Analysis of Flow around Robotic Fish by Three-dimensional Fluid-structure Interaction Simulation and Evaluation of Propulsive Performance

Yogo Takada\textsuperscript{a,}\textsuperscript{*}, Toshinori Ochiai\textsuperscript{a}, Noboru Fukuzaki\textsuperscript{a}, Tomoki Tajiri\textsuperscript{a} and Tomoyuki Wakisaka\textsuperscript{a}

\textsuperscript{a}Osaka City University, Japan

Abstract—The speed-up of robotic fish is hoped to aim at practical use. It is therefore important to develop methods for making such robots swim faster, and computer simulations are invaluable. In this study, a computational simulation model was developed for three-dimensional fluid-structure interaction analysis. The flow around the caudal fin can be analyzed by treating the fin as an elastic body. Experiments were also carried out and good agreement was found between the experimental results and those of the numerical analysis. Moreover, the effects of the flexibility of caudal fin on the robot’s propulsive performance were investigated using computational simulations.

Index Terms—Robotic fish, Swimming, Fluid-structure interaction analysis, Computational Fluid Dynamics.

I. INTRODUCTION

There has been a long history of research on the propulsion mechanism of underwater creatures in terms of the interaction of their caudal fins with the surrounding fluid \cite{1}\cite{2}\cite{3}. Caudal-fin-driven robotic fish that imitate live fish do not alarm other underwater creatures. Therefore, robotic fish are useful for purposes such as investigation of underwater ecosystems and environmental surveillance. Moreover, since such robotic fish do not use a screw propeller, they are unlikely to become entangled in water plants.

The swimming style of fish can be roughly divided into anguilliform, carangiform and ostraciiform locomotion. Moreover, carangiform fishes can be subdivided to two groups, of which carps and tunas are representative. Robotic fish that are based on carps, and use their caudal fin for propulsion, swim very slowly compared to live fish, who can swim a distance of three times their own length in one second. The propulsion performance of robotic fish is also much lower than that of live fish. For applications such as ecosystem investigation, it is necessary to achieve the same cruising speed as the fish being investigated, so improvements are necessary to current robots. We have previously reported that the use of a caudal fin with moderate flexibility can improve the propulsive performance of a robotic fish \cite{4}. However, a fin configuration that ensures high-speed swimming has not yet been determined. When robotic fish are constructed for trial purposes, because the optimum fin conditions depend on the type of fish, repeated experiments are required in order to determine this. Since this is a time-consuming process, a more attractive solution is to first optimize the fin conditions using computer simulations. Moreover, because the interaction of the caudal fin with the surrounding fluid can be examined in detail, the most efficient swimming mechanism can be determined for the robotic fish.

We have been carrying out numerical flow analyses of the movement of live and robotic fish \cite{5}. However, the motion of the caudal fin was investigated using a one-dimensional (1D) beam model, and deformations of the fin in three dimensions were not considered.

The purpose of the present study is to carry out a 3D fluid-structure interaction analysis in order to clarify the effects of the caudal fin flexibility on the propulsive performance of a robotic fish. For this purpose, a 3D elastic deformation analysis using the finite element method (FEM) was combined with existing 3D computational fluid dynamics (CFD) code \cite{4}. A weak coupling method was employed, in which multiple phenomena described by different governing equations are alternately solved. Using this approach, the propulsive performance of robotic fish that swim using carangiform locomotion was investigated for different fin configurations.

II. THREE-DIMENSIONAL FLUID-STRUCTURE INTERACTION SIMULATION

A. Method of flow analysis

The general conservation equation, describing either mass or momentum conservation in the flow, can be written as

$$\frac{\partial (\rho \phi)}{\partial t} + \nabla (\rho \vec{v} \phi - \Gamma_{\phi} \nabla \phi) = S_{\phi},$$  
where $\phi$ is the induced variable, $\Gamma_{\phi}$ is the diffusion coefficient and $S_{\phi}$ is the source term. Because the subject of analysis in this study is flow around a moving object such as a fish or robot, the fluid was analyzed as a moving boundary problem in which the computational grid moves. Under such circumstances, Eq. (1) can be expressed as

\textsuperscript{*} Corresponding author: 3-3-138 Sugimoto,Sumiyoshi-ku, Osaka-shi, Osaka, 558-8585, Japan, takada@mech.eng.osaka-cu.ac.jp.
\[
\frac{\partial (\rho \phi)}{\partial t} + \text{div} \left( \rho (\vec{v} - \vec{v}_g) \phi - \Gamma_\phi \text{grad} \phi \right) = S_\phi ,
\]
(2)

where \( \vec{v}_g \) is the movement velocity of the computational grid. Table 1 shows the induced variable \( \phi \), effective diffusion coefficient \( \Gamma_\phi \) and source term \( S_\phi \) in Eq. (2). Because the Reynolds number for the fluid in this study is very small, the flow can be considered to be laminar. Since the fluid around the robotic fish is normal water, an incompressible fluid at a constant temperature was used in the calculations.

Table 1 Induced variable \( \phi \), effective diffusion coefficient \( \Gamma_\phi \) and source term \( S_\phi \) in each conservation equation.

<table>
<thead>
<tr>
<th>Equation</th>
<th>( \phi )</th>
<th>( \Gamma_\phi )</th>
<th>( S_\phi )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass</td>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Momentum</td>
<td>( u ) ( \mu ) ( \frac{\partial p}{\partial x} + \frac{1}{3} \frac{\partial}{\partial x} (\mu_{\text{eff}} \text{div} \vec{v}) )</td>
<td>( v ) ( \mu ) ( \frac{\partial p}{\partial y} + \frac{1}{3} \frac{\partial}{\partial y} (\mu_{\text{eff}} \text{div} \vec{v}) )</td>
<td>( w ) ( \mu ) ( \frac{\partial p}{\partial z} + \frac{1}{3} \frac{\partial}{\partial z} (\mu_{\text{eff}} \text{div} \vec{v}) )</td>
</tr>
</tbody>
</table>

The GTT method used in this study adopted the technique of conversion into generalized curvilinear coordinates, based on the Tank and Tube method developed by Gosman [6] and Spalding [7]. In this method, a physical space expressed by a Cartesian coordinate system \((x,y,z,t)\) is first divided into small hexahedral volume elements of arbitrary shape. Each control volume is then converted into a cubic Cartesian coordinate system \((x,y,z)\). The allocation of data is done in the center of each control volume. The pressure correction equation is solved using the SIMPLER method [8], where the pressure is calculated by satisfying the continuity equation.

### B. Method of elastic deformation analysis

In this research, the caudal fin of a robotic fish is assumed to be a linear elastic body, and its deformation is modeled by an equilibrium equation considering the inertia force. The Newmark-\( \beta \) method [9] is used for time discretization and FEM is used for space discretization.

\[
\nabla \cdot \{\sigma \} + \{f\} = \rho \left\{ \frac{\partial^2 \vec{u}_b}{\partial t^2} \right\}
\]
(3)

Here, \( \nabla \) is the nabla differential operator, \( \{\sigma\} \) is the stress tensor, \( \{f\} \) is the body force vector and \( \rho \) is the density.

For the FEM calculations, the entire structure to be analyzed is divided into hexahedral elements of a certain serendipity family, each containing 20 nodes. Figure 1 shows an example of one such element. If the displacement vector at an arbitrary point in each element is defined as \( \{\vec{u}_b\}^T = \{u_B, v_B, w_B\} \), then \( \{\vec{u}_b\}^T \) can be expressed using Eq. (4) in terms of the displacement at each node.

\[
\{u_B\}^T = \sum_{i=1}^{20} N_i \{u_{B_i}\}^T \quad \text{for Eq. (7)}
\]
(4)

Here, \( N_i \) is the shape function for the hexahedral element, \( u_{B_i} \) is the displacement of node \( i \) along the \( x \)-axis, \( v_{B_i} \) is the displacement of node \( i \) along the \( y \)-axis and \( w_{B_i} \) is the displacement of node \( i \) along the \( z \)-axis. Therefore, the displacement of each element can be expressed in matrix form as

\[
\{u_B\} = [N]\{u_B^*\}
\]
(5)

where \([N]\) is expressed by the following matrix and \(\{u_B^*\} = [u_{B1}, v_{B1}, w_{B1}, u_{B2}, v_{B2}, w_{B2}, \ldots, u_{B20}, v_{B20}, w_{B20}]\).

If the virtual displacement \(\{\delta u_B\}\) is set as the weighting function, and the fundamental Eq. (3) is integrated for a single element, Eq. (6) is obtained. In addition, applying Green’s theorem to Eq. (6) yields

\[
\iiint_v \{\delta u_B\}^T \{\nabla \cdot \{\sigma\}\} dV + \iiint_v \{\delta u_B\}^T \{f\} dV = \iiint_v \{\delta u_B\}^T \rho \left\{ \frac{\partial^2 \vec{u}_b}{\partial t^2} \right\} dV
\]
(6)

\[
\int_s \{\delta u_B\}^T \{\sigma\} n dS - \iiint_v \{\delta e\}^T \{\sigma\} dV + \iiint_v \{\delta u_B\}^T \{f\} dV = \iiint_v \{\delta u_B\}^T \rho \left\{ \frac{\partial^2 \vec{u}_b}{\partial t^2} \right\} dV
\]
(7)

Using space discretization, Eqs. (8)-(11) can be derived from the first, second and third term on the left side of Eq. (7) and the term on the right, respectively:

\[
\int_s \{\delta u_B\}^T \{\sigma\} n dS = \{\delta u_B\}^T \{p^*\}
\]
(8)

\[
\int_v \{\delta e\}^T \{\sigma\} dV = \{\delta u_B\}^T \{k\} \{u_B\}
\]
(9)

\[
\int_v \{f\}^T \{\delta u_B\} dV = \{\delta u_B\}^T \{f^*\}
\]
(10)

where \([B]\) is the displacement-strain conversion matrix, and \([D]\) is the elasticity coefficient matrix.

The dynamic equation for an element is shown in Eq. (12), as a result of arranging after substituting Eqs. (8)-(11) for Eq. (7).

\[
[m] \left\{ \frac{\partial^2 \vec{u}_b}{\partial t^2} \right\} + [k]\{u_B\} = \{f^*\} + \{p^*\}
\]
(12)
If Eq. (12) is applied to all elements, the dynamic equation for all elements can be expressed as

$$[M] \frac{\partial^2 \mathbf{u}_t}{\partial t^2} + [K] \{\mathbf{u}_t\} = \{F\} ,$$  \hspace{1cm} (13)

where $[M]$ is the mass matrix for all elements, $[K]$ is the stiffness matrix for all elements, and $\{F\}$ is the external force vector.

When accelerated motion is added to the elastic body to swing the fin, the displacement vector $\{\mathbf{u}_g\}$ can be divided into the displacement of an elastic body $\{\mathbf{u}_d\}$ and the movement of a rigid body $\{\mathbf{u}_m\}$, as shown in Fig. 2.

![Fig. 2. Division into movement of rigid body and displacement of elastic body](image)

In this study, the movement of a rigid body and displacement of an elastic body are analyzed separately. Moreover, the displacement is updated based on incremental strain theory using an updated Lagrangian approach [10][11].

Thus, the dynamic Eq. (13) can be expressed as Eq. (14).

$$[M] \left[ \frac{\partial^2 \mathbf{u}_m}{\partial t^2} \right]_{t+\Delta t} + [M] \left[ \frac{\partial^2 \mathbf{u}_d}{\partial t^2} \right]_{t+\Delta t} + [K]\{\mathbf{u}_d\}_{t+\Delta t} = \{F\}_{t+\Delta t} - [K]\{\mathbf{u}_d\}_t$$  \hspace{1cm} (14)

$$\{\mathbf{u}_m\}_{t+\Delta t} = \{\mathbf{u}_m\}_t + \{\Delta \mathbf{u}_m\}_t$$  \hspace{1cm} (15)

$$\{\mathbf{u}_d\}_{t+\Delta t} = \{\mathbf{u}_d\}_t + \{\Delta \mathbf{u}_d\}_t$$  \hspace{1cm} (16)

Here, $\{\Delta \mathbf{u}_m\}$ and $\{\Delta \mathbf{u}_d\}$ are increments to $\{\mathbf{u}_m\}$ and $\{\mathbf{u}_d\}$ respectively.

Equation (13) does not describe the actual dynamic behavior because it does not contain a damping term. However, such damping is difficult to express because the damping mechanism in oscillatory phenomena is complex. For this reason, a proportional damping model [12] was used to take into account the effect of the damping force on the calculations; such a model is commonly utilized in the field of vibration and structural analysis. The dynamic equation is then transformed as follows:

$$[M] \left[ \frac{\partial^2 \mathbf{u}_m}{\partial t^2} \right]_{t+\Delta t} + [M] \left[ \frac{\partial^2 \mathbf{u}_d}{\partial t^2} \right]_{t+\Delta t} + [C] \left[ \frac{\partial \mathbf{u}_d}{\partial t} \right]_{t+\Delta t} + [K]\{\mathbf{u}_d\}_{t+\Delta t} = \{F\}_{t+\Delta t} - [K]\{\mathbf{u}_d\}_t$$  \hspace{1cm} (17)

$$[C] = C_M[M] + C_K[K]$$  \hspace{1cm} (18)

where, $C_M$ and $C_K$ are proportionality constants.

C. Experimental verification of elastic deformation calculation

The validity of the calculation method for elastic deformation based on Eq. (3) was next investigated experimentally. As described earlier, the FEM was used for spatial discretization and the Newmark-$\beta$ method for time discretization. Figure 3(a) shows a photograph of the experimental apparatus. As shown in Fig. 3(b), the beam consisted of an elastic section with a length of 400 mm and a rigid section with a length of 50 mm. The rigid section was connected at one end (rotation center O) to a rotating servo motor. A weight of 30 g was attached to one end of the elastic section (point A). The servo motor (RB995b; MiniStudio, Inc.) had a rated voltage of 4.8 V and a rated torque of 0.833 Nm. The beam rotated through an angle of $\pi$ rad. The behavior of the beam was investigated by measuring the position of point A using a CCD camera installed above the beam. The physical properties of the elastic beam are shown in Table 2. As shown in Fig. 3(b), the $x$-axis is towards the right and the $z$-axis is upward.

<table>
<thead>
<tr>
<th>Beam length</th>
<th>400 mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>30 g</td>
</tr>
<tr>
<td>Beam line density $\rho A$</td>
<td>0.0249 kg/m</td>
</tr>
<tr>
<td>Beam flexural rigidity $EI$</td>
<td>0.0213 Nm$^2$</td>
</tr>
</tbody>
</table>

The turning angle $\theta$ at point O between the rigid section of the beam and the $z$-axis was first increased from 0 to $\pi$ rad during a period of 2 seconds. The motor was then stopped to maintain this angle. This is expressed as

$$\theta = \pi$$  \hspace{1cm} (19)

$$\theta = \frac{\pi}{2} \sin \left( \frac{t}{2} - 0.5 \right) + 1$$  \hspace{1cm} $0 \leq t < 2$

$$\theta = \frac{\pi}{2}$$  \hspace{1cm} $2 \leq t$

Figure 3(c) shows simulated and experimental time course results for the $x$-coordinate of point A. The filled squares represent the experimental points, the dashed line represents the simulation results without damping, and the solid line represents the simulation results with damping. It can be seen that when damping is taken into consideration, there is good agreement between the experimental and simulation results.

D. Coupling of CFD and deformation calculations

The methods used for fluid analysis and elastic deformation analysis have been described in the foregoing section. In this section, the method by which these analyses are coupled is described. Either strong or weak coupling methods can be used.

In the weak coupling method, the governing equations for the fluid and the structure are solved independently. Calculations are then performed to ensure dynamic equilibrium and geometric continuity at the boundary between the fluid and the structure. This method is inferior to the strong coupling method in
terms of calculation accuracy and stability. However, different discretization techniques can be adopted to solve the governing equations for the fluid and structure.

In this study, 3D CFD code (GTT code) using the finite volume method is adopted for the fluid analysis. It was decided to use the weak coupling method because it makes it easier to combine the elastic deformation analysis with the existing GTT code. The pressure at the fluid-structure boundary surface determined by the fluid analysis is used in the elastic deformation analysis to provide dynamic boundary conditions for the beam. Moreover, the displacement of the boundary surface determined by the elastic deformation analysis is used in the fluid analysis to provide geometrical boundary conditions. This approach ensures dynamic equilibrium and geometric continuity at the fluid-structure boundary surface. A flow diagram of the fluid-structure interaction simulation used in this study is shown in Fig. 4. It involves the following steps:

[Step 1] The time is increased by an interval $\Delta t$.
[Step 2] The computational grid for the fluid analysis is created based on the displacement of the fluid-structure boundary surface determined in the former time step.

The Poisson equation with the Dirichlet boundary condition is solved to make the complex computational grid in this step, and the grid is then smoothed.

[Step 3] The fluid is analyzed by the GTT code using the computational grid created in Step 2. The calculations are iterated until the solution converges since the fluid exhibits nonlinear behavior.

[Step 4] The converged solution for the pressure obtained in Step 3 is used as a dynamic boundary condition for the structure, and the deformation analysis is carried out using the FEM.

The non-stationary fluid-structure interaction analysis is conducted by repeating Step 1 to Step 4.

III. EXPERIMENTAL ROBOTIC FISH

To verify the validity of the fluid-structure interaction simulation program used in this study, experiments were carried out using a robotic fish model and the results compared to the simulations. The amount of deformation of the caudal fin of the robotic fish was measured and the flow behind the robot was evaluated using particle image velocimetry (PIV).

Figure 5(a) shows a photograph of the robotic fish, and Fig. 5(b) shows a structural drawing. Since the purpose of this robot was to verify the results of the numerical analysis, it was made as simple as possible, and components such as driving motors and electrical circuits were positioned above the water surface. Servo motors were used to easily control joint angles in order to drive the robot. The servo motor for the front body joint (RB995b; MiniStudio Inc.) had a rated voltage of 4.8 V and a rated torque of 0.833 Nm. The servo motor for the rear tail joint (DS385; Japan Remote Control Co.) had a rated voltage of 4.8 V and a rated torque of 0.196 Nm.

The total length of the robot was 190 mm, and the length of the body part was 120 mm. There were two rotation axles, and the rear of the body and the caudal fin were rotated from side to side by a shaft installed on the servo motors. The shafts were restrained by the housings so that they did not move vertically or horizontally. All parts are rigid bodies from the head to Point A in Fig. 5(b), which is 20 mm behind joint R. The shape of the body part is NACA0012, which is a typical symmetrical airfoil. This shape is often used in
The part from point A to point B corresponds to the caudal fin and is elastically deformable. It was made from a polystyrene board with a thickness of 0.3 mm and a Young's modulus of 2.74 GPa. A PIC18F2520 from Microchip Technology Inc. was used to control the servo motors. A DC voltage of 5 V was supplied from a stabilized power supply (PW18-1.3A; Kenwood Co.), which was connected to the control circuit and servo motors.

The amount of deformation of the caudal fin was investigated under simulated robot movement. The robotic fish was fixed to a circulating water channel with a flow velocity of 0.2 m/s. The water flow in the vicinity of the fin was imaged at 200 frames per second from the lower side using a high-speed camera (k-II; Katokoken Corporation). The fin deformation was also measured from these images.

Particle image velocimetry was used to measure the water flow velocity distribution around the caudal fin in an area with a length of 150 mm in the travel direction and 184 mm in the lateral direction. The area around the robot was illuminated by a laser sheet produced by a LD-excited YAG laser at the side of the measurement chamber. The above-mentioned CCD camera and the software FlowExpert made by KatoKoken were used to measure the flow velocity distribution.

IV. ANALYSIS OF FLOW AROUND ROBOTIC FISH BY THREE-DIMENSIONAL FLUID-STRUCTURE INTERACTION SIMULATION

In this section, the validity of the fluid-structure interaction simulations is assessed by comparing the calculation and experimental results. Figure 6 shows the computational grid used for the calculation. It is divided into 62x50x159 cells, and is finer in the vicinity of the caudal fin and its downstream region.

Table 3 shows the calculation conditions used for the fluid and elastic body. As an upstream boundary condition, the water inlet velocity at the upstream end was fixed at 0.2 m/s. The density, Young’s modulus and moment of inertia of the fin, which determine its flexibility, are the same as those for the 0.3-mm-thick polystyrene board used in the experiment described in the foregoing section. Here, the method of deriving the Young’s modulus is described. The edge of the polystyrene board is horizontally fixed to a pillar. Then the load is given to the other edge. After the board bends, the state is taken with a camera. The deflection of the board is obtained by counting the pixel number in the camera image. The Young’s modulus is calculated from the displacement of board edge by using the theoretical formula of deflection.

The computational grid used for the FEM deformation analysis is shown in Fig. 7, and is the same as that used for the fluid analysis. If the grid numbers \(i, j\) and \(k\) correspond to the directions \(x, y\) and \(z\), respectively, the region defined by \(k = 36–63, i = 31–33, j = 19–33\) shown in Fig. 7 is calculated in the deformation analysis. The 20-node serendipity elements used for the fin deformation analysis described in section II are applied. The number of elements is 756 and the number of nodes is 4491. Each node has three degrees of translational freedom along the \(x, y\) and \(z\) directions. Because the 20-mm-long region with \(k = 36–40\) is rigid, the Young’s modulus of this region is set to 2.74x10^6 GPa so as to avoid any elastic deformation.
The center of rotation for the rear section of the robot body is at \( k = 26 \), and that for the caudal fin is at \( k = 36 \).

### Table 3 Calculation conditions

<table>
<thead>
<tr>
<th>Condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
<td>293 K</td>
</tr>
<tr>
<td>Water pressure</td>
<td>( 1.01325 \times 10^5 ) Pa</td>
</tr>
<tr>
<td>Water density</td>
<td>998.2 kg/m(^3)</td>
</tr>
<tr>
<td>Water viscosity</td>
<td>( 1.002 \times 10^{-3} ) Pa( \cdot )s</td>
</tr>
<tr>
<td>Inlet velocity</td>
<td>0.2 m/s</td>
</tr>
<tr>
<td>Density of elastic tail fin</td>
<td>1039 kg/m(^3)</td>
</tr>
<tr>
<td>Cross-section area</td>
<td>( 17.7 \times 10^{-7} ) m(^2)</td>
</tr>
<tr>
<td>Young's modulus</td>
<td>2.74 GPa</td>
</tr>
<tr>
<td>Geometrical moment of inertia</td>
<td>( 13.27 \times 10^{-14} ) m(^4)</td>
</tr>
</tbody>
</table>

Fig. 7. Computational grid for deformation analysis

When only one joint is operating, the rotation range at point R for the caudal fin is set to ±20°. When two joints are operating, the rotation ranges at point F and point R are set to ±10° and ±20°, respectively. In both cases, the right-left rotation of the body and fin is sinusoidal. The operating frequency is set at 1.97 Hz. If the motion is initiated at the maximum rotation amplitude \( \theta_{\text{max}} \), the pressure calculations diverge easily. To avoid this, the rotation amplitude is multiplied by an under-relaxation factor, and the peak magnitude \( \theta_{\text{max}} \) is increased gradually.

Fluid-structure interaction simulations were carried out using the above-mentioned computational grid and calculation conditions, and the numerical and experimental PIV results were compared. Because the numerical results were three dimensional and the PIV data were two dimensional, the results were compared only for a horizontal plane through the center of the robot.

Figure 8 shows the simulated vorticity distribution together with the experimentally obtained results for the case in which the robot moves only one joint. Figure 9 shows the equivalent results for two-joint motion. One fin locomotion cycle is 0.5 seconds. It can be seen that

![Fig. 8. Comparison of numerical and experimental vorticity distributions for one-joint motion](image)

![Fig. 9. Comparison of numerical and experimental vorticity distributions for two-joint motion](image)

for both one-joint and two-joint motion, the simulated and experimental flow patterns are similar.

Thus, it was confirmed that the experimental and numerical results for the flow pattern behind the robotic fish are in reasonably good agreement. The amount of fin deformation was next compared. Figure 10 compares the time results for the displacement in the x-direction of point A and point B. The open circles represent the experimental results, and the solid lines represent the numerical results. For both one-joint and two-joint motion, there is fairly good agreement between the experimental and numerical results, although there are some differences in the regions of maximum displacement. Thus, for both the flow pattern behind the fin and the amount of fin deformation, the fluid-structure interaction simulation program is shown to be valid.

### V. EVALUATION OF PROPELLIVE PERFORMANCE

Having established the validity of the fluid-structure interaction simulation code, it was next applied to simulate the effect of different fin configurations on the propulsive performance. The four different caudal fins that were investigated are shown in Fig. 11. Although the overall shape and area of each fin are the same, they have different flexibilities. In Case 1, the entire 70-mm-wide fin is rigid and has a very high Young’s modulus.
In Case 2, the fin is divided into a rigid section and an elastic section whose flexibility is the same as polystyrene. Finally, in Case 3 and Case 4, the fins have a rigid section and two separate soft sections that bend easily.

In Case 1 and Case 2, the water pressure on the fin surface is large when the robot swims with one-joint motion, so that both the net propulsive force and the side force are strong. In contrast, in Case 3 and Case 4, the net propulsive force and the side force are weak because the water pressure on the fin surface is small. In this study, net propulsive force of the fish robot was calculated by using the momentum principle. For a given control volume ABCDEFGH in Fig.12, the net propulsive force created in the control volume equals the change in the momentum flow rates.

Figure 13 shows the results for the net propulsive force, the side force and the propulsive performance [14] for the case of one-joint motion. The propulsive performance was calculated using Eq. (15). Because the side force gives rise to actuator energy loss in the robot, it should be weak for efficient swimming.

\[
R_s = \left| \frac{F_s}{F_p} \right|
\]  

(15)

It can be seen that the net propulsive force was strongest for Case 2, and the highest propulsive performance was obtained for Case 4.

Figure 14 shows the results for the net propulsive force, the side force and the propulsive performance for one-joint motion.
the case of two-joint motion. It can be seen that a remarkably high propulsive performance is obtained for Case 2. This is due to the fact that the net propulsive force is strong and the side force is weak. The simulated two-joint fin motion of the swimming robot in Case 2 with two-joint motion is very similar to that for a live fish.

The results of the present study indicate that for practical robotic fish, two movable joints are preferable. When the robot has two joints, the fin should not be too soft, although it should not be a rigid body. When two driving parts cannot be installed, it is preferable to use a soft material for the fin.

**ACKNOWLEDGEMENT**

The authors would like to express their gratitude to the Japan Society for the Promotion of Science who subsidized us (Grant-in-Aid for Scientific Research (C), No.24560301).

**REFERENCES**


