Numerical Simulation of Fish-like Bodies Swimming in Fluid

Ching-Biao Liao[1], and Cheng-Hsin Chen[2]

[1]Professor, Department of Water Resource Engineering and Conservation, Feng Chia University
No. 100 Wenhwa Rd., Seatwen, Taichung, Taiwan 40724, R.O.C., e-mail: cbliao@ fcu.edu.tw
[2]Ph.D. candidate, Institute of Civil & Hydraulic Engineering, Feng Chia University
No. 100 Wenhwa Rd., Seatwen, Taichung, Taiwan 40724, R.O.C., e-mail: p9317786@fcu.edu.tw

ABSTRACT

In this paper, the simplified immersed boundary method (IBM) is developed for modeling 2D fish-like bodies swimming in the fluid. The incompressible Navier-Stokes equations together with the IBM are discretized by finite difference method in simple staggered Cartesian grids. The numerical method is validated by the simulation of flow past a circler cylinder and circular cylinder immersed in lid-driven cavity. The versatility of the method is demonstrated by the numerical simulations of fish-like bodies swimming in the fluid. The results are shown a good agreement compared with previous computational results. In conclusion, our current method is efficient, straightforward, robust and successful in solving fluid-structure interaction problems.

Keywords: immersed boundary, fish-like

1. INTRODUCTION

In computational fluid dynamics, fluid-structure interaction problems have been a challenging subject for many years. The behaviors of buildings and bridges subject to wind loads, planes in flight, and undersea cables are a few examples, as is flow past a fixed or oscillating circular cylinder. When the immersed structure is complex and moving, it becomes difficult to accurately simulate and to treat in complex boundary. In this paper, we improve on these two difficulties by using immersed boundary method on a fixed Cartesian grid.

The immersed boundary method was first presented by Peskin (1977), who modeled blood flow in the heart. This method represents a body within a flow field via a forcing term added to the governing equations. Recently, some new methods based on the immersed boundary concepts of Peskin have been proposed. For example, Lai and Peskin (2000) developed a second-order accurate immersed boundary method that has been used to simulate flows over a circular cylinder and to study the effect of numerical viscosity on the accuracy of the computation by comparing the numerical results with those of a first-order method. Ye et al. (1999) proposed a method called the Cartesian Grid Method for simulating two-dimensional unsteady, viscous, incompressible flows over complex geometries. Fadlum et al. (2000) used discrete-time forcing, as suggested by Mohd-Yusof (1997), and showed that it is more efficient than the feedback forcing method for three-dimensional problems. Kim et al. (2001) proposed a method based on a finite-volume approach, using a staggered mesh, and simulated flows over immersed complex geometries.

The immersed boundary method specifies a body force in such a way as to simulate the presence of a surface without altering the computational grid. The main advantages of the immersed boundary method are memory and CPU savings and ease of grid generation when compared with unstructured grid methods. Uhlmann (2005) presented an improved method that allows for a smooth transfer between Eulerian and Lagrangian representations while avoiding strong restrictions of the time step. Chio et al. (2007) presented an immersed boundary method for time-dependent, three-dimensional, incompressible flows. They defined the immersed boundary surfaces as clouds of points. In their method, immersed boundary objects are rendered as level sets in the computational domains. They applied the method to the simulation of flow induced by realistic human walking. Feng et al. (2004) applied the immersed boundary-lattice Boltzmann method to simulate the sedimentation of a large number of circular particles in an enclosure.

The objective of this study is to develop an approach based on immersed boundary method without representation by a set of Lagrangian points for immersed boundary in fluid. The present method is based on a finite difference approach on a Cartesian grid system. In this paper, the numerical method is validated by the simulation of flow past a circler cylinder and circular cylinder immersed in lid-driven cavity. The versatility of the method is demonstrated by the numerical simulations of fish-like bodies swimming in the fluid.

2. Numerical Method and Physical Model

Governing equations

The method used for simulations was based on the finite difference formulation in the Cartesian grid system. The temporal evolution of an incompressible viscous fluid in a square domain containing an immersed solid body is governed by the Navier-Stokes equations:

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + (\mathrm{u} \cdot \nabla)\mathrm{u} = -\nabla p + \frac{1}{\mathrm{Re}}\nabla^2 \mathrm{u} + \eta f \tag{2}$$

written in a nondimensional form, where u represents Cartesian velocity components, p is pressure, Re = UL/ν refers to Reynolds numbers, U is a characteristic velocity, L is a characteristic length, and ν is kinematic viscosity of the fluid. $\vec{f}$ is an extra virtual force term. η denotes volume fraction of solid in each u and v cell. η can be as simple as 0 (indicating fluid) or 1 (indicating solid), or can be fractional in each cell through more elaborate computation to improve order of accuracy in space.

The momentum equation (2) is first expressed in explicit form by the application of the third-order Adam-Bashforth method. QUICK (Quadratic Upwind Interpolation of Convective Kinematics) scheme is applied to solve advection term, and second order central difference scheme is applied to solve diffusion term. The pressure Poisson equation derived by applying divergence operator to the momentum equations is solved by Fast Fourier Transform (Liao et al. (1999)), and the new pressure field is obtained.

Immersed Boundary Method with direct forcing

The method used for simulations was based on the finite difference formulation in the Cartesian grid system. The temporal evolution of an incompressible viscous fluid in a square domain containing an immersed solid body is governed by the Navier-Stokes equations:

$$\frac{u^{n+1}-u^{**}}{\Delta t}=\eta f^{n+1} \tag{3}$$

in fluid ($\eta=0$), $u^{n+1}=u^{**}$ and $f^{n+1}=0$;
in solid ($\eta\neq 0$), $u^{n+1}=u_s$. u^{**} is the intermediate velocity.
We can obtain the virtual force of solid f^{n+1} by

$$f^{n+1}=\frac{1}{\eta}\frac{u_s-u^{**}}{\Delta t} \tag{4}$$

where u_s is the solid velocity, $u_s=0$ for fixed solid body.

Physical Model for Fish-like Swimming

In this paper, the NACA0012 foil with a traveling wavy motion is considered. As shown in Figure 1, the length of the foil is L, and the free-stream velocity U_∞. The non-dimensional position of the foil is described as

$$y=A_m(x)\cos\left[2\pi\alpha(x-ct)\right],\qquad 0\le x\le 1 \tag{5}$$

where $\alpha=L/\lambda$, with λ being the wavelength of the traveling wave, A_m and c are the amplitude and the phase speed of the traveling wave, respectively.

The model the backbone undulation of fish swimming (Deng et al. (2006)) the amplitude is approximated to be in the form of a quadratic polynomial as

$$A_m(x)=C_0+C_1x+C_2x^2 \tag{6}$$

where C_0, C_1 and C_2 are adjustable coefficients.

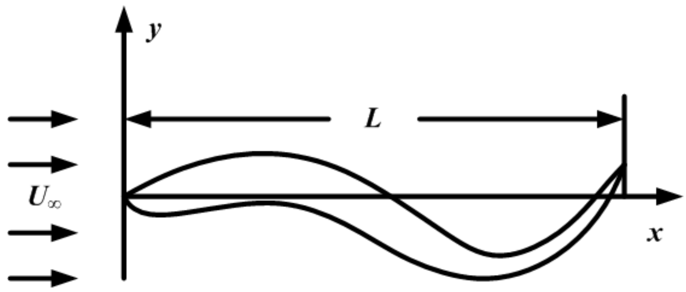


Figure 1 Schematic of flow configuration for traveling wavy foil

3. Computational Results and Discussion

Circular cylinder immersed in lid-driven cavity

In order to investigate the accuracy of our immersed boundary method, we carried out a grid convergence study for a test problem, which is the variation of solution error with grid refinement. In this case, we simulated a circular cylinder immersed in a lid-driven cavity. The domain size of the cavity was 1×1. The diameter of the cylinder was 0.5. This cylinder was placed at the centre of the cavity with Reynolds number Re=10. Calculations were carried out on six uniform grids: 96^2, 160^2, 224^2, 352^2, and 544^2. In the absence of an exact analytical solution to this flow problem, we took the results of 544^2 grid to be the exact solution to obtain the order of the accuracy of our scheme.

The results of the grid convergence study are shown in Figure 2, including the variation in L_1, L_2, and L_∞ norms of errors with grid size in logarithmic coordinate system. Two lines with a slope of 1 and a slope of 2 correspond to first-order and second-order accuracy, respectively. It is evident from Figure 2 that our method is over first-order accuracy.

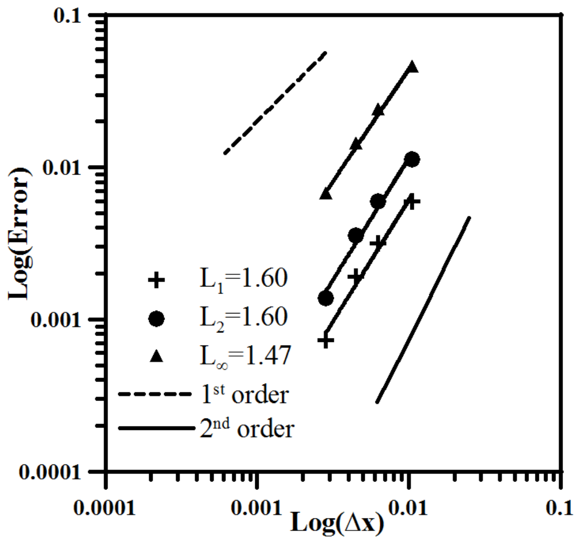


Figure 2 Norm error in u-velocity with Re=10

Flow past a fixed circular cylinder

Uniform flow past a fixed circular cylinder is a typical problem that has been widely investigated using experimental and numerical simulations. We also used this problem as a benchmark test for our numerical method. All simulations were performed in a 32*D*×16*D* rectangular domain, where *D* denotes the cylinder diameter, as shown in Figure 3. A uniform grid with resolution of 640×320 was used for all simulations.

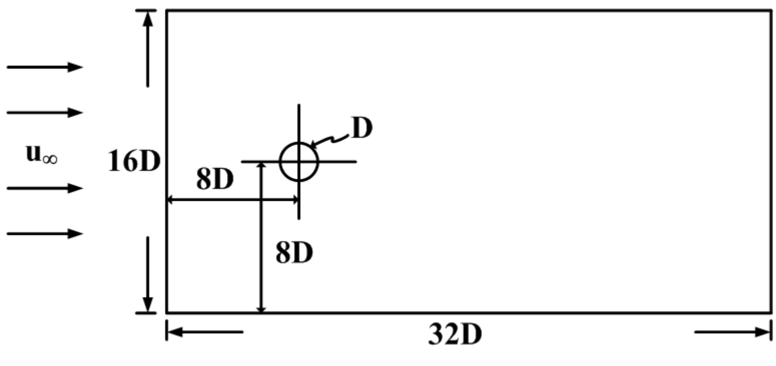


Figure 3 Geometry of flow past a fixed circular cylinder

Simulations were carried out at several Reynolds numbers in this study. Once the velocity and pressure fields were calculated, the drag coefficient (Cd) could be calculated directly from the force field. The comparison of the drag coefficient obtained in the present work with experimental and numerical results of other studies is presented in Table 1. Very good agreement was achieved.

Table 1 Comparison of drag coefficients in this study with those of other authors

Re	Present	Lima et al. (2003)	Russell and Wang (2003)	Deng et al.	Ye et al.
10	3.03	2.81	-	2.98	-
20	2.17	2.04	2.13	2.06	2.20
40	1.62	1.54	1.60	1.52	1.63
50	1.52	1.46	-	1.42	-
80	1.45	1.40	-	1.32	1.43
100	1.43	1.39	1.38	1.30	1.40
150	1.44	1.37	-	-	1.39

Single fixed fish-like body swimming in fluid

In this case, the computational domain is 12 × 6. A uniform grid with resolution of 512 × 256 was used. The coefficients C_0, and C_2 are set equal to zero, and C_1 is set to 0.2. According to the relevant study on the motion of an aquatic animal at intermediate Reynolds numbers (i.e., Re$\sim$O(10^2)), the Reynolds number is set as 500. The phase speed c range from 0.5 to 2.5. The traveling wavelength λ is set equal to the length of the fish body.

The drag force acting on the fish body and the power needed for it to be propelled are directly relevant to the study of fish-like locomotion. The total drag force (F_d) on the wavy fish-like body consists of a friction force (F_f) and a pressure difference force (F_p). The corresponding drag coefficients are defined as (Dong and Lu (2005))

$$C_{DF}=\frac{F_f}{1/2\rho U^2 L},\quad C_{DP}=\frac{F_p}{1/2\rho U^2 L},\quad C_D=\frac{F_d}{1/2\rho U^2 L} \tag{7}$$

$$F_d=F_f+F_p \tag{8}$$

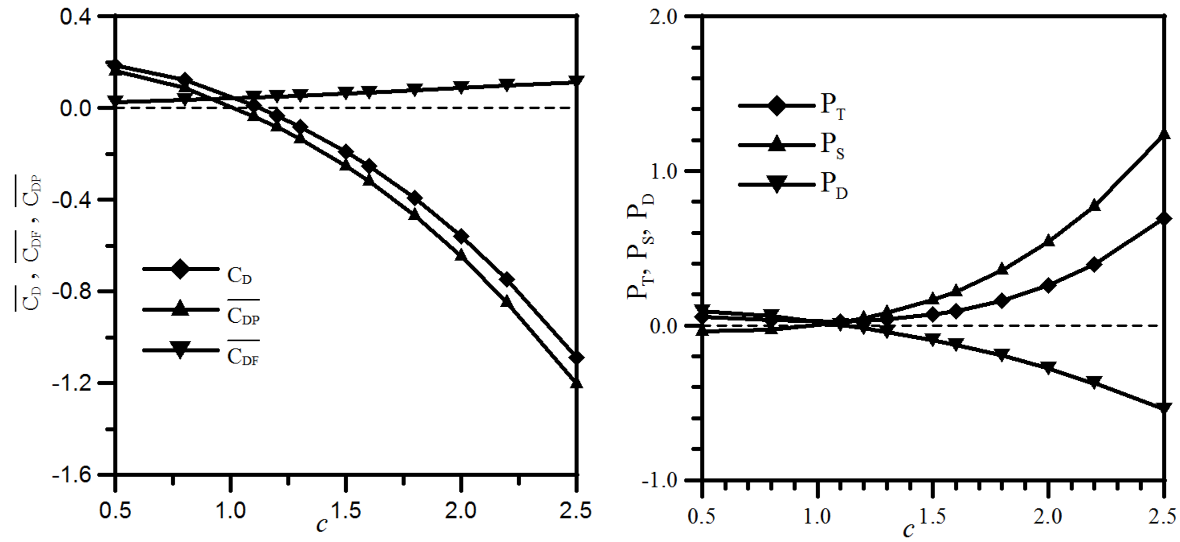
Base on the definition (Dong and Lu (2005), Lu and Yin (2005)), the total power (P_T) required for the propulsive motion of fish-like consists of two components. One is the swimming power, required to produce the vertical oscillation of traveling wave motion and is defined as

$$P_S=\int_0^L\left[\left(p_+-p_-\right)\frac{dy}{dt}\right]dx \tag{9}$$

where p_+ and p_- are pressure on the upper and lower surface of the fish-like body. The other power is needed to overcome the drag force and is represented as $P_D=UF_d$. Thus, the total power $P_T=P_S+P_D$.

The time-averaged drag force coefficients ($\overline{C_D}$, $\overline{C_{DF}}$, $\overline{C_{DP}}$) versus c are shown in Figure 4(a). With the increase of c, $\overline{C_D}$ and $\overline{C_{DP}}$ decrease, and $\overline{C_{DF}}$ increase to some extent. It can be obtained that $\overline{C_{DP}}$ becomes negative and act as thrust force when $c > 1.1$ approximately, and $\overline{C_D}$ becomes negative when $c > 1.1$. These results are also consistent with previous findings(Deng et al.).

The time-averaged power consumption ($\overline{P_T}$, $\overline{P_S}$, $\overline{P_D}$) versus c are shown in Figure 4(b). As c increases, $\overline{P_S}$ increases and becomes positive for $c > 1.0$. The negative value of $\overline{P_S}$ means that the fish motion can be actuated by the flow, and no power input is required to fluctuate it.



(a) drag force (b) power

Figure 4 Time-averaged drag force and power consumption versus c

The ratio of $\overline{P_D}$ to $\overline{P_S}$ can be denote the propelling efficient, i.e. $\eta = -\overline{P_D} / \overline{P_S}$. The relationship between the efficiency and the phase speed, a peak value is determined at $c = 1.5$, as shown in Figure 5, and the corresponding value of efficiency is $\eta = -0.569$. This result is different with Deng et al. Figure 6 shows the distribution of the vorticity contours for different phase speeds.

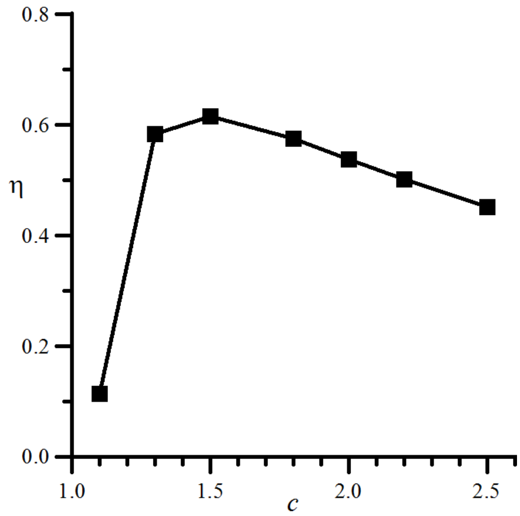
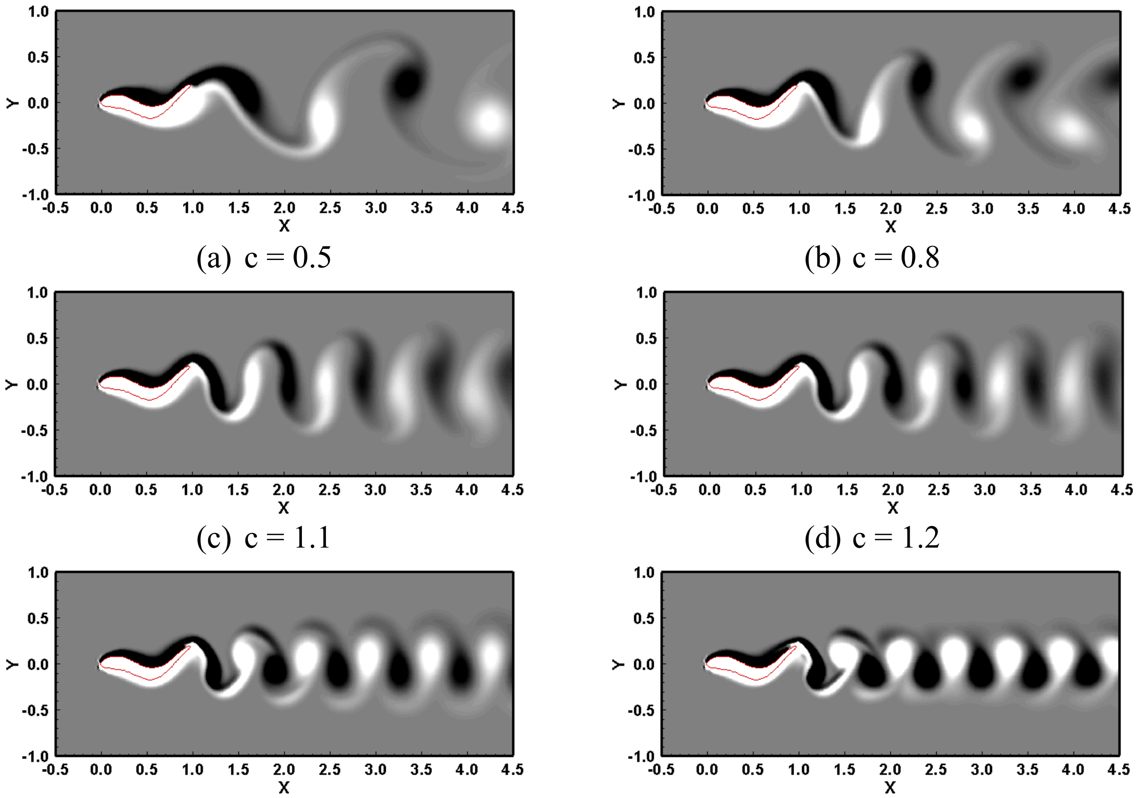


Figure 5 Propulsive efficiency η versus c



(a) c = 0.5 (b) c = 0.8

(c) c = 1.1 (d) c = 1.2

(e) c = 1.5 (f) c = 2.0

Figure 6 Vorticity contours with different phase speeds

Two fixed fish-like bodies swimming in tandem arrangement

In this case, the parameters are same as a single fish with c = 1.5. Several spacing between these two fishes are investigated, ranging from S/L = 1.1 to 5.0. Figure 7 shows that the drag coefficients with different spacing. It can be seen downstream $\overline{C_{DP}}$ increases suddenly and slightly when S/L = 1.8~2.5. It causes the efficiency of the downstream fish directly. The variation of efficiency with different spacing is shown in Figure 8. It can be observed that the efficiency of the downstream fish is higher that of the upstream fish, but is lower when S/L = 1.8~2.5. As common rule, the efficiency of downstream object should be higher then upstream object. The fishes in swimming maybe not follow this rule. Figure 9 shows the distribution of the vorticity contours for S/L= 1.8, 2.1, 2.5.

4. Conclusions

In this paper, the simplified immersed boundary method (IBM) is developed for modeling 2D fish-like bodies swimming in the fluid. The effects of the phase speed c at a C_1 = 0.2 are discussed. According to the distribution of drag force and power consumption with different c, it can be observed a critical value of c = 1.1. The efficiency with different c also investigate. Two fish-like bodies swimming in fluid in tandem arrangement are also discussed. It is observed that the efficiency of the downstream fish is higher that of the upstream fish, but is lower when S/L = 1.8~2.5.

It can be investigate behavior of 2D fish-like bodies swimming simply. Three dimensional simulations will carry out in our future work and observe more detail about fish swimming.

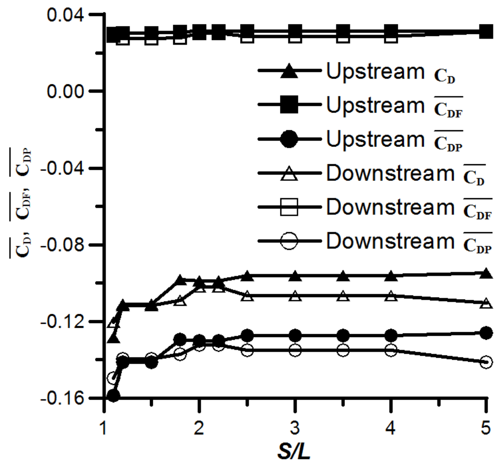


Figure 7 Time-averaged drag force versus different spacing

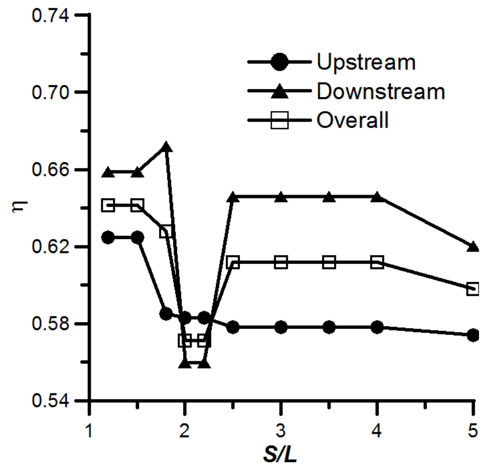
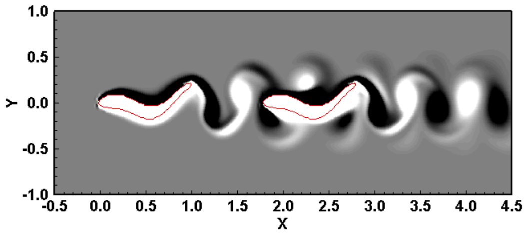
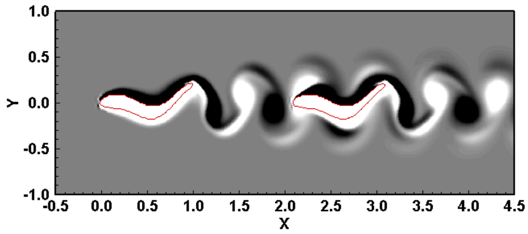


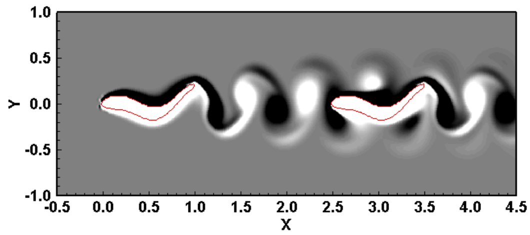
Figure 8 Propulsive efficiency η versus different spacing



(a) *S/L* = 1.8



(b) *S/L* = 1.8



(c) *S/L* = 1.8

Figure 6 Vorticity contours with different spacing

REFERENCES

Peskin, C.S. (1977), Numerical analysis of blood flow in the heart, *Journal of Computational Physics*, 25, pp.220-252.

Lai, M.C. and Peskin, C.S. (2000), An immersed boundary method with formal second-order accuracy and reduced numerical viscosity, *Journal of Computational Physics*, 160, pp.705-719.

Ye, T., Mittal, R., Udaykumar, H.S., and Shyy, W. (1999), An accurate Cartesian grid method for viscous incompressible flows with complex boundaries, *Journal of Computational Physics*, 156, pp.209-240.

Fadlun, E.A., Verzicco, R., Orlandi, P., and Mohd-Yousof, J. (2000), Combined immersed-boundary finite-difference methods for three dimensional complex flows simulations, *Journal of Computational Physics*, 161, pp.35-60.

Mohd-Yusof, J. (1997), Combined immersed boundaries/B-splines methods for simulations of

flows in complex geometries, CTR Annual Research Briefs, NASA Ames/Stanford University.
Kim, J., Kim, D., and Choi, H. (2001), An immersed-boundary finite-volume method for simulations of flow in complex geometries, *Journal of Computational Physics*, 171, pp.132-150.
Uhlmann, M. (2005), An immersed boundary method with direct forcing for the simulation of particulate flows, *Journal of Computational Physics*, 209, pp.448-476.
Choi, J.-I., Oberoi, R.C., Edwards, J.R., and Rosati, J.A. (2007), An immersed boundary method for complex incompressible flows, *Journal of Computational Physics*, 224, pp.757-784.
Feng, Z.-G., and Michaelides, E.E. (2004), The immersed boundary-lattice Boltzmann method for solving fluid-particles interaction problems, *Journal of Computational Physics*, 195, pp.602-628.
Liao, C.B., Jane, S.J., and Young, D.L. (1999), Numerical simulation of three-dimensional Couette-Taylor flows, *International Journal for Numerical Methods in Fluids*, 29, pp.827-847.
Deng, J., Shao, X.-M., and Ren, A.-L. (2006), Numerical study on propulsive performance of fish-like swimming foils, *Journal of Hydrodynamics*, Ser.B., 18(6), pp.681-687.
Lima, A.L.F., Silva, E., Silveira-Neto A, and Damasceno, J.J.R. (2003), Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, *Journal of Computational Physics*, 189, pp.351-370.
Russell, D., and Wang, Z.J. (2003), A Cartesian grid method for modeling multiple moving objects in 2D incompressible viscous flow, *Journal of Computational Physics*, 191, pp.177-205.
Dong, G.J., and Lu, X.Y. (2005), Numerical analysis on the propulsive performance and vortex shedding of foil-like traveling wavy plate, *International Journal for Numerical Methods in Fluids*, 48(12), pp.1351-1373.
Lu, X.Y., and Yin, X.Z. (2005), Propulsive performance of a foil-like traveling wavy wall, *Acta Mechanica*, 175(1-4), pp.197-215.

TECHNICAL CONTRIBUTIONS

Hydro Meteorology