Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
Ser.B, 2006,18(6): 681-687
sdlj.chinajournal.net.cn

NUMERICAL STUDY ON PROPULSIVE PERFORMANCE OF FISH-LIKE SWIMMING FOILS*

DENG Jian, SHAO Xue-ming, REN An-lu
Department of Mechanics, State Key Laboratory of Fluid Power Transmission and Control, Zhejiang University, Hangzhou 310027, China, E-mail: mecsxm@public.zju.edu.cn

(Received October 19, 2005)

ABSTRACT: Two-dimensional numerical simulations are performed to study the propulsive performance of fish-like swimming foils using the immersed-boundary method. A single fish as well as two fishes in tandem arrangement are studied. First, the effect of the phase speed on the propulsive performance of a single fish is analyzed. The wake structures and pressure distribution near the wavy fish are also examined. The results show good correlation with those by previous researchers. Second, two tandem fishes with the same phase speed and amplitude are studied. The results show that the fish situated directly behind another one endure a higher thrust than that of a single one.

KEY WORDS: biomechanics, fish-like swimming, traveling wavy foil, immersed-boundary method

1. INTRODUCTION

The study of the movement of fishes can be very informative in exploring mechanisms of unsteady flow control because movements in fish is a result of many millions of years of evolutionary optimization. Previous studies have shed light on the inviscid hydromechanics of fish-like propulsion, while predicting high propulsive efficiency. With the development of experimental equipments and novel numerical methods, there has been a better understanding of the principles of fish-like swimming. Some recent works have been reviewed in Refs. [1-3]. To better investigate the swimming ability of fishes, the wave-like swimming and flapping motions of the body are used as essential models to deal with the movement of fish. Previous researchers have shown the ability of caudal fin of a fish to produce a jet-like wake similar to that of a flapping foil [4-7]. The numerical investigation on a fish-like traveling wavy plate and a smooth wavy wall undergoing transverse motion has also been performed [8,9]. Recently, a NACA0012 foil has also been used as the profile of the body at an equilibrium position of undulating motion [10].

In this study, the immersed-boundary method is used to investigate the viscous flow over the body of fishes. The NACA0012 foil, which undergoes a traveling wave motion, is employed to represent the profile of the body of the fishes. The flow structures and propulsive performance of the fish for different swimming parameters are discussed. Two fishes in tandem arrangement with the same parameters are also examined.

2. PHYSICAL MODEL AND NUMERICAL METHOD

As shown in Fig. 1, the NACA0012 foil with a traveling wavy motion is considered in this article. The length of the foil is L, and the free-stream velocity is U. The motion of the foil is described as

$$y_s = A_m(x)\cos[2\pi\alpha(x-ct)],\ 0\le x\le 1 \quad (1)$$

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.

* Project supported by the National Natural Science Foundation of China (Grant No: 10472104) and National Laboratory of Hydrodynamics of China.
Biography: DENG Jian (1981-), Male, Ph. D. Student

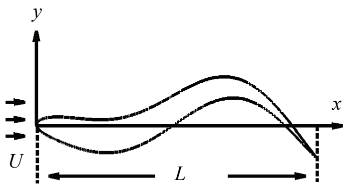

Fig.1 Physical model and definition of the coordinates system

As suggested by previous researchers [8,9], to model the backbone undulation of fish swimming, the amplitude is approximated to be in the form of a quadratic polynomial as

$$A_m(x) = C_0 + C_1 x + C_2 x^2 \tag{2}$$

In the simulations carried out in this study, the coefficients C_2 and C_0 are set equal to zero for simplification.

An immersed-boundary method-based solver proposed by Zou et al.[11] and Deng et al.[12] is employed to simulate the fluid flow over the foil. The advantage of this method is that the complexity and cost of generating a body-conformal mesh at each time-step is eliminated, thereby reducing amount of the resources required to perform such simulations. Inside the obstacle body, a body forces added at the grid point, which is directly calculated using the expression given by

$$\boldsymbol{F}^{n+1} = -\boldsymbol{RHS}^n + \frac{\boldsymbol{U}^{n+1} - \boldsymbol{u}^n}{\Delta t} \tag{3}$$

where $\boldsymbol{U}^{n+1}$ is the velocity of solid boundary point at current time level $t+\Delta t$ and $\boldsymbol{u}^n$ is the corresponding fluid velocity at t time level. The term $\boldsymbol{RHS}^n$ contains the convective, viscous, and pressure gradient terms in momentum equation at t time level. For the grid points outside the solid body, a certain interpolation procedure,

$$\boldsymbol{F}(x_f, y_f) = (1 - \mathrm{d}s / \mathrm{d}xy)\boldsymbol{F}(x_s, y_s)$$

is needed to weaken the added body-force. Here $\mathrm{d}xy$ is the diagonal length of a grid element. The subscript f denotes the grid point in the fluid, and the subscript s denotes a corresponding virtual point on the boundary, and $\mathrm{d}s$ is the distance between point f and point s. It should be noted that the body-force is equal to zero for the grid points where the condition $\mathrm{d}s > \mathrm{d}xy$ is satisfied.

The nondimensional Navier–Stokes equations for incompressible viscous flow are written as follows,

$$\nabla \cdot \boldsymbol{V} = 0, \frac{\mathrm{D}\boldsymbol{V}}{\mathrm{D}t} = -\nabla p + \frac{1}{Re}\nabla^2 \boldsymbol{V} + \boldsymbol{F}_{add} \tag{4}$$

where $\boldsymbol{F}_{add}$ is the added force. The Euler-explicit time discretization scheme is applied for convective terms and the second-order-implicit Crank–Nicholson scheme is used for viscous terms. Spatial derivatives are discretized by second-order central finite difference. The pressure variable is solved using the pressure Poisson equation derived by applying the divergence operator to the momentum equations.

A constant streamwise velocity has been used at the inlet and the lateral boundaries. The boundary condition at the outlet is set as $\partial \boldsymbol{u}/\partial t + u_a \partial \boldsymbol{u}/\partial n = 0$, where u_a is the averaged streamwise velocity at the outlet. A pressure Neumann condition is applied to inflow, far field, and outflow boundaries.

3. METHOD VALIDATION

To validate the numerical method, the flow around a circular cylinder is simulated. A rectangular domain is employed. The boundary conditions are imposed in such a way that the flow is from the left toward the right of the domain. A circular cylinder is placed inside the domain so that its center is 8D away from the inlet and 25D away from the outlet, where D denotes the cylinder diameter. The domain has a transverse dimension of 16D. These dimensions have been chosen to minimize the boundary effects on the flow development.

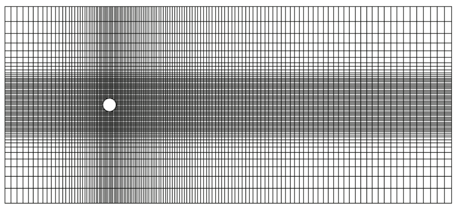

Fig.2 Nonuniform mesh used for the flow past a two-dimensional circular cylinder. Only alternate grid lines are shown in both the directions

Figure 2 shows the 275 × 156 nonuniform mesh used in the present studies, and the grid near the Cylinder is uniform and has a constant spacing. A

Download English Version:

https://daneshyari.com/en/article/1722966

Download Persian Version:

https://daneshyari.com/article/1722966

Daneshyari.com