CFD simulation of cooperative AUV motion

Muhamad Husaini*, Zahurin Samad1 & Mohd Rizal Arshad2

1 School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Seberang Perai Selatan, Pulau Pinang, Malaysia
[E-mail: zahurin@eng.usm.my]
2 Underwater Robotics Research Group (URRG), 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]

Received 14 July 2009, revised 11 September 2009

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 requirement since most of the power is spent for thruster propulsion. Drag force is known as the main parameter in resisting the body motion. In the present study, the behavior of this force is studied 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 investigated to represent the basic position arrangement of cooperative AUV. Second, the effect of different position arrangement was also investigated. The comparison between distance and position arrangement is discussed in this paper. Present study elucidates that the distance behind the leading AUV does not give much effect to the drag force, but the position arrangement indicated significant influence.

[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 capabilities1. The recent advances in sensing, communication and computation enable the implementation of cooperative missions. Multiple, highly autonomous systems are envisioned because they are capable for higher performance, lower cost, better fault tolerance, reconfigurability and upgradability2,3. The growth of cooperative AUV application increases the significant of optimum energy consumption of AUV. Most of the current AUV platform are powered by batteries that supply energy for sub-systems such of the propulsion system and electrical module. Underwater thruster is commonly used in AUV platform for propulsion system. This system consists of the motors and propeller. Many researchers are currently working on AUV thruster design and it’s performance2,3.

The basic concept of this type of cooperative AUV is that only one AUV will given the motion planning capability. This AUV will lead the other AUV’s trajectory. Due to the motion of first leading AUV while in operation, the following AUV will have to face the incoming fluid flow. This flow totally due to the turbulence generated by the leading AUV. The disturbance in fluid flow will lead to changes in the separation point of the AUV. Separation point is the position where the transition of fluid from laminar to turbulence occurs. LIU Zhen et. (2008) proved that the turbulent flow occurred around the submerge body will increase the drag force acting on the body4. The increase in the drag force will increase the thrust needed to move the AUV body5. New separation point is predicted to occur early than the leading 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 distances of follower AUV nose to the leading AUV tail. This relationship is important to further distance setting of the AUV, in terms of minimizing the energy consumption.
In order to understand the behavior of the following AUV which has been proposed earlier, the critical consideration is needed in mesh generation. LIU Zhen et. (2008) has reported in his work on fluid flow around the aerofoil, that the best quality of mesh will give a comparable result to the experimental data. In this study, a very good quality of mesh will ensure that the turbulence can be modeled properly, and close to the real condition. Turbulence flow is important in this study because the following AUV will be directly facing the turbulent flow from the leading AUV platform.

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

There are two types of corresponding drag for the body moving in fluid flow, viz. the pressure and the skin friction drags. The pressure drag refers to the component of force when measured in the drag direction. This force is due to the integral of pressure distribution over the body. Following the d’Alambert paradox, when the body is working in the inviscid fluid, this integral should be zero. This is because the integration around the closed body is symmetric. However, in 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 the integral. The skin friction drag is the component of integral of the shear stresses 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):

\[ \text{Viscous drag} = \text{skin friction drag} + \text{pressure drag}. \quad \ldots (1) \]

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 has been developed by applying the finite volume approach for discretization.

A 2D mesh was applied to water around the body for this study. Unstructured triangular mesh was chosen due to the complexity in the position order of multiple AUVs. There were 7197 node and 21297 face was generated by this method. The mesh scheme for the two cooperative AUVs are ascribed in Fig. 1. It shows the operation position arrangement of the AUVs. The distance between the AUVs was varied in this study to determine the variation of drag.

Amit Tyagi (2006) define the three parameter that characterize a computational grid is total number of grid points, location of outer computational boundaries and minimum spacing. Generally, minimum spacing is refer to \( y^+ \) value, a dimensionless parameter representing a local Reynolds’s number in the near-wall region.

\( y^+ \) value is defined by Schlichting:

\[ y^+ = \frac{yu}{\nu} \quad \ldots (2) \]

where, \( y = \) distance from the wall surface, \( u_* = \sqrt{\tau_w/\rho}, \tau_w = \) shear stress at the wall, \( \rho = \)density and \( \nu = \)kinematic viscosity.

The value of distance of boundary layer region from the wall can be determined by applying the flat plate boundary layer theory, the relationship can be derived below:

\[ y^+ = 0.172 \left( \frac{Y}{L} \right) \text{Re}^{0.9} \quad \ldots (3) \]

Fig. 1—Domain mesh for distance 0.2 length.
where, \(Re\) = Reynold’s number is based on the body length. In this paper the estimate minimum spacing is determined by setting the \(y^+ = 1\) equation above solved by setting the value of \(L\) is 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 show the boundary layer of the mesh geometry.

### Numerical method

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

**Mass conservation:**

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0
\]  

... (4)

**Momentum conservation:**

\[
\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \frac{\partial \tau}{\partial t} + \nabla \rho \vec{g}
\]

... (5)

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

\[
\frac{\tau}{\rho} = \mu \left[ (\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \vec{v} I \right]
\]

... (6)

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

**Mass conservation:**

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

... (7)

**Momentum conservation:**

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \frac{\partial \rho u_i}{\partial x_j} \delta_{ij} + \frac{\partial}{\partial x_j} (-\rho u_i u_j)
\]

... (8)

Here \(\delta_{ij}\) is the Kronecker delta, and \(-\rho u_i u_j\) the Reynolds stresses.

This governing equation will be discretized by using Finite volume approach which support by FLUENT package. The governing equation was solved using the boundary condition as shown in Fig. 3. The iteration was stop at 830 because of convergence criterion has meet. This simulation limited the convergence criterion to 1E-5. This value is comparable enough to define the accuracy of the solution.

### Results and Discussions

The simulation shows that the drag behavior of serial order AUVs is approximately constant with distance. The drag is not much affected by the serial arrangement. Only small difference of drag force for various distance are reported. Fig. 3 shows the relationship between drag and the distance from the leading AUV is presented in Fig. 4. The leading AUV...
always has the greater drag compared to the succeeding AUV (Fig. 4). This is because the leading AUV faced the maximum pressure caused by maximum velocity drop. The main contribution for total drag is pressure drag. The different in drag between leading and follower can be explained by pressure coefficient plot as shown in Fig. 5.

The variation of Pressure Coefficient (CP) was plotted along x axis n Fig. 5. The maximum pressure occurs at -0.125 or stagnation point of leading AUV (Fig. 5). The blue plot represents the CP behind the leading AUV. From there, the CP value increases until the new peak. This peak is the stagnation point of the follower AUV. The value of this peak is lower than the CP value at leading AUV. This condition occurs because of the variation of velocity along the X axis. At the body of both AUVs, the CP distribution is almost the same. The CP value is dependent on the shape position arrangement of the body, and location on the body. For this the shape and position arrangement, the body is identical. The CP contour around AUV body is presented in Fig. 6.

The simulation studies reveal that the drag behavior of AUV is not affected much by the distance with leading AUV. However, the drag force of AUV is reduced by the presence of the leading AUV. The main contribution of leading AUV in terms drag reduction, it can reduce 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 AUVs was simulated to understand the drag behavior of AUV body by different position. In this study, three AUVs were arranged as shown in Fig. 7.

The simulation has carried out for the velocity of AUVs in 2m/s for both arrangements. Table 1 shows the result of each arrangement. Cd1 and Cd2 in Table 1 represent the value of drag coefficient for position arrangement one and two respectively.

The value for Cd of leading and follower in position arrangement one is smaller than position arrangement 2 (Table 1). Important to note here that second position arrangement increase the value of Cd for leading and follower AUVs. But the Cd value different between leading and follower for position arrangement two is decrease compare to position arrangement one.

<table>
<thead>
<tr>
<th>AUV</th>
<th>Cd1</th>
<th>Cd2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leading</td>
<td>0.1630</td>
<td>0.2023</td>
</tr>
<tr>
<td>Follower</td>
<td>0.1020</td>
<td>0.1941</td>
</tr>
</tbody>
</table>
The Fig. 5 reveals that the succeeding AUV faces the inlet velocity directly and the leading AUV not much slowdowns the velocity. Compared to velocity contour of position arrangement one in Fig. 6, 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.

**Conclusion**

The study comprises, the distance between leading and follower AUVs give an effect to the CD value. he different in position arrangement, like position arrangement one and two also give the significant different in Cd value. But the drag for follower 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 the succeeding AUVs. Studies may 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 efficiency of AUV operation. The optimum position arrangement must be explore increase the cooperative AUV performance. This work endured that a 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), Malaysia, for providing funding to pursue our research in underwater system technology.

**References**
