Journal of Basic and Applied Engineering Research
p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 5, Issue 5; July-September, 2018, pp. 452-456

http://www.krishisanskriti.org/Publication.html

Pressure and Shear Stress Distribution over External Hull of an Autonomous Underwater Vehicle

Pritam Ghosh[1] and Pranibesh Mandal[2]

[1,2]Jadavpur University
E-mail: [1]pghosh_me@yahoo.com, [2]m.pran11@gmail.com

Abstract—*Under water vehicle design has been on the cards for long. Primary aspect of such vehicle design lies in proper geometrical design of the outer hull thereby providing minimum possible hull drag. However it is the pressure and shear stress distribution around the hull surface which dictates the hull drag and other forces applied on the hull. CFD analysis has been a useful tool to determine the same. The present work focuses on determination of such pressure and shear stress distribution over a 3D submarine hull geometry in simulation frame using ANSYS software platform.*

1. INTRODUCTION

The growing use of unmanned systems operating in air, on ground even under water are continually demonstrating new possibilities that can assist us in many ways in our daily life. The popularity of unmanned aerial vehicles which was initiated by US Department of Defense in 2005 and till date there are over 1000 UAS models being developed over 50 countries. In 2014 Cai et. al.[1] presented a brief overview on recent advances of small-scale unmanned aerial vehicles from three perspectives viz. platform, key elements and scientific research. A detailed survey has been conducted on 132 small-scale unmanned aerial vehicle models available worldwide. The small-scale unmanned aerial vehicles are classified into three categories bases on their collected data namely small-tactical, miniature, and micro. They have also presented the evolvement of key elements of a small-scale unmanned aerial vehicle including onboard processing units, navigation sensors, mission-oriented sensors, communication modules, and ground control station. An unmanned underwater vehicle is playing a vital role mainly in oceanic exploration, industrial and defense applications.

The HUGIN 1000 autonomous underwater vehicles (AUVs) has achieved great success in the civilian survey industry over past six years. Hagen et. al. [2] at San Diego, USA, has been presented a detail military applications of the HUGIN 1000 AUV. The HUGIN 1000 AUV has been developed jointly by the Norwegian Defense Research Establishment (FFI) and Kongsberg Simrad. Primary applications of HUGIN 1000 include mine countermeasures, rapid environmental assessment, anti-submarine warfare (ASW).

Yu et. al. [3] studied on ODYSSEY autonomous underwater vehicle which was deployed in September 1998 in Massachusetts and Cape Cod Bays for coastal optical and biological research. They also given the demonstration on AUVs which can be effectively used to study important coastal problems such as, concurrently measure bio-optical and physical properties and their interaction with physical process in coastal regions for ecosystem protection, proper utilization of oceanic resources, mapping and predicting underwater optical properties and visibility etc.

However the design of underwater vehicle is a complex and computationally expensive exercise and a big challenge too. To design of an underwater vehicle has long been an active area of research. While designing a submarine one needs to focus on various hydrodynamic parameters which play very significant role in submarine motion under water. Since under submerged condition energy is limited, so it is obvious that minimum resistance on the outer surface of the submarine is very necessary to attain and thus appropriate design of hull is an important criteria. Use of water channels have been a conventional method of finding an optimal hull shape in order to minimize the drag under water thereby reducing the engine power requirement. Recently with the development of the computational techniques costly water channel experimentations can be simulated in the computational domain. Submarine hull shape design using such computational methods have been also emerged now-a-days. In 2008 Karim et. al. [4] has presented a paper on numerical computation of viscous drag for axisymmetric underwater vehicles. In that paper mainly the simulated data and experimental data have been compared. Various types of hull have been used there of different L/D ratio ranging from 4 to 10. For the computation method finite volume method based on Reynolds-average Navier-Stoke (RANS) equations have been used for computing the viscous drag. For making the problem more realistic turbulent flow past the axisymmetric hull has been taken into account and to do so in computational model a Shear Stress Transport (SST) model has been used. Finally good agreement has been found between computational results and experimental results.

Another important requirement for autonomous underwater vehicles is the reduction of propulsive power. And this reduction is very much dependent on hull form design. So, for the reduction of propulsion power the hull design is a primary objective. But due to absence of reliable and sufficiently accurate experimental data large number of hulls are needed for experimental analysis and it is a very expensive and laborious work. But due to the development of Computational Fluid Dynamics (CFD) this problem becomes cost-effective and much accurate results can be obtained. In 1997 Sarkar et. al. [5] worked on four different axisymmetric hull forms. For obtaining the results general-purpose CFD software PHOENICS is used which helps for developing a new computationally efficient and numerically robust flow simulation technique. This computational method is then used for various geometries and it has found that the results obtained from this method is more accurate for designing purpose and significantly is reduces the cost during the earlier stage of design of hull form.

Maneuvering characteristics of a marine is also a major hydrodynamic factor and for that it is to be predicted during various design stage and validated after the construction of the vessel during trial tests. Abkowitz model is a much used model for predicting the maneuvering characteristic of a marine vehicle. Virtual maneuvering test in Computational Fluid Dynamic (CFD) has been performed by Hajivand and Mousavizadegan [6] where a DTMB 5512 model ship is used for maneuvering oblique towing test to obtain the linear and nonlinear velocity dependent damping coefficients in a CFD environment. Freely accessible Open FOAM library and two well-known turbulence models ($k-\varepsilon$ and SST $k-\omega$) are used in simulation and it has been found that SST $k-\omega$ model gives more accurate results because it works good to predict separate flow at high drift angles.

In 2014 Paz and Muñoz [7] also worked on submarine hull design. Their main focus was on multi-objective optimization of hull design. The term “multi-objective” is used here as they developed a synthesis model for the design concept of submarine consisting of three main factors viz. a parametric definition of the hull geometry, a maneuverability model based on slender-body and a resistance model with an optimization technique. The design variables are taken such which are adequate to define the geometry of the hull form. Upon optimization the resultant geometry is obtained and the resulting geometry showed good relationship between hull’s wetted surface and resistance.

However many of such exercises have been done considering two-dimensional simulation techniques. Applicability of such 2D techniques in practical field has been questionable. A proper 3D computational analysis has thus been of necessity to find out the simulated hull shape design in 3D frame that can be used for real-life manufacturing. Primarily 3D simulation analysis of water velocity distribution adjacent to the external surface of a moving submarine is very much required in order to find out the pressure variation around the same. This particular analysis actually comprises the present work.

2. NUMERICAL METHODOLOGY

Computational fluid dynamics (CFD) is extensively used to predict fluid flow, mass transfer, heat transfer and related phenomenon by solving mathematical models which govern the problem and by taking the advantage of the speed of computers, day by day it is becoming a powerful tool to the researchers. Also it is offers very cost-effective solution to many problems in underwater vehicles. However implementation of CFD on marine hydrodynamics depends on proper selection of turbulence model, boundary resolution and grid generation.

For this study an axisymmetric hull is used. The geometry is shown in Figure 1 and Figure 2.

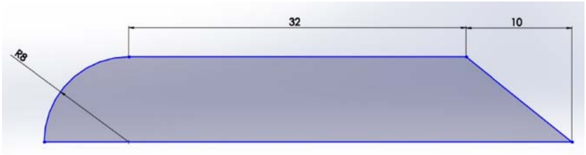

Figure 1: 2D Geometry of hull (all dimensions are in cm).

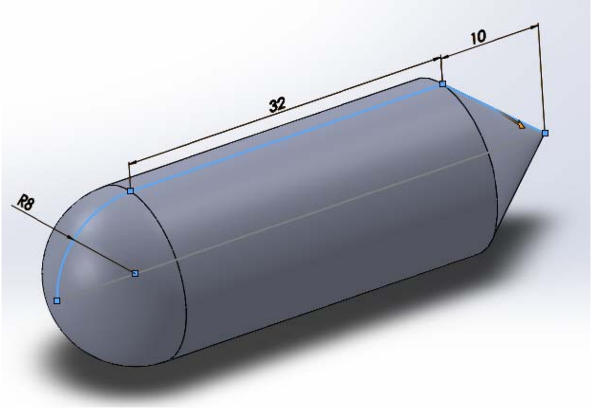

Figure 2. 3D Geometry of hull. (All dimensions are in cm).

The continuity equation for the flow past an axisymmetric hull form in cylindrical co-ordinate is given by:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}\left(\rho v_x\right)+\frac{\partial}{\partial r}\left(\rho v_r\right)+\frac{\rho v_r}{r}=S_m \tag{1}$$

Where ρ is fluid density, t is time, r is radial co-ordinate, v_x is the axial velocity, v_r is the radial velocity and S_m is the source term (taken as zero in this study). The axial and radial momentum equations are given as:

$$\frac{\partial}{\partial t}\left(\rho v_x\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(\rho r v_x v_x\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r v_x v_r\right)=-\frac{\partial p}{\partial x}+F_x \tag{2}$$

$$\frac{\partial}{\partial t}\left(\rho v_r\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(\rho r v_x v_r\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\rho r v_r v_r\right)=-\frac{\partial p}{\partial r}+F_r \tag{3}$$

where p is the static pressure and ρ is the external body force (taken as zero here) and

$$\nabla\cdot\vec{v}=\frac{\partial v_x}{\partial x}+\frac{\partial v_r}{\partial r}+\frac{v_r}{r} \tag{4}$$

3. THE SHEAR-STRESS TRANSPORT (SST) K-Ω MODEL

The shear-stress transport (SST) $k-\omega$ turbulence model was developed by Menter in 1994. It is a two-equation eddy-viscosity model. Combining the $k-\omega$ model in the near-wall region with the free-stream independence of the $k-\varepsilon$ model in the far field, SST model becomes much accurate and robust. The use of a $k-\omega$ formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer, hence the SST $k-\omega$ model can be used as a Low-Re turbulence model without any extra damping functions. The SST formulation also switches to a $k-\varepsilon$ behavior in the free-stream and thereby avoids the common $k-\omega$ problem that the model is too sensitive to the inlet free-stream turbulence properties. Authors who use the SST $k-\omega$ model often merit it for its good behavior in adverse pressure gradients and separating flow. The SST $k-\omega$ model does produce a bit too large turbulence levels in regions with large normal strain, like stagnation regions and regions with strong acceleration. This tendency is much less pronounced than with a normal $k-\varepsilon$ model though.

3.1 The Shear-Stress Transport (SST) k-ω model

Transport equations for the SST $k-\omega$ model are given by:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}\left(\rho k u_i\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+\tilde{G}_k-Y_k+S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_i}\left(\rho\omega u_i\right)=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial\omega}{\partial x_j}\right)+\tilde{G}_\omega-Y_\omega+D_\omega+S_\omega \tag{6}$$

In these equations, $\tilde{G}_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, $\tilde{G}_\omega$ represents the generation of ω, Γ_k and Γ_ω represent the effective diffusivity of k and ω, respectively, Y_k and Y_ω represent the dissipation of k and ω due to turbulence, D_ω represents the cross-diffusion term, S_k and S_ω are user-defined source terms.

3.2 Computational Method

For computational analysis commercial CFD software ANSYS Fluent has used to model flow over a 3D submarine hull. A rectangular envelop is taken as the computational domain within which the hull is placed. The computational domain is found large enough to capture the entire viscous-inviscid interaction and the wake development. For obtaining the solution of the Reynolds averaged Navier-Stokes equations finite volume method is used here. PISO algorithm is used here for pressure-velocity coupling and a second order upwind scheme is used for the convection term, Second Order interpolation scheme is used to compute face pressure and Second Order Upwind scheme is used for the discretization of the momentum equations.

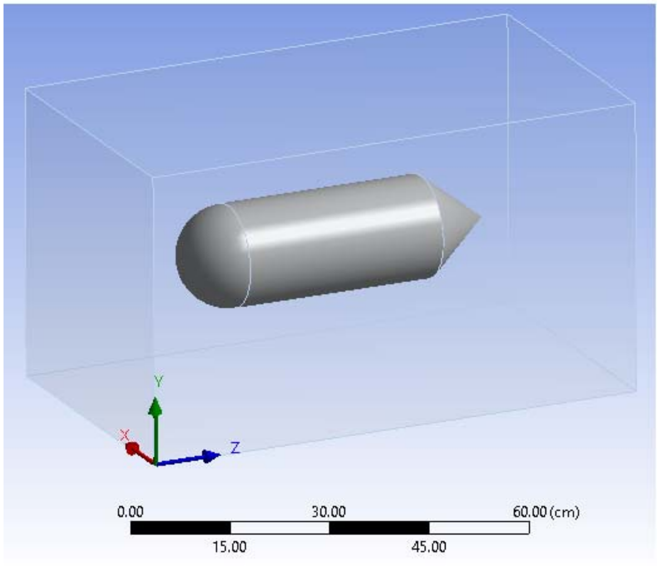

Figure 3: Computational domain.

Water is taken as the working fluid and other variables that are used in this computational method are given in Table 1. Aluminium is used as the hull material. In computational domain the wall of envelop in front of hull is considered as the inlet boundary. The wall of envelop backside of the hull is considered as the outlet. And the remaining walls of envelop are taken as wall having zero shear stress.

Table 1: Working fluid properties

Parameter	Value	Units
Water Density	998.2	kg/m^3
Water Viscosity	0.001003	kg/m-s
Inlet velocity	5	km/hr

4. RESULTS AND DISCUSSION

After running the simulation it has been observed that in front of the hull a high pressure zone, demarked as red colour, has been developed due to impact of water on the hull shown in figure 4. Following this high pressure zone a low pressure zone, demarked as blue colour, has also been developed as the stream lines are being elongated when it passes over the curved surface of the hull. The pressure again starts rising and remains more or less constant throughout the cylindrical hull shape or body of the submarine. This pressure again has suddenly decreased at the beginning of tail of the submarine due to separation of streamline. This variation of pressure around the hull is presented in a plot shown in figure 5.

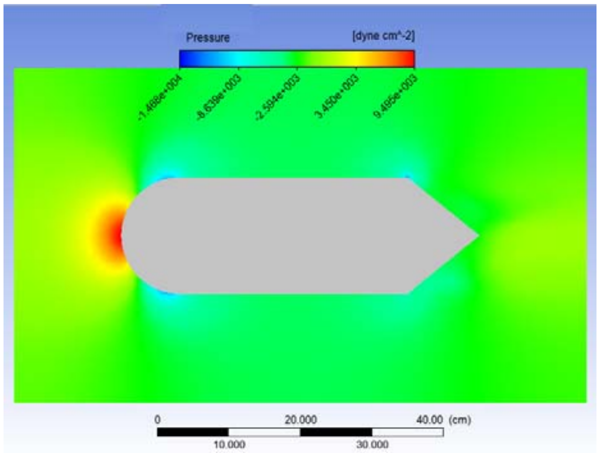

Figure 4: Contour of pressure around the surface of AUV

From figure 6 and figure 7 it is seen that just after the contact of water with the hull body, wall shear stress increases sharply till the water reaches the cylindrical part of the hull. When the water flows past the tail of the hull is has been observed that there is a drastic change of wall shear stress and at this position nothing can be concluded as there the flow is highly turbulent.

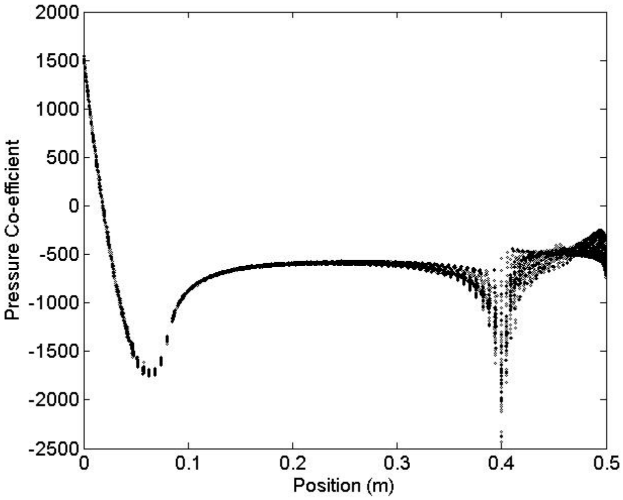

Figure 5: Variation of pressure coefficient around AUV

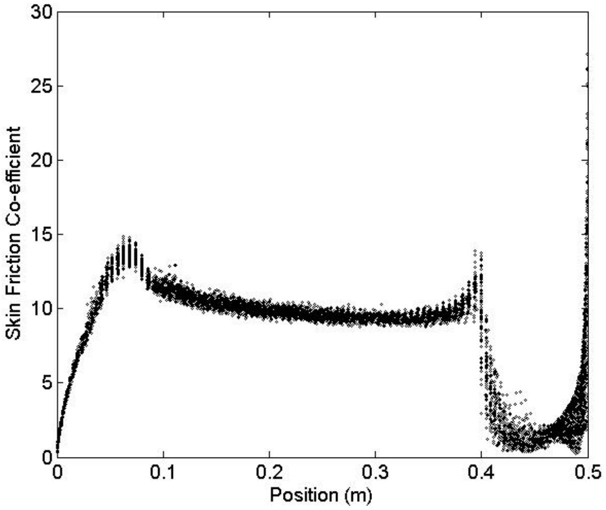

Figure 6: Variation of skin friction coefficient on AUV

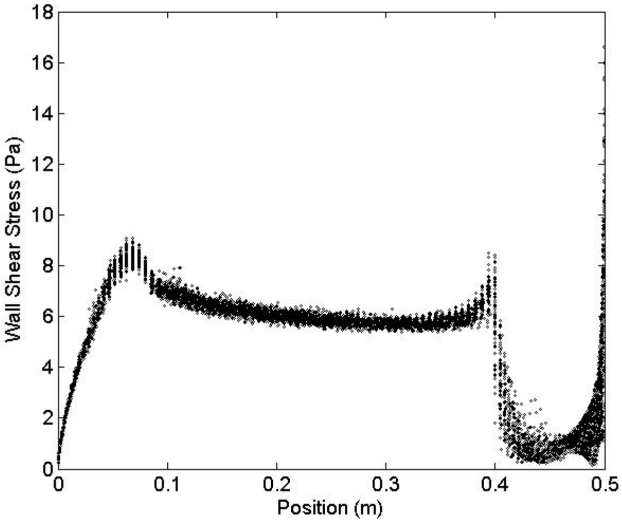

Figure 7: Variation of wall shear stress on AUV

5. CONCLUSION

The pressure and wall shear stress distribution over an autonomous underwater vehicle is obtained using commercial CFD software ANSYS. The numerical results conforms the theoretical knowledge.

REFERENCES

[1] Guowei Cai, Jorge Dias, and Lakmal Seneviratne, “A Survey of Small-Scale Unmanned Aerial Vehicles: Recent Advances and Future Development Trends”, *World Scientific Publishing Company, Unmanned Systems,* 2, 2, April 2014, pp. 1-25.

[2] Karstein Vestgård, and Per Kartvedt, “The HUGIN 1000 Autonomous Underwater Vehicle for Military Applications” *MTS/IEEE (Oceans 2003), San Diego, CA, USA,* September 2003.

[3] Xuri Yua, Tommy Dickeya, James Bellinghamb, Derek Manova, and Knut Streitlien, “The application of autonomous underwater vehicles for interdisciplinary measurements in Massachusetts and Cape Cod Bays”, *Continental Shelf Research*, 22, 2002, pp. 2225-22245.

[4] Md. Mashud Karim, Md. Mahbubar Rahman, and Md. Abdul Alim, “Numerical computation of viscous drag for axisymmetric underwater vehicles”, Journal Mekanikal, December 2008, pp. 9-21.

[5] T. Sarkar, P. G. Sayar and S. M. Fraser, “A Study of Autonomous Underwater Vehicle Hull Forms Using Computational Fluid Dynamics”, Int. J. Numer. Meth. Fluids, 25, 1997, pp. 1301–1313.

[6] Ahmad Hajivand and S. Hossein Mousavizadegan, “Virtual maneuvering test in CFD media in presence of free surface”, *Int. J. Nav. Archit. Ocean Eng.*, 7, March 2015, pp. 540-558.

[7] Jaime David Mora Paz and Oscar Darío Tascón Muñoz, “Multiobjective Optimization of a Submarine Hull Design”, *Ship Science & Technology,* 7, 14, January 2014, pp. 27-42.