 D},$$ (11)
where $F_y$ is force acted on the total circular cylinders in $y$ direction.

The drag coefficient ($C_x$) and lift coefficient ($C_y$) for each circular cylinders is calculated by the following equations:

$$C_x = \frac{2F_x}{\rho U_x^2 D}, \quad \text{and} \quad C_y = \frac{2F_y}{\rho U_x^2 D},$$

(12)

where $F_x$ and $F_y$ are force acted on each circular cylinder in $x$ and $y$ directions, respectively.

The force coefficient along the cylinder center’s line is calculated:

$$C_\alpha = -C_x \cos(\pi/9) + C_y \sin(\pi/9).$$

(13)

This force coefficient can reflect the interaction force between cylinders through the action of fluid medium.

**Fig. 27.** (a) The schematic of unconfined flow over eight equal side-by-side circular cylinders placed inclined for $L_D = 0.1D$. (b) The near-wall view of the mesh.

**Fig. 28.** The time average lift coefficients.

The time average lift coefficient ($\bar{C}_l$) of the eight cylinders at various $\alpha$ and $We$ in Newtonian and viscoelastic fluid are plotted in Fig. 28. In Newtonian fluid, $\bar{C}_l$ is 1.099. In viscoelastic fluid, $\bar{C}_l$ gradually decreases with an increase in $We$ when $We$ is higher than 0.5. The magnitude of lift could reflect the degree of asymmetric distribution of reaction force of the reflect flat plate upper and lower surfaces. The decreasing $\bar{C}_l$ indicates that the flow up and down asymmetry is
weakened near the cylinder’s wall. Moreover, the interaction force between cylinders through the action of fluid medium, as list in Tab. 2, which is similar to viscoelastic flow over two side-by-side circular cylinders reported in our previous publication. \(^{33}\) This is mainly due to the asymmetry of velocity distribution on the upper and lower sides of each cylinder.

| Cylinders | Cy-1 | Cy-2 | Cy-3 | Cy-4 | Cy-5 | Cy-6 | Cy-7 | Cy-8 |
|-----------|------|------|------|------|------|------|------|------|
| Newtonian | -6.499 | -1.220 | -1.073 | -0.950 | -0.818 | -0.652 | -0.419 | 0.288 |
| \(We = 0.1\) | -6.586 | -1.228 | -1.077 | -0.957 | -0.829 | -0.670 | -0.416 | 0.282 |
| \(We = 0.2\) | -6.464 | -1.240 | -1.072 | -0.949 | -0.818 | -0.652 | -0.415 | 0.277 |
| \(We = 0.5\) | -6.438 | -1.215 | -1.070 | -0.948 | -0.816 | -0.649 | -0.412 | 0.274 |
| \(We = 1\) | -6.162 | -1.212 | -1.057 | -0.934 | -0.799 | -0.629 | -0.393 | 0.260 |
| \(We = 2\) | -5.638 | -1.123 | -0.999 | -0.881 | -0.751 | -0.590 | -0.375 | 0.178 |
| \(We = 4\) | -4.984 | -0.971 | -0.871 | -0.749 | -0.614 | -0.463 | -0.357 | -0.113 |
| \(We = 6\) | -4.329 | -0.813 | -0.777 | -0.665 | -0.546 | -0.417 | -0.368 | -0.192 |
| \(We = 8\) | -3.943 | -0.735 | -0.668 | -0.603 | -0.497 | -0.378 | -0.315 | -0.247 |
| \(We = 10\) | -3.754 | -0.678 | -0.630 | -0.571 | -0.474 | -0.374 | -0.294 | -0.269 |

**Note:** Cylinders are numbered 1 to 8 from top to bottom in Fig. 27(a). The negative sign indicates the positive direction of the \(x\) axis.

**Fig. 29.** The instantaneous vorticity contours. \(\omega_z\) is normalized by \(\frac{U_\infty}{D}\).

**Fig. 30.** The typical instantaneous positions of vortex cores for Newtonian and viscoelastic fluids. The moments correspond to Fig. 29.
The instantaneous vorticity contours and the time average streamlines are shown in Fig. 29 and Fig. 30, respectively. The vortex core centers are extracted and drawn in Fig. 31. For both Newtonian and viscoelastic fluids, the flow resembles the flow around a single cylinder. In Newtonian fluid, the trajectory of vortex core center deviates from the centerline (dotted line in the figure) of the flow field obviously. In viscoelastic fluid, the vortex core center gradually concentrates on the center of the flow field as We increase when We is over 0.5. At the same time, the time-averaged wake bubbles are elongated and tend to the center line of the flow field.

Fig. 31. The time average streamlines.

Fig. 32. The instantaneous distributions of (a) $c_{xx}$ and (b) $c_{yy}$ for viscoelastic flow over eight equal side-by-side circular cylinders placed inclined closely at $L_D = 0.1D$ and $We = 10$.

Fig. 33. The distribution of time-averaged flow velocity magnitude ($\sqrt{u'^2 + v'^2}/U_{in}$) field for (a) Newtonian fluid and (b) viscoelastic fluid with $We = 10$ and $L = 100$, which is dimensionless with
incoming flow velocity ($U_\infty$). Isogram of $\sqrt{u^2 + v^2}/U_\infty = 0.1$ is marked with pink line in panel (b).

For viscoelastic flow over eight equal side-by-side circular cylinders placed inclined closely, the elastic stress is concentrated near the front edge of each cylinder, connects into one piece (similar to viscoelastic flow over side-by-side circular cylinders), and extends along the outer side of each cylinder towards the downstream of the cylinders, as shown in Fig. 32. The corresponding time-averaged flow velocity magnitude ($\sqrt{u^2 + v^2}/U_\infty$) distribution is shown in Fig. 33(b), which is compared with that of Newtonian fluid, as shown in Fig. 33(a). The isogram of $\sqrt{u^2 + v^2}/U_\infty = 0.1$ with pink line is marked in the figure. The flow velocity in its surrounding area is lower than this value (0.1), which could be regarded as a whole blunt body. The whole blunt body is symmetrical along the center line of the flow field, which results in symmetrical wake shedding trajectory along the flow filed center line. The velocity on the upper and lower sides of each cylinder is also asymmetrically weakened for low velocity distribution in the gap between cylinders, which results to low interaction force between cylinders through the action of fluid medium ($C_d$).

E. Common feature for these four serial cases

A common feature can be summarized from the above stress distributions for the four examples, i.e., an region with high elastic stress similar to shock wave appears near the frontal edge of the blunt body and extends downstream, as sketched by the red solid lines in Fig. 34. This region acts as a stationary shield to divide the flow into different regimes. Thus, the free stream resembles to pass this shield instead of the original blunt body. As this region is nearly symmetric against the horizontal center line of the corresponding flow configuration, the wake flow recovers symmetry. Specially, another high elastic stress regions surrounds the rotating circular, which makes the velocity distribution in the rotating boundary layer uniform along the circumferential direction. The whole rotating boundary layer behaves like a larger non-rotating cylinder (dotted line in Fig. 34(b)), resulting a symmetry downstream wake filed.
Fig. 34. Schematic diagram of regions with high elastic stress (the blue solid lines): (a) flow over an inclined flat plate with various angles of incidence, (b) flow over a rotating circular cylinder, (c) flow over a circular cylinder with asymmetrical slip boundary distribution, and (d) flow over eight equal side-by-side circular cylinders placed inclined closely.

IV. Conclusion Remarks

In this paper, we report that viscoelastic wake flow asymmetry in a free domain is no longer sensitive to asymmetry in geometry, boundary condition, or multi-body asymmetric distribution, through numerical simulation. Two-dimensional direct numerical simulations based on the FENE-P model (the finite-extensible nonlinear elastic model with the Peterlin closure) are conducted. Four asymmetric flows are considered in this study, i.e., flow over an inclined flat plate with various angles of incidence at $Re = 500$, flow over a rotating circular cylinder at $Re = 100$, flow over a circular cylinder with asymmetrical slip boundary distribution at $Re = 500$ and flow over eight equal side-by-side circular cylinders placed inclined closely at $Re = 100$. The four flows are the representative of geometric asymmetry, boundary condition asymmetry, and multi-body asymmetric distribution, respectively.

The lift coefficient can be used to describe the asymmetry degree of the flow field near the blunt body. The lift coefficient acted on blunt body in viscoelastic fluid becomes lower than that in Newtonian fluid. Accordingly, the viscoelastic wake flow tends to be symmetric with respect to the horizontal center line of the flow configuration for the inclined flat plate with an angle of attack, the rotating circular cylinder, a circular cylinder with asymmetrical slip boundary distribution or eight equal side-by-side circular cylinders placed inclined closely.

A careful examination of the elastic stress distributions of the four examples reveals a common feature, i.e., an region with high elastic stress similar to shock wave appears near the frontal edge of the blunt body and extends all the way downstream. The blunt body seems to be wrapped by this region, which retains a symmetric shape with respect to the horizontal center line of the flow configuration. The free stream cannot directly interact with the blunt body but flows around this region, as if the free stream passes an imaginary symmetric blunt body with larger characteristic length. Thus, the wake flow recovers symmetry. Specially, another high elastic stress regions surrounds the rotating circular, which makes the velocity distribution in the rotating boundary layer uniform along the circumferential direction. The whole rotating boundary layer behaves like a larger non-rotating cylinder, resulting a symmetry downstream wake filed.

The influence of elastic stress distribution on lift and flow symmetry discovered in this paper could be used for precise flow control. It would be interesting to investigate how to obtain the desired lift and vortex shedding trajectory by controlling the polymer concentration distribution around a hydrofoil to redistribute elastic stress in future. The flow asymmetry could cause higher flow-induced vibration response for a rotating circular cylinder. Xiong et al. proposed that polymer addition could restrain vortex-induced vibration of a non-rotating cylinder. It is reasonable to speculate that polymer addition could also effectively suppress vortex-induced vibration of a rotating circular cylinder, due to the flow asymmetry weakening discussed above, which entails future investigations.
Data Availability Statement

The data that support the findings of this study are available from the corresponding author upon reasonable request.

Acknowledgement

The author P. Yu would like to thank the financial support from Guangdong Provincial Key Laboratory of Turbulence Research and Applications (Grant No. 2019B21203001), the National Natural Science Foundation of China (NSFC, Grant Nos. 12172163, 12071367). This work is supported by Center for Computational Science and Engineering of Southern University of Science and Technology. The author Sai Peng would like to thank Dr. Jun-Qiang Zhang from Huazhong University of Science and Technology for useful discussion.

Reference

[1] Williamson, C. H. (1989). Oblique and parallel modes of vortex shedding in the wake of a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics, 206, 579-627.
[2] Gabbai, R. D., & Benaroya, H. (2005). An overview of modeling and experiments of vortex-induced vibration of circular cylinders. Journal of sound and vibration, 282(3-5), 575-616.
[3] Inoue, O., & Hatakeyama, N. (2002). Sound generation by a two-dimensional circular cylinder in a uniform flow. Journal of Fluid Mechanics, 471, 285-314.
[4] Qu, L., Norberg, C., Davidson, L., Peng, S. H., & Wang, F. (2013). Quantitative numerical analysis of flow past a circular cylinder at Reynolds number between 50 and 200. Journal of Fluids and Structures, 39, 347-370.
[5] Tritton, D. J. (1959). Experiments on the flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics, 6(4), 547-567.
[6] Perumal, D. A., Kumar, G. V., & Dass, A. K. (2012). Numerical simulation of viscous flow over a square cylinder using lattice Boltzmann method. International Scholarly Research Notices, 2012.
[7] Kouser, T., Xiong, Y., Yang, D., & Peng, S. (2021). Direct Numerical Simulations on the three-dimensional wake transition of flows over NACA0012 airfoil at Re= 1000. International Journal of Micro Air Vehicles, 13, 17568293211055656.
[8] Kang, S. (2003). Characteristics of flow over two circular cylinders in a side-by-side arrangement at low Reynolds numbers. Physics of Fluids, 15(9), 2486-2498.
[9] Stojković, D., Schön, P., Breuer, M., & Durst, F. (2003). On the new vortex shedding mode past a rotating circular cylinder. Physics of Fluids, 15(5), 1257-1260.
[10] Li, D., Li, S., Xue, Y., Yang, Y., Su, W., Xia, Z., ... & Duan, H. (2014). The effect of slip distribution on flow past a circular cylinder. Journal of Fluids and Structures, 51, 211-224.
[11] Sahin, M., & Owens, R. G. (2004). A numerical investigation of wall effects up to high blockage ratios on two-dimensional flow past a confined circular cylinder. Physics of fluids, 16(5), 1305-1320.
[12] Buford, W. E. (1954). Magnus Effect in the Case of Rotating Cylinders and Shell (No. 821). Aberdeen Proving Ground.
[13] White, C. M., & Mungal, M. G. (2008). Mechanics and prediction of turbulent drag reduction with polymer additives. Annu. Rev. Fluid Mech., 40, 235-256.
[14] Kouser, T., Xiong, Y., & Yang, D. (2021). Contribution of Superhydrophobic Surfaces and Polymer Additives to Drag Reduction. ChemBioEng Reviews.

[15] Xiong, Y., Peng, S., Zhang, M., & Yang, D. (2019). Numerical study on the vortex-induced vibration of a circular cylinder in viscoelastic fluids. Journal of Non-Newtonian Fluid Mechanics, 272, 104170.

[16] Karathanassis, I. K., Trickett, K., Koukouvinis, P., Wang, J., Barbour, R., & Gavaises, M. (2018). Illustrating the effect of viscoelastic additives on cavitation and turbulence with X-ray imaging. Scientific reports, 8(1), 1-15.

[17] Naseri, H., Trickett, K., Mitrogoulou, N., Karathanassis, I., Koukouvinis, P., Gavaises, M., ... & Wang, J. (2018). Turbulence and cavitation suppression by quaternary ammonium salt additives. Scientific reports, 8(1), 1-15.

[18] Li, G. D., Deng, S. S., Guan, J. F., & Yao, S. (2019). Numerical analysis on cavitation effects in submerged water jet added with turbulent drag-reducing additives of CTAC. Chemical Engineering Science, 196, 391-401.

[19] Azad, S., Lotfi, H., & Riasi, A. (2019). The effects of viscoelastic fluid on the cavitation inception and development within a centrifugal pump: An experimental study. International Communications in Heat and Mass Transfer, 107, 106-113.

[20] Raei, B., Shahrai, F., & Peyghambarzadeh, S. M. (2018). Experimental study of the effect of drag reducing agent on heat transfer and pressure drop characteristics. Experimental Heat Transfer, 31(1), 68-84.

[21] Minaeian, A., Nili-Ahmadabadi, M., & Norouzi, M. (2020). Forced convective heat transfer of nonlinear viscoelastic flows over a circular cylinder at low Reynolds inertia regime. Communications in Nonlinear Science and Numerical Simulation, 83, 105134.

[22] Sahin, Ç., & Atalık, K. (2019). Effects of polymer/surfactant additives on forced convective heat transfer in vortex shedding flow past a circular cylinder. International Journal of Thermal Sciences, 145, 106031.

[23] Traore, B., Castelain, C., & Burghelea, T. (2015). Efficient heat transfer in a regime of elastic turbulence. Journal of Non-Newtonian Fluid Mechanics, 223, 62-76.

[24] Yao, G., Zhao, J., Shen, X., Yang, H., & Wen, D. (2020). Effects of rheological properties on heat transfer enhancements by elastic instability in von-Karman swirling flow. International Journal of Heat and Mass Transfer, 152, 119535.

[25] Tatsumi, K., Nagasaka, W., Kimura, R., Shinotsuka, N., Kuriyama, R., & Nakabe, K. (2019). Local flow and heat transfer characteristics of viscoelastic fluid in a serpentine channel. International Journal of Heat and Mass Transfer, 138, 432-442.

[26] Haward, S. J., Toda-Peters, K., & Shen, A. Q. (2018). Steady viscoelastic flow around high-aspect-ratio, low-blockage-ratio microfluidic cylinders. Journal of Non-Newtonian Fluid Mechanics, 254, 23-35.

[27] Haward, S. J., Hopkins, C. C., & Shen, A. Q. (2020). Asymmetric flow of polymer solutions around microfluidic cylinders: Interaction between shear-thinning and viscoelasticity. Journal of Non-Newtonian Fluid Mechanics, 278, 104250.

[28] Haward, S. J., Hopkins, C. C., Varchanis, S., & Shen, A. Q. (2021). Bifurcations in flows of complex fluids around microfluidic cylinders. Lab on a Chip.

[29] Poole, R. J., Alves, M. A., & Oliveira, P. J. (2007). Purely elastic flow asymmetries. Physical Review Letters, 99(16), 164503.
[30] Arratia, P. E., Thomas, C. C., Diorio, J., & Gollub, J. P. (2006). Elastic instabilities of polymer solutions in cross-channel flow. Physical review letters, 96(14), 144502.
[31] Oliveira, P. J. (2003). Asymmetric flows of viscoelastic fluids in symmetric planar expansion geometries. Journal of Non-newtonian Fluid Mechanics, 114(1), 33-63.
[32] Xiong, Y., Peng, S., Yang, D., Duan, J., & Wang, L. (2018). Influence of polymer additive on flow past a hydrofoil: A numerical study. Physics of Fluids, 30(1), 013104.
[33] Peng, S., Xiong, Y. L., Xu, X. Y., & Yu, P. (2020). Numerical study of unsteady viscoelastic flow past two side-by-side circular cylinders. Physics of Fluids, 32(8), 083106.
[34] Kurtulus, D. F. (2015). On the unsteady behavior of the flow around NACA 0012 airfoil with steady external conditions at Re = 1000. International Journal of Micro Air Vehicles, 7(3), 301-326.
[35] Liu, Y., Li, K., Zhang, J., Wang, H., & Liu, L. (2012). Numerical bifurcation analysis of static stall of airfoil and dynamic stall under unsteady perturbation. Communications in Nonlinear Science and Numerical Simulation, 17(8), 3427-3434.
[36] Norouzi, M., Varedi, S. R., & Zamani, M. (2016). Wake instability of viscoelastic flows past an unconfined inclined square cylinder. Physics of Fluids, 28(2), 023101.
[37] Gao, Y., Yu, D., Tan, S., Wang, X., & Hao, Z. (2010). Experimental study on the near wake behind two side-by-side cylinders of unequal diameters. Fluid dynamics research, 42(5), 055509.
[38] Sumner, D., Price, S. J., & Paidoussis, M. P. (2000). Flow-pattern identification for two staggered circular cylinders in cross-flow. Journal of Fluid Mechanics, 411, 263-303.
[39] Richter, D., Iaccarino, G., & Shaqfeh, E. S. (2010). Simulations of three-dimensional viscoelastic flows past a circular cylinder at moderate Reynolds numbers. Journal of Fluid Mechanics, 651, 415-442.
[40] Richter, D., Shaqfeh, E. S., & Iaccarino, G. (2011). Numerical simulation of polymer injection in turbulent flow past a circular cylinder. Journal of fluids engineering, 133(10).
[41] Peng, S., Tang, T., Li, J., Zhang, M., & Yu, P. (2022). Numerical Study of Viscoelastic Upstream Instability. arXiv preprint arXiv:2203.09239.
[42] Bird, R. B., & Wiest, J. M. (1995). Constitutive equations for polymeric liquids. Annual review of fluid mechanics, 27(1), 169-193.
[43] P-T. Nhan, Understanding Viscoelasticity (first ed.), Springer, New York (2002).
[44] Page J., Dubief Y., & Kerswell R. R. (2020). Exact traveling wave solutions in viscoelastic channel flow, Physical Review Letters 125, 154501.
[45] Dubief, Y., Page, J., Kerswell R. R., Terrapon, V. E., & Steinberg, V. (2020). A first coherent structure in elasto-inertial turbulence. arXiv preprint arXiv:2006.06770.
[46] Sid, S., Terrapon, V. E., & Dubief, Y. (2018). Two-dimensional dynamics of elasto-inertial turbulence and its role in polymer drag reduction. Physical Review Fluids, 3(1), 011301.
[47] Oliveira, P. J. (2002). An exact solution for tube and slit flow of a FENE-P fluid. Acta Mechanica, 158(3), 157-167.
[48] Yang, D., Pettersen, B., Andersson, H. I., & Narasimhamurthy, V. D. (2013). Floquet stability analysis of the wake of an inclined flat plate. Physics of Fluids, 25(9), 094103.
[49] Bourguet, R., & Jacono, D. L. (2014). Flow-induced vibrations of a rotating cylinder. Journal of Fluid Mechanics, 740, 342-380.
[50] Navier, L. M. H. (1823). Mémoire sur les lois du mouvement d. fluids. Mémoires de l'Acad. Royale des Sci. de l'Inst. France, 6, 389-440.
[51] Legendre D, Lauga E, Magnaudet J. Influence of slip on the dynamics of two-dimensional wakes. Journal of fluid mechanics, 2009, 633: 437-447.