

UC San Diego

UC San Diego Previously Published Works

Title

Locomotion of a tank-treading-inspired swimmer in a pipe

Permalink

<https://escholarship.org/uc/item/8j7826qs>

Journal

Journal of Fluids and Structures, 142

ISSN

0889-9746

Author

Zhu, Qiang

Publication Date

2026-04-01

DOI

10.1016/j.jfluidstructs.2026.104523

Peer reviewed



Locomotion of a Tank-Treading-Inspired Swimmer in a Pipe

Qiang Zhu

Department of Structural Engineering, University of California San Diego, La Jolla, California 92093-0085, USA.

Abstract

A computational model has been developed to simulate the dynamics and energetics of a novel locomotion method inside a pipe during straightline swimming. This method relies on shear stress on the solid surface for force generation. Inspired by the tank-treading responses of red blood cells and vesicles in shear flow, this swimmer features a barrel-shaped body with an open-ended chamber surrounded by a wall. The wall is made of a flexible membrane enwrapping fluid inside. During operation, the body geometry remains unchanged while the membrane circulates in a tank-treading style, generating a net force through the shear stress exerted by the surrounding fluid on it. The fluid-structure interaction problem is simulated with a model based on the immersed-boundary approach and a finite-difference algorithm to solve the axisymmetric Navier-Stokes equations. Systematic simulations have been conducted with various pipe sizes and the Reynolds number ranging from 1 to 200. The results indicate that in comparison with the performance of the swimmer in open flow, the propulsive force it generates is enhanced inside the pipe due to the hydrodynamic interaction with the inner wall of the pipe, whereas the swimming speed is reduced as a result of the increased drag. Nevertheless, since the drag increment is partially compensated by the thrust enhancement, the speed reduction inside the pipe is mitigated. It also causes minimal disturbance to the flow field. Two parameters have been used to characterize the energy efficiency of this system, the propulsive efficiency and the cost of transport. Furthermore, examination of the energy dissipation in the flow suggests that the energy efficiency of the system may be significantly improved by mitigating energy dissipation in the interior flow field.

1. Introduction

In recent years, robots have been developed to perform tasks such as inspection, maintenance and repair in pipelines [1]. To navigate and maneuver inside pipes, various propulsion mechanisms have been proposed and tested, including passive motion driven by pressure gradient [2], walking with multiple-degrees of freedom legs [3], moving on the inner wall of the pipe through wheels or tracked wheels [4, 5], and inchworm-type motion that relies on the relative motion between its front and rear modules [6]. Compared with these traditional designs, swimming robots, often inspired by living creatures such as fish, offer unique advantages in terms of their maneuverability and reduced footprint as they do not have direct contact with the pipe [7, 8].

Micro- to nano-scale robots are also used for biomedical purposes. These robots, propelled magnetically or acoustically, travel through blood vessels to perform tasks such as clot removal, drug delivery or micro surgery [9, 10]. The working conditions of these robots are often more complicated as they have to swim in highly unsteady flow fields with time-dependent boundary conditions.

For the aforementioned industrial and biomedical applications, it is necessary to develop robots capable of swimming efficiently in pipes with pronounced boundary effect. Towards this end, the performance of various propulsion

Email address: qizhu@ucsd.edu (Qiang Zhu)

methods (both conventional and unconventional) in pipes must be investigated. In the current study we concentrate on the locomotion capacity of a novel swimming method inspired by tank-treading responses of red blood cells inside a pipe. This recently proposed method relies mostly on shear stress rather than pressure for force generation [11].

Stemming from the hypothetical creature called the ‘Purcell’s toroidal swimmer’ [12], shear-driven swimming is an unconventional locomotion method in which the momentum exchange between the swimmer and the fluid environment occurs mostly in the shear direction. As suggested by its name, the original design of this swimmer is characterized by a torus-shaped body. Unlike bioinspired swimming methods that rely on deformations of the body and appendages, this swimming mode does not involve any shape change in the body itself. Instead, the outer surface of the body circulates in the poloidal direction similar to the motion of a smoke ring.

In the Stokes flow regime, theoretical analyses based on asymptotic approach [13, 14] have been conducted [15, 16]. According to these investigations, when the ratio between the major radius and minor radius of the torus is reduced, the swimming speed can be increased. In the experimental side, a torus-shaped swimmer made of liquid crystal elastomer has been recently developed and tested [17]. This system absorbs energy from light for activation. To make sure that the inertia effect is negligibly small, in this experiment a high-viscosity material (polydimethylsiloxane, or PDMS) was used as the surrounding fluid to compensate for the fact that the size of the swimmer was not sufficiently small.

At finite Reynolds numbers, the performance of the toroidal swimmer has also been explored through numerical studies [18, 19]. These studies demonstrate the existence of two locomotion modes with opposite swimming directions, shear-generated motion and jet-generated motion. Depending on physical parameters such as the aspect ratio of the body and the Reynolds number, the actual swimming activity of the system is dominated by one of these modes. Another numerical investigation has been conducted to illustrate the locomotion performance of a pair of counter-rotating cylinders [20]. Although this study is strictly two dimensional, it bears an uncanny similarity to the toroidal swimmer in geometry, kinematics, and underlying physics. This two-dimensional swimmer also displays the ability of swimming in multiple directions driven by the shear stress and the repulsive force from the jet it creates.

Due to the blunt-body design of the toroidal swimmer, significant drag can be generated during forward motion. The peak swimming speed of such a system was found to be 60-70% of the speed of the skin (hereafter referred to as the membrane circulation speed). In a recent study, we have proposed an innovative design featuring a streamlined body for drag reduction and increased surface area parallel to the swimming direction to enhance generation of the propulsive force via shear stress [11]. The internal structure of this design resembles that of a red blood cell - it includes a highly deformable membrane enveloping fluid inside. It is propelled by circulatory motion of its membrane, which is inspired by the tank-treading response of red blood cells [21]. Initial tests have demonstrated that this tank-treading-inspired swimmer achieves a swimming speed of 80% of the membrane circulation speed, faster than the toroidal swimmer. It was also shown that this speed is insensitive to the Reynolds number. Compared with existing bio-inspired locomotion method using deformable bodies, e.g., swimmers inspired by fish, marine mammals, or cephalopods, this design has the following advantages:

1. Steady speed: Unlike existing designs in which the force generation is achieved by body undulation, fin flapping, or pulsed jetting, the swimming speed of this swimmer is steady as long as the circulation speed of its membrane remains unchanged.
2. Straightline swimming: Since there is no lateral force generation, the swimmer can swim along a perfectly straight path.
3. Reduced footprint: As the body is streamlined and the momentum exchange between it and the fluid occurs primarily in the shear direction, it is expected to leave minimal disturbance in the flow field. The noise generation may also be mitigated, although investigations in this direction is necessary to prove this.
4. Trans-medium capacity: This locomotion mechanism has the potential in working in various environments, e.g., crawling on ground or burrowing in granular media [22]. It is thus a promising locomotion method for trans-medium operations.

The purpose of the present study is to investigate the locomotion performance of the tank-treading-inspired swimmer inside a pipe. According to the Couette flow problem, the fluid shear stress on a plate is increased when it travels parallel to a flat solid boundary due to the increase in the shear rate near it. The implication is that when swimming inside a pipe, the thrust force on the tank-treading-inspired swimmer can be amplified due to its hydrodynamic interaction with the inner wall of the pipe. On the other hand, the drag force on the swimmer can also be increased since it

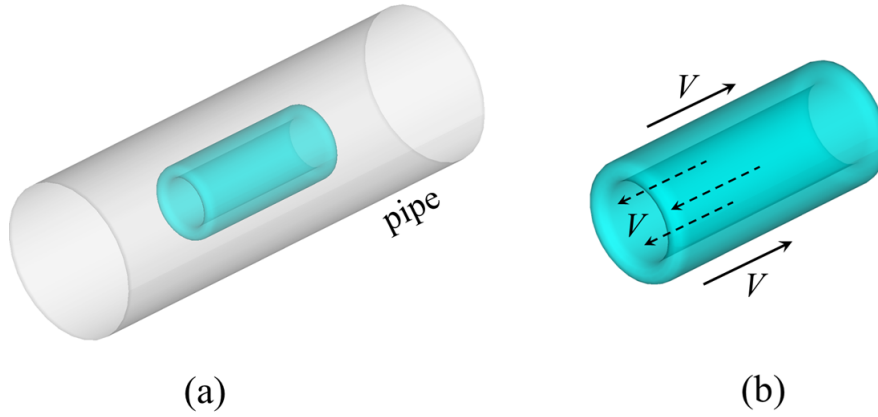


Figure 1. (a) The locomotion of a tank-treading-inspired swimmer inside a pipe. (b) The swimmer is propelled by the circulation motion of its membrane.

is harder for flow to go around it in a confined space. To quantitatively understand the net effect of the pipe wall on the swimming performance of this system, we will conduct systematic simulations to examine the force generation, swimming speed, and energy consumption of such a swimmer in a pipe.

The rest of the paper is organized as follows. The next section contains a depiction of the physical system to be studied, including the geometry and the kinematics of the system. This is followed by a brief description of the mathematical formulation and the numerical algorithm. The numerical results are then presented. Finally, conclusions are drawn.

2. Physical problem

As summarized in Fig. 1a, the problem includes a straight and uniform pipe with circular cross section. A tank-treading-inspired swimmer [11] moves along the central axis of the pipe. The swimmer is characterized by a barrel-shaped body with both ends open. The side wall of the barrel is made of a fluid (hereafter referred to as the interior fluid) enwrapped within a highly flexible membrane. For simplicity, the interior fluid is assumed to have the same physical properties (density ρ and dynamic viscosity μ) as the exterior fluid. The central axis of the swimmer overlaps the one of the pipe.

During locomotion, the geometry of the swimmer remains unchanged, whereas its membrane circulates around the wall in a manner illustrated in Fig. 1b. This membrane circulation is similar to the tank-treading response of red blood cells or vesicles in shear flows [21, 23, 24]. We assume that the circulation speed of the membrane is V at every location.

In an infinite and still flow field, the inner side membrane of the wall moves leftwards during circulation while the outer side membrane moves rightwards (Fig. 2). The shear forces exerted on the inner side membrane and the outer side membrane are thus in opposite directions. Within the parametric space considered in this study, the force on the outer side membrane is larger in magnitude due to the following reasons: 1) The outer side membrane has larger surface area due to the three-dimensional geometry. 2) The outer side membrane moves in an open flow fields, whereas the inner side membrane moves in a closed space (the chamber shown in the figure). Subsequently, the net force generated by fluid-structure interaction points towards left.

The problem is assumed to be axisymmetric and considered within a meridional plane (x, r) as shown in Fig. 3a. Here the longitudinal coordinate x overlaps with the axis of symmetry and r is the radial direction. Two scenarios are considered, tethered and free swimming. In the tethered scenario, the center of mass of the swimmer is not allowed to change. The focus is on the force-generation capacity and power expenditure of the swimmer in the confined space within the pipe. In the free-swimming scenario, on the other hand, the swimmer undergoes self-propelled motion inside the pipe. The swimming speed, power expenditure, and efficiency will be recorded for analysis. Figure

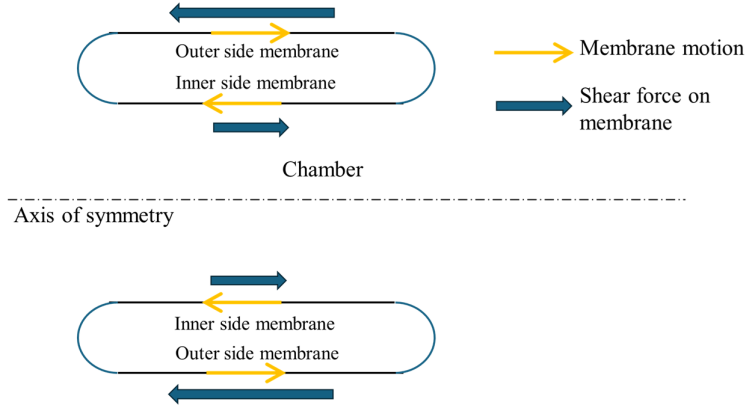


Figure 2. Schematic depiction of the membrane circulation and the subsequent shear stress on it due to the surrounding fluid.

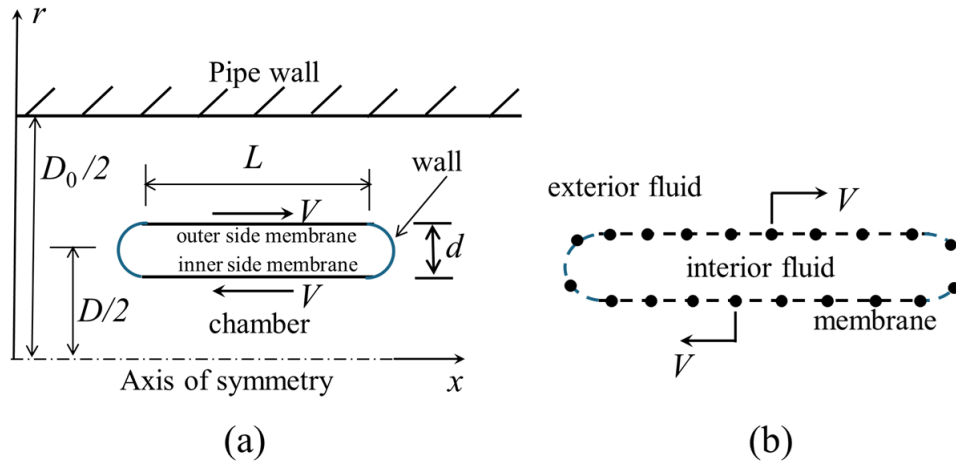


Figure 3. (a) Rendition of the physical problem in a meridional plane (x, r) . (b) The numerical achievement of the circulatory motion of the membrane in the immersed-boundary method.

3b shows that with the immersed-boundary formulation utilized in the current study, the membrane circulation is achieved by the prescribed motions of the structural grids (see §3.3 for more details).

The geometric parameters are also displayed in Fig. 3a. As shown in this figure, the body length is L , the diameter of the barrel is D , and the thickness of the wall is d (the two ends of the wall are closed with semicircles whose diameter is d). Correspondingly, the diameter of the chamber in the barrel is $D - d$. The diameter of the pipe is D_0 .

When activated, the geometry of the body remains unchanged, whereas the membrane circulates around the wall in a tank-treading manner with a constant speed V (see Figs. 1b, 2 and 3a). This circulatory motion transfers momentum into the surrounding flow field through shear stress and disturbs both the exterior and the interior flows. Although it costs energy, the motion of the interior fluid does not participate in thrust generation as the force it exerts on the membrane is an internal force. The propulsion effect is thus attributed to the momentum transferred to the exterior fluid, which includes two parts, the outer side flow and the inner side flow (i.e., flow in the chamber). In the outer side flow field, the membrane generates a flow in the $+x$ direction so that the force on the body is in the $-x$ direction (Fig. 2). This part of the axial force is called the shear-generated force, as it is exerted on the body mostly through the shear stress on the outer side membrane. In the inner side flow field, the jet flow created by the membrane motion is in the $-x$ direction. The repulsive force associated with this jet is thus in the $+x$ direction. Even though this repulsive force also acts through the shear stress on the membrane, it is called the jet-generated force to distinguish it from the one on

the outer side membrane. Previous studies on toroidal swimmers [18, 19, 20] suggest that the relative importance of these two forces depends on the Reynolds number and the size of the chamber - the jet-generated force dominates in cases with high Reynolds number and large chamber, otherwise the shear-generated force is dominant. We concentrate on cases in which the shear-generated force dominates and the swimming direction is $-x$. In this swimming mode, the jet-generated force points towards the opposite direction of swimming so that it needs to be mitigated for better performance.

For simplicity, it is assumed that the swimmer is neutrally buoyant and the pipe it swims in is horizontal so that no gravity effect is considered.

3. Mathematical formulations and numerical approach

3.1. Governing equations

Let $\mathbf{u} \equiv (u_x, u_r)$ be the flow velocity, p be the pressure, and $\mathbf{f} \equiv (f_x, f_r)$ be the force exerted by the solid object on the fluid, by invoking the immersed-boundary approach the fluid dynamics is governed by the axisymmetric Navier-Stokes equations as

$$\begin{aligned} \rho \left(\frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} + u_r \frac{\partial u_x}{\partial r} \right) + \frac{\partial p}{\partial x} - \mu \left[\frac{\partial^2 u_x}{\partial x^2} + \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_x}{\partial r} \right) \right] - f_x &= 0, \\ \rho \left(\frac{\partial u_r}{\partial t} + u_x \frac{\partial u_r}{\partial x} + u_r \frac{\partial u_r}{\partial r} \right) + \frac{\partial p}{\partial r} - \mu \left[\frac{\partial^2 u_r}{\partial x^2} + \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_r}{\partial r} \right) - \frac{u_r}{r^2} \right] - f_r &= 0, \\ \frac{\partial u_x}{\partial x} + \frac{1}{r} \frac{\partial (ru_r)}{\partial r} &= 0. \end{aligned} \quad (1)$$

In the immersed-boundary method, $\mathbf{f}$ is determined via the following relation through the Dirac delta function

$$\mathbf{f}(\mathbf{x}, t) = \int_{\Gamma} \mathbf{F}(s, t) \delta(\mathbf{X}(s, t) - \mathbf{x}) ds, \quad (2)$$

where $\mathbf{x} \equiv (x, r)$ is the location vector. Γ represents the contour where the membrane is located, with s being a Lagrangian coordinate along this contour. $\mathbf{F} \equiv (F_x, F_r)$ is the force exerted by the fluid on the solid object. $\mathbf{X}$ is the location of a point on the contour measured in the (x, r) coordinate system.

In this approach, $\mathbf{F}$ is obtained by enforcing the no-slip and no-flux conditions at the fluid-solid interface through a penalty method. Specifically, to prevent neighboring fluidic and structural particles from drifting away from each other, the fluid-structure interaction force is specified as

$$\mathbf{F}(s, t) = \alpha \int_0^t [\mathbf{U}(s, \tau) - \mathbf{V}(s, \tau)] d\tau + \beta [\mathbf{U}(s, t) - \mathbf{V}(s, t)], \quad (3)$$

where $\mathbf{V}$ is the structural velocity, and $\mathbf{U}$ is the fluid velocity at the fluid-structure interface. Again, by using the Dirac delta function $\mathbf{U}$ is evaluated as

$$\mathbf{U}(s, t) = \int_{\Omega} \mathbf{u}(\mathbf{x}, t) \delta(\mathbf{x} - \mathbf{X}(s, t)) d\mathbf{x}, \quad (4)$$

where Ω is the fluid domain. α and β are numerical parameters whose values have no effect on the results if they are sufficiently large. Based on numerical tests, the values of these two parameters are -10^5 and -50 , respectively.

3.2. Dynamic analysis

The swimming direction of the system is $-x$, thus the fluid dynamic force on the body in this direction is defined as the body force F_b . This force is obtained by integrating F_x (the component of $\mathbf{F}$ in the x direction) over the contour of the body, i.e.,

$$F_b = - \int_{\Gamma} 2\pi r(s) F_x(s, t) ds. \quad (5)$$

The power expenditure P is obtained as

$$P = \int_{\Gamma} 2\pi r \mathbf{F} \cdot \mathbf{U} ds. \quad (6)$$

In the free-swimming scenario, the forward motion x_b (defined as the displacement of the center of the mass in the $-x$ direction) is determined through Newton's law as

$$m_b \frac{d^2 x_b}{dt^2} = F_b, \quad (7)$$

where m_b is the mass of the swimming body. The simulation is carried out until the swimming speed $\dot{x}_b$ reaches a constant value U .

The body force F_b is usually considered as a superposition of thrust F_T (the force in the direction of swimming) and drag F_D (the force in the opposite direction of swimming), i.e., $F_b = F_T - F_D$. At the free-swimming speed U , the thrust force and the drag force balances each other so that $F_b = 0$.

A widely used measure to quantify the locomotion efficiency of a swimming object is the propulsive efficiency η , defined as $F_T U / P$. The difficulty in this approach is that in deformable swimmers with no clear distinction between their bodies and propellers, it is hard to accurately decompose thrust and drag from the net force on the body, which is often the only output from experiments or simulations. To resolve this issue, several approximate methods have been proposed [25], including:

1. The method based on the drag force on a towed rigid body: In this approach, the drag force on the swimmer which undergoes a towed motion at the same speed as its free-swimming speed without any deformation is used as the approximation of the drag in free swimming [26]. The accuracy of this method may be questionable if significant body deformation is involved in free swimming.
2. The method based on the direction of force components: This method separates the longitudinal force on the swimmer into two parts based on their origin, pressure force and viscous force. The time histories of these parts are obtained. At each time instant, if any one of these is in the swimming direction, it contributes to thrust. Otherwise, it contributes to drag [27]. This method cannot be used in cases in which the force on the body is steady.
3. The method based on the physical origin of a force component: This method was developed for a jet-propelled swimmer [28]. By utilizing control-volume analysis, the components of the jet-related forces are found to be the jet momentum flux, the normal stress at the exit plane, and the flow acceleration inside the body. The thrust force is obtained by summarizing these components. It is still not clear if this approach can be used for a wider range of swimmers.

In the problem we study, there is no geometric deformation of the body. The only change on the body is that its membrane circulates in a tank-treading manner. We thus employ the first approach to obtain the thrust force and the propulsive efficiency η . In our model, the drag force F_D in the free-swimming scenario is calculated as the net longitudinal force on the swimmer when the circulation is terminated and the swimmer is towed at the free-swimming speed U .

An alternative parameter to measure the energy efficiency of a swimmer is the *Cost of Transport*. The classical definition of this parameter is $P/(m_b g U)$, where g is the gravitational acceleration [29]. For generality, the present study will be conducted in a dimensionless way, in which the normalized value of g is not known until the exact values of the length scale and the time scale are determined. Besides, the body mass m_b is not going to be changed in our simulations. For this reason, we use a slightly modified definition of Cost of Transport, $CoT \equiv P/U$.

3.3. Numerical method

The fluid-structure interaction problem formulated in §3.1 is solved in a rectangular computational domain in the $x - r$ space (Fig. 4). The bottom side of this domain corresponds to the axis of symmetry $r = 0$. The top side of the domain is the inner wall of the pipe. The boundary conditions for flow velocity $\mathbf{u}$ and pressure p are all included Fig. 4.

A finite difference algorithm has been developed to solve the differential equations in §3.1. This method includes a second-order method for spatial discretization. For high resolution, near the solid body a denser mesh is used with grid sizes of Δx and Δr in x and r directions, respectively. Out of this zone a coarser mesh is used for computational efficiency. The Crank-Nicholson method is employed for time integration. The time step is Δt .

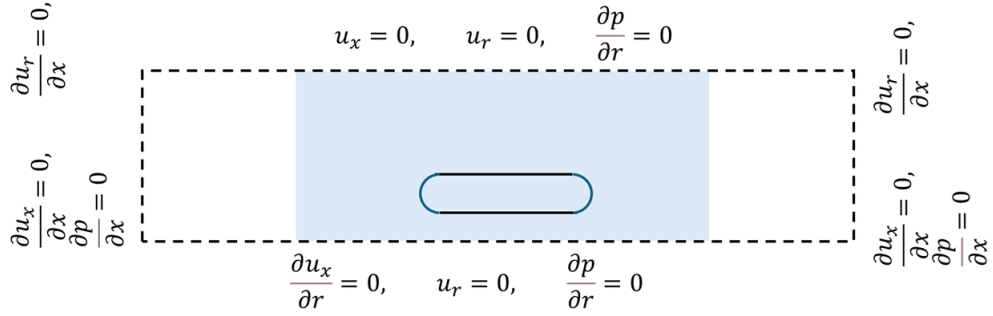


Figure 4. The computational domain and boundary conditions. The shaded area represents the region where a denser mesh is used.

Name	$\Delta x (= \Delta r)$	number of structural grids
coarse	4.8×10^{-3}	730
medium	3×10^{-3}	1167
fine	2×10^{-3}	1751

Table 1. Properties of different computational meshes.

In the immersed-boundary approach, the solid structure (hereby the zero-thickness membrane) is represented by isolated grids. The grid size in the s direction is Δs . To achieve the circulatory motion of the membrane, these grids circulates along the contour of the membrane in a prescribed motion at the given speed V (see Fig. 3b). This drives the motion of both the exterior and the interior fluids.

Detailed descriptions of the numerical method as well as validations can be found in previous publications [30, 31, 32]. For example, this method has been used to study the dynamics of flow around a circular cylinder [31]. The predictions closely match benchmark results from literature. Further validations have been conducted in our recent study with a configuration similar to the one used here [11]. In that investigation, the numerical model has been used to find the drag force on a fixed toroidal object in a uniform flow field. The predictions agree perfectly with the results reported in literature [33]. Numerical sensitivity tests with various configuration and kinematics have also been conducted with satisfying results, indicating the validity and accuracy of this method. Additional sensitivity tests will be conducted in the following section for the choice of numerical parameters.

4. Results

The problem is normalized by using the fluid density ρ , the body length L , and the speed of membrane circulation V as the repeating variables. This way, in the following results the lengths are normalized by L , the speed by V , the forces by $\rho L^2 V^2$, the power and the energy dissipation rate by $\rho L^2 V^3$, and the vorticity by $L^{-1} V$. For convenience, we do not change the symbols of the variables defined earlier, although hereafter they are all dimensionless.

The important parameters in this problem are: a) the body diameter D , b) the wall thickness d , c) the pipe diameter D_0 , and d) the Reynolds number $\text{Re} \equiv \rho V L / \mu$. In this study, we focus on a case in which $D = 0.4$ and $d = 0.2$. The range of D_0 is between 0.9 and 1.8. Four values of Reynolds number between 1 and 200 are selected.

4.1. Numerical sensitivity test

To further prove the validity of the numerical method, and to choose the numerical parameters for the following simulation, we carry out a numerical sensitivity test by choosing three different computational meshes (coarse, medium and refined) with properties detailed in Table 1 while keeping the length of the computational domain as 10 and the length of the zone with denser mesh as 4. Two time steps, $\Delta t = 1 \times 10^{-4}$ and $\Delta t = 5 \times 10^{-5}$, are also examined. A representative case in the tethered swimming scenario with $\text{Re} = 100$ and $D_0/D = 0.6$ is used for this test.

	$\Delta t = 1 \times 10^{-4}$	$\Delta t = 5 \times 10^{-5}$
coarse	0.517	0.516
medium	0.501	0.497
Refined	0.493	0.492

Table 2. Sensitivity of the predicted body force F_b to computational mesh and time step.

	$\Delta t = 1 \times 10^{-4}$	$\Delta t = 5 \times 10^{-5}$
coarse	0.826	0.820
medium	0.808	0.802
Refined	0.798	0.794

Table 3. Sensitivity of the predicted power expenditure P to computational mesh and time step.

Tables 2 and 3 show the variations of the body force F_b and power expenditure P predicted with different numerical parameters. It is seen that these numerical results are not sensitive to both the computational mesh and the time step. Other tests have been performed to check the sensitivity of the results to the sizes of the computational domain and the zone with high mesh density. It has been discovered that these factors have almost no effect on the results so that these tests are not presented here.

Balancing numerical accuracy and computational efficiency, in the following simulations we choose the medium mesh and $\Delta t = 1 \times 10^{-4}$. According to the sensitivity tests, this set of numerical parameters may lead to errors less than 2% in F_b and P . In the tethered scenario, the length of the computational domain is chosen to be 10 and the length of the denser mesh zone is 4. In the free-swimming scenario, to prevent the swimmer from exiting the computational zone, the length of the computational domain is increased to 18 and the length of the denser mesh zone is increased to 9.

4.2. Tethered scenario

In the tethered scenario the forward motion of the swimmer is disabled so that it is fixed at the center of the computational domain. Meanwhile, the membrane circulates around in the given speed so that a body force F_b determined by Eq. (5) is generated.

Figure 5 demonstrates the predicted values of F_b and the power expenditure P in this mode as functions of the pipe size represented by the ratio D_0/D at four values of Re , 1, 10, 100, and 200. The corresponding values of F_b and P when the swimmer operates in an open flow without boundary are also plotted for reference. These results are obtained by using the approach reported in our recent publication [11].

Comparing the results at $Re = 1$ and 10, it is seen that both F_b and P are almost linearly proportional to $1/Re$. This is explained by the fact that within this range of Reynolds number, when other parameters are fixed the shear stress on a flat plate in a flow parallel to it is proportional to the viscosity μ [34]. At higher Reynolds numbers, however, the dependence of F_b (or P) upon Re becomes more complicated as the relation between shear stress and μ drifts towards the Blasius solution. This is reflected in the cases at $Re = 100$ and 200 in Fig. 5.

The most pronounced feature in Fig. 5 is that both F_b and P increase in narrower pipes with smaller D_0 . For physical insight, in Fig. 6a we plot the longitudinal flow velocity (u_x) profiles at a cross section that separate the swimmer into two equal parts (referred to as the ‘center line’ in the caption of the figure) at two representative values of D_0/D , 4.5 and 2.25. It is seen that above the outer side membrane, at $D_0/D = 2.25$, u_x has a much larger slope, indicating larger shear rate, and thus larger shear stress in the $-x$ direction. An interesting finding is that below the inner side membrane the flow is in the $-x$ direction in both cases, and the flow speed surpasses the circulation speed of the membrane so that the shear stress exerted on the inner side membrane by the flow is also in the $-x$ direction. This contradicts the scenario shown in Fig. 2, in which the direction of the shear stress on the inner side membrane is assumed to be in the $+x$ direction. This is due to the fact that in this case there is neither an incoming flow or forward speed of the body. The tank treading motion of the membrane creates clockwise circulatory flow around the body (see Figs. 6 b and c). When this circulatory flow enters the chamber its speed is increased due to conservation of mass. This acceleration effect is amplified when the chamber size is reduced, increasing the propulsive force at the cost of more energy consumption.

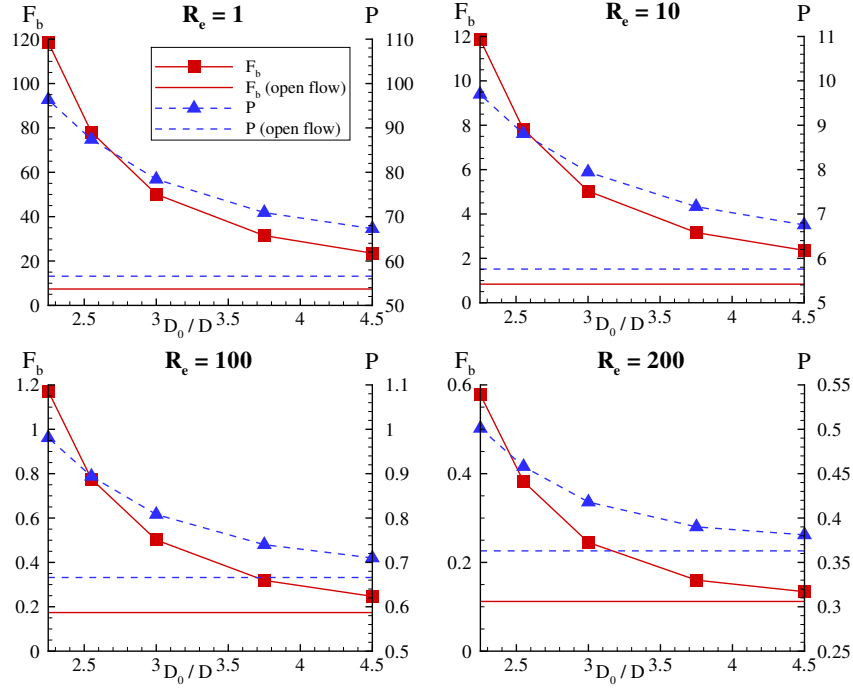


Figure 5. The force generation F_b and power expenditure P at various values of Re and D_0/D in the tethered scenario.

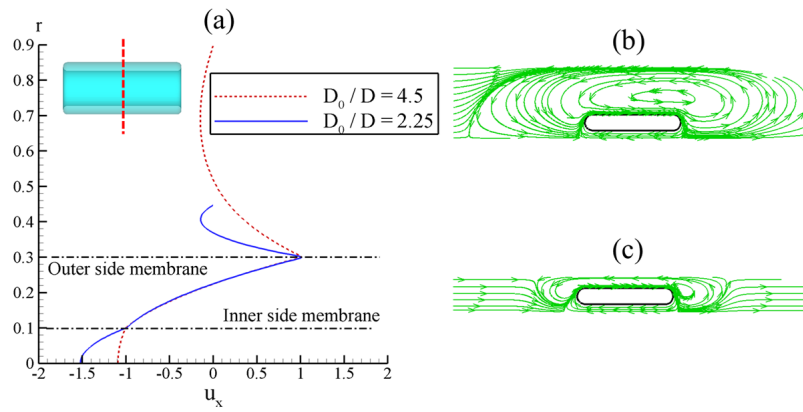


Figure 6. (a) Profiles of the longitudinal flow velocity u_x along the center line (see the dashed line in the inset) of the swimmer with $D_0/D = 4.5$ and 2.25 . (b) and (c) Near-body flow fields visualized via the streamlines with $D_0/D = 4.5$ and 2.25 . $Re = 100$.

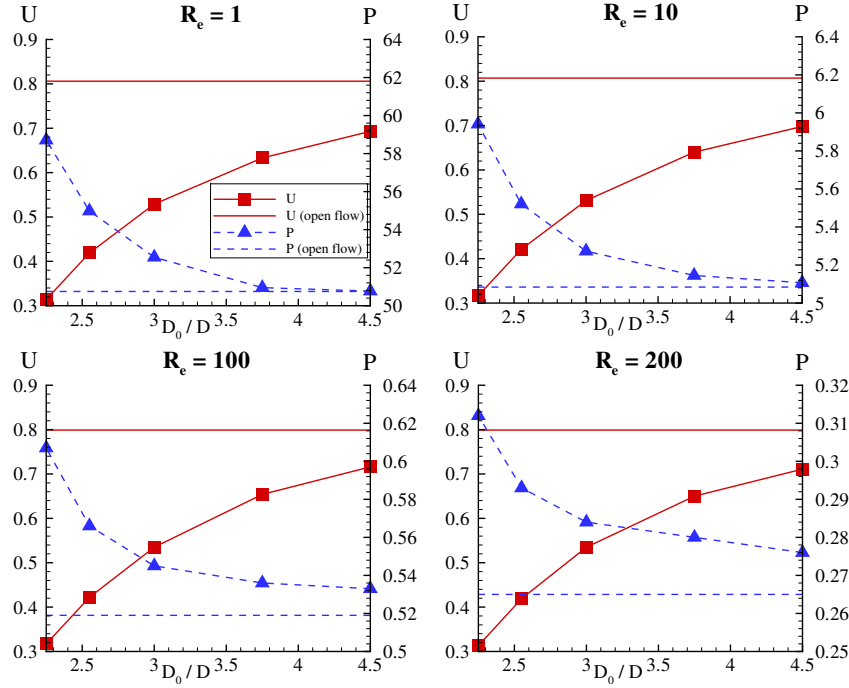


Figure 7. The free-swimming speed U and power expenditure P at various values of Re and D_0/D in the free-swimming scenario.

4.3. Free-swimming scenario – dynamics

In the free-swimming scenario, the forward motion of the body is simulated by using Eq. (7). The simulation continues until the swimming speed $\dot{x}_b$ reaches a steady-state value U , which is defined as the free-swimming speed. At this state, the corresponding power expenditure is recorded as P . In these simulations the body mass m_b is chosen to be 10^{-3} . This value does not affect U and P , although it does affect the acceleration process towards the steady state.

Figure 7 displays the free-swimming speed U and the power expenditure as functions of the diameter ratio D_0/D at different values of Re . It shows that despite the enhanced propulsive force inside a pipe, the swimming speed is decreased in comparison with its value in an open flow. This is explained by the increase of the drag effect inside a pipe. For illustration, in Fig. 8 we plot the drag force F_D on the swimmer when the membrane circulation is turned off. Instead, the swimmer is towed at its free-swimming speed U . It is seen that in all the cases the swimmer experiences a larger drag force F_D inside a pipe, and the value increases when the diameter of the pipe is reduced. This is due to the fact that inside a tube it is more difficult for the fluid to go around a solid object so that the blocking effect is increased.

Unlike D_0/D , the effect of the Reynolds number Re upon U appears to be weak, as can be seen by comparing the four sub-figures in Fig. 7 with different values of Re . In these simulations, the geometry of the object, the fluid density, and the circulation speed of the membrane are kept unchanged. The variation of Re is achieved by choosing different values of viscosity μ . Based on approximations such as the Blasius solution, it is reasonable to assume that the shear stress on the membrane can be written as $f_1(\mu)f_2(\mathbf{X}, \dot{x}_b)$, where $\mathbf{X}$ is the location vector of the point on the membrane and $\dot{x}_b$ is the forward speed. f_1 and f_2 are unknown functions. In the lower range of the Reynolds number considered in this study ($Re = 1 - 10$), $f_1(\mu) \sim \mu$ [34]. Above this range, it shifts towards the Blasius solution of $f_1(\mu) \sim \mu^{1/2}$. The function f_2 depends on the geometry of the swimmer and the size of the tube, while f_1 does not. Subsequently, the net force on the body F_b , which is dominated by the shear stress, can be approximated by integrating the shear stress on the body so that we have $F_b \sim f_1(\mu)F_2(\dot{x}_b)$, where F_2 is another unknown function that depends on the geometry of the swimmer. It is easy to see that with this approximation, when the geometric parameters are fixed the free swimming speed U is the solution of $F_2(\dot{x}_b) = 0$, which does not depend on μ and thus Re .

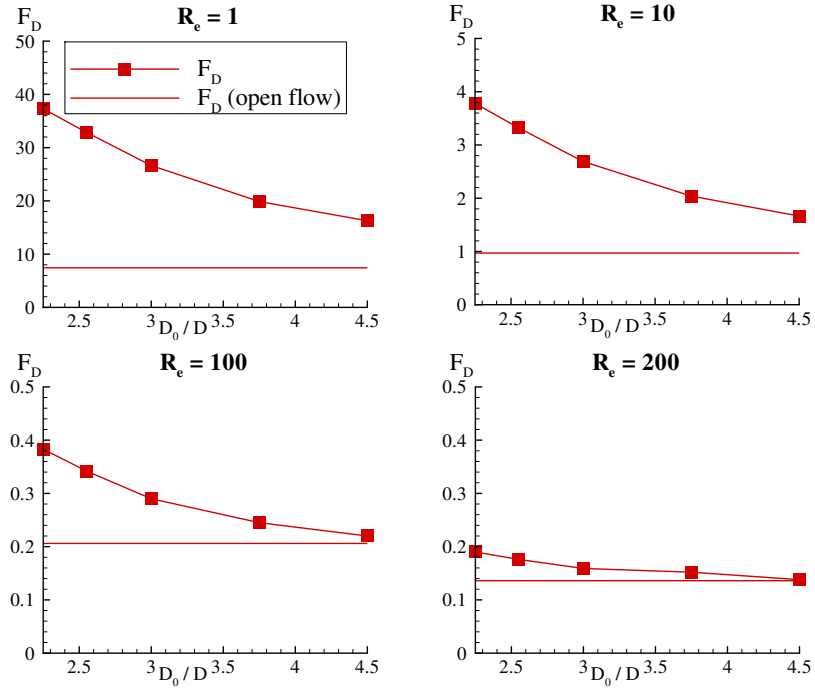


Figure 8. The drag force F_D on the body without membrane circulation towed at the free-swimming speed U at various values of Re and D_0/D .

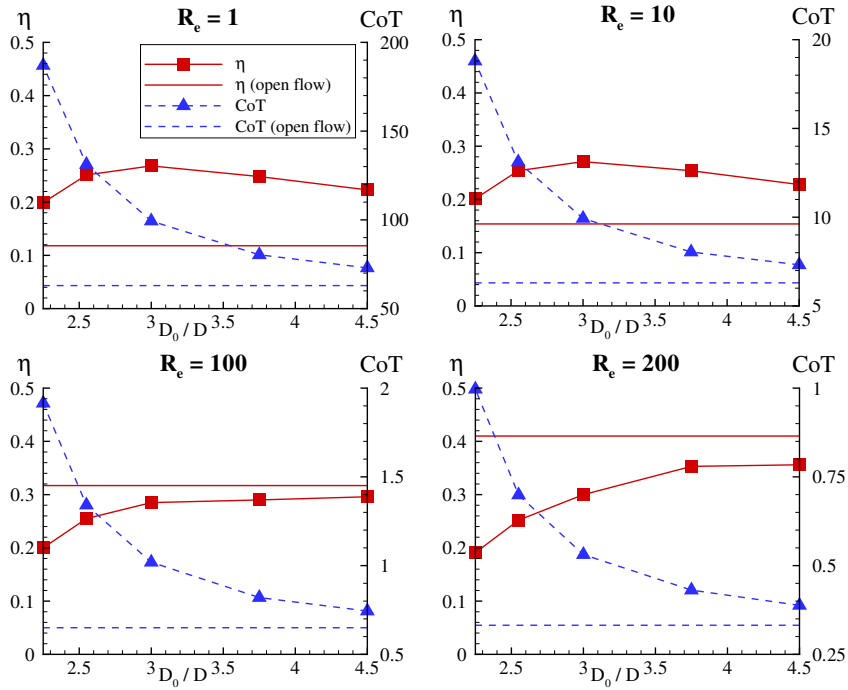


Figure 9. The propulsive efficiency η and cost of transport CoT at various values of Re and D_0/D in the free-swimming scenario.

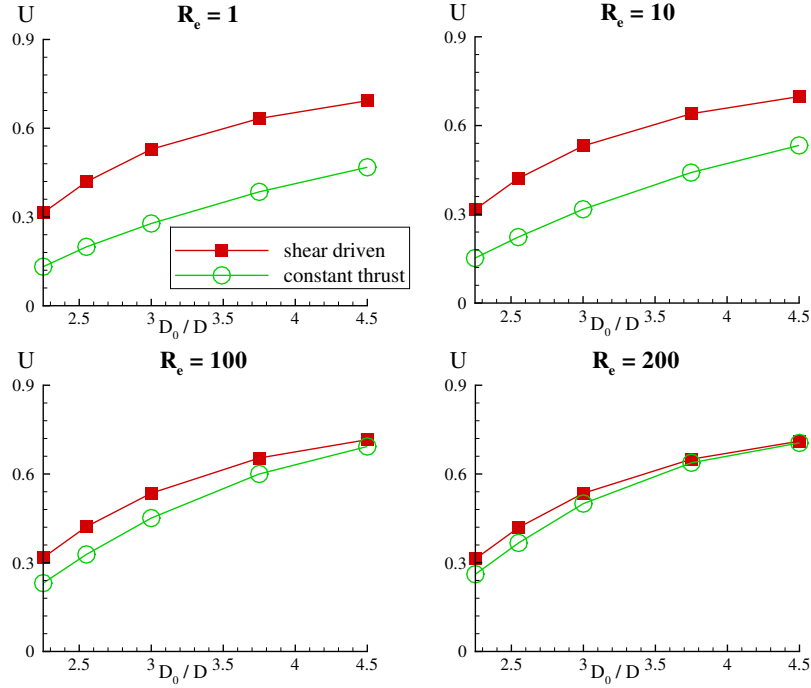


Figure 10. Comparison between the swimming speed of the current tank-treading-inspired swimmer and that of a swimmer with constant thrust at different values of Re and D_0/D .

The effect of the pipe on the power expenditure P does depend on the Reynolds number. It turns out that when Re is small, the effect of the pipe on P is reduced. For example, as shown in Fig. 9, when $Re = 1$ and $D_0/D = 4.5$, the power expenditure remains almost unchanged in comparison with its value in an open flow. However, at the same $D_0/D = 4.5$, when Re is raised to 100, there is a 2.7% increase in P compared with the open flow case. When Re is 200, the increase in P is 4.2%. However, based on these results this effect is quantitatively insignificant. Indeed, it is so subtle that it is not reflected in any notable characteristics in the surrounding flow field.

The two parameters characterizing the efficiency of the swimmer, the propulsive efficiency η and cost of transport CoT , are plotted in Fig. 9 as functions of D_0/D at the four values of Re considered. They predict different trends in terms of the effect of the pipe on the energetics.

Inside the pipe the swimmer has higher propulsive efficiency if $Re = 1$ or 10 when compared with the values in open flow. This is attributed to the significant increase in the drag force F_D (and, subsequently, the thrust F_T) inside a pipe as shown in Fig. 8, which overcomes the decrease in swimming speed U and the increase in the power expenditure P (see Fig. 7). At these two values of Re , there is a peak value of η around 0.27, reached when D_0/D is 0.3. This peak value is significantly higher than the propulsive efficiencies of 0.12 at $Re = 1$ and 0.15 at $Re = 10$ in open flow. In the cases when $Re = 100$ or 200, the rise of the thrust inside a pipe is not sufficient so that the propulsive efficiency η there remains below the value in open flow. This value drops monotonically when the pipe diameter is reduced.

Since no thrust is included in the definition of the cost of transport, this property is not affected by the increase in F_D inside a pipe. Thus, at all the Reynolds numbers considered CoT increases with smaller pipes. Besides, at no combination of Re and D_0/D its value goes below the open flow one, suggesting diminished efficiency inside a pipe.

As the propulsive force generated by the shear-driven mechanism is expected to increase in narrower pipes (see Fig. 5), it is expected to offset the effect of drag increasing in these pipes (Fig. 8) to certain extent and benefit the free-swimming speed U . To quantitatively illustrate this effect, we hereby consider a swimmer which has the same geometry as the tank-treading-inspired swimmer so that it experiences the same drag force F_D in the same conditions (forward speed, Reynolds number, and pipe diameter), yet the thrust force F_T on this hypothetical swimmer

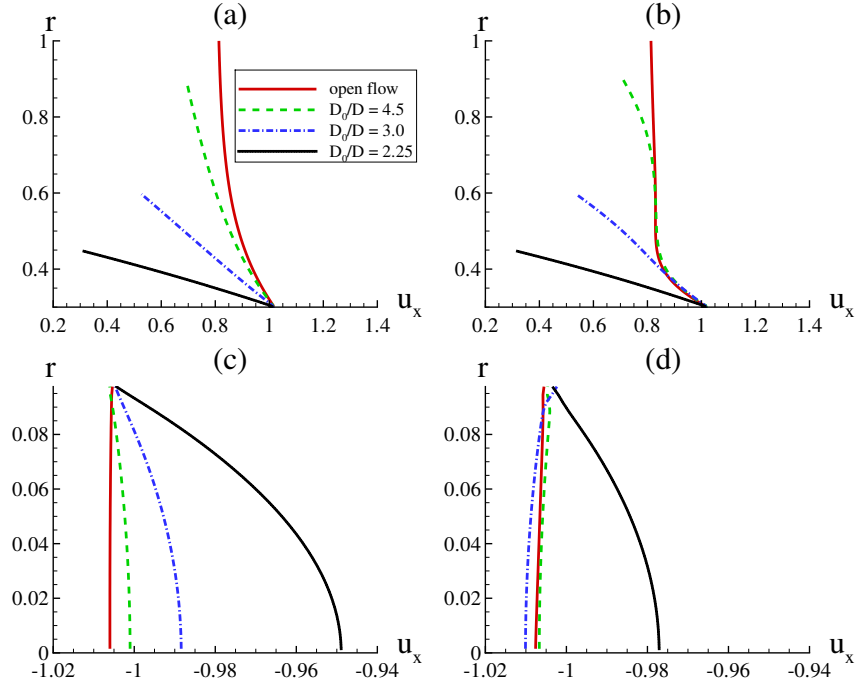


Figure 11. Profiles of the longitudinal flow speed u_x measured in a body-fixed reference system along the center line at $Re = 1$ ((a) and (c)) and 100 ((b) and (d)) in the free-swimming scenario.

is independent of the pipe diameter and the forward speed – it remains the same as the one in the free-swimming condition in open flow at the given Reynolds number. This value is denoted as F_{T0} . Hereafter this case is called the *swimmer with constant thrust*. The swimming speed U of this swimmer in a pipe is obtained by determining the speed at which the drag force on it is balanced by the given thrust F_{T0} , i.e., $F_D(U) = F_{T0}$. The results are plotted in Fig. 10, where they are compared with the free-swimming speed of the tank-treading-inspired swimmer at the same conditions. It is clearly seen that the tank-treading-inspired swimmer wins the race in all the cases, and its advantage is more pronounced at lower Reynolds numbers.

4.4. Free-swimming scenario – flow fields

The near-body flow fields are illustrated by the distribution of the longitudinal flow speed u_x along the center line of the body (see Fig. 5 for the definition of the center line) and the distribution of the vorticity $\partial u_x / \partial r - \partial u_r / \partial x$ around the body.

Figure 11 plots the longitudinal flow speed u_x along the center line at two values of Reynolds number, 1 and 100. The speed is measured in a body-fixed coordinate system which follows the forward motion of the swimmer. Two regions are plotted: $r > 0.3$ (in the gap between the outer side membrane and the inner wall of the pipe) and $r < 0.1$ (inside the chamber). For comparison, the corresponding flow speed profiles in open flow are also shown. According to this figure, in the gap the distribution of u_x is almost linear when the pipe is narrow (e.g., $D_0/D = 2.25$). When D_0/D decreases, the slope of the $u_x \sim r$ curve on top of the outer side membrane which lies at $r = 0.3$ is increased at both Reynolds numbers so that the shear stress there is raised. Inside the chamber, the distribution of u_x is not far from -1, the speed of the inner side membrane. The implication is that the shear stress on the inner side of the membrane is much lower than the one on the outer side membrane.

The distributions of vorticity are shown in Fig. 12. It is seen that aside from the strong vorticity in the interior flow field, the concentration of vorticity is mostly within a small region close to the body and in the gap between the outer side membrane and the inner wall of the pipe if the pipe diameter is sufficiently small. There is no prominent flow vorticity anywhere else. According to the flow field illustrated by the streamlines (Fig. 13), near the ends of the swimmer there are regions in the vicinity of the central axis of the pipe where there is certain disturbance caused by

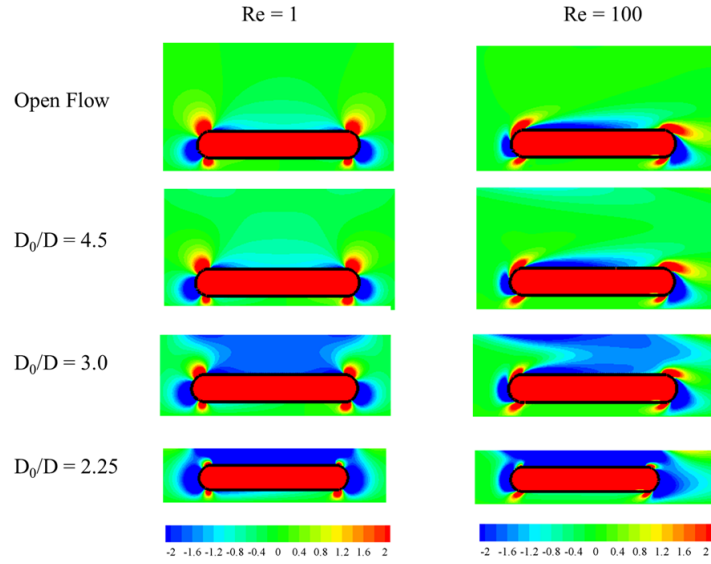


Figure 12. Near-body flow fields visualized through vorticity at $Re = 1$ and 100 in the free-swimming scenario.

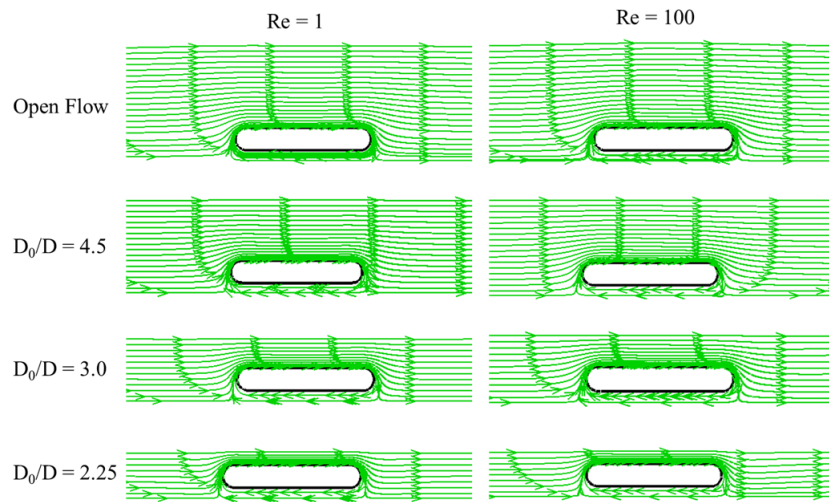


Figure 13. Near-body flow fields visualized through streamlines at $Re = 1$ and 100 in the free-swimming scenario. The flow is measured in a body-fixed reference system.

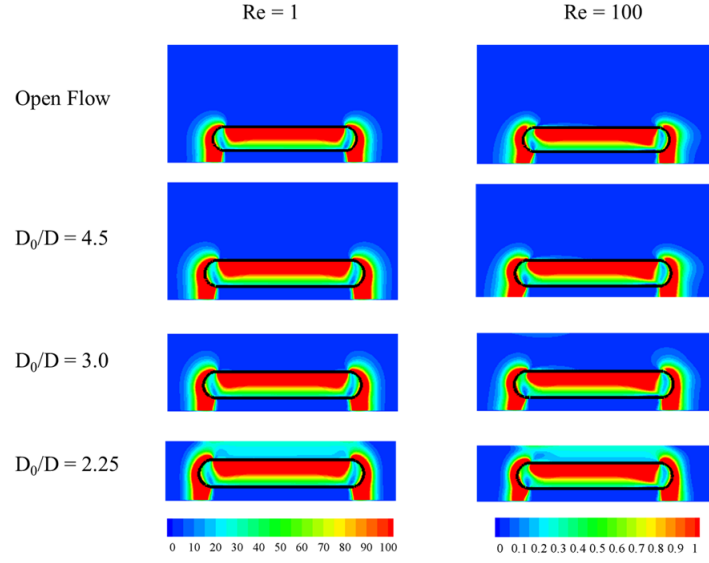


Figure 14. Distribution of the dissipation rate per unit volume of fluid ϕ at $Re = 1$ and 100 in the free-swimming scenario.

the swimmer. Behind the swimmer, however, the effect of the swimmer disappears after a very short distance from it. Based on these results, we conclude that the disturbance caused by the swimmer during self-propelled locomotion is localized and there is no long-lasting wake behind it as in other locomotion methods.

4.5. Process of energy transformation

During the swimming process, mechanical energy is transferred from the membrane to both the interior and the exterior flow fields. Eventually, the energy is dissipated by the viscous effect. The rate of energy dissipation per unit volume of fluid is defined as ϕ , which is determined as

$$\phi = 2\mu \left[\left(\frac{\partial u_r}{\partial r} \right)^2 + \left(\frac{u_r}{r} \right)^2 + \left(\frac{\partial u_x}{\partial x} \right)^2 \right] + \mu \left(\frac{\partial u_x}{\partial r} + \frac{\partial u_r}{\partial x} \right)^2. \quad (8)$$

By examining Fig. 14, it is easy to see that significant amount of the energy dissipation occurs in the interior flow field. This part of energy loss is balanced by the work done by the membrane to the interior fluid. For better illustration, in Fig. 15 we plot the portion of power expenditure spent to activate the interior flow field Φ_I/P , where $\Phi_I = \int_{\Omega_I} \phi d\mathbf{x}$ is the rate of energy dissipation in the interior flow field (Ω_I is the volume occupied by the interior fluid). In the cases we study, the range of Φ_I/P is between 0.55 (when $D_0/D = 2.25$ and $Re = 200$) and 0.65 (when $D_0/D = 4.5$ and $Re = 1$). A general trend is that Φ_I/P decreases at smaller values of D_0/D . This is consistent with the distribution of ϕ shown in Fig. 14, where we see that in narrower pipes more energy dissipation happens in the exterior flow field between the outer side membrane of the swimmer and the inner wall of the pipe.

The interior flow field does not participate in the generation of propulsive force – the force it exerts on the membrane is an internal force with zero net effect. This means that most of energy consumed by the membrane for its circulatory motion is wasted. If this part of energy is saved, the energy efficiency of the system will be greatly improved. A possible way to achieve this is by using an interior fluid with smaller viscosity. However, in practice it may not be straightforward to employ this strategy. First, it is not always easy to find an interior fluid with the desired physical properties such as density and viscosity. Even if such a fluid is available, there is a need to completely separate it from the exterior fluid, which might be a challenge in mechanical design. Furthermore, since the density of the interior fluid is most likely different from that of the exterior one, there is a need to balance buoyancy effect. From the numerical point of view, if the viscosity of the interior fluid is lower than the range considered in this study, there is a possibility that the interior flow field may not be axisymmetric anymore due to instabilities that leads to the growth of disturbances in the azimuthal direction. For example, to keep the flow around a sphere axisymmetric, the Reynolds

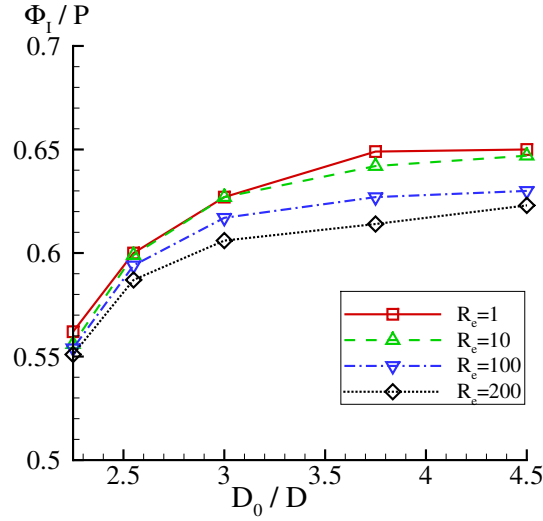


Figure 15. The portion of power expenditure dissipation in the interior flow field at various values of R_e and D_0/D in the free-swimming scenario.

number has to be below 210. If the Reynolds number in the interior flow field is sufficiently low, there could even be turbulence effect. In these scenarios, a more sophisticated numerical model is necessary.

5. Conclusions and discussion

With a fluid-structure interaction model based on the immersed-boundary framework, the dynamics of a novel propulsion method utilizing the shear stress on a solid surface exerted by the surrounding flow have been investigated. The focus is on the locomotion performance of this system inside a straight and uniform pipe with circular cross section. Different combinations of pipe size and Reynolds number have been considered in both the tethered and the free-swimming scenarios.

By fixing the geometry of the swimmer and the circulation speed of its membrane, it has been illustrated that in the tethered scenario when the body is not allowed to move, both the propulsive force F_b and the power expenditure are increased in the pipe due to the hydrodynamic interaction between the circulating membrane and the inner wall of the pipe. When the Reynolds number is within the range between 1 and 10, F_b and P are almost linearly proportional to $1/R_e$. Above this range, there is certain deviation from this linear relation.

In the free-swimming scenario, on the other hand, there is a reduction in swimming speed inside a pipe despite the increased propulsive force. This is attributed to the rise in drag force in the pipe. Although the swimming speed is significantly affected by the size of the pipe, its dependence on the Reynolds number is much weaker. In terms of the energy efficiency, using different criteria may lead to different conclusions. According to the propulsive efficiency, if the Reynolds number is small the swimming efficiency inside a pipe may be higher than the one in open flow, while if the Reynolds number is relatively large the opposite trend is observed. On the other hand, if the cost of transport is chosen as the criterion, the confinement effect inside a pipe diminishes the swimming efficiency in all the cases considered. The key difference between these two criteria is whether or not the thrust force is included in the definition.

The cost of transport used in this study is actually not dimensionless – it has the dimension of force. Such parameter may be more meaningful in characterizing the energy performance of the same system in different conditions. The original definition of the cost of transport is normalized by the weight of the swimmer. However, the body weight does not have direct relation with the swimming speed, as it is determined by the balance between thrust and drag. Similarly, it is not directly associated with the power expenditure, which is the power transferred from the swimmer to the surrounding flow during locomotion. Thus the physical basis of this definition may be questionable. The propulsive efficiency, on the other hand, is a dimensionless parameter. It may thus be a better tool in comparing the energy

performance of different systems. However, the evaluation of this parameter in the free-swimming scenario involves the somewhat controversial step of thrust-drag decomposition so that it has to be used with caution.

Visualizations of the surrounding flow field show highly localized vorticity distribution resulting from the locomotion activity of the swimmer. Therefore, this method is a good option in applications where minimal disturbance to the flow field is preferred. The dissipation rate in the flow field due to viscosity has also been studied. Most of the dissipation occurs in the interior flow field. This part of energy is completely wasted as the interior flow has nothing to do with the generation of propulsive force. Therefore, by improving the design of the system to mitigate energy dissipation in the interior flow, the energy efficiency of the system may be significantly enhanced. Further studies in this direction are necessary.

The current study is purely computational. Future experiments are needed to validate the numerical predictions and pave the way for the development of prototypes. Toward this end, it is necessary to design activation systems to create the skin circulation. A possible way to do this is by adopting the mechanical system developed by Eken et al. [22]. Driven by a motor, this device is able to achieve continuous skin eversion similar to the circulatory motion in our system. Robotic systems based on this design will have industrial and biomedical applications in which locomotion inside pipes or pipe-like structures such as blood vesicles is necessary. These robots can be used in operations such as monitoring, issue detection/repairing, delivery, and micro surgery.

Data Availability Statement

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

References

- [1] J. Kahnouei and M. Moallem, "A comprehensive review of in-pipe robots," *Ocean Eng.*, vol. 12, no. 3, p. 030801, 2024.
- [2] X. Zhu, S. Z. X. Li, D. Wang, and D. Yu, "Numerical simulation of contact force on bi-directional pig in gas pipeline: At the early stage of pigging," *J. Nat. Gas Sci. Eng.*, vol. 23, pp. 127–138, 2015.
- [3] F. Pfeiffer, T. Rossmann, and K. Löffler, "Control of a tube crawling machine," in *2nd International Conference. Control of Oscillations and Chaos*, pp. 586–591, 2020.
- [4] T. Oya and T. Okada, "Development of a steerable, wheel-type, in-pipe robot and its path planning," *Adv. Robot.*, vol. 19, no. 6, pp. 635–650, 2005.
- [5] J. Nagase and F. Fukunaga, "Development of a novel crawler mechanism for pipe inspection," in *IECON 2016-42nd Annual Conference of the IEEE Industrial Electronics Society*, pp. 5873–5878, 2016.
- [6] W. Jeon, J. Park, I. Kim, Y. Kang, and H. Yang, "Development of high mobility in-pipe inspection robot," in *IEEE/SICE International Symposium on System Integration*, pp. 479–484, 2011.
- [7] W. Liu, X. Jia, F. Wang, and Z. Jia, "An in-pipe wireless swimming microrobot driven by giant magnetostrictive thin film," *Sensors and Actuators A*, vol. 160, pp. 101–108, 2010.
- [8] Y. Wu, A. Niel, D. Kim, K. Youcef-Toumi, and R. Ben-Mansour, "Design of a maneuverable swimming robot for in-pipe missions," in *IEEE/RSJ International Conference on Intelligent Robots and Systems*, (Hamburg), pp. 4864–4871, 2015.
- [9] Y. Cao, K. Michel, F. Alimardani, and Y. Wang, "A review on recent progress of biodegradable magnetic microrobots for targeted therapeutic delivery: materials, structure designs, and fabrication methods," *J. Micro Nano Sci. Eng.*, p. 114260, 2023.
- [10] F. Landers et al., "Clinically ready magnetic microrobots for targeted therapies," *Science*, vol. 390, pp. 710–715, 2025.
- [11] Q. Zhu and Q. Xiao, "Shear-driven swimming in laminar flow inspired by tank treading," *Phys. Rev. Fluids*, vol. 10, p. 054101, 2025.
- [12] E. Purcell, "Life at low reynolds number," *Am. J. Phys.*, vol. 45, no. 1, pp. 3–11, 1977.
- [13] L. Payne and W. Pell, "The stokes flow problem for a class of axially symmetric bodies," *J. Fluid Mech.*, vol. 7, pp. 529–549, 1959.
- [14] W. Pell and L. Payne, "On stokes flow about a torus," *Mathematika*, vol. 7, pp. 78–92, 1978.
- [15] R. Thakkar, H. Schiessel, and I. Kulic, "Hydrodynamics of a rotating torus," *Eur. Phys. J. B*, vol. 60, pp. 325–336, 2007.
- [16] A. Leshansky and O. Kenneth, "Surface tank treading: propulsion of purcell's toroidal swimmer," *Phys. Fluids*, vol. 20, p. 063104, 2008.
- [17] Z. Deng, K. Li, A. Priimagi, and H. Zeng, "Light-steerable locomotion using zero-elastic-energy modes," *Nature Materials*, vol. 23, pp. 1728–1735, 2024.
- [18] N. Moshkin and P. Suwansri, "Self-propelled motion of a torus rotating about its centerline in a viscous incompressible fluid," *Phys. Fluids*, vol. 22, p. 113602, 2010.
- [19] N. Moshkin and P. Suwansri, "Two regimes of self-propelled motion of a torus rotating about its centerline in a viscous incompressible fluid at intermediate reynolds numbers," *Phys. Fluids*, vol. 24, p. 053603, 2012.
- [20] W. van Rees, G. Novati, and P. Koumoutsakos, "Self-propulsion of a counter-rotating cylinder pair in a viscous fluid," *Phys. Fluids*, vol. 27, p. 063102, 2015.
- [21] T. Fischer, M. Stöhr-Liesen, and H. Schmid-Schönbein, "The red cell as a fluid droplet: tank tread-like motion of the human erythrocyte membrane in shear flow," *Science*, vol. 202, pp. 894–896, 1978.

- [22] K. Eken, N. Gravish, and M. Tolley, “Continuous skin eversion enables an untethered soft robot to burrow in granular media,” in *IEEE International Conference on Soft Robotics*, (Singapore), pp. 1–6, 2023.
- [23] D. Barthès-Biesel, “Motion of a spherical microcapsule freely suspended in a linear shear flow,” *J. Fluid Mech.*, vol. 100, pp. 831–853, 1980.
- [24] R. Tran-Son-Tay, *A study of the tank-treading motion of red blood cells in shear flow*. PhD thesis, Washington University, St. Louis., 1983.
- [25] Q. Zhu and Q. Xiao, “Physics and applications of squid-inspired jetting,” *Bioinspir. Biomim.*, vol. 17, p. 041001, 2022.
- [26] D. Barrett, *Propulsive Efficiency of a Flexible Hull Underwater Vehicle*. PhD thesis, Massachusetts Institute of Technology, 1996.
- [27] I. Borazjani and F. Sotiropoulos, “Numerical investigation of the hydrodynamics of carangiform swimming in the transitional and inertial flow regimes,” *J. Exp. Biol.*, vol. 211, pp. 1541–58, 2008.
- [28] X. Bi and Q. Zhu, “Efficiency of pulsed-jet propulsion via thrust-drag decomposition,” *Phys. Fluids*, vol. 33, p. 071902, 2021.
- [29] R. O’dor and D. Webber, “Invertebrate athletes: trade-offs between transport efficiency and power density in cephalopod evolution,” *J. Exp. Biol.*, vol. 160, pp. 93–112, 1991.
- [30] K. Kim, S. Baek, and H. Sung, “An implicit velocity decoupling procedure for the incompressible navier–stokes equations,” *Int. J. Numer. Meth. Fluids*, vol. 38, pp. 125–138, 2002.
- [31] X. Bi and Q. Zhu, “Pulsed-jet propulsion via shape deformation of an axisymmetric swimmer,” *Phys. Fluids*, vol. 32, p. 018902, 2020.
- [32] X. Bi and Q. Zhu, “Free swimming of a squid-inspired axisymmetric system through jet propulsion,” *Bioinspir. Biomim.*, vol. 16, p. 066023, 2021.
- [33] G. Sheard, K. Houriga, and M. Thompson, “Computations of the drag coefficients for low-reynolds-number flow past rings,” *J. Fluid Mech.*, vol. 52, pp. 257–275, 2005.
- [34] E. Janssen, “Flow past a flat plate at low reynolds numbers,” *J. Fluid Mech.*, vol. 3, no. 4, pp. 329–343, 1958.