Hull Form Design of Underwater Vehicle Applying CFD (Computational Fluid Dynamics)

Tomoya Inoue*, Hiroyoshi Suzuki**, Risa Kitamoto**, Yoshitaka Watanabe*, Hiroshi Yoshida*

*JAMSTEC (Japan Agency for Marine-Earth Science and Technology)
Yokosuka, 237-0061 Japan

**Osaka University
Suita, 565-0871 Japan

***Abstract*- CFD is recognized as a possible advanced tool to understand the performance of the underwater vehicle such as a hull resistance, a thrust force, motion in waves and so on. Focusing on the hull resistance, we conducted the investigation of the water resistance of the AUV PICASSO utilizing CFD and also conducting experiments to confirm the CFD simulations. After that, designing the hull form within the limitation of equipment arrangement to improve the hull resistance was conducted utilizing the CFD. And the experiment was conducted to confirm.**

I. INTRODUCTION

Japan Agency for Marine-Earth Science and Technology (JAMSTEC) developed and is developing AUVs. The AUV URASHIMA was developed [1] as a long-range cruise AUV and has been operated for survey on the seabed. The AUV PICASSO (Plankton Investigatory Collaborating Autonomous Survey System Operon) [2] as shown in Figure 1 has been developing. The PICASSO is a small unmanned submersible developed for study of the deep-sea Plankton and Marine Snow up to a depth of 1,000m. It is able to be equipped with a high-resolution digital camera, a visual Plankton Recorder and a high-intensity light, so that the high-resolution pictures of deep-sea life forms such as Plankton are available. Table 1 shows principal specifications of the PICASSO.

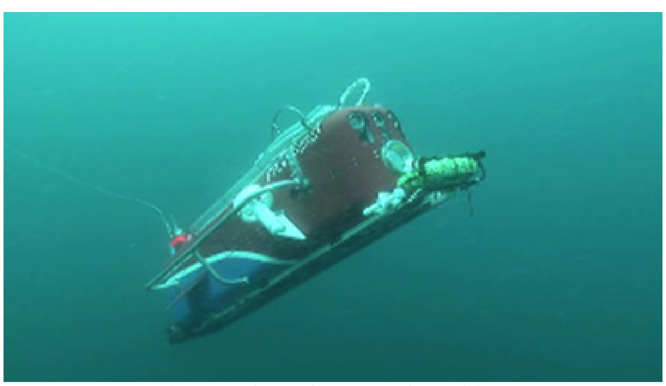

Figure 1. Photo of ABISMO in Cruising.

The hull performance such as a propulsive performance and maneuverability is required according to the mission and the concept of the AUV. On the other hand, it is very important for design of the hull form to keep enough and adequate area inside the hull to arrange the equipment and device. Also the length may be limited from the view point of handing it onboard. So, a fair trade-off is important.

We have started to re-design the hull form of the PICASSO to improve the hydrodynamics performance. Some researches on the CFD (Computational Fluid Dynamics) to estimate the hydrodynamic performance were conducted [3] to leverage it as a design tool. We also developed a in-house CFD code for AUVs [4][5][6][7]. However, re-design here is conducted to utilize a commercial CFD code. And focusing on the hull resistance, we conducted the investigation of the water resistance of the AUV PICASSO utilizing CFD and also conducting experiments to confirm the CFD simulations. After that, designing the hull form within the limitation of equipment arrangement to improve the hull resistance was conducted utilizing the CFD. And the experiment was conducted to confirm it.

The purposes of the investigation here is not only to improve the hydrodynamics performance of the PICCASO but also to lead the numerical approach which is CFD simulation to a possible design tool of the hull form of AUVs.

TABLE 1
PRINCIPAL SPECIFICATIONS OF THE AUV PICASSO

Items	Specification
Dimensions	<Overall> 2.58 (L) x 0.8 (W) x 0.8 (H) m <Fuselage> 2.13 (L) x 0.45 (W) x 0.526(H) m
Weight in air	200 kgf
Working Depth	1,000 m
Propulsion	2 x Lateral thruster 1 x Vertical thruster
Cruising Speed	2 knots (max 3 knots)

978-1-4244-5222-4/10/$26.00 ©2010 IEEE

II. WATER RESISTANCE OF THE ORIGINAL HULL FORM OF THE AUV PICASSO

The original hull form of the PICASSO was mainly designed to arrange components such as pressure vessels, cameras, sensors and so on and to minimize the overall size in order to expand the variety of the ship which operates the PICASSO.

In this section a water resistance of the original hull form was investigated by a numerical approach and an experimental one not only to know the current hull resistance characteristics but also to establish and verify the numerical approach.

A. Numerical Simulation

Although we established the in-house CFD code, in this investigation FLUENT6.3 was used to solve the flow field around a model of the PICASSO. And GAMBIT2.4 was used to generate the model and grids.

The initial settings and schemes used in this computation are shown below;

1. Velocity-Pressure coupling: SIMPLE
2. Turbulence Model: Standard k-ε turbulence model with standard wall function
3. Grid System: Unstructured Grid System
4. Advection term: 1st order upwind scheme

The Computational model of the PICASSO and an example of the computational grid around the PICASSO are shown in Figure 2 and Figure 3, respectively. As shown in the figures, appendages equipped on the fuselage observed in Figure 1 such as thrusters, fins and so on are not considered in the computation.

It is well known that the use of boundary layer mesh around an object can get the more accurate numerical solution than tetra mesh. However, to generate boundary layer mesh is so difficult. Therefore, in this simulation tetra mesh to the whole computational domain was used. It should be noted that the number of computational elements is about 650,000 for each computation.

Since the mission of the PICASSO is to investigate the planktons as described above and to track them, omnidirectional motions including the rotating motion and its response are all important. However, here the simulations of two motions, one is the forward motion at 1.0m/s and the other is the lateral motion at 0.5m/s, were conducted. Figure 4 shows the pressure distribution of the hull in the forward motion.

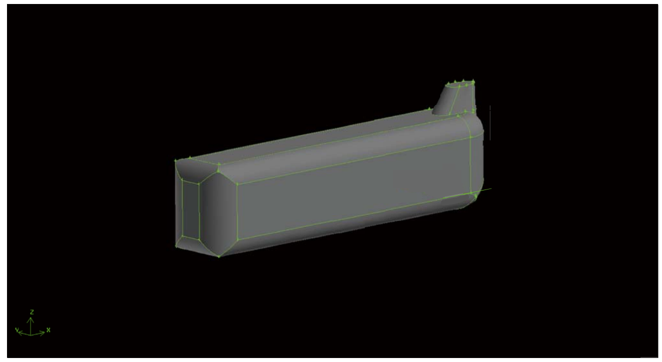

Figure 2. Computational model of the PICASSO.

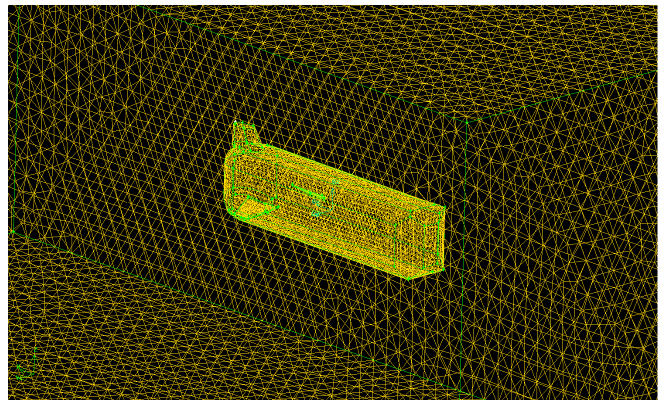

Figure 3. Computational grid.

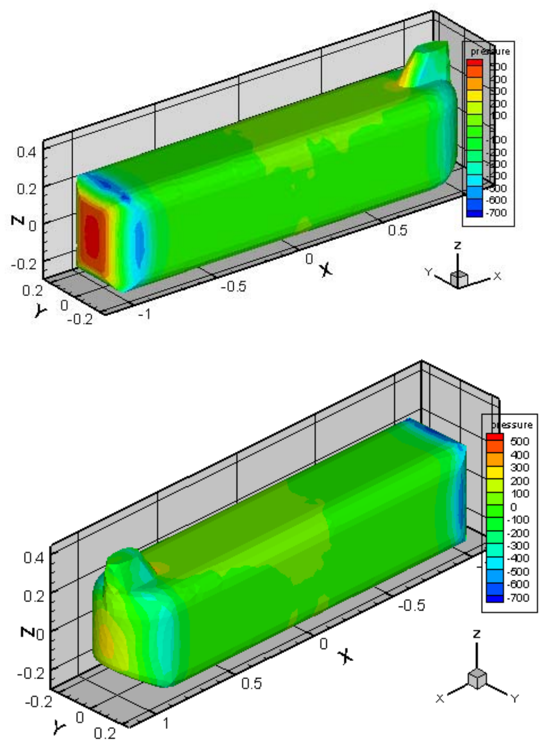

Figure 4. Pressure distributions on the hull surface.

B. Experimental Investigation

Experimental investigation to measure the water resistance of the original hull of the PICASSO was conducted using a full scale model at the towing tank of Osaka University to check the accuracy of numerical simulation in addition to know the hull resistance characteristics of the original hull form of the PICASSO. The model was submerged to a certain depth with a cylindrical strut within the allowable range of a load cell to eliminate the effect of the water surface on the model as far as possible.

C. Resistance Characteristics of the Original Hull Form

Figure 5 represents the results of the numerical simulation and the experiments. Here, the drag coefficient is defined as follows:

$$C_D = \frac{F_x}{\frac{1}{2}\rho S V^2} \qquad (1)$$

Where F_x is the dimensional resistance, ρ is density of the water, S means the projected area defined as breadth of fuselage $b \times$ depth d and V is the velocity of ground speed of towing carriage.

As is known, the numerical simulation is affected by the simulation condition such as a mesh size. The mesh size was set so that the simulation result of moving forward at 1.0m/s corresponds to the experiment result. And the mesh size in other simulations and further simulation described later was made same. As shown in Figure 5, CFD simulations are in excellent agreement with the experiment results.

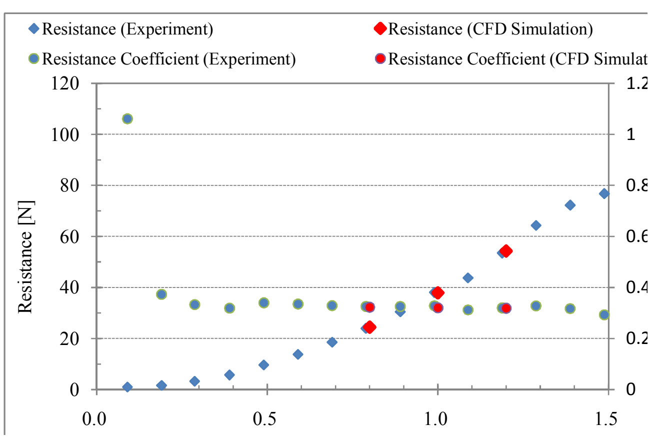

Figure 5. Results of the simulations and the experiments of the resistance on the original hull form of the PICASSO.

III. HULL DESIGN TO IMPROVE THE PROPULSIVE PERFORMANCE

In designing the hull form of AUV, it is necessary to consider not only the propulsive performance or maneuverability but also the efficient arrangement of equipment and device inside the hull.

Even a sectional area will also be designed considering it. In this paper, to improve the propulsive performance the investigation of changing the hull form of a fore part and an aft part which will greatly influence on it is conducted.

A. Fore Part Hull Form

Considering 4 elliptic columns are formed with base planes which are Ellipses perpendicular to the y and z axes having a radius of *c* as shown in Figure 6. Then the fore part hull form is formed with the elliptic columns.

CFD simulations described the previous section were conducted for the hull with fore part variation. Figure 7 and 8 represent the results.

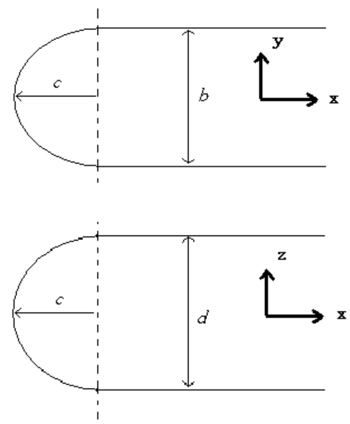

Figure 6. Fore part hull form variation.

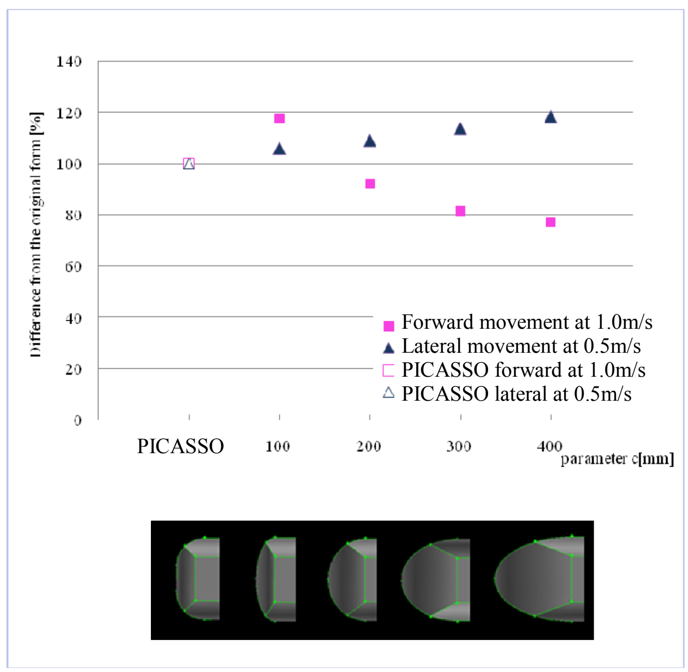

Figure 7. Hull resistance for the fore part hull form variations.

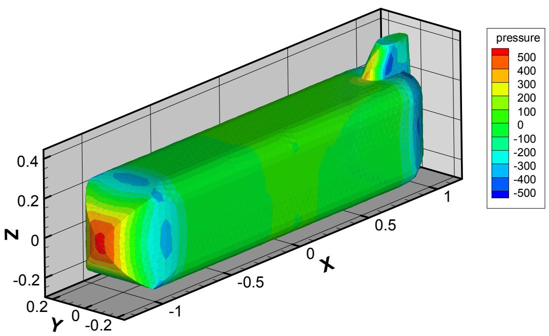

Figure 8. Pressure distributions on the hull of c=300mm.

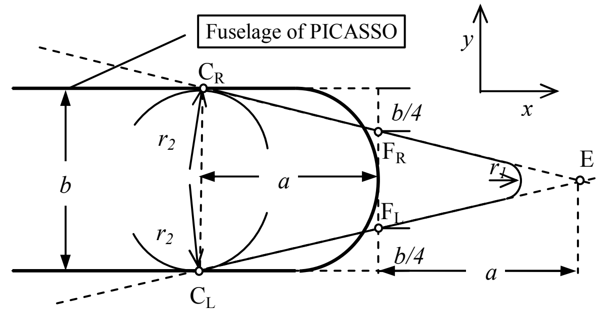

Figure 9. Aft part hull form variation.

B. Aft Part Hull Form

Now, we suppose the fuselage of PICASSO as a rectangle (break line) in Figure 9. In case that points F_L and F_R are determined on the vertical line of the rectangle, the distance from the both side are b/4 and these points are fixed. The following are made to be point C_L: A straight line which passes through points F_L and intersection point of the hull. C_R is determined as same manner as C_L using F_R. A distance from the point C_R or C_L to stern end of original PICASSO along longitudinal direction is a and this is as the parameter of aft part variation of the hull form. The cross point of above two straight lines is determined as point E, this point means the virtual end point of the hull. After this, molding were carried out around the points C_L, C_R and E and the molding radius r_1 and r_2 are determined as follows:

$$r_1 = \frac{a}{2\sqrt{1+\left(\frac{4a}{b}\right)^2}} \tag{2}$$

$$r_2 = \frac{1}{2}b \tag{3}$$

Using above variation of form parameters, we can generate series of hull forms.

CFD simulations described the previous section were conducted for the hull with aft part variation. Figure 10 and 11 represent the result.

C. Experimental Investigation

The experiments to measure the water resistance were conducted using models being combined with the fore part hull form of c=200mm and/or the aft part hull form of a=400. Thus, models are one with the fore part hull form of the c=200mm and the original aft hull form, one with the original fore hull form and the aft hull form of a=400mm, and one with the fore part hull form of the c=200mm and the aft hull form of a=400mm. Figure 12 represents the results of the experiments

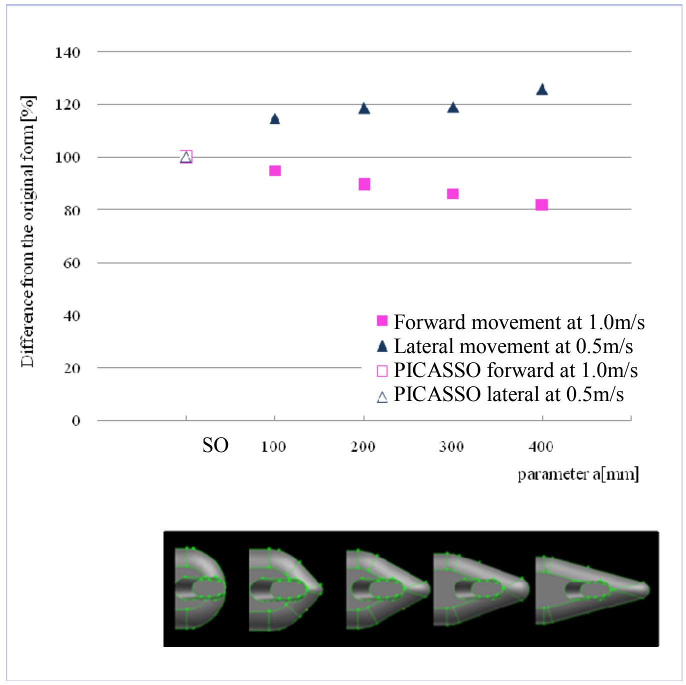

Figure 10. Hull resistance for the aft part hull form variations.

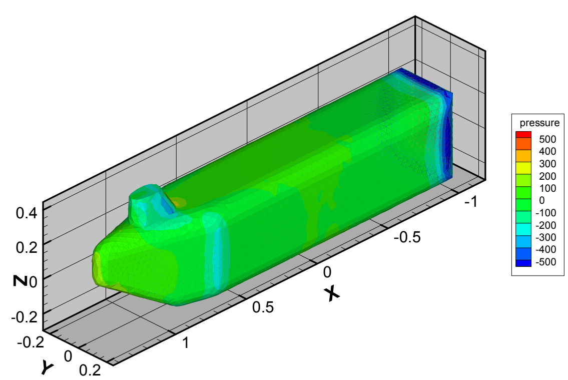

Figure 11. Pressure distributions on the hull of a=400mm.

and the CFD simulations. The results reveal that modifying the aft part form greatly improved the propulsive performance.

However, as shown in Figure 12 the results of CFD simulations are comparatively different from the ones of the experiments. Although of course the CFD simulation method and condition should be investigated from the view points of accuracy and adequacy of the CFD simulations, the accuracy of the model and the accuracy of the measurements should be also discussed. The models were manufactured so that it could be divided into three parts, the fore part, the middle part and the aft part, and the fore part and the aft part were replaced. The assembly accuracy was slightly not flat. Of course it is not the reason why the result of the experiment is lower than the one of the simulation. Another possible reason is an effect of the strut. The model was submerged into about 400 mm below water surface with the cylindrical strut generating the waves a lot. Although of course the resistance of the strut was measured by towing only the strut and its resistance was subtracted, the measurement condition may be slightly changed because the water resistance on the model of the PICASSO would cause a greater moment on the strut and make it inclined.

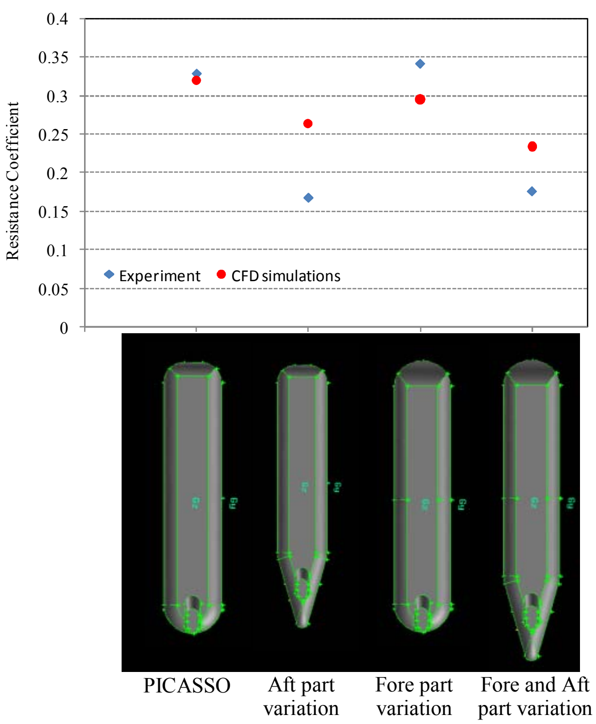

Figure 12. Comparison the CFD simulations with experiments for the hull form with fore or/and aft part variations.

IV. Conclusions

To improve the propulsive performance of the PICASSO, the water resistance was investigated by the numerical and the experimental approaches. Another purpose of the investigation is to lead the numerical approach which is CFD simulation to a possible design tool of the hull form of AUVs.

Changing the fore and the aft hull form can improve the propulsive performance considerably. However, actual design of AUV can be conducted considering the arrangement of the equipment and device and overall length from the view point of handling it on board. So, within the limitations, the hull form should be designed in fair trade-off. In the pre-design of the hull form the numerical simulation using CFD can be leveraged.

We will continue to develop the CFD simulation method to leverage it as a design tool.

Acknowledgment

Authors would like to appreciate to Marine Division of Kowa Corporation for their support in conducting the experiments.

References

[1] S. Tsukioka, T. Aoki et al., "Experimental Results of an Autonomous Underwater Vehicle URASHIMA," *Oceans Conference*, 2003.

[2] H. Yoshida and D. Linsay, "Development of the PICASSO System at the JMSTEC," *Japan Deep Sea Technology Association Report*, Vol.54, pp.5-10, 2007.

[3] A. Phillips, M. Furlong and S.R. Turnock, "The Use of Computational Fluid Dynamics to Assess the Hull Resistance of Concept Autonomous Underwater Vehicle," *Oceans Conference*, *Vancouver*, 2007.

[4] H. Suzuki, N. Kato, T. Katayama and Y. Fukui, "Numerical Study on Unsteady Flow Around an Underwater Vehicle with Two Pairs of Mechanical Pectoral Fins," *Proceedings of 2nd PAAMES and Advanced Maritime Engineering Conference (AMEC),Jeju,* pp. 225-2432006.

[5] H. Suzuki, T. Shimamura, K. Miura and n Kato, "Numerical Motion Simulation of Submerged Body in Waves," *Proceedings of 3rd PAAMES and Advanced Maritime Engineering Conference (AMEC), Makuhari,* pp.677-684, 2008.

[6] Y. Fukui, H. Suzuki and N. Kato, "CFD-Based Motion Simulator with Control Algorithm for Biomimetic Underwater Vehicle,", *Proceedings of 3rd PAAMES and Advanced Maritime Engineering Conference (AMEC), Makuhari,* pp.685-691, 2008.

[7] H. Suzuki and N. Kato, "Computation of Unsteady Flow Around Underwater Vehicle with Mechanical Pectoral Fins," *Proceedings of International Symposium of Offshore and Polar Engineers (ISOPE)*, pp.273-278, 2006.