Proceedings of 2013 IEEE
International Conference on Mechatronics and Automation
August 4 - 7, Takamatsu, Japan

# ANSYS FLUENT-based Modeling and Hydrodynamic Analysis for a Spherical Underwater Robot

Chunfeng Yue[1] Shuxiang Guo[2, 3]

*1. Graduate School of Engineering, Kagawa University*
*2217-20, Hayashi-cho, Takamatsu, 761-0396, Kagawa, Japan*
s12d502@stmail.eng.kagawa-u.ac.jp

guo@eng.kagawa-u.ac.jp

Maoxun Li[1]

*2. College of Automation, Tianjin University of Technology, China.*
*3. Faculty of Engineering, Kagawa University*
*2217-20, Hayashi-cho, Takamatsu, 761-0396, Kagawa, Japan*

s12g537@ stmail.eng.kagawa-u.ac.jp

*Abstract* – **For an underwater robot, hydrodynamic characteristics are very important. This paper focuses on the research of the hydrodynamic analysis of a spherical underwater robot with three motions, horizontal motion, vertical motion and yaw motion. Firstly, the prototype of related second generation spherical underwater robot (SUR-II) was developed. In order to analyze the hydrodynamic characteristics of the spherical underwater robot exactly, CATIA software was employed to establish the 3D models of the flow field. For the complex structure of the developed underwater robot causing the limitations on meshing and hydrodynamic analysis, we simplified the 3D models properly. Finally, we used ANASYS FLUENT to analyze the three models and compare the simulation results to the theoretical values. It showed that the error was less than 3%. The pressure contours and velocity vectors showed the detail of the flow field when the robot implemented the basic motions.**

***Index Terms* – *3D Model, Spherical Underwater Robot, Hydrodynamic Analysis, Computational Fluid Dynamics (CFD).***

## I. Introduction

Thousands of underwater robots have been developed to satisfy the requirement of underwater tasks. The streamline shape is usually used in high speed underwater robots and the special shape is always used in remotely-operated vehicle (ROV) to carry out some real time underwater operations [1]-[3]. For a kind of special shape, spherical shape shows a good performance on water resistance and the spherical underwater robot is easy to realize 0 turning radius no matter in what motion state of the robot. Due to the advantages, some researchers proposed their spherical underwater robots. For example, the University of Manchester and University of Oxford co-developed a kind of μ-AUV [4]. This robot employed 6 propellers as propulsion system and equipped them on the equator of the spherical hull. The development purposes of this micro robot were monitoring the nuclear storage ponds and waste water treatment facilities to prevent leakage. There is another typical spherical underwater robot, ODIN-III, which is developed by University of Hawaii [5]. In our laboratory, a novel spherical underwater robot with vectored water-jet thruster is developed [6]-[13].

Due to the importance of hydrodynamic characteristics, a lot of researchers did study on spherical underwater robots. Leroyer *et al.* analyzed the DTMB5415 bare hull using a computational fluid mechanics (CFD) method, and proposed two numerical procedures that sped up the Reynolds-averaged Navier–Stokes (RANS) solvers [14]. After compared to a classical simulation the author obtained that, these two method numerical solutions can up to four times faster. Mylonas and Sayer predicted the forces acting on a yacht keel based on the large-eddy simulation (LES) and detached-eddy simulation (DES) solutions [15]. Propulsion systems have also been a main research subject. Wei *et al.* predicted the propeller-excited acoustic response of a submarine structure using a numerical method [16]. Cheng *et al.* analyzed the hydrodynamic characteristics of an unconventional propeller with an endplate effect and compared the results to those of a conventional propeller [17]. In my research, the water resistance is difficult to get from experiment, so we want to get the water resistance and other important features according to modeling and hydrodynamic analysis.

The paper is organized as follows. The section II briefly introduced the modelling and meshing of the SUR-II for hydrodynamic analysis. And then in section III, 3 major motion states are analysed by FLUENT. Finally, the conclusions and future work are pointed in section IV.

## II. Modeling of the Flow Field

### *A. Prototype of the SUR-II*

The prototype of the SUR-II is shown in Fig. 1. We adopted 3 vectored water-jet thrusters as propulsion system. In order to reduce the effect of the propulsion system on the robot flexibility and prevent the propulsion system from the impact force, the propulsion system is assembled inside of the spherical hull. The robot is also a kind of open frame structure. Water is easy to go through the robot, and all of the control parts are contained in the water proof box. This design will reduce the risk of leakage. The waterproof treatment of the servo motors and water-jet thrusters were also done respectively. The structure of SUR-II is too complicated to carry out hydrodynamic analysis because there are two much faces and parts. For example, 12 screws are used for fastening the water proof box and some wires are used to connect the servo motors and water-jet thrusters to the control circuit. Although we can build the 3D model by the software including all of these parts, they will affect the result of hydrodynamic analysis. Worse still, the ANSYS FLUENT cannot get a result due to the complexity. And the complicate structure will cost a long computational time.

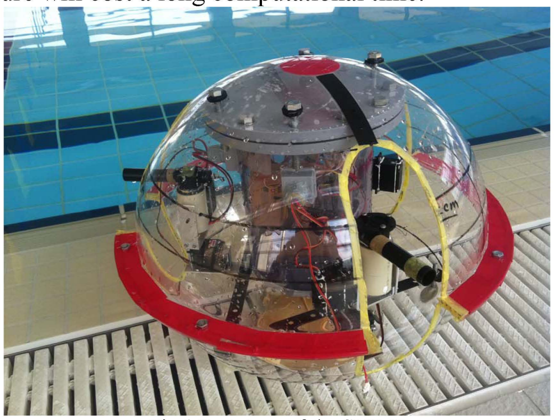
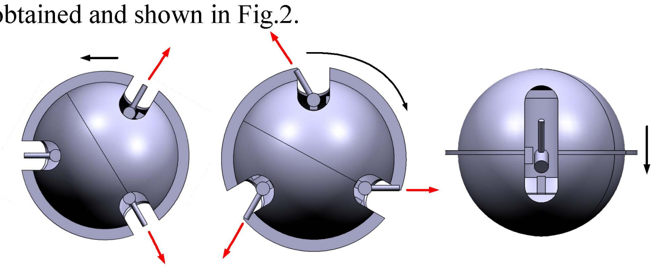


Fig.1 Prototype of the SUR-II

### B. *Modeling of the SUR-II*

The 3D model must be simplified for reducing the computation time and getting more effective result. The simplification standard is to reduce the amount of surfaces and parts. Therefore, some unimportant parts and surfaces will be simplified, including the screws, the wires and the support frame of the servo motors. All the ignored parts are small and complicated. Besides that, we simplify the shape of servo motor to cubical shape and the water-jet thruster to cylindrical shape. Finally, three models for the three basic motions are obtained and shown in Fig.2.



(a) Horizontal motion (b) yaw motion (c) vertical motion

Fig.2 Simplified models for 3 basic motions

### C. *Modeling of the flow field*

For the hydrodynamic analysis, we just care about the influence of the flow field from the robot, so the flow field must be built based on the 3D model of robot. The size of flow field should be big enough to ensure that the wall of flow field cannot affect the results of hydrodynamic analysis. Generally, the size and the shape of flow field are decided by the robot. If the speed of robot is relatively high and have a big effect on the flow field, we should choose a big flow field. In this research, the cylindrical flow field with a radius of 1 m and a length of 4 m. After decided the 3D model and flow field, Boolean operation is carried out between the 3D model and flow field. Then we use the flow field minus the 3D model of robot. Finally, the hydrodynamic analysis object is obtained as shown in Fig.3. We use the same method get three 3D models. Totally, there are three objects should be obtained because three motions must be analysed respectively.

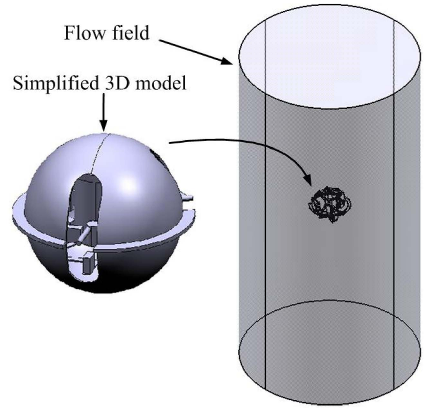
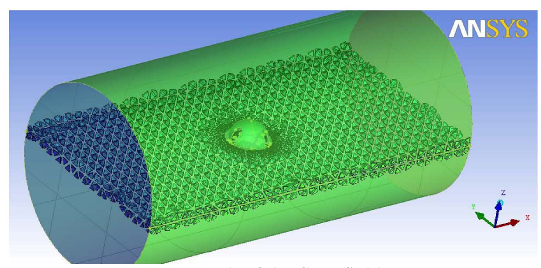



Fig.3 Simplified 3D model in the flow field

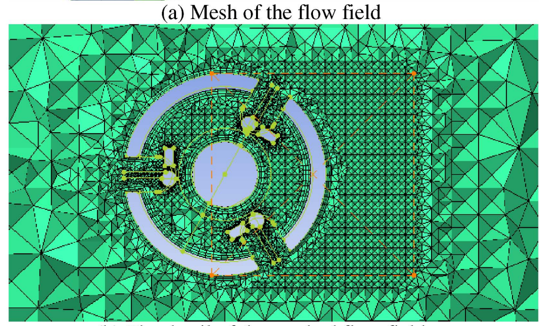
### D. *Mesh of the flow field*

Mesh of the flow field is the most important factor to the hydrodynamic analysis. The amount of mesh decides the performance of hydrodynamic analysis and computational complexity.

Fig. 4 shows the mesh of the flow field. In this research, a total of 1.5 million mesh elements were used. The Fig. 4(b) depicts the detail of the mesh around the robot. This is just an example for horizontal motion. For the complex structure of the robot, the size of element should be very small to get a good hydrodynamic analysis result. Near the wall of robot, three layers are set and mesh density is higher. The mesh is carried out in the ANASYS ICEM. Finally, we output the mesh files as a previous work of hydrodynamic analysis.




(a) Mesh of the flow field



(b) The detail of the meshed flow field

Fig.4 Mesh of the 3D model

## III. HYDRODYNAMIC ANALYSIS OF THE SUR-II

Due to the commercial software FLUENT combined into ANSYS, all the analysis works are carried out by using this software. In total, there are 3 meshing files which will be analysed for the three motions. The steps of CFD are as follows:

1. Import the meshing file to ANASYS FLUENT;
2. Set the condition of flow field;
3. Carry out solution;
4. Get the results.

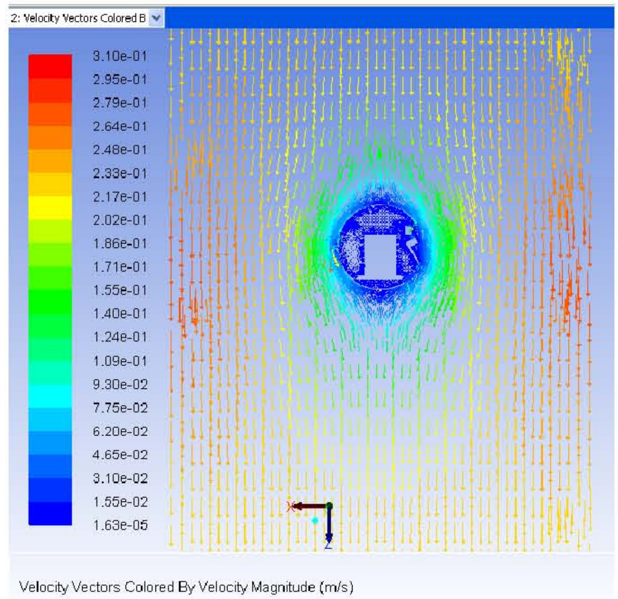
In the second step, the material and boundary condition should be set. In this research, k- ε model is selected to describe the fluid situation, while turbulence occurs when the robot moves at a speed of 0.3m/s. The maximum speed of robot is about 0.3m/s [18, 19]. There are two methods to describe the kinematic relationship between the fluid and the robot. The first method is dynamic mesh, which set the water to be static and the wall of robot to move. The second method is to assume that the water moves at a speed of 0.3m/s and the wall of robot is static. Because the second method is very simple and easy to be realized, we choose the second method [20]-[22].

### *A. CFD simulation for horizontal motion and vertical motion*

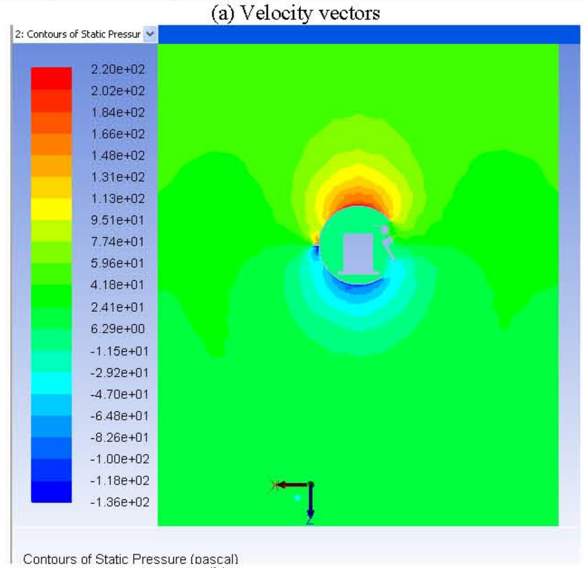
The robot and fluid move relative to each other, so the robot was set as a static wall while the fluid was set as a constant velocity flow. Fig. 5 presented how both the velocity and pressure were affected by the fin that was fixed on the equator of the robot. Thus, the fin cannot be ignored when the robot was moving in vertical direction. But the effect of the holes was not obvious. The velocity of the fluid inside the robot was the same as the velocity of the robot. Therefore, the fluid inside the robot can be assumed as part of the robot and the robot can be assumed as a sphere. However, for horizontal motion (Fig. 6), the fin can be ignored and the holes must be considered, because water flows into the robot through the front holes, and then out of the robot through the back holes, as shown in Fig.6. Thus, the robot cannot be assumed as a closed sphere. In summary, the fins cannot be ignored for vertical motion and the holes cannot be ignored for horizontal motion. Because the robot is set as static and the fluid is moving, the result of Fig 5(a) and Fig 6(a) are relative velocity. The blue colour stands for high speed part. The out boundary is set as pressure output.

### *B. Verify the drag coefficient by CFD simulation for horizontal motion and vertical motion*

The drag coefficient ($C_d$) is an important hydrodynamic characteristic. It is different in vertical direction and horizontal direction because there are three holes in the horizontal direction. As a sphere object in vertical direction, the $C_d$ is equal to 0.4. In horizontal direction, we can calculate the $C_d$ according to (1).




(a) Velocity vectors




(b) Pressure contours

Fig. 5 Influence of the robot to fluid during sinking down

$$F_d = \frac{1}{2} C_d(R_e)\rho V^2 A \qquad (1)$$

Where,

$F_d$: water resistance, it is equal to thrust force $T$;

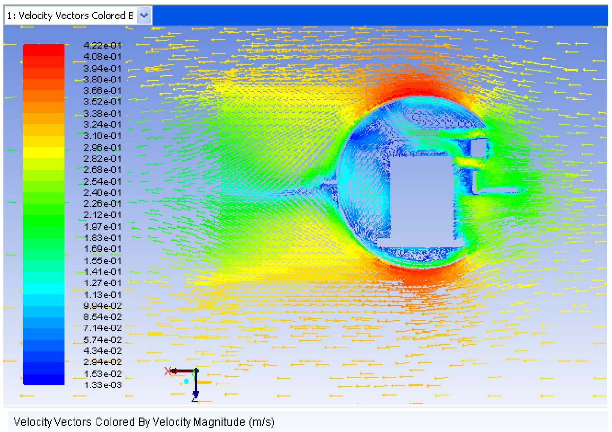
$C_d$ : the drag coefficient;

$R_e$ : the Reynolds number that reflect the flow characteristics;

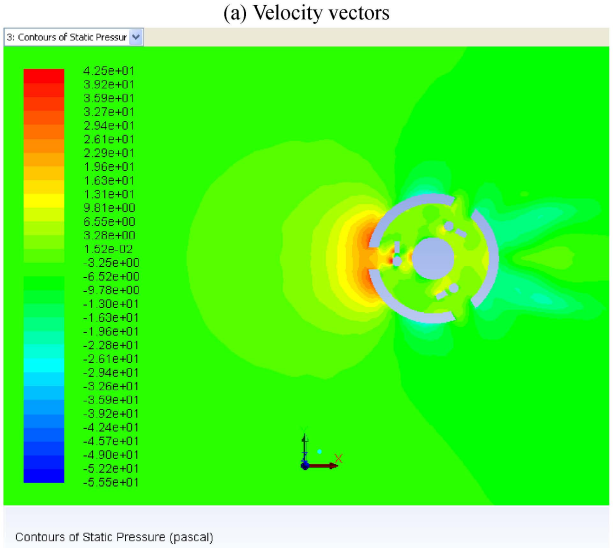
$V$ : the relative velocity of spherical underwater robot to the fluid;

$A$: the cross-sectional area;

$\rho$ : the density of the fluid.




(a) Velocity vectors




(b) Pressure contours

Fig. 6. The influence to fluid when the robot is moving in horizontal direction

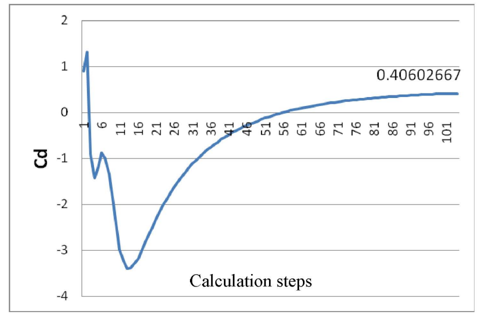
These parameters are easy to obtain, therefore,

$$C_d = \frac{2T}{\rho V^2 A} = 0.61. \qquad (2)$$

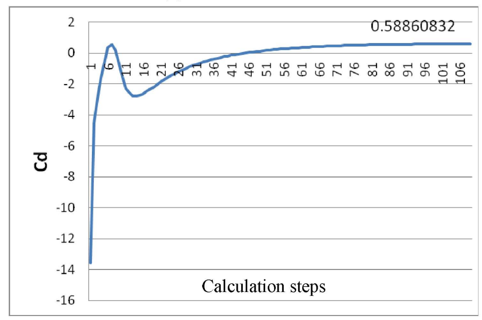
Besides the theoretical calculations, we can also get the drag coefficient from the CFD simulation, which are shown in Fig.7. The drag coefficient for vertical motion converged to a constant $C_d = 0.41$, similar to the value calculated in Section 3. For horizontal motion $C_d = 0.59$, which indicates a 3% error compared to the calculated value. Therefore, the results of the CFD analysis are acceptable.

### C. yaw motion

In addition to vertical and horizontal motion, rotational motion was also simulated. Usually, dynamic mesh and reference motion can be used to simulate the rotation motion, in this paper, because the simulation condition is constant, we selected reference motion to simulate the motion of robot. The velocity and pressure results indicated that the influence which rotational motion exercises on the fluid was negligible. The interaction between the fluid and the robot was caused by the propulsion system, as shown in Fig. 7, and a pressure surface was generated around the propulsion system Fig. 7 (b).




(a) Vertical motion




(b) Horizontal motion

Fig. 7 The drag coefficient

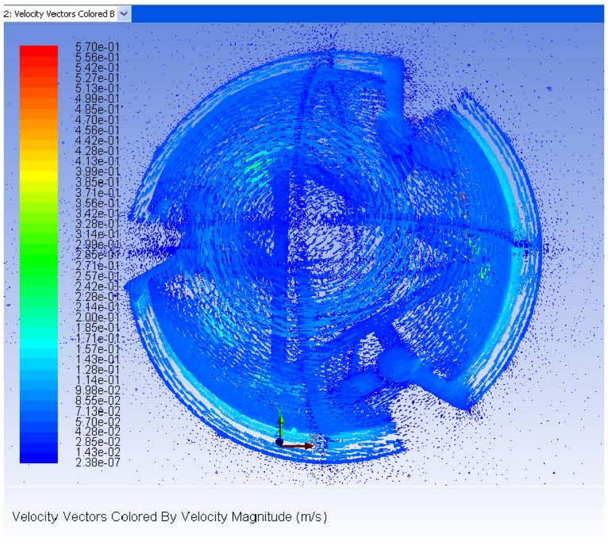
## CONCLUSIONS AND FUTURE WORK

This paper presented a hydrodynamic analysis for the second generation Spherical Underwater Robot (SUR-II). First of all, the mechanical structure of the SUR-II was presented. Due to the complexity of the SUR-II, it was difficult to get a good simulation result, so we simplified the 3D model of SUR-II. And then the 3D models were meshed by ANSYS ICEM. Finally, the mesh files were imported to ANSYS FLUENT to carry out the CFD simulation. Three main basic motions were analysed, including horizontal motion, vertical motion and yaw motion. The drag coefficient in vertical direction and horizontal direction were obtained and compared to the results of calculation. The simulation results showed that the drag coefficient error is fewer than 3%. The velocity vector and pressure contours clarified the hydrodynamic features and provided important evidence to confirm the assumptions that the robot could be assumed as a sphere in vertical motion.

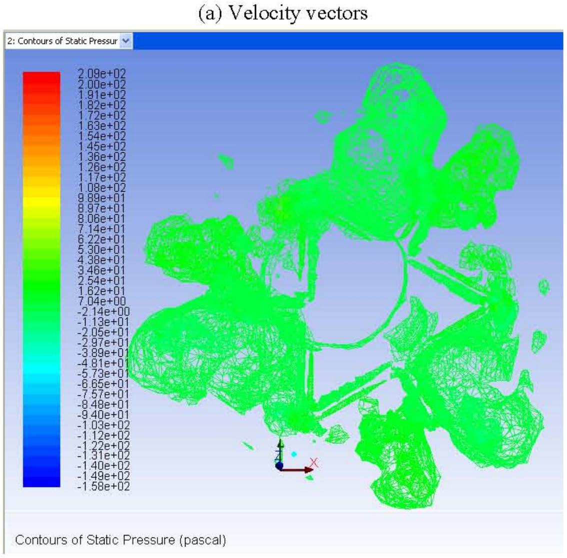
In the future, we will focus on some hybrid motions to carry out the hydrodynamic analysis.

## ACKNOWLEDGMENT

This research is partly supported by Tianjin Key Research Program of Application Foundation and Advanced Technology (the Natural Science Foundation of Tianjin) and Kagawa University Characteristic Prior Research Fund 2012.




(a) Velocity vectors




(b) Pressure contours

Fig. 7 The results of CFD analysis for yaw motion

## REFERENCES


[1] B. W. Hobson, A. D. Sherman and P. R. McGill, "Imaging and sampling beneath free-drifting icebergs with a remotely operated vehicle", *Deep-Sea Research II*, vol. 58, No. 11-12, pp. 1311-1317, 2011.

[2] R. Panish and M. Taylor, "Achieving high navigation accuracy using inertial navigation systems in autonomous underwater vehicles", *Proceedings of OCEANS'11 IEEE Santander Conference*, pp. 1-7, Santander, Spain, 2011.

[3] S. X. Wang, X. J. Sun, Y. H. Wang, "Dynamic Modeling and Motion Simulation for A Winged Hybrid-Driven Underwater Glider", *China Ocean Eng.*, Vol. 25, No. 1, pp. 97-112, 2011.

[4] S.A. Watson, D. Crutchley and P. N. Green, "The mechatronic design of a micro-autonomous underwater vehicle (μAUV)", *International Journal of Mechatronics and Automation,* vol. 2, No. 3, pp. 157-168, 2012

[5] H.T. Choi, A.Hanai, S.K. Choi and J.Yuh, "Development of an underwater robot, ODIN-III", *Proceedings of the 2003 IEEE International Conference on Intelligent Robots and Systems.* pp. 836-841, Lag Vegas, USA, 2003.

[6] X. Lin and S. Guo, "Development of a spherical underwater robot equipped with multiple vectored water-jet-based thrusters", *Journal of Intelligent and Robotic Systems*, vol. 67, pp. 307-321, 2012.

[7] S. Guo, J. Du, X. Ye, R. Yan and H. Gao, "The computational design of a water jet propulsion spherical underwater vehicle", *Proceedings of the 2011 IEEE International Conference on Mechatronics and Automation*, pp. 2375-2379, Beijing, China, 2011.

[8] S. Guo, X. Lin, K. Tanaka and S. Hata, "Development and control of a vectored water-jet-based spherical underwater vehicle", *Proceedings of the 2011 IEEE International Conference on Information and Automation*, pp. 1341 - 1346, Shenzhen, China, 2010.

[9] X. Lin, S. Guo, K. Tanaka and S. Hata, "Underwater experiments of a water-jet-based spherical underwater robot", *Proceedings of the 2011 IEEE International Conference on Mechatronics and Automation*, pp. 738-742, Beijing, China, 2011.

[10] J. Guo, S. Guo, N. Xiao, X. Ma, S. Yoshida, T. Tamiya and M. Kawanishi, "A novel robotic catheter system with force and visual feedback for vascular interventional surgery", *International Journal of Mechatronics and Automation,* vol. 2, No. 1, pp. 15-24, 2012

[11] B. Gao, S. Guo and X. Ye. "Motion-control analysis of ICPF-actuated underwater biomimetic microrobots", *International Journal of Mechatronics and Automation,* vol. 1, No. 2, pp. 79-89, 2011

[12] Q. Pan, S. Guo and T. Okada, "A novel hybrid wireless microrobot", *International Journal of Mechatronics and Automation,* vol. 1, No. 1, pp. 60-69, 2011

[13] X. Lin, S. Guo, C. Yue and J. Du, "3D modelling of a vectored water jet-based multi-propeller propulsion system for a spherical underwater robot", *International Journal of Advanced Robotic Systems,*vol.10,pp.1-8, 2013

[14] A. Leroyer, J. Wackers, P. Queutey, E. Guilmineau, "Numerical strategies to speed up CFD computations with free surface application to the dynamic equilibrium of hulls". *Ocean Engineering,* Vol. 38, pp. 2070–2076, 2011.

[15] D. Mylonas, P. Sayer, "The hydrodynamic flow around a yacht keel based on LES and DES". *Ocean Engineering*, Vol.46, pp. 18–32, 2012

[16] Y. Wei, Y. Wang, S. Chang, J. Fu, "Numerical prediction of propeller excited acoustic response of submarine structure based on CFD", FEM and BEM. *Journal of hydrodynamic*. Vol.2 No.24, pp.207-216, 2012

[17] H. Cheng, Y. Chien, C. Hsin, K. Chang, P. Chen, "A numerical comparison of end-plate effect propellers and conventional propellers". *Journal of hydrodynamic*. Vol.5, No22, pp.495-500, 2010.

[18] C. Yue, S. Guo, L. Shi. "Hydrodynamic Analysis of the Spherical Underwater Robot SUR-II", *International Journal of Advanced Robotic Systems,* pp.1-12, 2013.

[19] S. Guo, J. Du, X. Ye, H. Gao and Y. Gu, "Real-time Adjusting Control Algorithm for the Spherical Underwater Robot", *Information,* vol. 13, No. 6, pp. 2021-2029, 2010.

[20] X. Han and M. Neubauer, "A research on the switching control laws for synchronised switch damping on inductor technique", *International Journal of Mechatronics and Automation,* vol. 2, No. 3, pp. 207-216, 2012

[21] Z. Liu, S. Guo, H. Li and X. Lin, "An Improved 3D Modeling of Water-jet Propellers for a Spherical Underwater Robot", *Proceeding of the 2011 IEEE International Conference on Mechatronics and Automation,* pp. 319-324, Beijing, China, 2011

[22] Y. Jiang, S. Wang, K. Ishida, T. Ando, M. G. Fujie, A novel direction control method for walking support with an omnidirectional walker, *International Journal of Mechatronics and Automation,* vol. 1, No. 3/4, pp. 244-252, 2011.