CFD Simulation of Cooperative AUV Motion

Muhamad Husaini[2], Zahurin Samad[1], Mohd Rizal Arshad[2]

[1] School of Mechanical Engineering
Universiti Sains Malaysia, Engineering Campus
14300 Nibong Tebal, Seberang Perai Selatan, Pulau Pinang, Malaysia
Tel: +604-5996312, Fax: +604-5941025, E-mail: zahurin@eng.usm.my

[2] USM Robotics Research Group, School of Electrical and Electronic Engineering
Universiti Sains Malaysia, Engineering Campus
14300 Nibong Tebal, Seberang Perai Selatan, Pulau Pinang, Malaysia
Tel: +604-5937788 ext. 6002, Fax: +604-5941023,
E-mail: rizal@eng.usm.my, husaini_pge8078@yahoo.com.my

Abstract

Cooperative AUV performance and efficiency is directly related to its power efficiency. The power consumption for this type of underwater vehicle is influenced by its motion needed because most of the power is spent for thruster to produce thrust. Drag force is known as main contribution to resist the body motion. In this paper, the behavior of this force is study by using Computational fluid dynamic approach (CFD). Two position arrangement of cooperative AUV was chosen to study the drag variation. First, the distance effect between two AUV was studied to represent the basic position arrangement of cooperative AUV. second position arrangement is studied to look at the effect of different position arrangement. The comparison between distance and position arrangement was discuss in detail in this paper. The study show that the distance behind the leading AUV does not give much effect to drag force, but the position arrangement show significant important to drag force.

Keywords: Cooperative AUV, CFD, hydrodynamic coefficient.

Introduction

The concept of multiple Autonomous Underwater Vehicles (AUVs) cooperatively performing a mission offers several advantages over single vehicles working in a non-cooperative manner such as increased efficiency, performance, robustness and the emergence of new capabilities.[3] The recent advances in sensing, communication and computation enable the conduct of cooperative missions. Multiple, highly autonomous systems are envisioned because they are capable of higher performance, lower cost, better fault tolerance, reconfigurability and upgradability.[1]

The growth of cooperative AUV application increase the significant of optimum energy consumption of AUV. Most of current AUV powered by battery that supplied energy for propulsion system, electrical circuit etc.. Underwater thruster is commonly used in AUV for propulsion system. This system consists of motor and propeller. Many researchers currently work on motor design to achieve the best performance of AUV thruster [1, 5, 7]. Because of the limitation in energy, certain capability of AUV was disable and this will affect the final performance of AUV.

From this problem, the idea of leading follower AUV was emerge. Basic concept of this type of cooperative AUV is only one AUV will be install the motion planning capability. This AUV will lead the other's AUV trajeractory. Due to the series motion of AUV while its operation, the following AUV will face the different fluid flow because of leading AUV. This flow totally effected by disturbance generated by front AUV. The disturbance in fluid flow will lead to changing in separation point of AUV. Separation point is the position where the transition of fluid from laminar to turbulence occur. LIU Zhen et. proved that the turbulent flow occurred around the submerge body will increase the drag force acting on the body [6]. The increasing in the drag force will increase the thrust needed to move the AUV body[5]. new separation point predicted occur early than leading or first AUV.

The objective of this study is to determine the value of hydrodynamic force acting on the follower body due to disturbance of fluid flow in certain position arrangement. Also included in this study is the relationship between the distance of follower AUV nose to leading AUV tail. This relationship is important to further distance setting of AUV in term of minimizing the energy consumption.

For understanding the behavior of the following AUV that proposed earlier, the critical consideration is needed in mesh generation. LIU Zhen report in his work on fluid flow around the aerofoil, best quality of mesh will give the comparable value to experiment data. In this study the good quality of mesh will ensure the turbulence can be modeled properly and close to real condition [6]. Turbulence flow is important to this study because the following AUV will be directly facing the turbulent flow of leading AUV.

Approach and Methods

The effect of the wake produced by leading AUV motion will be studied by varying AUV distance. For this purpose the AUVs were arrange in series position arrangement as shown in Figure 1. The distance between front and back AUV will be varied every 10% from length of the body. Drag of the back AUV will be observed for different distance. In this study the angle of attack of body is set to zero to represent the motion along X axis only. This setting is to avoiding the effect of variation in flow direction.

For the body moving in fluid flow, there are two types of drag correspond to the body. These are the pressure drag and the skin friction drag. The pressure drag refers to the component of force where measure in drag direction. This force is due to integral of pressure distribution over the body. Following the d'Alambert paradox, when the body working in the inviscid fluid this integral should be zero because integration around closed body is symmetry [6]. However, in the real condition the pressure distribution decreases from the inviscid prediction in the regions of separated flow and consequently gives to rise non-zero values of integral. The skin friction drag, is the component of integral of the shear stresses $(\boldsymbol{\tau}-\boldsymbol{\omega})$ over the body surfaces, that is measured in the drag direction. In this paper the 2D viscous drag is considered and given by equation (1):

Viscous drag = skin friction drag + pressure drag. (1)

Grid generation

Geometry meshing in this study was generated by using Gambit software. The mesh file will be imported to FLUENT for numerical study. FLUENT is a commercial CFD package for running the complex differential equation by numerical approach. The solver used in FLUENT basically developed by applying the finite volume approach for discretization.

For this study 2D mesh was applied to Domain and geometry. Unstructured triangular mesh was chosen due to the complexity in position order of multiple AUVs. 7197 node and 21297 face was generated by this method. Figure 1 show the mesh scheme for the two cooperative AUVs. This position arrangement of AUV position is call position arrangement one in this study. This figure shows the operation position arrangement of AUV. The distance between the AUVs was varied in this study to determine the variation of drag.

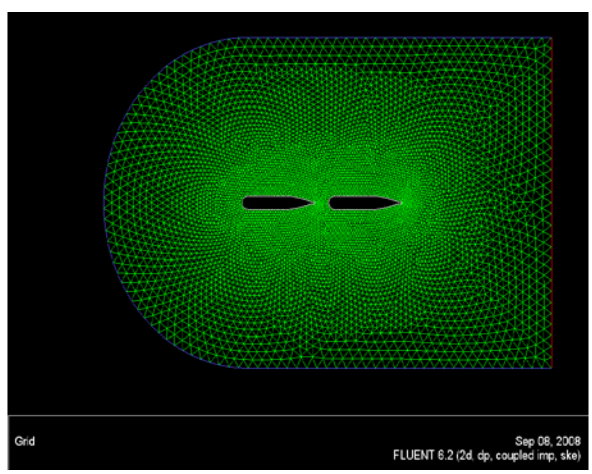


Figure 1: Domain mesh for distance 20% length of the body

Amit Tayagi define the three parameter that characterizing a computational grid is total number of grid points, location of outer computational boundaries and minimum spacing [2] . Generally, minimum spacing is refer to y^+ value, a dimensionless parameter representing a local Reynolds's number in the near-wall region.

y^+ value defined by Schlichting [2] is given by equation (2) :

$$y^+ = yu_* \qquad (2)$$

Where, $y =$ distance from the wall surface, $u_* = \sqrt{\tau_w/\rho}$, τ_w = shear stress at the wall, $\rho =$ density and $v =$ kinematic viscosity.

The value of distance of boundary layer region from the wall can be determined by applying the flat plate boundary layer theory,[2] the relationship can be derived in equation (3) :

$$y^+ = 0.172\left(\frac{y}{L}\right)Re^{0.9} \qquad (3)$$

where, Re = Reynold's number based on body length. In this paper the estimate minimum spacing is determined by setting the $y^+ = 1$ The equation above is solved by setting the value of L to 1.5. Important to note here that the value of y^+ is just estimated based on flat plate theory. The real value of y^+ is varies over the surface according to the flow in the boundary layer.

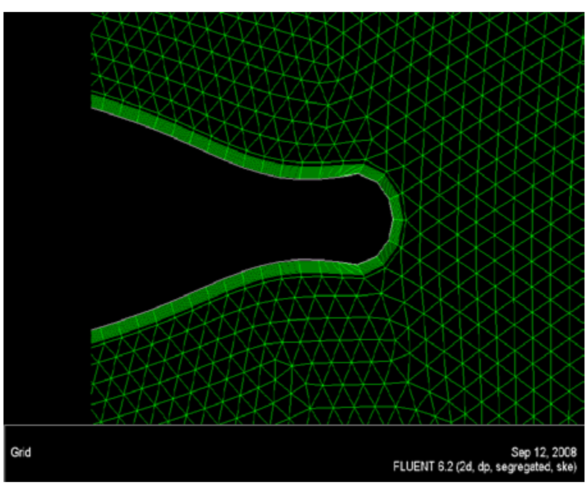


Figure 2: Boundary layer at tail of AUV

Numerical method

The governing equations for mass and momentum are written as below:
Mass conservation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = 0 \qquad (4)$$

Momentum conservation:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla\cdot(\rho\vec{v}\vec{v}) = -\nabla\rho + \nabla\cdot(\bar{\bar{\tau}}) \qquad (5)$$

In this equation, $\vec{v}$ represent the velocity vector in Cartesian coordinate. P the static pressure and $\bar{\bar{\tau}}$ the stress tensor given by equation (6) :

$$\bar{\bar{\tau}} \equiv \mu\left[\left(\nabla\vec{\mathcal{V}} + \nabla\vec{\mathcal{V}}^{\mathrm{T}}\right) - \frac{2}{3}\nabla\cdot\vec{\mathcal{V}}\right. \qquad (6)$$

μ is molecular viscosity, ***I*** the unit tensor and the second term of right hand sight is volume dilation. In this paper k-ε turbulent model was used. This turbulent model are based on Reynolds averaging approach. After apllied Reynolds averaging term, the Navier Stoke equation can be written in Cartesian Form as :

Mass conservation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (7)$$

Momentum conservation:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \qquad (8)$$

Here δ_{ij} is the Kronecker delta, and $-\rho\overline{u_i' u_j'}$ the Reynolds stresses.

This governing equation will be descretized by using Finite volume approach which support by FLUENT package.

Result and discussion

The simulation reports that the drag behavior of series order AUV is approximately constant with distance. This show that the drag is not much effect by series arrangement. Only small difference of drag force for various distance reported. Figure 3 shows the relationship between drag and the distance from leading AUV. The figure 3 show that the leading AUV always has the greater drag compare the follower AUV. This because the leading AUV facing the maximum pressure caused by maximum velocity drop. For the translational motion, the main contribution for total drag is pressure drag. The different in drag between leading and follower easily explain by pressure coefficient plot as shown in figure 4.

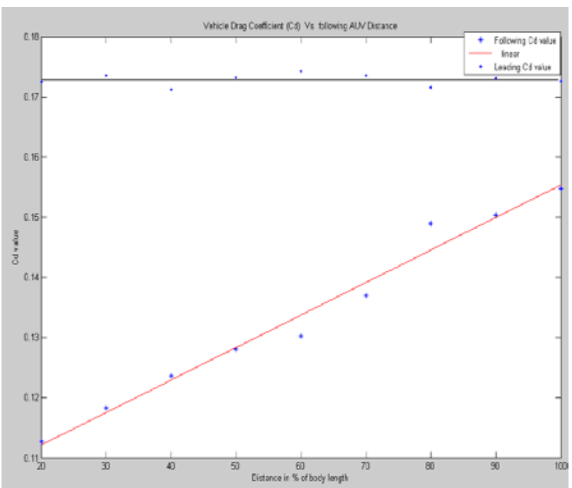
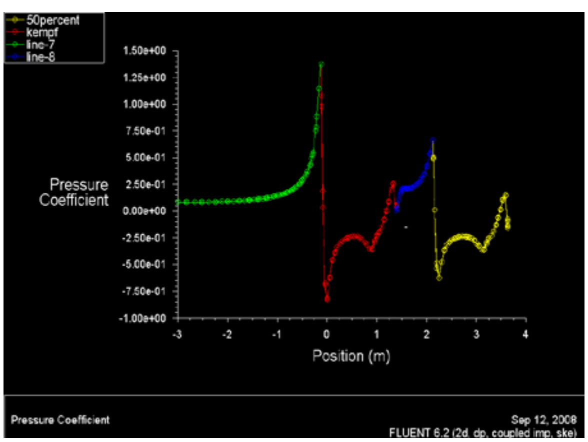


Figure 3: Cd vs distance



Graph 2: Graph CP VS position

In figure 4 the variation of Pressure Coefficient (CP) was plotted along x axis. From this figure the maximum pressure occur at -0.125 or stagnation point of leading AUV. The blue plot represent the CP behind the leading AUV, from there, the CP value increase until the new peak. This peak is stagnation point of follower AUV. From the graph, the value of this peak is lower than CP value at leading AUV. this condition occur caused of the variation of velocity along the X axis. At the body of both AUV, the CP distribution almost same. The CP value are depend on the shape position arrangement of the body and location on the body. for this the shape and position arrangement of the body is identical. Figure 5 show the CP contour around AUV body.

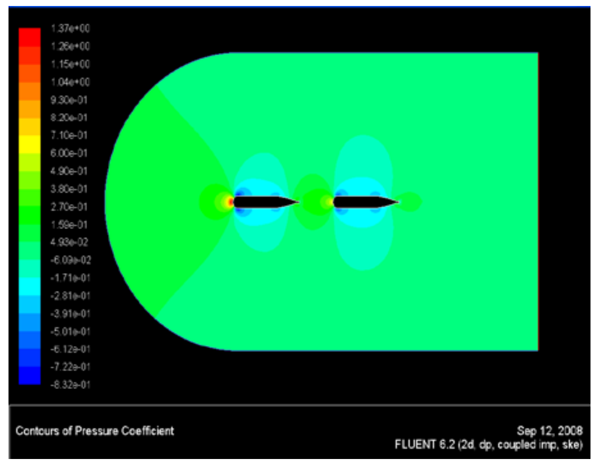


Figure 3: Pressure coefficient contour.

From this simulation the drag behavior of AUV is not affected much by the distance with leading AUV. However the drag force of AUV is reduced by present of leading AUV. The main contribution of leading AUV in term to reduce the drag is it can slow down the velocity of fluid. So that the following AUV only facing the smaller fluid velocity rather than front AUV. This small velocity lead to small velocity drop at stagnation point. This condition leads to reducing the pressure drag.

Second position arrangement of AUV position was simulated to understand the drag behavior of AUV body by different position. In this study, three AUVs were arranged as shown in figure 6. This position arrangement is called position arrangement two in this study.

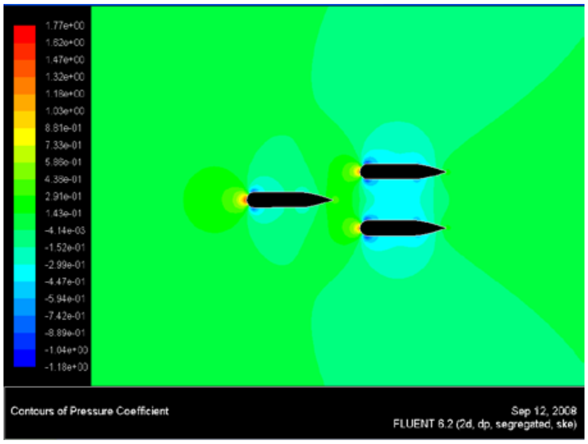


Figure 3: CP contour plot.

Simulation for 2m/s motion velocity of position arrangement one and two give the result for drag as show in Table 1. Cd1 and Cd2 represent the value of drag coefficient for position arrangement one and two respectively.

Table 1: Cd value of the position arrangement.

AUV	Cd1	Cd2
Leading	0.1630	0.2023
Follower	0.1020	0.1941

From the table the value for Cd of leading and follower in position arrangement one is smaller than position arrangement two. Important to note here that second position arrangement increase the value of Cd for leading and follower. But the Cd value different between leading and follower for position arrangement two is decrease compare to position arrangement one.

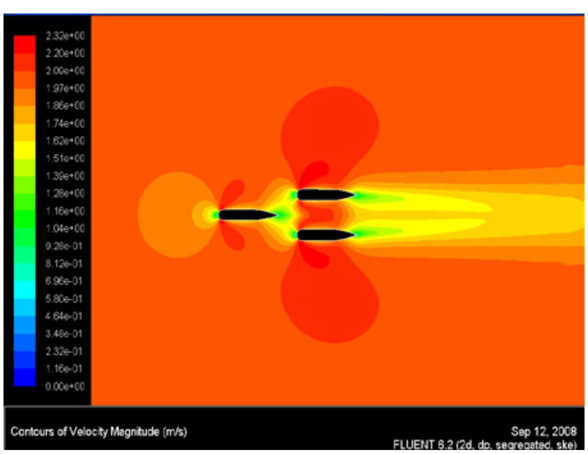


Figure 4: velocity contour for position arrangement 2.

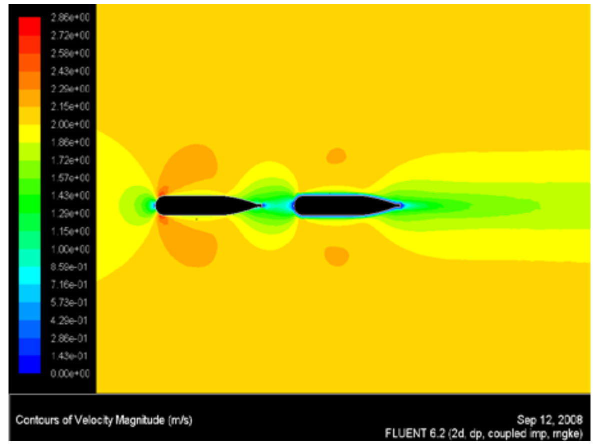


Figure 5 velocity contour for position arrangement 1.

As shown in Figure 4, the follower AUV faces the inlet velocity directly and the leading AUV not much slowdowns the velocity. Compared to velocity contour of position arrangement one in Figure 5, the velocity around the nose of the follower is low due to leading effect. So that the pressure drop around the body is increase and the value of pressure drag also increase. Increasing in Cd value of leading will reduce the efficiency of cooperative operation in term of energy saving.

Conclussion and future work.

From the study, the distance between leading and follower is not much affect to Cd value. But the different in position arrangement, like position arrangement one and two give the significant different in Cd value. The Cd value of different distance behind the leading AUV almost constant so it is for following AUV. but the drag for folower AUV in position arrangement one always lower than leading drag. For the second position arrangement, this order lead to increasing the drag value for leading and follower.further study must be conducted to optimize this position arrangement for decrease the drag. This result shows that the position arrangement of AUV order in cooperative is important factor for increase the effeciency of AUV operation. The optimum position arrangement must be explore increase the cooperative AUV performance. From this work, new position arrangement must be study to find the effect of other position arrangement to the AUV drag.

Acknowledgements

We are very grateful to National Oceanographic Directorate (NOD) and Ministry of Science, Technology and Innovation (MOSTI) for providing funding to pursue our research in underwater system technology.

References

1) T.Sarkar, P.G. Sayer, and S.M. Fraser, 1997. A Study of Autonomous Underwater Vehicle Hull Forms Using Computational Fluid Dynamics. International Journal For Numerical Methods in Fluids 25:1301-1313.

2) Amit Tyagi, and Debabrata Sen, 2006. Calculation of Transverse Hydrodynamic Coefficients Using Computational Fluid Dynamic Approach. Ocean Engineering 33(2006):798-809.

3) Hongwei Zhang, and Shuxin Wang, 2007. Modelling and Analysis of an Autonomous Underwater Vehicle Via Multibody System Dynamics. 12th IFToMM World Congress, Besançon (France), June18-21,2007.

4) Erfu Yang, and Dongbing Gu, 2007. Nonlinear Formation-Keeping and Mooring Control of Multiple Autonomous Underwater Vehicles. IEEE/ASME Transactions on Mechatronics, Vol. 12, NO. 2, April 2007.

5) Qiuling Jia, and Guangwen Li, 2007. Formation Control and Obstacle Avoidance Algorithm of Multiple Autonomous Underwater Vehicles (AUVs) Based on Potential Function and Behavior Rules. Proceedings of the IEEE International Conference on Automation and Logistics August 18 - 21, 2007, Jinan, China.

6) Liu Zhen, Hyun Beom-Soo, Kim Moo-Rong and Jin Ji-Yuan. 2008. Experimental and Numerical Study For Hydrodynamic Characteristics of an Oscillating Hydrofoil. Journal of Hydrodynamics, 2008,20(3): 280-287