THE EXPERIMENTAL METHOD AND CFD STUDY OF THE BARE HULL FORM OF UNDERWATER VEHICLE

Thu Han Tun

Professor/Head, Marine Mechanical Department, Myanmar Maritime University

Thanlyin, Yangon Region, The Republic of the Union of Myanmar, e-mail: dr.thuhantun@gmail.com

Abstract

Hull form selection, resistance and powering are important in designing underwater vehicle. An underwater vehicle bare hull form is based on the five parameters due to the interaction between the propeller and the hull. When they are running on the surface condition, there will be problems likely as surface vessel, but the main hull of the underwater vehicle is below the waterline with low freeboard. The underwater vehicles are operating with high speed at a high Froude Number. Therefore, the wave making component becomes important in surface resistance. The flow around the ship's hull is complicated, so that model experiments are still the most reliable data source on ship resistance determination. The bare hull form of underwater vehicle resistance is based on the model experiments and CFD results. The towing mechanism arrangement should be considered at model. Therefore, towing mechanism is designed for model testing. This paper discusses the towing method and result between model test and CFD. This paper also makes comparison of wave formation Towing tank test and CFD at various speeds. The model was tested with bare hull form of underwater vehicle in the Ship Model Towing Tank at Marine Hydrodynamics Centre, Myanmar Maritime University. CFD analysis is also carried out and the results are compared for surface condition. The software packages FINEMarine and Solidworks are used for CFD simulations. The comparison of results shows that the coefficient differences are less staggered based on the speed.

Keywords: Underwater Vehicle, Model Experiment, Resistance, CFD, Towing Tank

NOMENCLATURE

LOA	Length overall (m)
LWL	Length waterline (m)
L_B	Bow length (m)
L_C	Cylinder length (m)
L_{CS}	Conical Stern length (m)
γ	Conical Stern angle (degree)
D_H	Hull Diameter (m)
T	Draft (m)
Δ	Displacement (tons)
V	Speed of vessel (m/s)
R_t	Total resistance (N)
C_T	Total resistance coefficient
C_F	Frictional resistance coefficient
C_R	Residual resistance coefficient
Re	Reynolds number
ρ	Density of water (ton/m^3)
S	Wetted surface of ship (m^2)

Introduction

Hull form selection, resistance and powering are important in designing underwater vehicle. Model testing is carried out traditionally to predict the resistance and power at the near surface condition. Model is bare hull form without any appendages. Appendages resistance also should be considered for prediction of the resistance.

Nowaday CFD tools are widely used for prediction of ship's resistance and power. However the flow around the ship's hull is complicated, so that model experiments are still the most reliable data source on ship resistance determination. The model experiments will be carried out in the Ship Model Towing Tank at Marine Hydrodynamics Centre, Myanmar Maritime University[1]. Towing tank size established 2011 made in UK (CUSSON Technology) is 60m in length, 4m in breadth and 4m in depth. Maximum carriage speed is 4 m/s. CFD codes are also used in design step, validation of the results is carried out by comparing the model test results.

Objective

This paper focuses on towing mechanism which arranged for model testing with bare hull form, comparing the total resistance of underwater vehicle model and to make verification for model tests results with CFD results.

Study area

Underwater vehicles are designed to work under the water but they have to work at the water surface for many reasons. So the resistance of the underwater vehicle operating at the water surface is one of the major problems in designing that kind of vehicles. Their resistance behavior at the water surface is similar to those of normal surface ships, but underwater vehicles have low freeboard with majority of the displacement is below the water surface. The operating speed of the underwater vessels is high. They are also operating at high Froude Number, so the wave making component becomes dominant in surface resistance [2].

By towing scale models, the resistance can be measured and using appropriate scaling laws, the resistance of the full scale vessel can be predicted.

Model of 1.5 m in length is fabricated with wood and will be tested in model basin at Myanmar Maritime University. Total resistance of the model will be calculated from the towing test results by using Froude's Law of Comparison. In order to calculate the underwater vehicle model total resistance non-dimensional coefficients will be used [3]. The residual resistance is found by the pressure distribution about the hull because of the waves and eddies generated by the vessel's motion. Therefore, the wave making resistance of the underwater vehicle model at surface condition will be analysed by using CFD tools. Friction component of resistance will be calculated by using ITTC'57 correlation line and coefficient of total resistance is counted by as below.

$$C_T = C_F + C_R \tag{1}$$

$$C_F = \frac{0.075}{(\mathrm{Log}_{10}\mathrm{R_e} - 2)^2} \tag{2}$$

In general, the resistance of a vessel to motion through water can be calculated in the following formulation:

$$R_t = \frac{1}{2} C_T \rho S V^2 \tag{3}$$

Methodology of Study

Determining the Main Dimensions

An underwater vehicle bare hull form is based on the following parameters due to the interaction between the propeller and the hull –

- The fineness ratio
- Prismatic coefficient
- Nose radius
- Tail angle and
- The position of the maximum section

Dimensions of underwater vehicle model in this paper are shown in Table 1 with parallel middle body form. Relation L/D is equal to 8.3 because of limitation of testing arrangement and model making by hand. Main hull is cylindrical shape and stern is conical as shown in Figure 1.

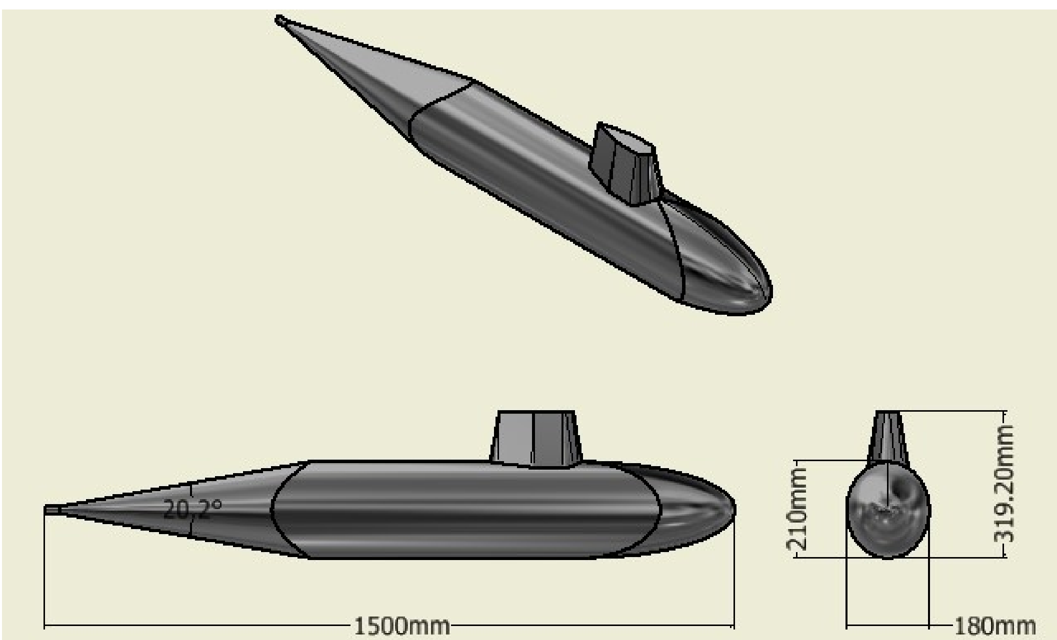


Figure 1 Model of underwater vehicle

Table 1: Main Particulars of Model at Design Condition

Main Particulars	Unit
Overall length, LOA (m)	1.500
Hull Vertical Diameter, D_{VH} (m)	0.210
Hull Horizontal Diameter, D_{HH} (m)	0.180
Displacement, Δ (tons)	0.24
Bow length, L_B (m)	0.333
Cylinder length, L_C (m)	0.590
Conical Stern length, L_{CS} (m)	0.576
Draft, T (m)	0.172
Conical Stern angle, γ (degree)	20.2

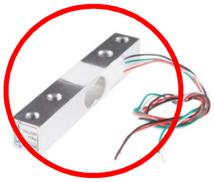
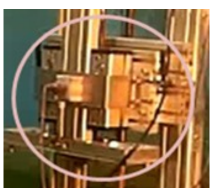
Model Making and Towing Mechanism

Fabricated wooden model without appendages are shown in Figures 2 below.

Figure 2 Fabricated wooden model

When model without appendages is testing in towing tank at near surface condition, even keel condition cannot be controlled. Therefore, the towing mechanism arrangement should be considered at model as shown in Figure 3. Elements of towing mechanism are listed in detail as the following Table 2 –

Table 2: List of Towing Mechanism

Symbol	Items	Function
	Load cell (sometimes called a strain gauge)	To measure the downforce or lift force
	Guide arm fit pins	To ensure the straight line position on flow (2 numbers for bow and stern)
	Load cell (sometimes called a strain gauge)	To measure the resistance of model (attached to model)
	Vertical slipway or towing post	To measure the heaving motion and tow the model by carriage.

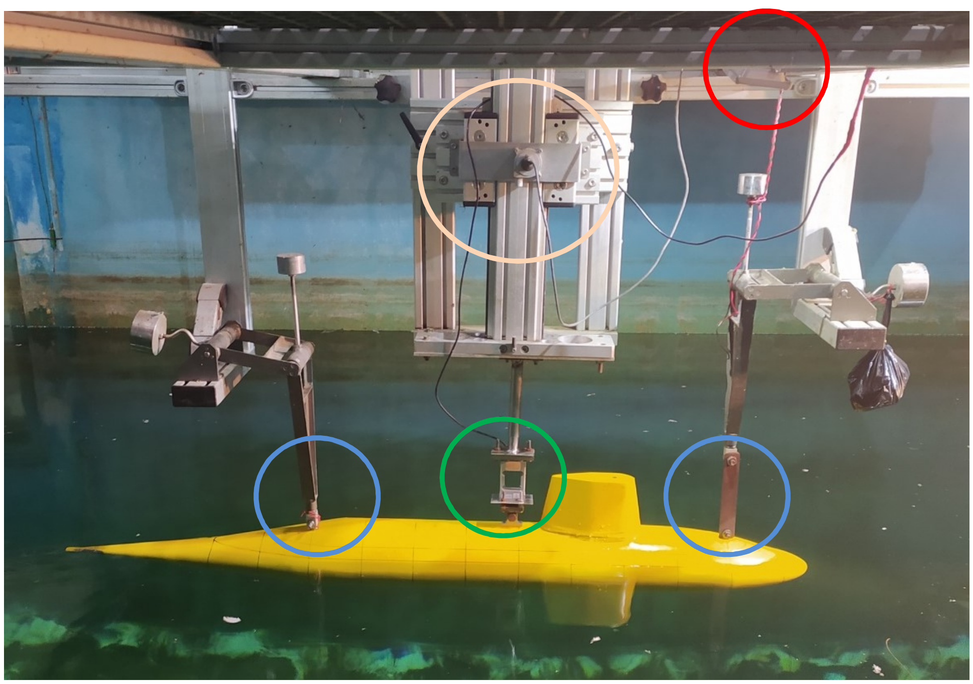


Figure 3 Towing mechanism arrangements

Experimental Approach

Two test case studies will be carried out for the speed range of 0.6 to 1.8 m/s. There are results in Table 3 below for near surface condition.

Table 3: Model Resistance Test

Speed of Vessel V (m/s)	Total Resistance of Model R_t(N)	Down Force (N)
0.6	0.800	0.720
0.8	2.187	2.764
1.0	4.522	6.129
1.2	7.937	10.796
1.4	7.457	10.446
1.6	12.497	8.794
1.8	18.015	7.363

Numerical Approach

The complexity of the flow around ship's hull, model experiments are still the most reliable data source on ship resistance determination; nevertheless, numerical methods have strongly advanced in this field, so that a combined use of both model tests and CFD codes can be very useful for ship design and for understanding the ship hydrodynamics [4].

Numerical computations are carried out around vessel using the two software packages FINEMarine with the turbulence model k-omega (SST-menter) and Solidworks 2020 Flow Simulation with turbulence only. The initial mesh is X axis 24, Y axis 16, Z axis 12 and total cells 843146 presented in Figure 4(a and b).

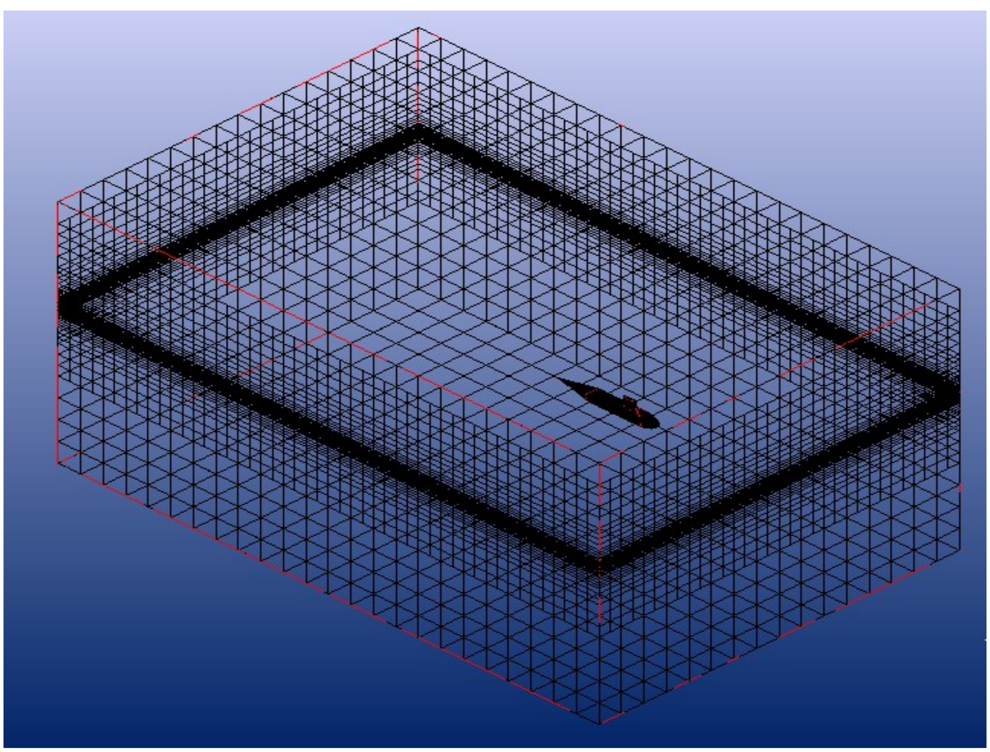


Figure 4a Generated mesh of the whole model

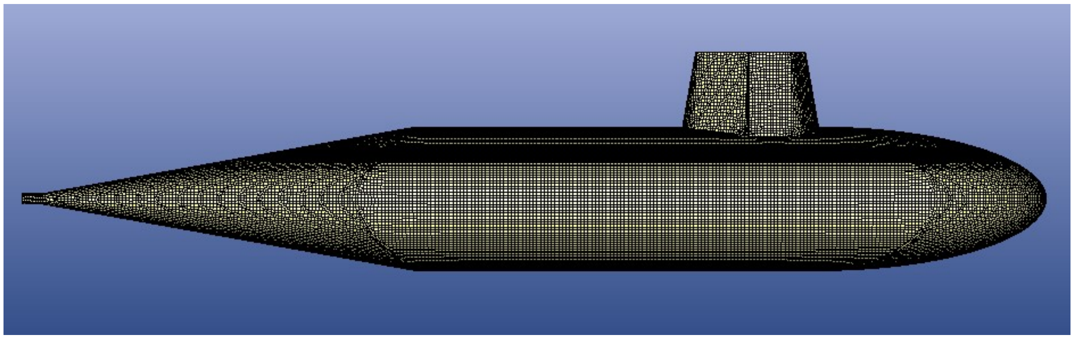


Figure 4b Generated mesh of model

As the model test was carried out with fixed condition (locked for pitching motion), the CFD simulation was also carried out with fixed model to get the fixed trim and sinkage. CFD simulation is carried out for the model from the speed range of 0.6 to 1.8 m/s. Figure 5 (a to c) shows the comparison of wave formation of model test and CFD simulation at various speeds.

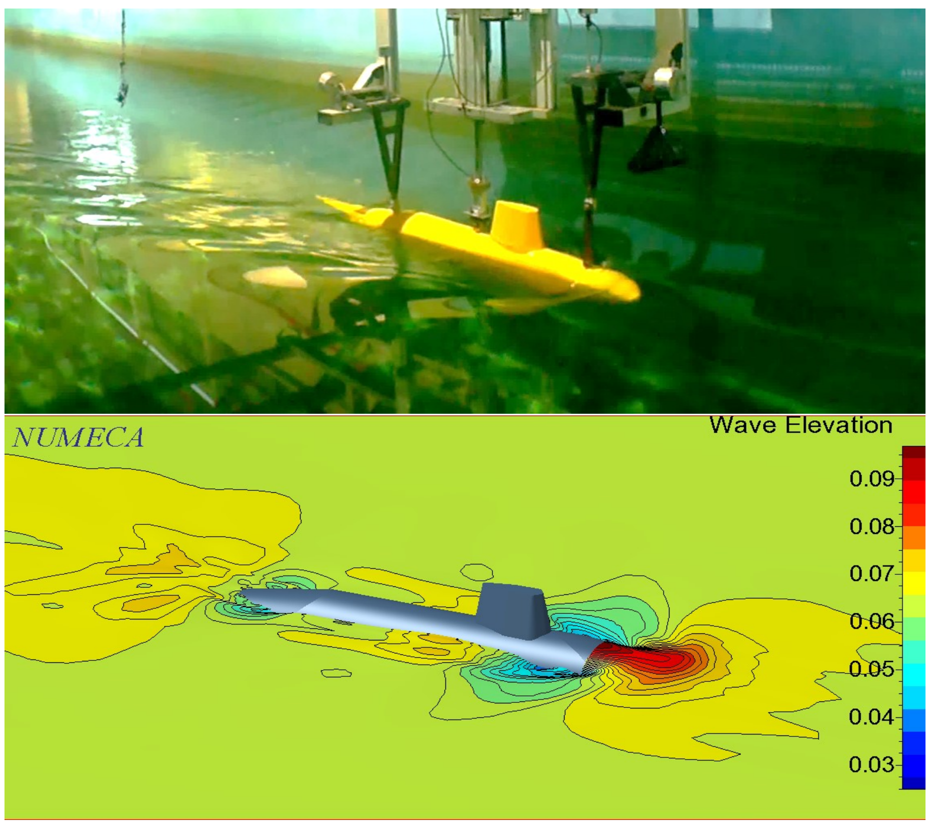


Figure 5(a) Towing tank test and CFD simulation with 0.8m/s

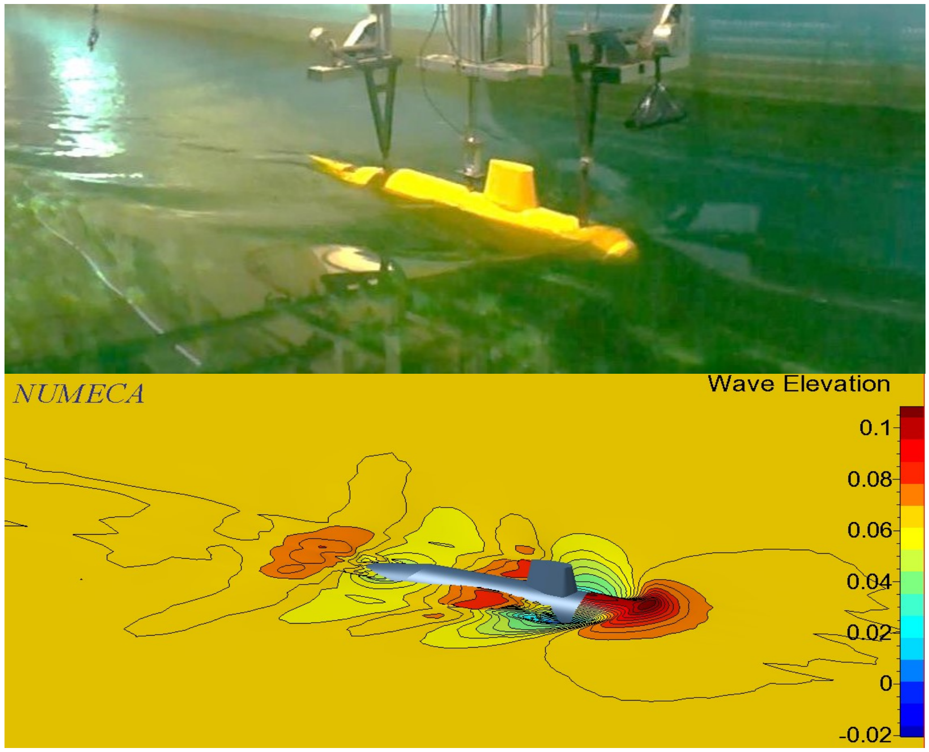


Figure 5(b) Towing tank test and CFD simulation with 1m/s

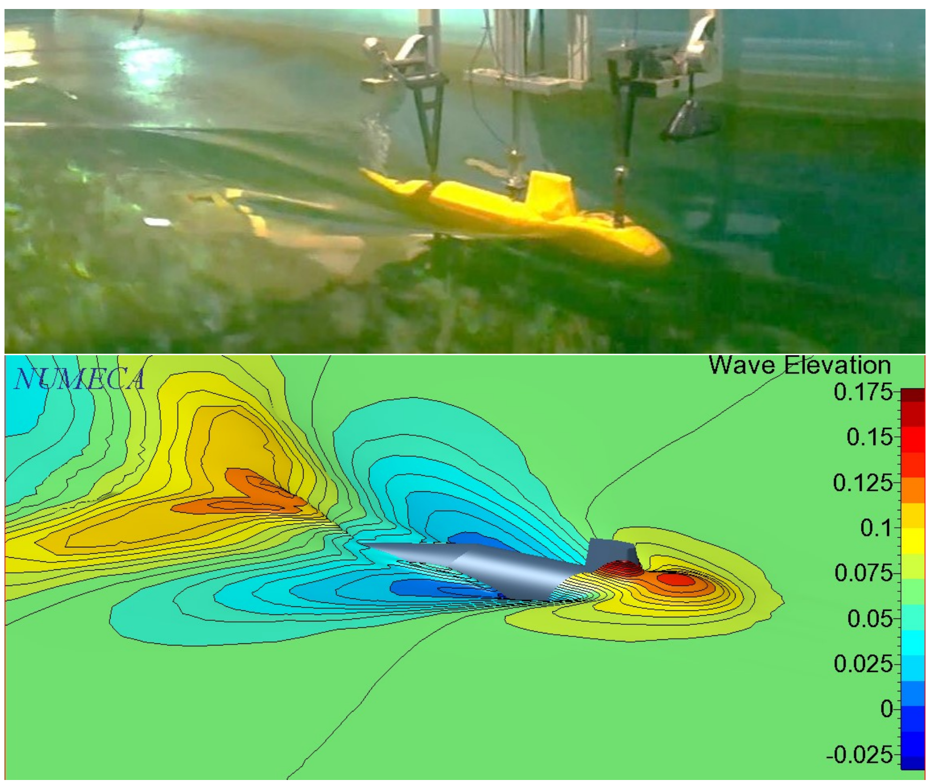


Figure 5(c) Towing tank test and CFD simulation with 1.8m/s

Figure 5 Comparison of wave formation Towing tank test and CFD at various speeds

Result Comparison

The resistance comparison of underwater vehicle is carried out for the speed of 0.2 to 1.8 m/s. At the Froude's number of about 0.33, bow wave creates a trough at the stern and increases the stern wave system and also increase the wave making resistance and resistance hump will be result. The results are compared in Figure 6. According to the results, the resistance behaviour for the two software packages CFD result are differing at Fn=0.33.

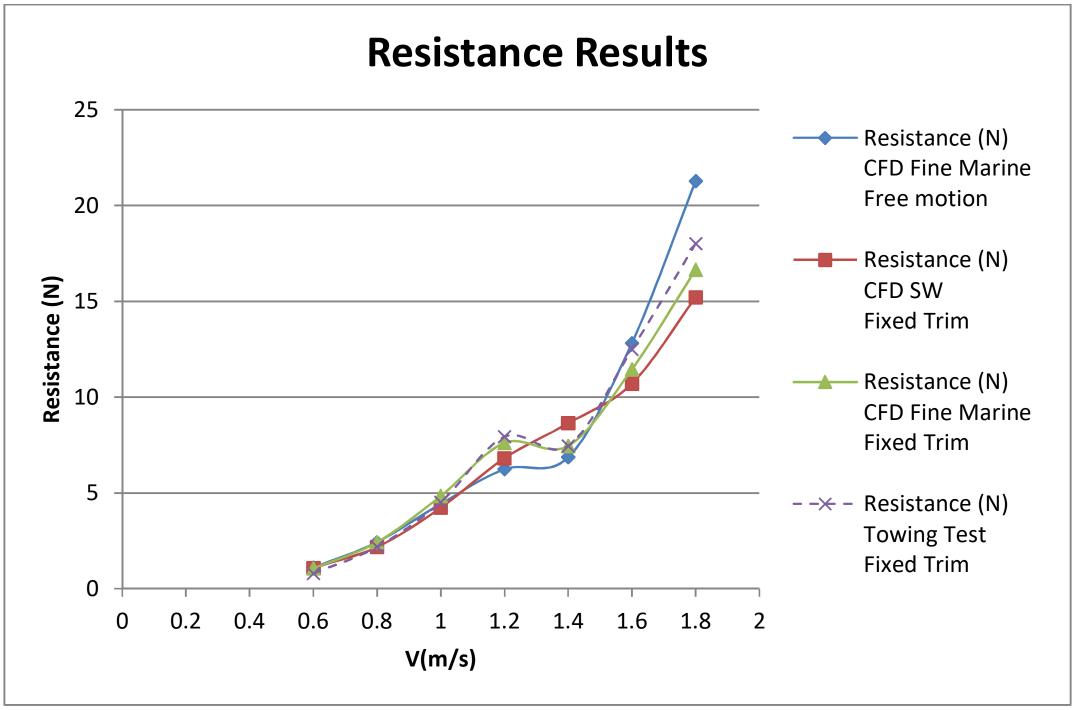


Figure 6 comparison of CFD and Towing test of ship resistance without appendages.

Conclusions

Towing test analysis was carried out for the model with fixed trim and pitch condition. Then CFD analysis was also carried out with fixed model condition. The model test results are compared with CFD results and the comparison shows that experimental results and CFD results have good agreement for near surface condition. At the higher speed, hydrodynamic forces acting on the vehicle will become larger and so, it's still needed to consider the hydrodynamic effect of the model in freely moving condition presented in Figure 7. After calculating the hydrodynamic effect of the model, hydroplane should be attached to ensure for fixed trim and pitch condition.

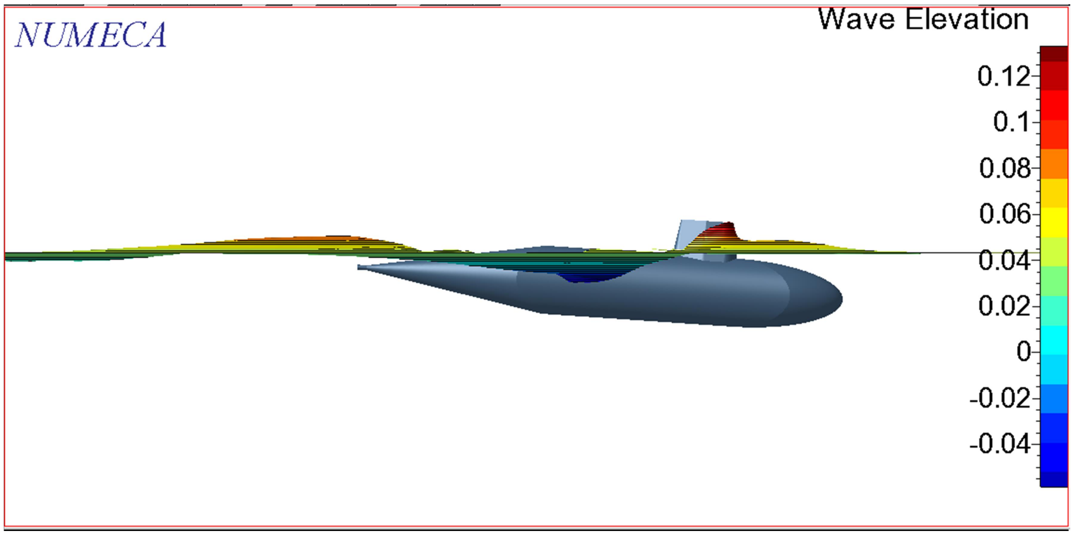


Figure 7 Hydrodynamic effect of the model (down force)

Acknowledgements

The authors would like to express their thanks to the staffs for all their support during the experimental model tests.

References

[1] ITTC 7.5-02-02-01, (2011). Recommended Procedures and Guidelines, Resistance Test, Revision 03.

[2] Roy Burcher and Louis Rydill, Concepts in Submarine design, Cambridge ocean technology series, 1995.

[3] C.B. Barrass, Ship Design and Performance for Masters and Mates, Elsevier Butterworth-Heinemann, 2004.

[4] David G.M. Watson., Practical Ship Design, Elsevier Science Ltd, 1998.

[5] Francesco Capizzano, Amedeo Migali, Salvatore Miranda and Claudio Pensa, Experimental and Numerical Study on the Efficiency of Trimaran Configuration for High-Speed Very Large Ship, 6^{th} Symposium on High Speed Marine Vehicle(HSMV), Baia, Naples, Italy, September 2002.

[6] A. Dey, "A study of a two dimensional flow around a hydrofoil by Rankine source panel method using a computer program based on C++," B.Sc. thesis, Dept. Naval Arch. and Marine Eng., BUET, Dhaka, Bangladesh, 2013.

[7] M. S. Nizam, "Compuation of wave-making resistance of multihulls using a surface panel method," M.Sc. thesis, Dept. Naval Arch. And Marine Eng., BUET, Dhaka, Bangladesh, 2011.

[8] M. M. Karim and N. Naz "Computation of Hydrodynamic Characteristics of Ships Using CFD" International Journal of Materials, Mechanics and Manufacturing, Vol. 5, No. 4, November 2017.

[9] Md. Mashud Karima,∗, Nabila Naza "Computation of Flow Field around Ship Hull including Self Propulsion Characteristics at Varying Rudder Positions" 10th International Conference on Marine Technology, MARTEC 2016.

[10] Choi JE, Min KS, Kim JH, et al. (2010). Resistance and propulsion characteristics of various commercial ships based on CFD results. Ocean Engineering; 37: 549-566.

[11] Tarafder S, Suzuki K. (2008). Numerical calculation of free-surface potential flow around a ship using the modified Rankine source panel method. Ocean Engineering; 35: 536-544.