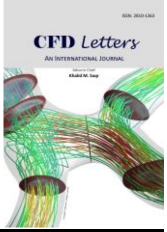
CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363



Matlab Implementation using Holtrop and Mennen Method of Bare Hull Resistance Prediction for Surface Combatant Ship Coupled with CFD

Alaaeldeen Mohamed Elhadad[1,*], Ahmed Mokhtar Abo El-Ela[1], Mohamed Mostafa Hussien[1]

[1] Shipbuilding Engineering Department, Military Technical College, Cairo, Egypt

ARTICLE INFO

Article history:
Received 4 April 2023
Received in revised form 6 May 2023
Accepted 3 June 2023
Available online 1 October 2023

Keywords:
Combatant ship; Total resistance; Holtrop Method; CFD

ABSTRACT

Holtop's & Mennen's method is one of the most widely used technique in resistance and powering prediction for tankers, general cargo ships, container ships and combatant surface ships. Holtrop's Statistical Analysis method is utilized, as in comparison to the other options, more complete and closer to the actual results. The main purpose of this paper is to develop a Matlab application using Holtop's & Mennen's method for studying and research the total resistance of DTMB 5415 combatant ship. The solid modelling is developed using Maxsurf. The computational fluid dynamic (CFD) technique is applied to analyze the hydrodynamic behaviour of hull form and predict the total resistance (Rt) to provide a description of the other prediction methods. The simulation process was executed using ANSYS software based on fluid flow (STAR-CCM+) solver. The variation of computational grid used in computation is SST (Shear Stress Transport) coarse and fine mesh size for hull speed up to 32 kn. RT resulted from CFD computation is validated with results from Holtrop that has been developed by Matlab implementation. The larger the grid meshing size, the better the validation of the results. The CFD technique demonstrated good agreement with Holtrop formulae in predicting the (RT) of DTMB 5415.

1. Introduction

The most important factor to consider when designing a ship's power requirements is the ship's resistance, or the sea drag forces acting on the ship. It is critical to estimate ship resistance when designing the propulsion system because the power required to overcome sea drag forces contributes to propulsion system losses. To calculate ship resistance, three methods are used: statistical methods such as the Holtrop-Mennen (HM) method, numerical analysis or Computational Fluid Dynamics (CFD) simulations, and model testing in towing tank [1].

In the preliminary design stage, total ship resistance is predicted using statistical data or empirical formulas as an important first step before calculating ship resistance using a scaled model tested in a towing tank. It is also possible to predict several methods of ship resistance based on the ship type and method limitations. The HM method is based on regression analysis of full-scale data that is already available. It is a very useful tool for estimating ship resistance early in the ship design process

* *Corresponding author.*
E-mail address: dr.aladdinahmed@gmail.com (Alaaeldeen Mohamed Elhadad)

https://doi.org/10.37934/cfdl.15.10.111

[2]. However, it is important to understand that statistical methods provide only a rough estimate and may differ significantly from actual ship resistance values.

These computations' algorithms were created. Using the developed algorithms, simple MATLAB scripts were written, and attempts were made to compare the programs with the supply vessel specifics and standard commercial software, the MAXSURF resistance module. Once the design is complete, ship model tests in towing tanks are carried out to determine the ship's resistance. Scaled model and testing in towing tanks are both costly and time-consuming procedures. Furthermore, the results of the model tests must be scaled to the actual size of the ship, a process fraught with uncertainty [3]. Due to the availability of powerful computational resources and advancements in the field of CFD, CFD analysis is becoming the preferred method over model tests for calculating ship resistance.

The primary goal of this paper is to investigate and study ship resistance for a 142-meter combatant vessel. A CFD simulation procedure is used to calculate the calm water ship resistance using a case study of a DTMB 5415. Star CCM+ is the software used. Ship resistance was calculated at various ship speeds ranging from 5 knots on average to 32 knots at maximum. Because the mesh has an impact on the results of any CFD simulation, multiple meshes were used to test the mesh's sensitivity [4]. All simulation results were compared to Matlab calculations using the HM method. Star CCM+'s ability to perform ship resistance analysis was demonstrated.

2. Holtrop-Mennen Method

2.1 Hull Geometry and Ship Particulars

The HM method is arguably the most popular method for estimating displacement-type ship resistance and powering. This method employs the regression approach and allows for the analysis of various resistance components, such as the viscous frictional effect of sea water, the appendages of the vessel that house the anchors and the bilge, the possible resistance effects of the wave, the components that arise from the bulbous bow, and the theoretical ship. It is the only method that employs the ITTC form factor k. The hull-propeller interaction parameters thrust deduction, full-scale wake fraction, and relative rotative efficiency is also estimated using HM formulas [5].

The MATLAB implementation of the HM method is based on equations modelled using regression principles and is as follows:

$$R_{total} = R_f\,(1+k_1) + R_{App} + R_w + R_B + R_{TR} + R_A \quad (1)$$

Where R_f is the frictional resistance calculated using the ITTC-1957 friction formula, and $1+k_1$ is the form factor describing the viscous resistance of the hull form in relation to Rf. R_{App} stands for appendage resistance. R_W is the resistance of making and breaking waves. R_B denotes the additional pressure resistance of the bulbous bow near the water's surface. R_{TR} is the additional pressure resistance of immersed transom stern. R_A denotes the model-ship correlation resistance.

The effective power required to overcome a given resistance, R_{tot}, at a given speed, V, is given as:

$$P_E = R_{tot}\,V \quad (2)$$

Where P_E is in KW, R_{tot} in KN and V in ms^{-1}. The shaft power required to overcome a given resistance, R_{tot} with a given speed V is given as:

$$P_S = P_E/\eta \quad (3)$$

Where P_S is in KW, η is the efficiency.

The MATLAB implementation requires the ship's length, beam, draught, and speed to be entered. Water density and kinematic viscosity are also required. This method is an efficient and accurate way to estimate a ship's resistance in calm water [6].

A database was created that contained the model test results of the DTMB 5415 combatant ship. The calculations are performed sequentially for the various resistance components and propulsion factors. Figure 1 and Table 1 show and list the hull geometry and main ship particulars for speeds up to 32 knots [7].

Fig. 1. DTMB #5415 Model geometry

Table 1
Main particulars of combatant Ship DTMB 5415

Particulars	Ship	Particulars	Ship	Particulars	Ship
L_{OA} (m)	153.300	T (m)	6.150	GM (m)	1.938
L_{PP} (m)	142.200	V (m^3)	8424.4	LCG (m)	70.137
B_{OA} (m)	20.540	Δ (t)	8636	C_B	0.505
B_{WL} (m)	19.082	KM (m)	9.493	C_P	0.616
D (m)	12.470	KG (m)	7.555	C_M	0.815

2.2 Source Code Implementation

The below mathematical models are best implemented using functional implementations [8]. Figure 2 depicts only the frictional component of total resistance, as represented by the r f.m file.

```
%%Calculations%%
% SApp=sum(double(Appen(:,2)));
% Keq=sum(double(Appen(:,2)).*double(Appen(:,3)))/SApp-1;
Rn=Vs.*(Dens*LWL/Visc);
Fn=Vs./sqrt(9.81*LWL);
Tm=(Ta+Tf)/2;
CB=Disp/(LWL*B*Tm);
Cp=CB/CM;
C13=1+0.003*Cstern;
if Tm/LWL >0.05
    C12=(Tm/LWL)^0.2228446;
elseif 0.02<Tm/LWL && Tm/LWL<0.05
    C12=48.20*(Tm/LWL - 0.02)^2.078 + 0.479948;
else
    C12 =0.479948;
end
lcb=LCB;
LR=LWL*(1-Cp+0.06*Cp*lcb/(4*Cp-1));
K1=C13*(0.93+C12*((B/LR)^0.92497)*((0.95-Cp)^(-0.521448))*((1-Cp+0.0225*lcb)^0.6906))-1;
% WSA=LWL*(2*Tm+B)*sqrt(CM)*(0.453+0.4425*CB-0.2862*CM-0.003467*B/Tm+0.3696*CWP)+2.38*ABT/CB;
WSA=2976.7;
Cf=0.075./((log10(Rn)-2).^2);
if B/LWL <0.11
    C7=0.229577*((B/LWL)^(1/3));
elseif B/LWL>0.11 && B/LWL< 0.25
```

Fig. 2. Source code implementation

2.3 Matlab Results

The DTMB 5415 total resistance was calculated in the first preliminary step using a MATLAB implementation of the HM method with speeds ranging from 1 to 32 Kn, or Fn 0.01 to 0.43. The calculated drag on the hull was measured, plotted, and compared with the Maxsurf results to assess the convergence of the solution depicted in Figure 3.

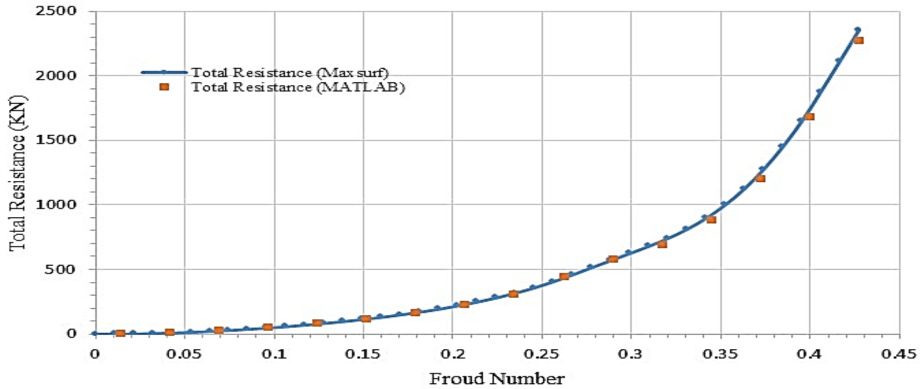


Fig. 3. Resistance curve by MATLAB for DTMB 5415 comparing with Maxsurf

Figure 4 depicts graphs of total and residual resistance coefficients (C_{TM} and C_R vs. Fr, respectively). The ITTC 57 formula has also been used to calculate viscous resistance. C_{TM} gradually decreases between 0.01 and 0.15, remains constant (C_{TM}=0.0028) between 0.15 and 0.225, gradually increases between 0.225 and 0.35, and rapidly increases when Fr exceeds 0.35. C_R is essentially piecewise linear, with a rising slope for Fr values ranging from 0.1 to 0.35. In both cases, the presence of humps and hollows is minimal. Figure 5 depicts the effective power curve versus Fr for the Maxsurf and Matlab implementations. When we tested the Maxsurf resistance module against this formulation, we obtained the very good agreement shown below. Finally, Figure 6 depicts the free surface wave and volume fraction (water) contours of the hull model at various Froude numbers

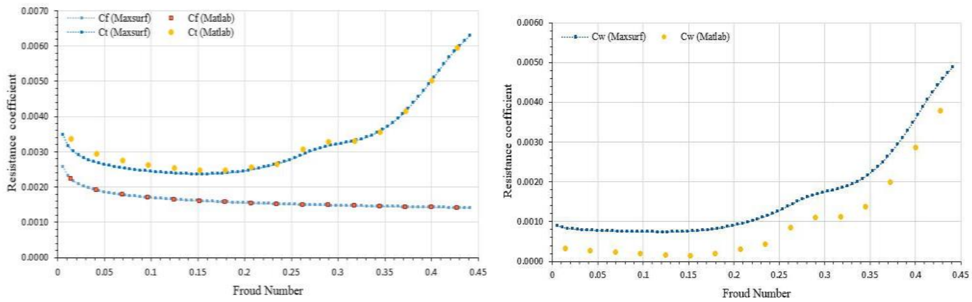


Fig. 4. The curves of total and residual resistance coefficient C_{TM} and C_R

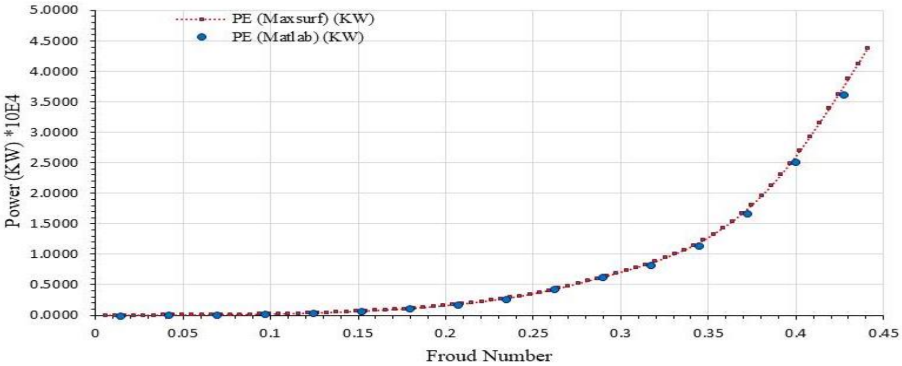


Fig. 5. The effective power against Fr

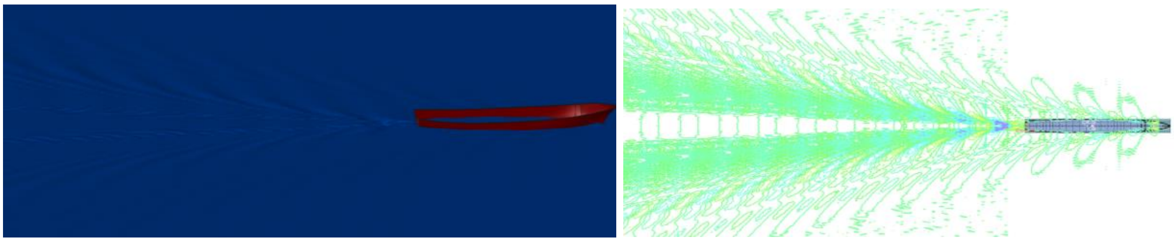


Fig. 6. Free surface wave and volume fraction contours at various Froude numbers Fn = 0.40

3. Resistance Computation on STAR-CCM+

3.1 Hull Geometry

The surface combatant DTMB model 5415 is a well-known test model, with numerous experimental and CFD test results made public and addressed at numerous workshops and conferences. Figure 7 and Table 2 depict the main dimensions and data used to conduct simulation tests on this ship. For computations and experiments, the bare hull condition is used [9, 10].

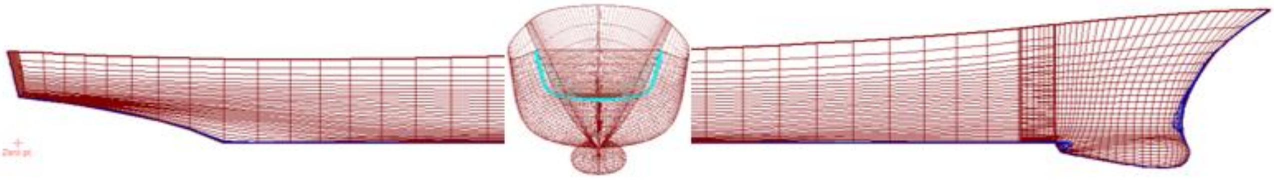


Fig. 7. Hull form of studied case

Table 2
Main particulars of DTMB #5415 (ITTC, 2005)

L_{OA} (m)	6.167	D (m)	0.502	KM (m)	0.186
L_{PP} (m)	5.72	T (m)	0.248	KG (m)	0.148
B_{OA} (m)	0.826	V (m^3)	0.549	GM (m)	0.038
B_{WL} (m)	0.767	Δ (t,kg)	0.549	LCG (m)	1.375

It is commonly regarded as the best test sample for validating a new numerical method on this combatant hull [11]. The grid structure and combatant ship profile corresponding to a scale of 24.48 were used for CFD calculations.

3.2 Governing Equations

The Reynolds-averaged equations for mass and momentum conservation are as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{4}$$

$$\frac{\partial u_i}{\partial t} + u_i\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\left(\frac{\partial P}{\partial x_i}\right) + \nu\frac{\partial^2 u_i}{\partial x_j^2} - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{5}$$

where *i*, *j* = 1, 2 and x_1, x_2 denote the horizontal and the vertical dimensions, respectively, u_1 and u_2 are the mean velocity components, $\overline{u_i' u_j'}$ defines the Reynolds stress component with u_i' is the fluctuating part of the velocity, *p* represents the dynamic pressure, and ρ is the fluid density [12].

By using the Boussinesq approximation, the Reynolds stress component was defined in terms of a turbulent viscosity, ν_T, and the mean flow gradients as follows:

$$-\overline{u_i' u_j'} = \nu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \quad (6)$$

where δ_{ij} denotes Kronecker delta function, and k denotes turbulent kinetic energy., three-dimensional numerical URANS simulations for the flow past a ship model were performed In this study. A two-layer form of the k-ε model, developed by Star-CCM + software, was used to predict the numerical simulations [13].

3.3 Computational Domain, Boundaries and Mesh Generation

A rectangular box defined the numerical computation domains. The dimensions of the computational domain were determined in accordance with the ITTC regulations and recommendations. The size dimensions are listed in Table 3, and Figure 8 depicts the fluid flow computational domain, with the size of the computational field proportional to the size of the model. The arrows on the domain's left (inlet) side indicate the flow direction [14].The finite volume method was used in this study to discretize the domain into a set of adjoining cells. Furthermore, a fixed mesh method was used, in which it is assumed that the fluid moves at the same speed as the ship model, where the ship model is static [15].

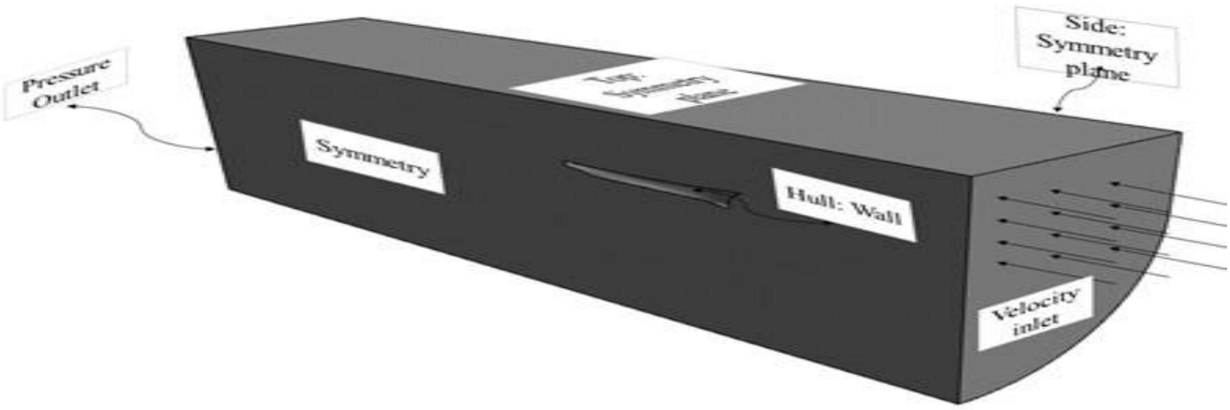


Fig. 8. Computational domain and boundary conditions of the DTMB 5415

Table 3
Computational domain dimensions

Upstream	1.0 L_{OA}	front of the bow
Down stream	3.0 L_{OA}	from the stern
Top	0.5 L_{OA}	depth of air zone
Bottom	1.0 L_{OA}	depth of water zone
Transverse	1.0 L_{OA}	width of both zones

Figure 9 depicts perspective views of the domain as well as slice views of the refinement areas. Grids are finer in regions surrounding the ship model near the free surface and over a longitudinal slice of the ship bow and stern, allowing the generation of inflation layers around the free surface to adequately capture the hull's high-velocity gradients and waves [16].

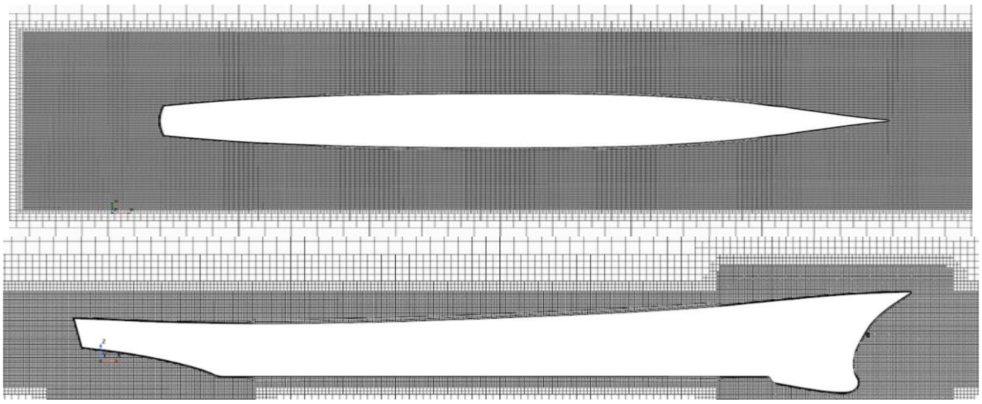


Fig. 9. Top and side view of the computational mesh 2 on the medium grid

In order to use second-order temporal discretization, we implemented the SIMPLE algorithm in code Star-CCM+. While the k-ε turbulence approach was used for the calculations, the second-order upwind scheme was used. The outlet was placed at the right boundary, where the gradients of outflow velocity in the streamwise direction were set to zero and pressures were assigned a reference value of zero [17]. A slip condition was assumed on the bottom boundaries, while a symmetry condition was assumed on the side boundaries. On the model surface, a non-slip boundary condition was assumed.

A volume of fluid (VOF) model is used to simulate free-surface flow. A conservation equation for the volume fraction of water is used to derive the water surface position implicitly [18]. This equation is discretized using specific compressive discretization schemes to keep the transition between water and air close to a discontinuity. Fine meshes are required at the surface to accurately resolve the volume fraction [19].

The numerical simulations were run on three different grids, each with a finer mesh and a refinement factor of c = 1.25 uniformly specified for all spatial directions. To estimate the discretization errors, one method was used. This method produced good, consistent predictions for a variety of transient problems, including those investigated here [20]. The method demanded that the same refinement factors be applied to all dimensions and that spatial and temporal refinements are performed concurrently [21]. It considered uniform refinement in all directions (space and time) with a constant CFL number. The one-dimensional CFL number, which is a good measure for convection-dominated flow, was written as follows:

$$r = \frac{\Delta x_{i+1}}{\Delta x_i} = \frac{\Delta t_{i+1}}{\Delta t_i} \Rightarrow \mathrm{CFL} = u\frac{\Delta t}{\Delta x} = \mathrm{const} \tag{7}$$

Where Δx and Δt are the spatial and time-step sizes, respectively, the index i denotes the level of refinement, r is the refinement factor, and u is the fluid velocity. To ensure numerical stability in all cases, the y+ value was set to one. Mesh 3 represented the coarse grid, mesh 2 the medium grid, and mesh 1 the fine grid, all of which were well-matched with the factor.

4. Results and Discussion

4.1 Numerical Calculations

The calm water resistance of the DTMB 5415 model was tested using CFD against model Fn, ranging from 0.01 to 0.45. The experimental results of the ship's towing tank test are required to validate the results obtained by Star-CCM+ using specific mesh densities and the conditions applied to boundaries and solver. Resistance tests for the bare hull at different speeds up to 3.75 m/sec were run for 9 days on a computer with 16 cores and a mesh density of 2.1 million cells. To estimate the hydrodynamic performances for the CFD mesh investigation, three different mesh sizes were used. To evaluate the convergence of the solution as depicted in Figure 10, the calculated drag on the hull was measured, plotted, and compared with the experimental results. The findings for the coarse, medium and fine grid results are extremely similar and the difference ε = (medium - coarse) / (fine - medium), where 0< ε <1 is small and acceptable as shown in Table 4.

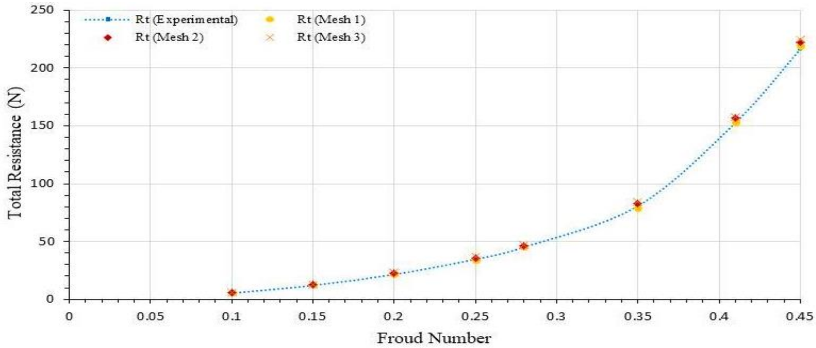


Fig. 10. Resistance curve for DTMB #5415-51 comparing three different sized grids with Maxsurf

Table 4
The CFD results for DTMB 5415 model

Fn	0.1	0.15	0.2	0.25	0.28	0.35	0.41	0.45
R_T (N) (C)	5.98	12.88	23.33	36.98	47.32	84.75	157.9	224.48
R_T (N)(M)	5.85	12.81	22.78	35.82	46.32	82.77	156.64	222.32
R_T (N) (F)	5.68	12.72	22.1	34.3	45.3	78.8	152.9	218.91
Difference ε	0.764706	0.777778	0.808824	0.763158	0.980392	0.498741	0.336898	0.633431

According to the bare hull results, the mesh generation approach can be used to estimate hydrodynamic performances using CFD, and numerical convergence shows that the overall strategy is suitable for resistance prediction. The results for the coarse and medium grids are very similar, and the difference between the medium and fine grids is larger but still manageable. As a result, the fine grid provides the best match for the calculations and displays the most appropriate results. Table 5 displays the results of the verification and validation exercises for these three grids' total resistance force at a flow stream velocity of 3.37 m/s. After this checking of STAR-CCM+ results, it is possible now move to calculate drag resistance with bulbous using this setting of mesh and solver parameters' which can be considered as reliable.

Table 5
Grid convergence for a DTMP-5415 model obtained from different grids

Element	Δx [m]	No. of cell x106	Δt [sec]	RT (Exp.)	RT (Num.)	Δ RT (%)
Coarse	0.3	0.542178	0.01	216.13 N	224.48	3.86
Medium	0.21	1.439452	0.0071	216.13 N	222.32	2.86
Fine	0.15	3.711726	0.005	216.13 N	218.91	1.29

Figure 11 depicts the total resistance time histories at Fr 0.38 for the 40 seconds, at which the simulation was run. It is possible to see how the free surface causes the graphs to oscillate. Finally, Figure 12 depicts the wave pattern and water volume fraction hull model for the coarse grid at Froude number 0.40.

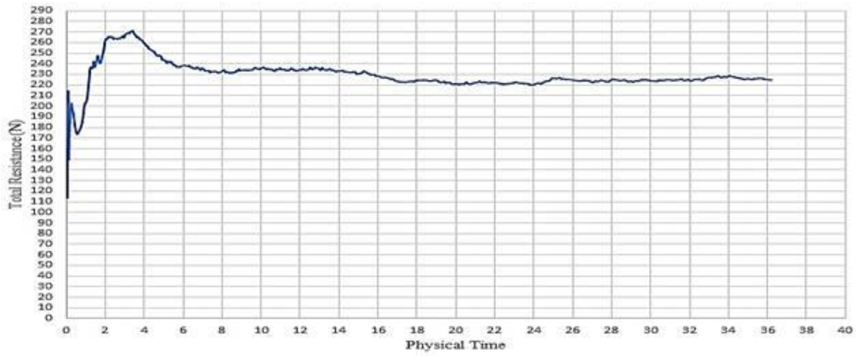


Fig. 11. Time history of resistance as generated in CFD at Fr=0.38

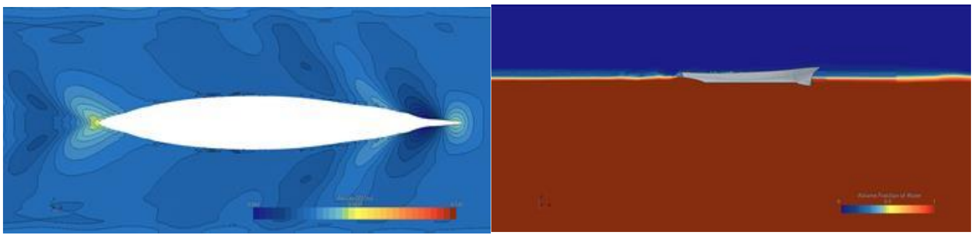


Fig. 12. Wave pattern and Water volume fraction on the ship hull and its center symmetry plane at Fn = 0.40 - coarse grid

4.2 Comparison of Resistance Results

The scale model ship was used to investigate the full similarity technique (FST) after the validation study. The resistance components at the hull model were calculated, and the results were presented using both Froude and Reynolds conditions for the naval combatant. Because increasing element numbers result in decreasing wall y+ values, numerical C_F values approach the ITTC-1957 formula. In other words, because the boundary layer grid has a low resolution, increasing the wall y+ value degrades the calculation of C_F in full-scale resistance prediction. As the number of elements increases, the C_R values of full-scale ships at various grid sizes reach the value of model-scale ships. The functionality of FST on the prediction of a full-scale ship's C_T at model scale dimensions was investigated. The previous results were taken and represented on a graph for validation purposes.

Figure 13 depicts the numerical results of STAR-CCM+ and the predicted method of HM for full-scale ship total resistance. Instead of experimental results for this ship that have yet to be performed, these results are for a bare hull without appendages (twin screws, bow thrusters, and two rudders).

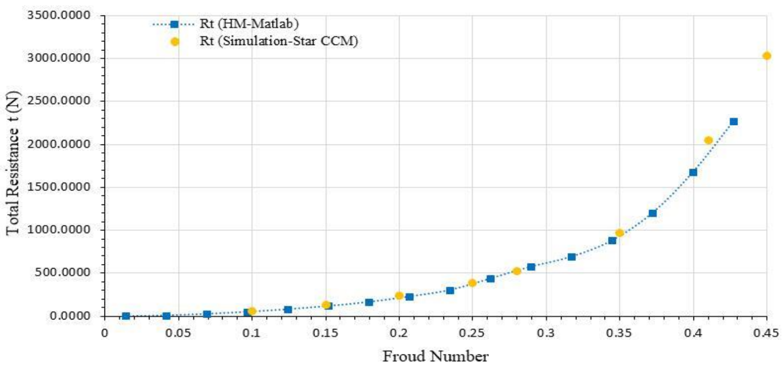


Fig. 13. Comparisons between Results of Star-CCM+ and HM-Matlab for Bare Hull

At lower speeds, the agreement between the CFD simulation calculations and the Matlab data deteriorates; at Fr = 0.35, the CFD values deviate significantly from the Matlab data. In general, the results show that the resistance values at various speeds for the fine mesh agree with the HM data very well, with an error of less than 5%. Between the two methods, total resistance has the best match. By being in the range of the two, we can see a good deal of consistency in the results for higher speeds for STAR-CCM+ with the HM method, but there is less agreement for 20 knots.

5. Conclusion

To overcome the disadvantages of CFD tools and the difficulties of towing tank tests, this paper attempts to provide a method that provides a fast and reliable estimate of resistance during the initial design stage. Among the hydrodynamic characteristics, resistance evaluation during the initial design stage is critical. The current research contributes total resistance for combatant geometry (DTMB 5415).

This study looked at the mathematical model formulation for computing the vessel's bare hull resistance using the HM method. The algorithm and implementation of these formulations were then developed using the functional programming approach in MATLAB, a scientific computing language.

A significant portion of the current data set was cross-validated using DTMB experimental data, yielding an excellent outcome. Extensive comparisons of our Star-CCM+ calculations and experimental results for DTMB 5415 demonstrate that the method is acceptable and that the mesh refinement method is capable of helping estimate the model's hydrodynamic performance. Many different meshes, time steps, volume mesh domains, and boundary conditions were tested. After comparing the total resistance grid obtained results through the CFD, we reach the conclusion that the fine grid is best suited to the calculation and produces the best results because convergent results are obtained as mesh size decreases, so the fine grid will be used.

The developed algorithms were then compared with a standard Star-CCM+ resistance module and found to have very good agreement and sufficient accuracy. The HM statistical analysis method is more comprehensive and accurate.

Acknowledgement
This research was not funded by any grant.

References
[1] Zhang, Zhi-rong, Liu Hui, Song-ping Zhu, and Z. H. A. O. Feng. "Application of CFD in ship engineering design practice and ship hydrodynamics." *Journal of Hydrodynamics, Ser. B* 18, no. 3 (2006): 315-322. https://doi.org/10.1016/S1001-6058(06)60072-3.

[2] Deshpande, Sujay, P. Sundsbø, and Subhashis Das. "Ship resistance analysis using CFD simulations in Flow-3D." *The International Journal of Multiphysics* 14, no. 3 (2020): 227-236. https://doi.org/10.21152/1750-9548.14.3.227.

[3] Elhadad, Aladdin, Wen Yang Duan, Rui Deng, and H. Elhanfey. "Numerical Analysis for Resistance Calculations of NPL as a Floating Hull for Wave Glider." *Applied Mechanics and Materials* 619 (2014): 38-43. https://doi.org/10.4028/www.scientific.net/AMM.619.38.

[4] Rosemurgy, William J., Deborah O. Edmund, Kevin J. Maki, and Robert F. Beck. "A Method for resistance prediction in the design environment." In *11th International Conference on Fast Sea Transportation FAST*. 2011.

[5] Nikolopoulos, Lampros, and Evangelos Boulougouris. "A study on the statistical calibration of the holtrop and mennen approximate power prediction method for full hull form, low froude number vessels." *Journal of Ship Production and Design* 35, no. 01 (2019): 41-68.https://doi.org/10.5957/JSPD.170034.

[6] Augustine, Elakpa Ada, Olusegun Samuel Dare, and Ogbe Emmanuel Ediba. "Bare hull ship resistance computation of anoffshore supplyvessel using the Holtrop and menenn method: A MATLAB implementation." https://doi.org/10.35629/5252-0310678683.

[7] Ahmed, Y., and C. Guedes Soares. "Simulation of the flow around the surface combatant DTMB model 5415 at

different speeds." In *13th Congress of Intl. Maritime Assoc. of Mediterranean, Istanbul, Turkey*, pp. 307-314. 2009.

8] S Leksono; Muryadin; DKartikasari. Comparative Study of Ship Resistance between Model Test and Empirical Calculation of 60 GT Fishing Vessel. *Int J Sci Res* 2018;7:1077–82.

[9] Zhao, Feng, Song-Ping Zhu, and Zhi-Rong Zhang. "Numerical experiments of a benchmark hull based on a turbulent free-surface flow model." *Computer Modeling in Engineering and Sciences* 9, no. 3 (2005): 273-285.

[10] Elhadad, Aladdin, Wen Yang Duan, and Rui Deng. "Comparative investigation of an automated oceanic wave surface glider robot influence on resistance prediction using CFD method." *Applied Mechanics and Materials* 710 (2015): 91-97. https://doi.org/10.4028/WWW.SCIENTIFIC.NET/AMM.710.91.

[11] Diez, Matteo, Andrea Serani, Emilio F. Campana, Omer Goren, Kadir Sarioz, D. Bulent Danisman, Gregory Grigoropoulos et al. "Multi-objective hydrodynamic optimization of the DTMB 5415 for resistance and seakeeping." In *SNAME International Conference on Fast Sea Transportation*, p. D021S005R012. SNAME, 2015. https://doi.org/10.5957/FAST-2015-034

[12] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA journal* 32, no. 8 (1994): 1598-1605. https://doi.org/10.2514/3.12149.

[13] Akbarzadeh, Pooria, Pouya Molana, and Mohammad Ali Badri. "Determining resistance coefficient for series 60 vessels using numerical and experimental modelling." *Ships and Offshore Structures* 11, no. 8 (2016): 874-879. https://doi.org/10.1080/17445302.2015.1081779.

[14] Ahmed, Alaaeldeen Mohamed Elhadad. "Resistance Evaluation for the Submerged Glider System using CFD Modelling." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 29, no. 3 (2023): 147-159. https://doi.org/10.37934/araset.29.3.147159

[15] Ahmed, Alaaeldeen ME, and Wenyang Duan. "Resistance and Seakeeping Investigation for Optimization of the Floating Hull of Wave Glider." *World Journal of Engineering and Technology* 4, no. 3 (2016): 235-249. https://doi.org/10.4236/wjet.2016.43d029.

[16] Youssef, M., Ould El Moctar, and Thomas E. Schellin. "Two Turbulence Models For URANS Simulations Of A Circular Cylinder Subject To Vortex-Induced Vibration." In *ISOPE International Ocean and Polar Engineering Conference*, pp. ISOPE-I. ISOPE, 2021.

17] Pacuraru, F. "Validation of potential flow method for ship resistance prediction." In *IOP Conference Series: Materials Science and Engineering*, vol. 591, no. 1, p. 012113. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/591/1/012113.

[18] Toda, Y., F. Stern, I. Tanaka, and V. C. Patel. "Mean-flow measurements in the boundary layer and wake of a series 60 CB= 0.6 model ship with and without propeller." *Journal of Ship Research* 34, no. 04 (1990): 225-252. https://doi.org/10.5957/jsr.1990.34.4.225.

[19] Elhadad, Aladdin, Wen Yang Duan, and Rui Deng. "A computational fluid dynamics method for resistance prediction of the floating hull of wave glider." *Advanced Materials Research* 936 (2014): 2114-2119. https://doi.org/10.4028/www.scientific.net/AMR.936.2114.

[20] ITTC, ITTC. "Recommended procedures and guidelines." *Resistance Test* (2011).

[21] Bahatmaka, Aldias, Muhammad Yusuf Wibowo, Andi Abdullah Ghyfery, Muhammad Harits, Samsudin Anis, Deni Fajar Fitriyana, Rizqi Fitri Naryanto et al. "Numerical Approach of Fishing Vessel Hull Form to Measure Resistance Profile and Wave Pattern of Mono-Hull Design." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 104, no. 1 (2023): 1-11. https://doi.org/10.37934/arfmts.104.1.111