Abstract- The hull resistance of an autonomous underwater vehicle (AUV) is an important factor in determining the power requirements and range of the vehicle. AUV hull profiles are traditionally cylindrical with propulsion through the means of externally placed thrusters in order to optimise production cost. This design of AUVs has been known to increase drag on the system. This paper discusses the design method using Computational Fluid Dynamics (CFD) for determining the hull resistance of an AUV and the effects of an internally placed thruster system as compared to the traditionally placed external thrusters.

IndexTerms- hull, thruster, drag, duct, CFD

I. INTRODUCTION

Autonomous robots not only represent the next great milestone for science, but their practical uses span a wide range. AUVs are such an example. In particular, AUVs are being used to explore hostile environments far too hazardous for humans or manned vehicles. These environments are of particular interest to researchers studying extremophiles, microorganisms capable of surviving in extraordinary circumstances. The hull resistance of an AUV is an important factor in determining the power requirements and range of the vehicle. This paper describes a design method using Computational Fluid Dynamics (CFD) for determining the hull resistance of an AUV under development. The CFD results reveal the distribution of hydrodynamic values (velocity, pressure, etc.) of the AUV with a ducted propeller. The optimization of the AUV hull profile for reducing the total resistance is also discussed.

This paper demonstrates that shape optimization in conceptual design is possible by using a commercial CFD package. The optimum designs to minimize the drag force of the AUV were carried out, for a given object function and constraints. The base of any AUV is the hull. A hull is the main body of the vehicle that houses the electronics and supports the thrusters. Basic structure of any AUV that we observe is cylindrical or ellipsoidal. Hence, to get an optimized design, analysis was carried out for the two.

Mini AUVs generally possess two or more thruster for propulsion and maneuvering. Traditionally, these thrusters are placed external to the hull. Upon placing the thrusters internal to the hull with ducts providing for thruster functionality, the frontal projected area is reduced, also variations in the drag experienced occur. The same was modeled and tested for through CFD simulations.

II. MODELS

The geometries for the simulation were rendered using Solidworks 2014 whereas the analysis for the same was conducted on ANSYS FLUENT 15.0. While comparing the cylindrical profiled hull and the elliptical profiled hull, the two structures were made volumetrically similar so as to ensure similar payload. 

$$\frac{4}{3}\pi r^3 + \pi l = \frac{4}{3}\pi a^2b + \pi abl \quad (1)$$

Where, r is the radius of cylindrical profile, l is the length of the extruded section, a and b are the major and minor radii of the elliptical profile.

Using equation (1) and fixing the values for the cylinder, l = 50 mm, r = 25 mm, values of b and a of the ellipse were found for various aspect ratios, i.e. b/a.

<table>
<thead>
<tr>
<th>Aspect ratio</th>
<th>a (mm)</th>
<th>b (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>33.20</td>
<td>16.6</td>
</tr>
<tr>
<td>0.6</td>
<td>30.86</td>
<td>18.516</td>
</tr>
<tr>
<td>0.7</td>
<td>28.97</td>
<td>20.28</td>
</tr>
<tr>
<td>0.8</td>
<td>27.422</td>
<td>21.938</td>
</tr>
<tr>
<td>0.9</td>
<td>26.12</td>
<td>23.58</td>
</tr>
<tr>
<td>1.0</td>
<td>25</td>
<td>25</td>
</tr>
</tbody>
</table>

Fig. 1. Cylindrical Hull profile model
To model the thruster positioning, two identical elliptical profile hulls were taken. For the external thruster system, 2 thrusters were mated to the exterior of the hull, while for the internal thruster system, thrusters were placed inside extruded ducts along the length of the hull. The ducts were designed such that the velocity gradient within the duct is gradual and occupies the least volume.

III. SOLVER MODEL

The Realizable K-epsilon model is chosen for simulation. It is one of the most common turbulence models, and has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results in cases where mean pressure gradients are small.

Following equations govern the flow profile around the models simulated:

\[
\frac{\partial}{\partial t}((\rho k) + (\rho u_j u_j)) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + F_k + P_k - \rho e - Y_M - \beta_k
\]

\[
\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial e}{\partial x_j} + p C_p \right] = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] - \rho \frac{S^2}{2} + \rho \frac{S^2}{2} - \frac{C_{C_1}^2 C_{C_2}}{\sqrt{S^2}} - \frac{C_{C_3} C_{C_4} C_{C_5} S}{\sqrt{S^2}}
\]

Where,

\[ C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \quad \eta = \frac{S^2}{\epsilon}, \quad S = \sqrt{2S_{ij}S_{ij}} \]

IV. MESHING AND SETUP

A cylindrical enclosure with equal cushion of 100mm is created for each of the geometries and Tetrahedral Meshing is used for the analysis. A quarter model is used during the simulation as the model is symmetric about the two planes along the length. This is an efficient approximation that is accurate and saves calculation time. Every model is simulated at medium coarseness, with sizing and inflation properties such that the mesh is highly refined at the region around the geometry.
V. RESULTS AND DISCUSSIONS

The Cd values for all the b/a ratios generated were plotted and it was realized that for a ratio of 0.8, the Cd value obtained was the least. Force report for each model is rendered and the table as below is obtained:

<table>
<thead>
<tr>
<th>b/a ratio</th>
<th>Force (N)</th>
<th>COP</th>
<th>Cd</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.0963</td>
<td>0.0449</td>
<td>0.1115</td>
</tr>
<tr>
<td>0.6</td>
<td>0.0971</td>
<td>0.0468</td>
<td>0.1069</td>
</tr>
<tr>
<td>0.7</td>
<td>0.0985</td>
<td>0.0485</td>
<td>0.1069</td>
</tr>
<tr>
<td>0.8</td>
<td>0.0998</td>
<td>0.0497</td>
<td>0.1058</td>
</tr>
<tr>
<td>0.9</td>
<td>0.1029</td>
<td>0.0529</td>
<td>0.1066</td>
</tr>
<tr>
<td>1.0</td>
<td>0.1052</td>
<td>0.0539</td>
<td>0.1074</td>
</tr>
</tbody>
</table>

From the above results it can be noted that although the resistance is least for a ratio of 0.5, the drag is high. For a ratio of 0.8 satisfying values are obtained for both the force resistance and the drag.

FIG. 6. Coefficient of drag (cd) plotted against hull profile aspect ratios

FIG. 7. Pressure coefficient of drag of External and Internal thruster systems

From figure 7 it can be seen that the pressure drag associated with system is lower in the case of and internal thruster system. However the viscous drag is slightly higher, and is contributed largely by duct friction. Study on the effect duct friction produces on the propulsion of the AUV is matter for further study.

CONCLUSIONS

The study showed that the ellipsoidal hull profile of similar volume produced lesser axial fluid drag, as compared to the cylindrical hull profile. AUVs with internally ducted thruster systems showed lesser fluid drag due the reduction in frontal projected area, in comparison to an externally placed thruster system. The effective system of an ellipsoidal hull with internally placed thruster can increase service hours due to the reduced power consumption to overcome fluid drag. There is further scope in this study by varying the design of the duct to optimize outlet velocity and thrust force.

ACKNOWLEDGMENT

Acknowledge with thanks C.S.Prasad, Professor, Department of Mechanical Engineering, RV College of Engineering, Bangalore for guiding in the progress of our project.

REFERENCES


