Computational Fluid Dynamics Simulations of a Biomimetic Underwater Robot

Madis Listak, Deivid Pugal, Maarja Kruusmaa

***Abstract*—This paper represents a comparative study of hydrodynamic properties of a biomimetic underwater robot. The hydrodynamic properties are modeled with methods of computational fluid dynamics. In particular, we measure wake turbulence created by natural and man-made objects, an ideal cone, a dolphin, manta ray and the biomimetic robot. Our research is aiming at creating a biomimetic robot operating in shallow turbid waters near the bottom with volatile sediments and therefore we are interested in minimizing the turbulent flow created by the robot. The results of the computational fluid dynamics simulations show that the turbidity in a steady flow created by the model of our biomimetic robot is within the same range with fish and considerably better than of a non-streamlined object. It also gives us further guidelines for improving the design of our biomimetic underwater robot.**

I. Introduction

This paper represents a computational fluid dynamic (CFD) model of a biomimetic underwater vehicle and investigates its properties compared to a CFD model of a non-streamline object and fish. The aim of this study is to get comparative data about hydrodynamic properties of those objects and guidelines to improve the design of the biomimetic underwater vehicle.

A hydrodynamic model of an underwater robot permits us to determine the properties of the vehicle and facilitates the development of the control algorithms. The shape of the body determines the viscous drag of the vehicle and to a great extent the controllability of the robot. On the other hand the available technology sets constraints to the mechanical design of the robot. The constraints are set by the physical properties of the components, like weight and shape, and therefore in practice it is not possible to build a vehicle with ideal hydrodynamic properties. The irregularities and asymmetries of the body can change the hydrodynamic properties, in some cases, by even improving them [8]. We therefore view the design of the underwater vehicle as cyclic process where the hydrodynamic properties of the prototype are investigated theoretically, which in turn give suggestions for further improvements of the robot.

Due to the rapid increase of computational efficiency, the practical 3D CFD modeling is becoming increasingly popular as an efficient tool of investigating the hydrodynamic properties of robots [1]. Similar studies of comparing natural objects with artificial objects or their components to investigate their hydrodynamic properties are also reported previously [6].

In this work we use free GPL software Gerris [2, 3]. This software has been, for example, used to study ship turbulence [4]. It is a new generation software that exploits semi-structured quadtree/octree mesh models to increase computational speed without the loss of precision by permitting computation of the components of the model with a different precision. In this paper, we use a time-adaptive mesh to discretize the solution dynamically. This allows us to study time-dependent 3D Euler incompressible turbulent flow.

Mostly CDF modeling is performed using Reynolds averaged Navier-Strokes equations (RANS) and averaging is carried out in space and time. This solution has limited information about turbulence characteristics, because effects of turbulence are time-averaged.

In our simulations the averaging is done only spatially (Large Eddy Simulations-LES) and computational domain is discretized using cubic finite volumes that are organized as a spatial octree. This allows dynamic adaptation of the spatial resolution to follow the evolving flow structures. For example, in the vicinity of the robot the resolution is finer and in less interesting regions, only large structures are computed.

II. The robot

Fig. 1. The robot

The robot is a fin actuated biomimetic robot presented in Fig. 1. It is actuated by a tail fin and two side fins, equipped with a color camera and sonars, compressed air supply and air chambers to change the buoyancy. It is

This work was supported in part by Estonian Science Foundation Grants No. 6765 and 7000, Estonian Environmental Investment Center and Netbell Ltd.

M. Listak is with Tallinn University of Technology, Dept. of Computer Engineering, Raja 15, Tallinn, Estonia ; e-mail: madis.listak@mail.ee.

D. Pugal is with the Mechanical Engineering Department, University of Nevada, Reno, NV, U.S.A

M. Kruusmaa is with the Tartu University Institute of Technology, Nooruse 1, 50411, Tartu, Estonia e-mail: maarja.kruusmaa@ut. ee.

designed to be used in a towed mode as well as in an autonomous mode. In the towed mode the side fins are used as horizontal rudders to change the altitude. In the autonomous mode the robot moves forward by the tail fin propulsion. The detailed description of the robot can be found in [11].

The design considerations of the vehicle are task-specific. The vehicle is designed for visual inspection in shallow waters with low visibility and volatile sediments. Shallow water coastal regions often fall into this category, also inland water bodies (rivers, lakes, ponds, bogs) that often have a muddy bottom and low visibility. In these conditions it is important that the vehicle used for visual inspection creates as little turbulence as possible since the turbulent flow would beat up the silt from the bottom and decrease visibility.

Therefore we are interested in designing a vehicle that creates possibly little turbulent flow. The first choice is therefore to use fin propulsion, apart from the classical choice of propelled locomotion.

Besides the means of propulsion the vorticity of the wake also depends on the hydrodynamic properties of the vehicle itself. Theoretically, we would like to design a vehicle with ideal hydrodynamic properties. Practically, we are constrained by the available technology, components, task specification, end user requirements, and cost factors. These CDF simulations permit us to evaluate the hydrodynamic properties of our vehicle and compare them to natural objects as well as to man-made non-streamlined objects. Furthermore, it gives us guidelines to improve our vehicle's design.

III. MODELS

We are interested in comparative assessments of the wake turbulence. For this purpose we model and measure the wake turbulence of the following objects: the biomimetic underwater robot described in the previous section, an ideal cone, a dolphin and a manta ray.

To get a realistic 3D model of the underwater vehicle, the vehicle represented in Fig. 1 is photographed in the up-, side- and front view. The colored photos are converted to black and white images and converted to stl (stereolitography) format compatible with Gerris. The stereolitography images are converted to gtl (GNU Triangulated Surface) format by dividing the body to polygons. The model is corrected by hand in order to assure that there are no holes and gaps between the polygons.

The model of the dolphin and manta ray are retrieved from [4]. The Blender models are converted to stereolitography format and corrected by creating a new mesh with a polymender tool to compensate inaccuracies caused by conversions. Then the files are converted to gts format. The cone is created with Blender and saved to stl format. The resulting Bender CAD models of the objects are depicted in Fig. 2.

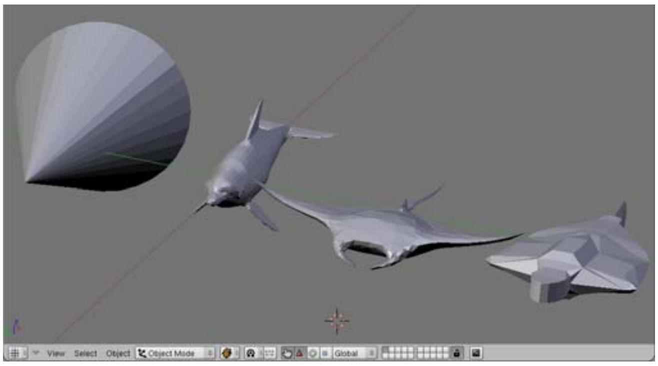

Fig. 2. The Blender models of the bodies.

As water flows around an object, the uniform velocity field gets disturbed. The disturbance of the flow basically depends on the shape of the object and parameters of the flowing liquid. By keeping the flow parameters equal and changing the shape of the object we can get comparative estimates about the hydrodynamic properties of them. In particular, we are interested in the velocity field formed behind the vehicle, in the wake, because the disturbances in this region are most likely to beat up some silt from the bottom. For that reason, some simulations of water flow have been done to determine the circulation of the velocity of water behind an object, caused by motion with a constant velocity. The velocity field is found with Gerris simulation software, as shown in Fig. 3. The output is a grid with velocity vectors known at each square of the grid. To find the circulation of velocity (further denoted also as a circulation), a contour behind the object is chosen in x-y plane, and the following integral is calculated:

$$\oint \vec{v} \cdot d\vec{l} \ , \qquad (1)$$

The numeric result of the integral is a velocity circulation by the definition and it is related to local vorticities as follows:

$$\oint \vec{v} \cdot d\vec{l} = \int_S curl(\vec{v}) \cdot d\vec{s} \ , \qquad (2)$$

where the term $curl(\vec{v})$ is a local vorticity and vector ds is a local infinitesimally small surface area with the direction of normal to the surface.

For comparability, all models are scaled down to the same length. . Table I describes the measures of the bodies.

The water velocity of the simulations is set to 0,5 m/s, water viscosity is set to 1, temperature is 20 degrees Celsius.

In addition, we also measure the velocity and pressure in the area right ahead and behind the body to find out if there are any changes and differences in these parameters.

TABLE I
MEASURES OF THE BODIES

	CONE	MANTA RAY	DOLPHIN	ROBOT
Max length	0,2	0,2	0,2	0,2
Max width	0,112	0,21	0,08	0,112
Max thickness	0,112	0,04	0,07	0,03

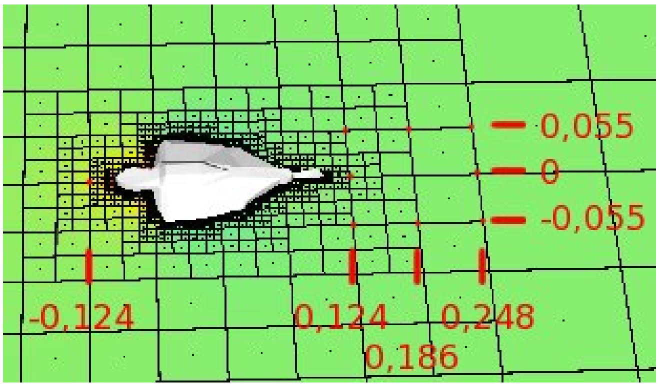

Fig. 3. The object, mesh of the velocity field and the coordinates of the points used for calculations.

The cone is chosen for modeling as an object creating large vorticities. It can be used as a reference to compare the characteristics of the robot to a non-streamlined body. The actual value of the circulation depends on the chosen contour in (1). We tried to choose as similar contours as possible for all the objects. Since the cone creates turbulences larger at the sides and behind of the object, and less turbulence straight behind, we have defined the circulation to be measured between the points shown in Fig. 3 (the rectangular area behind the object determined by the given coordinates).

As the biomimetic robot resembles a flatfish, the manta ray is chosen to be modeled as a species with a similar topology to the robot. Comparing those simulation results would give us n estimate how much can the hydrodynamic properties further improved and guidelines for doing it.

Other parameters we use for comparison are the pressure and the pressure difference in the front and behind of the object. For these measurements, we use only four points shown in Fig. 3, one in the front of the object and three points behind the object.

IV. RESULTS

This section represents the experimental results. The cone was chosen as an object with poor hydrodynamic properties. This object is used as a reference to compare the simulation data. Fig. 4 shows the 3D simulation of the cone. As mentioned above, for his object, the largest vortices are created quite far from the axis of symmetry.

Fig. 5 shows the pressure change of the cone in four points with the given coordinates. Fig. 6 shows the velocity circulation of the cone calculated over a rectangular trajectory behind the object defined by coordinates in Fig. 3

It can be seen that the circulation created by the cone varies heavily over time. This is the reason of the large variance of the velocity.

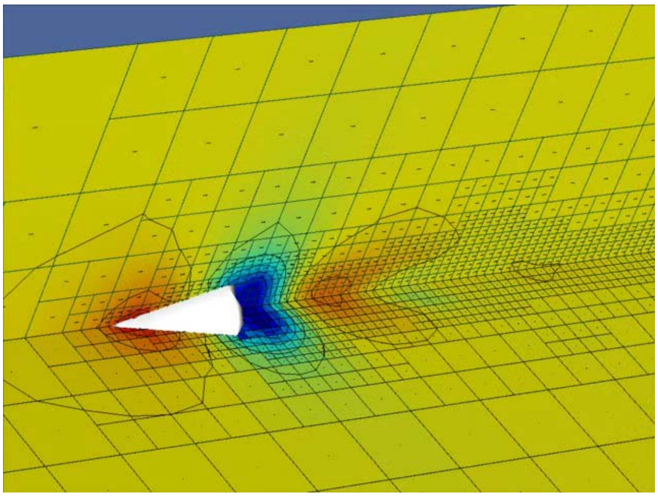
Fig. 4. The 4D simulation of the cone.

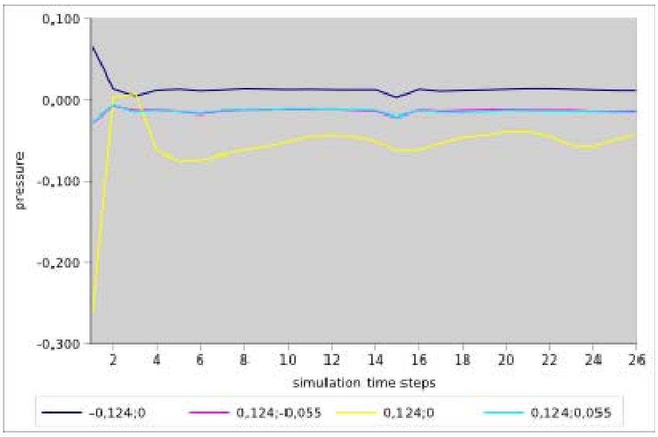

Fig. 5 The pressure measurements of the cone.

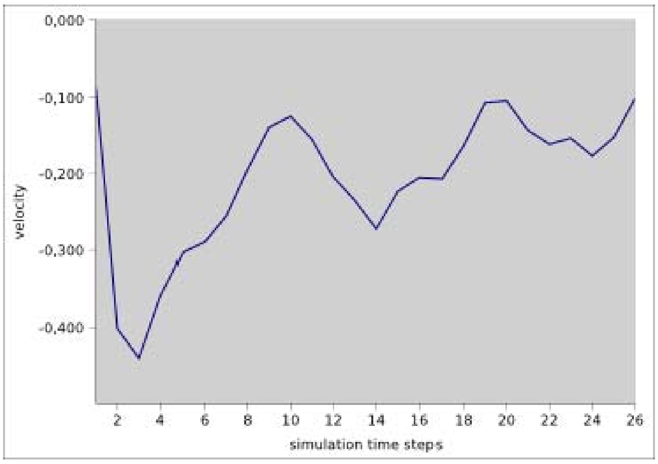

Fig. 6. The velocity changes around the cone

Figures 7-9 show the same simulation data for the model of a dolphin and Figures 10-12 for the manta ray.

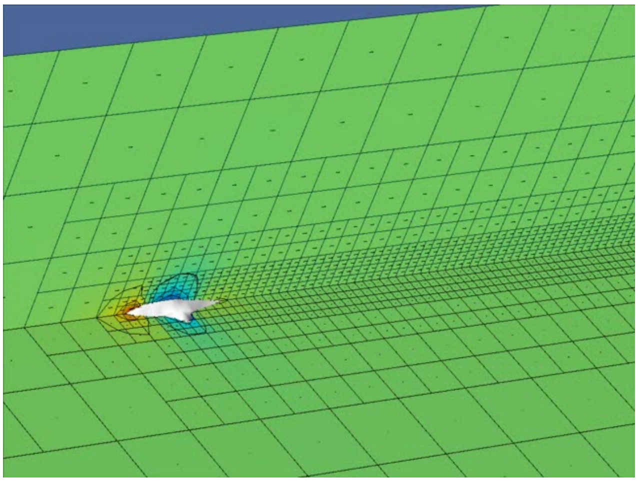

Fig. 7 3D simulation of a dolphin

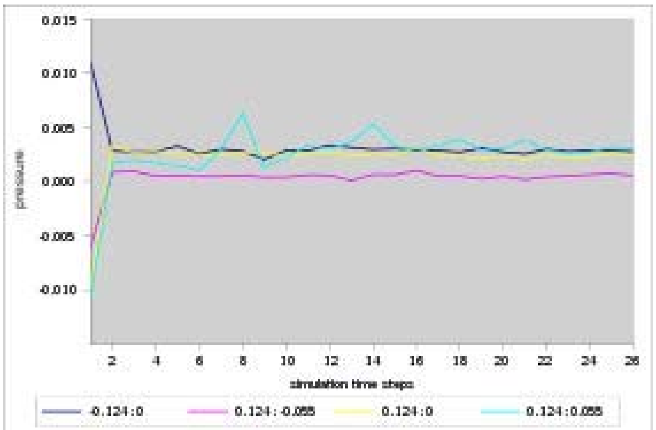

Fig. 8. Dolphin pressure changes

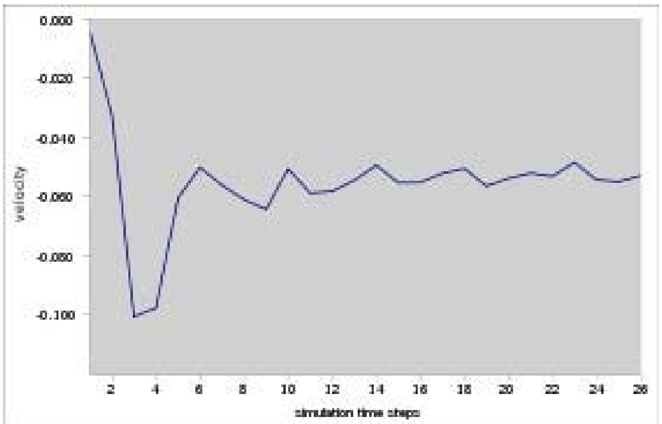

Fig. 9 Dolphin circulation

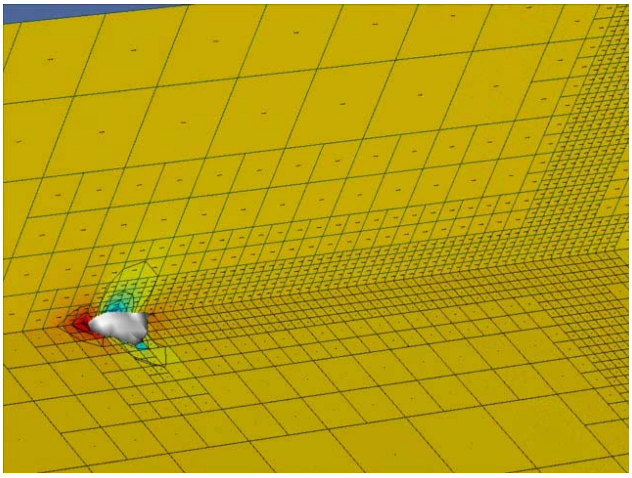

Fig. 10. 3D simulation of a manta ray

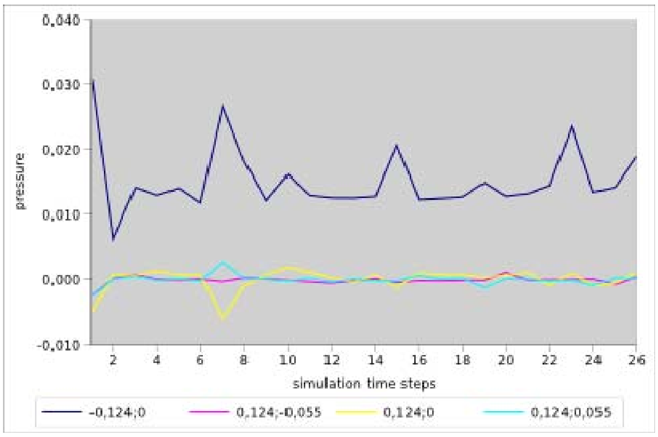

Fig. 11. Manta ray pressure changes

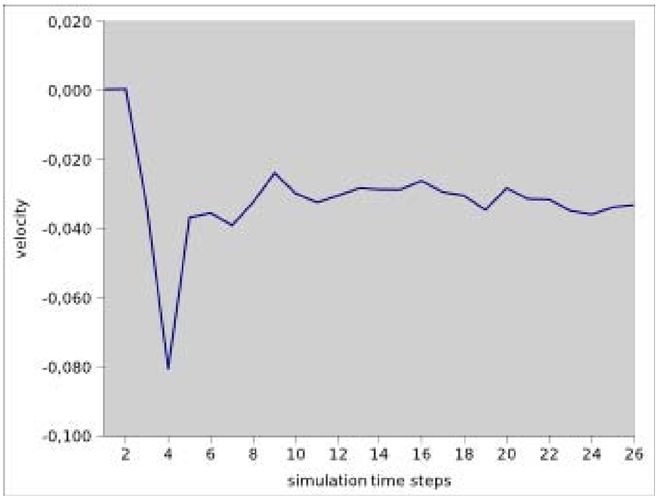

Fig. 12. Manta ray circulation

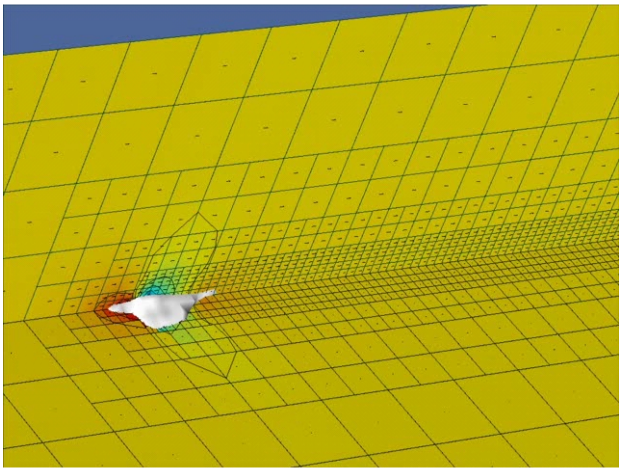

Fig. 13 3D simulation of the robot

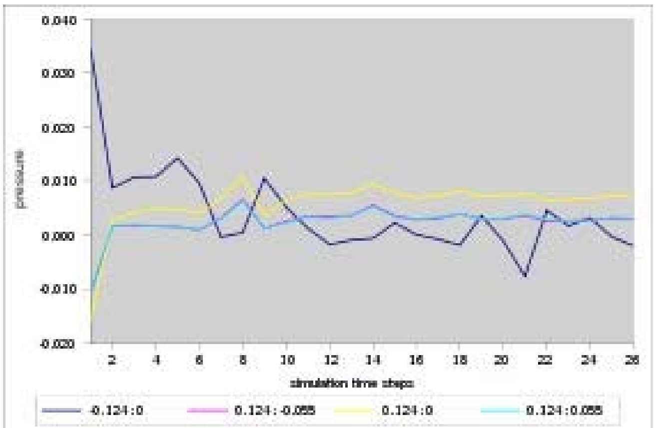

Fig. 14 Robot pressure changes

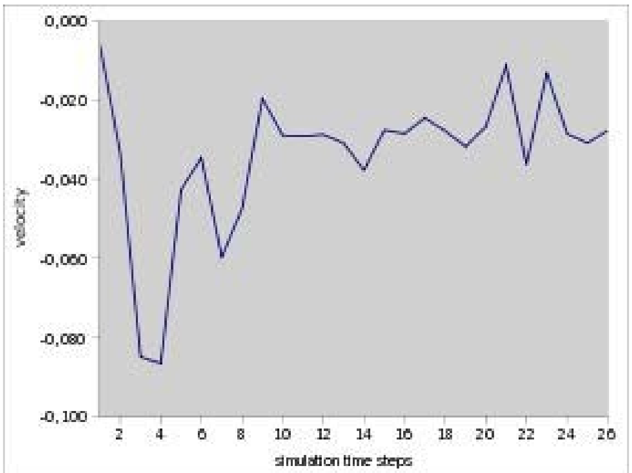

Fig. 15. Robot circulation

The same simulation data for the robot is represented in Fig. 13-15.

The difference with the previous objects is that the pressure in front of the robot and behind of the robot now differs, being slightly less in front of the robot. It has been investigated and demonstrated experimentally that fish are passively moving forward against its own drag, being able to extract additional energy from the environment [10]. As the experiments have also been conducted with dead fish, it suggests that the effect occurs not solely due to their motion patterns but also due to their body shape. The pressure difference (with the pressure being less in front of the robot) might indicate that due to the shape of the object, the Bernoulli force could act on the body, slightly increasing the drag. The force is created which induces a positive, upstream drag, thus making the swimming more efficient. Although, Bernoulli equation is applicable for ideal fluids and steady flow, we can still get some estimations of the force, because most of the flow of the water around the robot is quite steady. The dolphin and manta ray simulations, however, do not revile that such an effect is taking place. Apparently the shape of the dolphin, which is widest at the middle, does not create the asymmetry necessary for creating Bernoulli force.

Table II summarizes the numerical values of velocity circulations. It shows that the circulation of the robot is slightly lower than that of the dolphin and considerably lower than that of the cone (the sign here shows the direction of circulation and the absolute value shows the amount of circulation). At the same time, the manta ray is creating slightly less circulation than the robot.

TABLE II
NUMERICAL VALUES OF CIRCULATIONS

Model	Circulation
Cone	-5,358
Dolphin	-1,439
Robot	-0,886
Manta ray	-0,810

V. CONCLUSION

This paper presents comparative CFD simulations of an ideal cone, fish and a biomimetic robot. The velocity circulation of the water, caused by the robot, is less that of a dolphin and much less than of a non-streamline object chosen for a reference, although slightly larger than of a manta ray. The model of the robot is also shown to create Bernoulli force while the cone, the manta ray and the dolphin do not create such an effect.

The simulation results give us confidence that the underwater robot has acceptably good hydrodynamic properties. The manta ray is very close to the robot in terms of the topology of the body and their circulation values differ by less than 10%.

Manta ray has evolved from the bottom dwelling flat fish. It has apparently developed to create possibly little turbulence to hide and stay unnoticed near the bottom covered with silt. For example, it creates much less turbulence than the dolphin that is developed for swimming in mid-waters and near the surface.

The design of the robot can apparently be approved somewhat further by changing the configuration of the tail and the side fins closer to the manta ray. However, we conclude that the mechanical design of the robot is sufficiently good and comparable with fish that have evolved to create little turbulence. Further improvements of the mechanical design would be rather incremental.

However, the simulations results have to be treated with some criticism since the model of the robot, though created from the real object, is to some extent idealized. Our next immediate goal is to compare the simulation results to the experimental results of the real robot.

The impact of the Bernoulli force to the swimming efficiency needs to be investigated further before drawing conclusions about the significance of this effect. Although there is evidence suggesting that some fish seem to benefit from such a phenomenon, it was not found at simulations of neither the manta ray nor the dolphin. It is possible that its impact can be either beneficial of disadvantageous depending on the locomotion style of the fish.

This study shows that the hydrodynamic properties of the robot are good when we compare the circulation values around steady objects in a steady flow. However, the turbulence also depends on the locomotion style of the robot. Our further goal is to investigate various locomotion patterns, both in simulations and in pool tests. By comparing swimming styles of the fish and the robot, similar to this study we can choose actuation mechanisms and motion patterns most suitable to our task considerations.

REFERENCES

[1] R. Mittal, „Computational modeling in bio-hydrodynamics-trends, challenges and recent advances," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2003)*, 2003.

[2] S. Popinet, "The Gerris Flow Solver Software," http://gfs.sf.net, "Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries." *J. Comput. Phys.*, 190:572-600, 2003.

[3] S. Popinet, "Gerris: a tree-based adaptive solver for the incompressible Euler equations in complex geometries," J. of Computational Physics, 190:572-600, 2003

[4] S. Popinet, M. Smith, C. Stevens, "Experimental and Numerical study of the turbulence characteristics of air flow around a research vessel," Ocean Atm, Tech., 21:1574-1589, 2004

[5] http://toucan.web.infoseek.co.jp/3DCG/3ds/FishModelsE.html

[6] R. Bandyopadhyay, M. Boller Convergence in Underwater Swimming between Nature and Engineering; Bandyopadhyay , ," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2005)*, 2005.

[7] Su Yu-min, Dong Tao, Pang Young-jie, Huang Sheng, Xu Yu-ru, Numerical Calculation of Hydrodynamic Characteristics of Rigid and Flexible Oscillating Tuna-Tail ," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2003)*, 2003.

[8] I.K. Bartol. M .S. Gordon, M. Gharib,D . Weihs,P. Webb, „Role of carapace morphology in dynamic stability of rigid-bodied boxfishes- , ," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2001)*, 2001.

[9] R. Mittal1, I Akhtar, M. Bozkurttas, F. Njjar, Towards a Conceptual Model of a Bio-Robotic AUV - Pectoral Fin Hydrodynamics, ," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2003)*, 2003

[10] D.N.Beal, J.C.Liao, G.V.Lauder, F.S.Hover, M.S.Triantafyollou, "Passive energy extraction in the wake of bluff objects by fish," *In Proc. of the International Symposium on Unmanned Untethered Submersible Technology (UUST 2003)*, 2003

[11] M. Listak, D. Pugal, M. Kruusmaa, "Biomimetic fish-like underwater robot for shallow water applications",submitted to ICAR 2007.