Paper ID: FM-75

Numerical Predictions of Calm Water Resistance of a Modern Surface Combatant

Doyal Kumar Sarker[1], Md. Safwan Ahsanullah[2], Samiul Azam[3] and Md. Mashiur Rahaman[4]

Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000
E-mail: [1]doyalkumar94@gmail.com, [1]safwanahsanullah@gmail.com, [3]samiul775@gmail.com, [4]mashiurrahaman@name.buet.ac.bd

Abstract

An important part of designing any marine vessel is to predict resistance, sinkage and trim during the design stage to obtain an optimal and economical design that will satisfy the necessary IMO regulations. Traditional tools for these predictions include carrying out towing tank tests on a downscaled version of the ship and then extrapolating the obtained results for the full scale. However, these methods are time consuming, expensive and often offer inaccurate results due to scaling problems, since Reynolds similarity is not fulfilled in model tests. With the advent of numerical techniques in recent times, it has become a widely used tool in ship hydrodynamics because of its flexibility and ability to be applied on model and full scale with great details of flow field. In present study, finite volume based commercial code STARCCM+ is used to predict the calm water resistance of a surface combatant. Present numerical results were compared with the available experimental results, which show a very good consistency between the two results.

Keywords: resistance, calm water, numerical, surface combatant.

1. Introduction

Reducing a ship's total resistance is one of the fundamental objectives for naval architects, to obtain an optimum power requirement and fuel consumption for a desired speed and thus to benefit the ship owner. Calculation of resistances has traditionally relied upon model scale experiments carried out in towing tanks. In addition, there are empirical formulations based on regression analysis of available experimental data, to predict the overall resistance, based on the shape of the hull. Two such empirical methods that are widely used are Holtrop and Mennen's method[1], used for displacement hulls, and Savitsky's method[2], used for planing hulls. However, these formulations are only useful for hull shapes that are similar to the ones that are used to develop these empirical models. Also, model tests are long dated and expensive.

The recent development of high performance computational facilities, along with intricate and precise numerical algorithms for solving problems using these computers, has introduced an important dimension in the studies of fluid dynamics. Computational Fluid Dynamics (CFD) has revolutionized the way fluid dynamics is studied and practiced today, offering results that are analogous to the traditional towing tank test results, but at a smaller cost, in both money and time. It also allows modifications of hull shape and extrapolation of model scale test results to a full scale analysis with ease and limited cost of time and money.

In the present study CFD computations are used to predict the resistance, trim and sinkage of a modern surface combatant hull with transom stern, ONR Tumblehome. The resistance results are compared to available EFD data, and a good agreement is found. Same computations are then repeated with different mesh resolutions, applying a constant grid size refinement to predict the resistance of the same hull, to determine numerical uncertainties through verification studies. The surfaces of the hull was pre-processed and prepared, in CAD Software Rhinoceros 3D. Subsequently they were imported into CFD software StarCCM+ where the mesh was generated, and upon prescribing proper boundary conditions, simulations were generated.

2. StarCCM+ overview

Star-CCM+ is a commercial CFD analysis and simulation software, produced by CD-Adapco™, which was first released in 2004, and currently a version 12 is available. This software code is based on object-oriented

programming technology, which is uniquely designed to deal with large models quickly and efficiently using a client–server architecture that generates meshes and carries out CFD analyses and simulations, with post-processing. The object-oriented nature of this code can be seen in a user interface. An object tree is provided for modeling and running the simulation, containing object representations of all the data associated with the simulation.
There are a range of various options available in the Star-CCM+ software package for solving the Navier-Stokes equations. These include Reynolds Averaging, Large Eddy Simulation (LES), Detached Eddy Simulation (DES), and inviscid potential flow. Within the RANS solution approach to the solution of Navier-Stokes equations, a wide variety of turbulence modeling options is also offered, including: Spalart-Allmaras, k-ε, k-ω, Reynolds Stress Transport etc. Star-CCM+ uses the Volume of Fluid model exclusively to model free surface flows. In this model, the various fluid phases are assumed to be immiscible and all phases share velocity and pressure fields. This VOF model is a segregated flow model, with the pressure and velocity fields coupled using an implementation of the SIMPLE algorithm originally proposed by Caretto et al. (1973)[3]. The Star-CCM+ software package includes an extensive collection of post-processing tools to analyze and visualize simulation results with.

3. Geometry and Computational domain

The present numerical simulations are carried out for the ONR Tumblehome model 5613, which is a preliminary design of a modern surface combatant fully appended with skeg, bilge keels and rudders. The geometry model of ONR Tumblehome without propellers and shaft brackets is shown in Figure 1, and its principle parameters are listed in Table 1. The ship model is used as one of the benchmark cases in Tokyo 2015 CFD workshop in ship hydrodynamics.

Fig 1. Geometry model of ONR Tumblehome (from Tokyo 2015 CFD Workshop)

Table 1. Principle dimensions of fully appended ship

Main particulars		Model scale	Full scale
Length of waterline	L_{WL} (m)	3.147	154.0
Maximum beam of waterline	B_{WL} (m)	0.386	18.78
Depth	D (m)	0.266	14.50
Draft	T (m)	0.112	5.494
Displacement	Δ (kg)	72.6	8.507e6
Wetted surface area	S_0 (m²)	1.5	NA
Block coefficient (CB)	$\frac{\nabla}{(LWL\,.BWL\,.T)}$	0.535	0.535
LCB	aft of FP (m)	1.625	NA
VCG (from keel)	KG (m)	0.156	NA
Metacentric height	GM (m)	0.0422	NA
Moment of inertia	K_{XX}/B_{WL}	0.444	0.444

Discretization of the governing flow equations necessitates the subdivision of the complete computational domain into a number of sufficiently small cells, as shown in Figure 2. This grid system required for carrying out the CFD calculations is generated, in this study, in StarCCM+. Figure 3 shows the unstructured mesh obtained for the domain, which also has a refinement near the hull body surfaces, and further refinements at stern and bow to account for the sudden and sharp changes in geometry, and at the free surface capture the interface between two fluids and the wake produced by the motion of fluid past the vessel, with desired precision. Unstructured mesh offers a certain degree of flexibility in geometry and grid generation. For the hull body and whole domain, approximately 2.4M cells are used.

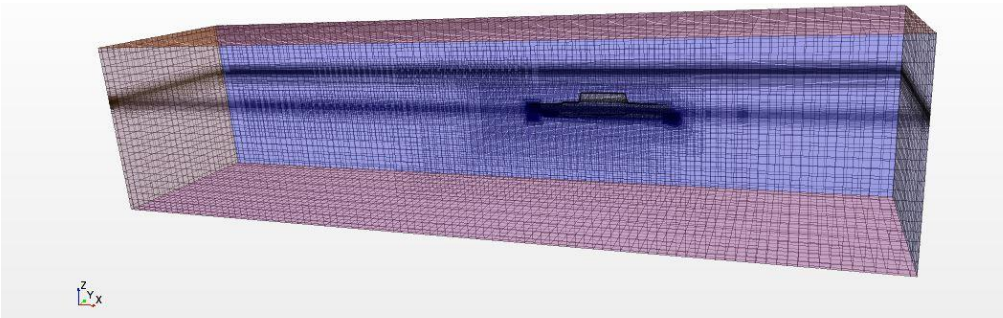

Fig 2. Grid distribution within the computational domain

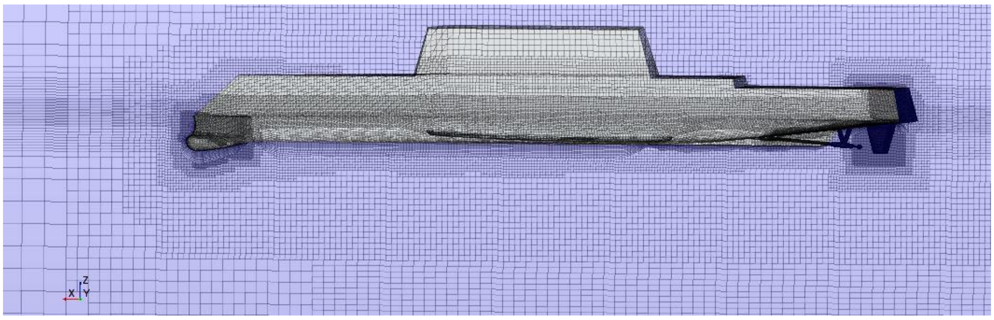

Fig 3. Grid distribution around the hull

To decrease the runtime, this simulation is carried out only for the half-body of the vessel, since the vessel is symmetrical about its centerline plane.
The coordinate system is fixed at the stern of the vessel, on the centerline, where z = 0 is at the designed waterline of the hull and the direction of heave, x = 0 is located at the end of the stern and is the direction of surge and y = 0 is in the middle of the hull, on the centerline and is the direction of sway. The extent of the computational domain agrees well with the minimum requirements prescribed in ITTC regulations [4]. For this study, the domain is taken to be similar to the domain taken by Wan et al. (2016) [5], which is as follows – $-4 L_{WL} < x < 2.5 L_{WL}$, $0 < y < 1.5 L_{WL}$, $-1.0 L_{WL} < z < 0.5 L_{WL}$, with reference to the aforementioned coordinate system.

Boundary conditions

Following boundary conditions are applied on the faces of the domain and the hull surface:

Inlet	:	Forward, top and bottom of the domain, uniform flow is given, 1.11 m/s
Outlet	:	Backward of the domain, pressure outlet
Walls	:	Hull body, no-slip condition
Symmetry	:	Centerline boundary and Side of the domain

4. Results and Discussions

The simulation of towing condition is carried out for fully appended condition, without propeller, for an advancing speed of 1.11 m/s, which corresponds to Fr = 0.20. The domain consists of a total 2.4M grids. During the computation, a release time of 1 second is set, which releases the hull one second after the start of the simulation, to allow some time for the fluid flow to initialize. After this period, the body motion calculations are carried out with four degrees of freedom constrained, and two allowed, which are rotation about y-axis (pitch) and translation along z-axis (heave). This enables the calculation of trim and sinkage values for this simulation.
A comparison between the obtained numerical results and the available IIHR EFD data (fully appended) are listed in Table 2. In addition, to further validate our numerical results, grid uncertainty analysis is performed for the towing condition, which will be described in the grid uncertainty analysis part.
It is noteworthy that none of the parameters except R_T can be compared with the measured data, since the only EFD data available is the IIHR (fully appended) EFD value of total resistance. Since experimental values other parameters are not available, only numerical results for them are presented in Table 2.

Figure 4 shows the plot of total resistance and its components against time, which shows that, the simulation start converging after approximately 10 seconds, and shear force component has dominance over pressure resistance force.

Table 2. Numerical results for ship motions and resistance, at Fr = 0.20

Parameters	EFD (fully appended)	CFD
Total Resistance (N)	4.54	4.836
Pressure Resistance (N)		1.296
Shear Resistance (N)		3.540
Sinkage* (m)		1.02E-4
Trim* (degree)		-0.476

* Trim is taken positive for bow up position, and to measure sinkage, z-axis is taken positive in downwards direction

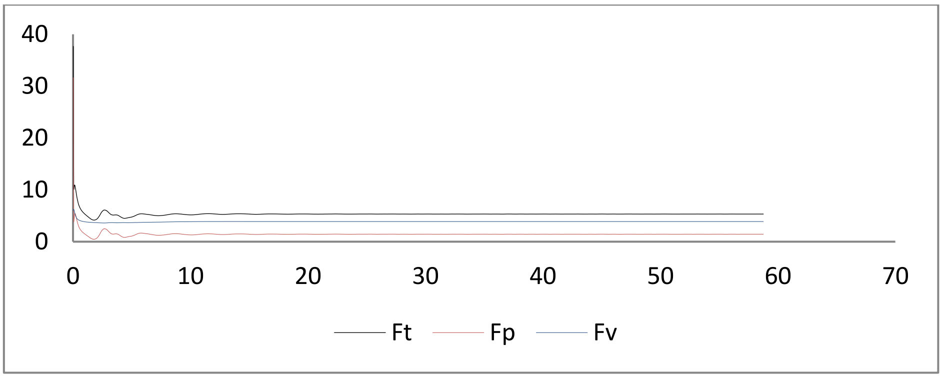

Fig 4. Time histories of the ship resistance at Fr = 0.20

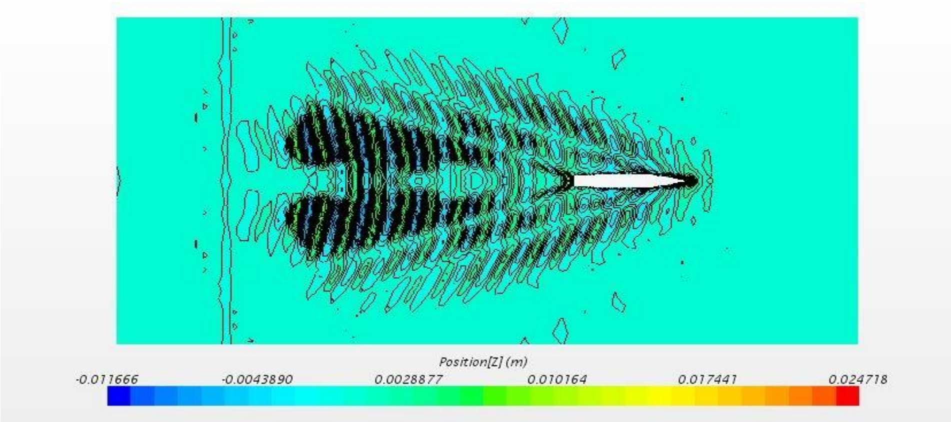

Fig 5. Wave pattern produced by the vessel at Fr = 0.20

Figure 5 shows the pattern of Kelvin wake generated in present simulation due to the motion of the water past the vessel, at Fr = 0.20. Gravity waves generated by an object moving at constant speed at the water surface form a specific pattern commonly known as the Kelvin wake, which is obtained due to the relative motion of vessel with respect to water in the present simulation.

5. Uncertainty analysis

Uncertainty analysis in the present work is done following the verification methodology described in Stern et al. (2006) [6]. Numerical uncertainty is obtained from grid uncertainty (U_G) and iterative uncertainty (U_I), according to (1)

$$U_{SN}^2 = \sqrt{U_G^2 + U_I^2} \quad (1)$$

The convergence solution (R_G) of the different solutions (S_i , at least three) is defined as:

$$R_G = \frac{S_2 - S_1}{S_3 - S_2} \quad (2)$$

where S_i, $i = 1$, 2 and 3 correspond to solutions with fine, medium, and coarse grid respectively.
According to this formulation, three convergence conditions are possible:

(i) $0 < R_G < 1$: Monotonic Convergence
(ii) $R_G < 0$: Oscillatory Convergence
(iii) $R_G > 1$: Divergence

For condition (i), generalized Richardson extrapolation (RE) is used to estimate grid uncertainty U_G. For condition (ii), uncertainties are estimated simply by attempting to bind the error based on oscillation maximums S_U and minimums S_L, i.e. $U_G = \frac{1}{2}(S_U - S_L)$. While for condition (iii), errors and uncertainties cannot be estimated.
In the present study, three grids with a refinement ratio of $\sqrt{2}$ in each direction are carried out for the grid convergence analysis. The results of the grid uncertainty are listed in Table 3.

Table 3. Grid uncertainty results for towing condition at Fr = 0.20

Grid	ID	Grid Size (M)	C_T (x10³)	Error
EFD			4.92	---
Fine	S1	2.4M	5.23	6.3 (%D)
Medium	S2	1.8M	5.77	10.3 (%S1)
Coarse	S3	0.6M	5.45	-5.55 (%S2)
R_G			-1.71	
U_G (%S2)			-1.91	
Convergence Type			Oscillatory	

Figure 6 shows the time of the convergence of total resistance results obtained from three different simulations carried out at three different grid densities.

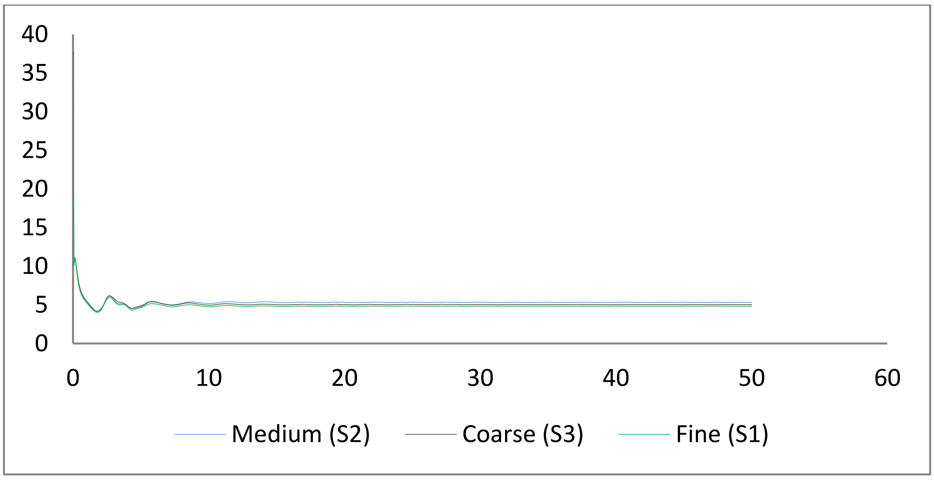

Fig 6. Plot of total resistance against time, for three different grids

The results have good convergence as shown in Table 3. The grid uncertainty of total resistance coefficient C_T is only 1.91%, which indicates that the grid density has very limited effect on the resistance within the current range of grid size, presented in this study. Also, according to Stern et al. (2006) [6] iterative uncertainty can be neglected while calculating numerical uncertainty, since U_I is very small.

6. Conclusion

This paper presents the towing condition simulations for ONR Tumblehome ship, carried out using commercial CFD software StarCCM+ at Fr = 0.20. Predicted total resistance is compared with the experimental results and the results were found to have good consistency. Furthermore, grid uncertainty analysis is also carried out at Fr=0.20. The total force coefficient shows oscillatory convergence and the grid uncertainty of C_T is 1.91%, indicating that the grid density has very limited influence upon the total resistance within the current range of grid size.

7. References

[1] J. Holtrop, and G. G. J. Mennen, “A Statistical Power Prediction Method”, *International Shipbuilding Progress*, 25(290), pp. 253–256, 1978.

[2] D. Savitsky, and P. W. Brown, “Procedures for Hydrodynamic Evaluation of Planing Hulls in Smooth and Rough Water”, *Marine Technology*, 13(4), pp. 381–400, 1976.

[3] L. Caretto, A. Gosman, S. Patankar, and D. Spalding, “Two calculation procedures for steady, three-dimensional flows with recirculation”, *Proceedings of the Third International Conference on Numerical Methods in Fluid Mechanics*, volume 19 of *Lecture Notes in Physics*, pp. 60–68, 1973.

[4] 26th ITTC Specialist Committee on CFD in Marine Hydrodynamics, *Practical Guidelines for Ship CFD Simulations*, Technical report 7.5-03-02-03, Revision 01, International Towing Tank Conference (ITTC), 2011.

[5] J. H. Wang, W. W. Zhao, and D. C. Wan, “Self-propulsion Simulation of ONR Tumblehome Using Dynamic Overset Grid Method”, *Proc 7th Int Conf Comput Method*, 2016.

[6] F. Stern, R. Wilson, and J. Shao, “Quantitative V&V of CFD simulations and certification of CFD codes”, *International Journal for Numerical Methods in Fluids*, 50(11), pp. 1335–1355, 2006