VII International Conference on Computational Methods in Marine Engineering
MARINE 2017
M. Visonneau, P. Queutey and D. Le Touzé (Eds)

SIMULATION OF FISH ESCAPE AND SWIMMING TOWARD A PREDEFINED GOAL

Seyed-Amin Ghaffari*, Stéphane Viazzo††, Kai Schneider† and Patrick Bontoux††

* Société Hydro-Mécanique Marine, 13127 Vitrolles, France
e-mail: s.amin.ghaffary@gmail.com

† I2M - UMR 7373 - CNRS, Aix-Marseille Université
Centre de Mathematiques et d'Informatique
39 rue Joliot-Curie, 13453 Marseille, France
e-mail: kai.schneider@univ-amu.fr

†† M2P2 - UMR 7340 - CNRS, Aix-Marseille Université
38 rue Joliot-Curie, 13451 Marseille, France
Web page: http://www.m2p2.fr

Key words: Fluid interaction with forced deformable bodies, Compact finite difference, Volume penalization, Fish escape, Wake of a swimming fish

Abstract. We present an immersed boundary method for numerical simulation of a swimming fish. The vorticity transport equation is solved on a Cartesian grid using compact finite differences. A Lagrangian structured grid defines the fish body and is moving in the surrounding incompressible flow due to the exerted hydrodynamic forces and the torque. An efficient law determining the curvature of a swimming fish is presented which is based on the geometrically exact theory of nonlinear beams and quaternions. Validation of the solver shows the efficiency and expected accuracy of the algorithm for swimming fish simulations. The structure of the wake of a swimming fish is studied, and some common features with the wake of a flapping plate are demonstrated.

1 INTRODUCTION

Investigation about swimming can help researchers to provide insights into nature, biology, fluid mechanics, fluid structure interaction, etc. On the other hand, this can help engineers to design fast boats, innovative bioinspired propulsion systems, like robofishs and kayaks fitted with new propulsion systems. As an example a Mirage Drive peddle system is used in a new kayak, in which two underwater flexible flippers moving to right and left, producing a forward propulsion. In this regard, many experimental and numerical studies were performed considering the hydrodynamics of swimming.

Three-dimensional numerical simulation of swimming have only become possible in the last decade. Even if due to the shape and deformation style of the fish-like swimmers the surrounding flow is fully three-dimensional, most of the fundamental features of swimming are included in two-dimensional analyses. Therefore, in the present investigation we study the rotation and escape of a two-dimensional swimmer. The well accepted mathematical model for incompressible flows are the Navier–Stokes equations. The presence of nonlinearity in the governing equations restricts the analytical solutions to simple domains. In recourse to numerical simulations of the Navier–Stokes equations, according to many researchers the vorticity stream-function formulation proved to be efficient for two-dimensional simulations.

A common discussion in numerical simulations would be the grid spacing in accordance with the Reynolds number. For high Reynolds number flows a direct numerical simulation (DNS), i.e., using a grid spacing capable to capture all length scales present in the flow field, would not be a possible practice. Moreover, the nonlinearity promotes the creation of small scales. The small scales (subgrid scales) need to be modeled to avoid blow-up (instabilities) in the flow field. A common practice is to use a turbulence model via adding an eddy viscosity. Upwinding, adding artificial dissipation or applying a low-pass filter to the flow field are equivalent tricks. In the present methodology, an approach similar to implicit LES is applied to the flow field, to dissipate the transfered energy toward high wave numbers. We apply a low-pass spatial-filtering in accordance with the high-order compact differencing to the equations. This enables us to use a tailored grid when high Reynolds flow simulations are envisaged.

Simulation of swimming lead to a fluid-structure interaction problem, i.e., from numerical view point we have a moving boundary problem. By using an immersed boundary method (IBM), see e.g. Mittal and Iaccarino [2] and Schneider [7], a Cartesian grid and therefore a high-order and efficient method can be used. This considerably increases the accuracy far from the solid boundaries, on the other hand the accuracy near moving boundaries remains acceptable, i.e., between first and second order accuracy can be achieved. However, some researchers developed higher-order IBMs, see for example Linnick and Fasel [3]. In the present investigation, the motion of the fish is imposed, thus we reconstruct the boundaries of the fish at each time step. We then use Newton–Euler equations to move and rotate the body of the fish around the reference point. The boundary of the fish is described exactly within the grid space accuracy.

In solving the incompressible Navier–Stokes equations an elliptic Poisson equation is typically encountered which is the most time consuming part of the algorithm. In the vorticity stream-function formulation, an elliptic equation has to be solved with Dirichlet boundary condition for vorticity and stream-function. We are using the fourth-order direct method proposed by Ghaffari et al. in [6] for an efficient solution of the Poisson equation.

2 METHODOLOGY

The governing equations of incompressible flows are the Navier–Stokes equations. For two-dimensional flows, by taking the curl of the Navier–Stokes equations, the vorticity transport equation can be derived as follows:

$$\partial_t \omega + (\mathbf{u} \cdot \nabla)\omega = \nu \nabla^2 \omega + \nabla \times \mathbf{F} \;\;, \;\; \mathbf{x} \in \Omega \in \mathbb{R}^2 \tag{1}$$

where $\omega(\mathbf{x}, t) = \nabla \times \mathbf{u} = v_x - u_y$ denotes the vorticity, Ω is the spatial domain of interest, $\mathbf{u}(\mathbf{x}, t)$ is the velocity field, $\nu = \mu/\rho_f > 0$ is the kinematic viscosity of the fluid, ρ_f is the density

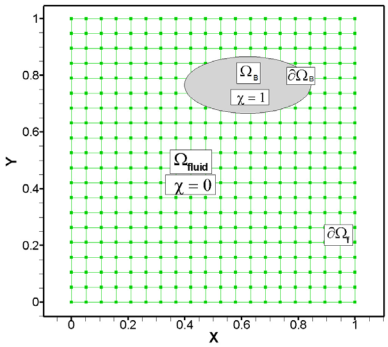


Figure 1: Domain of the solution and the immersed body, $\Omega = \Omega_f \cup \Omega_B$.

and $\mathbf{F}(\mathbf{x}, t)$ is a force function. For a complete description of a particular problem, the above equations need to be complemented to describe an initial/boundary value problem. The velocity components are $(u, v) = (\partial_y\psi, -\partial_x\psi)$, with ψ satisfying the Poisson equation

$$-\nabla^2\psi = \omega \tag{2}$$

To take into account a deformable body which moves in the solution domain the volume penalization method is used. The volume penalization method belongs to the family of immersed boundary methods, we refer to Mittal and Iaccarino [2] and Schneider [7] for more details. The forcing function is representative of the immersed body and is defined as follows:

$$\mathbf{F} = -\eta^{-1}\chi(\mathbf{u} - \mathbf{u}_B) \tag{3}$$

where $\mathbf{u}_B(\mathbf{x}, t)$ is the velocity field of the immersed body. The penalization parameter η represents the permeability coefficient of the immersed body. The mask (or characteristic) function χ is dimensionless and describes the geometry of the immersed body

$$\chi(\mathbf{x}, t) = \begin{cases} 1 & \mathbf{x} \in \Omega_B \\ 0 & \mathbf{x} \in \Omega_f \end{cases} \tag{4}$$

where Ω_f represents the domain of the fluid and Ω_B represents the immersed body in the domain of the solution (see Fig. 1). The solution domain $\Omega = \Omega_f \cup \Omega_B$ is governed by the Navier-Stokes equations in the fluid regions and by a Darcy–Brinkmann law in the penalized regions, in the limit when $\eta \to 0$. For fluid/solid interaction problems the simulations start with the body $\mathbf{u}_B(\mathbf{x}, 0) = 0$, and fluid at rest, i.e., $\omega(\mathbf{x}, 0) = \psi(\mathbf{x}, 0) = 0$ and free-slip boundary conditions are imposed at the surrounding walls ($\psi|_{\partial\Omega} = \omega|_{\partial\Omega} = \partial p/\partial\mathbf{n}|_{\partial\Omega} = 0$). For time integration we are using classical fourth-order Runge–Kutta method. The spatial discretization and filtering are based on compact finite differences, we refer to Ghaffari et al. [6] for more details.

3 APPLICATION AND RESULTS

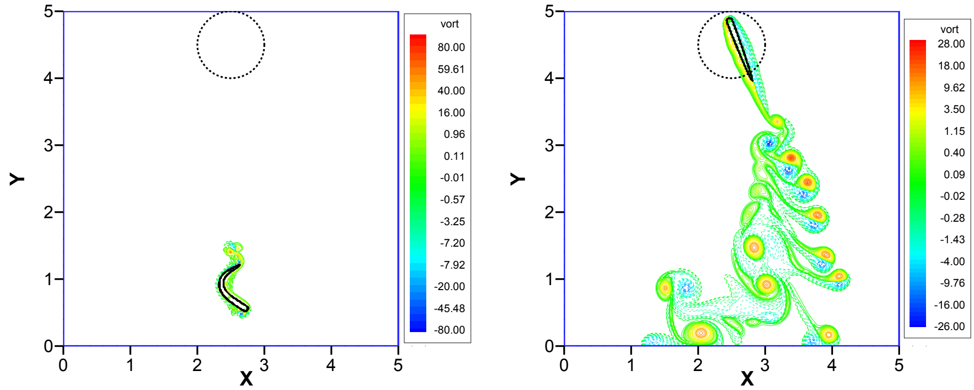
To perform simulations of a fish swimming toward a predefined goal, a domain of $(x, y) \in [0, 5l_{\text{fish}}] \times [0, 5l_{\text{fish}}]$ is chosen, with $l_{\text{fish}} = 1$. The Reynolds number is defined as $Re = \overline{U}l_{\text{fish}}/\nu$, where $\overline{U}$ is the average swimming speed and ν is the kinematic viscosity of the fluid. The backbone of the fish is performing an imposed undulatory periodic motion according to

$$y(x,t) = (a_0 + a_1 x + a_2 x^2)\sin(2\pi(x/\lambda + ft)) \tag{5}$$

where $\lambda = f = 1$, $a_2 = 0$, $a_1 = 0.12\overline{12}$, $a_0 = 0.00378\overline{78}$. Considering the fact that curvature is a more convenient framework for geometry construction, from two successive derivatives of Eq. (5) the needed curvature for the propulsion mode can be derived. We refer to Boyer et al. [4] for details of the backbone definition with the use of quaternions. The motion of the backbone is starting smoothly with a cosine type rise function $x \in [\pi, 2\pi]$, in the first period and tends to zero in the same manner after reaching the goal. The geometry of the fish is defined by the half width of the body along its arclength, which is superimposed on the curve of the backbone given by Eq. (5). We refer to Carling et al. [1] for details of geometry definition and to Ghaffari et al. [6] for details on rotation control. An important point in swimming is to identify the origins and evolution of the vortices. A street of single and pair vortices constructs the morphology of the wake. The vortices are stronger near the swimming fish and are weaker away from the fish because of the damping effect of fluid viscosity. The motion of the vortices is a self induced movement. They repulse or attract each other. They can merge according to their sign and surrounding vortices. The weaker ones are more subjected to merge with the stronger ones. Depending on the Reynolds and Strouhal numbers, the tail-beat amplitude and the trajectory of the fish, different wakes can be observed. According to Schnipper et al. [5] from 2 up to 16 vortices per oscillation period can be created in pair or single configurations. As a test case, a simulation of a fish swimming toward a predefined goal, located exactly in its backside, is considered ($Re \approx 3500$). In our simulations, in each stroke (one tail-beat) approximately 4 vortices are created. We observe two strong and two weak vortices. Within 15 strokes ($t = 15$), 10 pairs (dipole) and 10 single vortices can approximately be identified (see Fig. 2). In the second test case ($Re \approx 5000$) the fish is swimming toward a predefined goal located in the front. Within 20 strokes ($t = 20$), 12 pairs (dipole) and 15 single vortices can approximately be identified (see Fig. 3). Some weaker vortices are merging with the stronger ones just after their creation in the wake. The pair (dipole) vortices are stronger than the single ones, thus persisting longer in the flow field. The wake is stable during the swimming and becomes unstable after reaching the goal and stopping the stroke.

4 CONCLUSIONS

- The structure of the wake of a swimming fish is studied, and some common features with the wake of a flapping plate are demonstrated. A street of single and pair vortices constructs the morphology of the wake.
- In each stroke (one tail-beat) approximately 4 vortices are created. We observe two strong and two weak vortices.
- Some weaker vortices are merging with in the stronger ones just after their creation in the wake.
- The pair (dipole) vortices are stronger than the single ones and thus persist in the wake even after stopping the stroke.



(a) $t = 1.25$ (b) $t = 15$

Figure 2: Vorticity fields of a fish swimming toward a predefined goal, located in its backside, $Re \approx 3500$. Within 15 strokes ($t = 15$), 10 pairs (dipole) and 10 single vortices can approximately be identified.

- The wake is stable during the swimming and becomes unstable after reaching the goal and stopping the stroke.

REFERENCES

[1] J. Carling, T. L. Williams and G. Bowtell, Self-propelled anguilliform swimming: simultaneous solution of the two-dimensional Navier-Stokes equations and Newton's laws of motion. *Journal of Experimental Biology*, Vol. 201, 3143-3166, 1998.

[2] R. Mittal and G. Iaccarino, Immersed boundary methods. *Annual Reviews of Fluid Mechanics*, Vol. 37, 239-261, 2005.

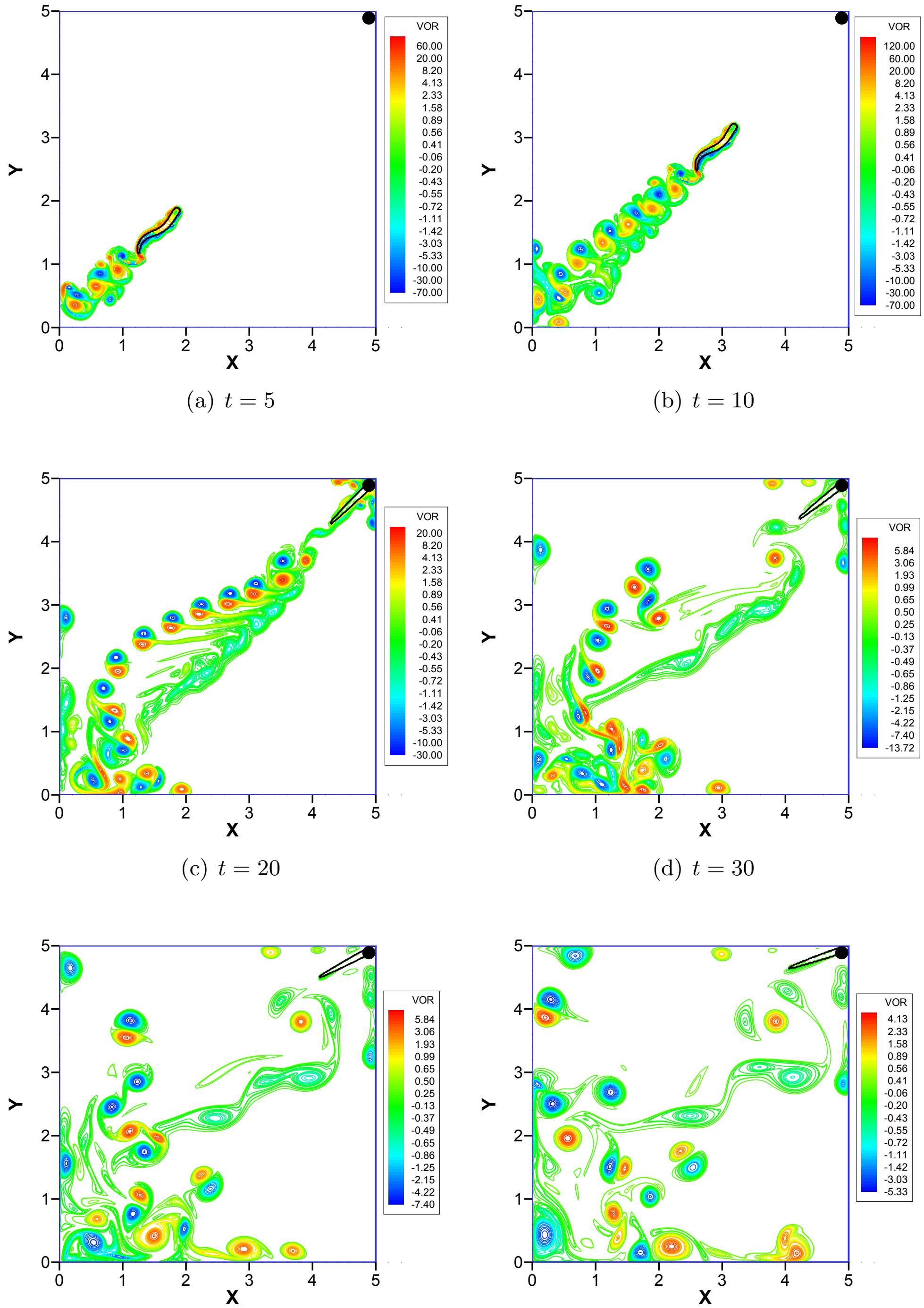
[3] M. N. Linnick and H. F. Fasel, A high-order immersed interface method for simulating unsteady incompressible flows on irregular domains. *Journal of Computational Physics*, Vol. 204, 157-192, 2005.

[4] F. Boyer, M. Porez and W. Khalil, Macro-continuous computed torque algorithm for a three-dimensional Eel-like robot. *IEEE Transactions on Robotics*, Vol. 22, No. 4, 763-775, August 2006.

[5] T. Schnipper, A. Andersen and T. Boher, Vortex wakes of a flapping foil. *Journal of Fluid Mechanics*, Vol. 633, 411-423, 2009.

[6] S. A. Ghaffari, S. Viazzo, K. Schneider and P. Bontoux, Simulation of forced deformable bodies interacting with two-dimensional incompressible flows; Application to fish-like swimming. *International Journal of Heat and Fluid Flow*, Vol. 51, 88–109, 2015.

[7] K. Schneider, Immersed boundary methods for numerical simulation of confined fluid and plasma turbulence in complex geometries: a review. *J. Plasma Phys.*, Vol. 81(6), 435810601, 2015.



(a) $t = 5$ (b) $t = 10$

(c) $t = 20$ (d) $t = 30$

(e) $t = 40$ (f) $t = 50$

Figure 3: Vorticity fields of a fish swimming toward a predefined goal, located in the front, $Re \approx 5000$. Within 20 strokes ($t = 20$), 12 pairs (dipole) and 15 single vortices can approximately be identified.