Hydrodynamic Analysis of Ship Resistance following ITTC Method and Star CCM$^+$

[1]Sharadindu Bikash Majumdar, [2]Dr. Asis Mazumdar

[1]*Research Scholar in Engineering, SWRE, Jadavpur University, India*
[2]*Dean of Interdisciplinary Studies, Law and Management, Jadavpur University, Kolkata & Director, School of Water Resources Engineering & Coordinator, Regional Centre, NAEB (MOEF, GoI), Jadavpur University, Kolkata – 700 032, INDIA*

Abstract - *In ship design, hydrodynamics play a vital role which is conventionally solved by towing tank experiment in laboratory followed by ITTC methods for resistance calculations, causing a higher expense. But nowadays, due to development in the technological R&D, these experiments can also be performed in CFD simulation software in computers creating more financial savings by reducing the experimental setup in the laboratory. In the present study, an attempt has been taken to compare results obtained analytically from model test in the towing tank in a marine laboratory with computational results of the model obtained from simulation software, Star CCM$^+$. It has been found that there is a fairly satisfactory similarity in the results obtained from the analytical and the computational methods practised in the present study.*

Keywords - *Hydrodynamics, Towing Tank, ITTC, CFD, Star CCM$^+$*

I. INTRODUCTION

The investigation of a ship in steady rectilinear motion at the free surface of a calm sea is a classic problem in ship hydrodynamics [1], where ship waves and wave resistance are subjects of utmost importance. The satisfactory solution of this problem is of great importance to naval architects and engineers. The ship designer has to ensure that a proposed ship achieves the desired speed with a minimum of required power. For most ships the viscous resistance cannot be significantly reduced by changing the hull form and this leaves the ship designer more or less free to choose a suitable hull form (from a resistance point of view). Optimal ship forms are those which generate the smallest waves and it is therefore highly desirable to develop a theoretical tool to analyze the relationship between wave resistance and the geometry of the ship's hull. The renowned English engineer, William Froude established a formula (now known as the Froude number) by which the results of small-scale tests could be used to predict the behavior of full-sized hulls. The first Ship model basin was built, at public expense, at his home in Torquay, England. The geometry of the test model has been adopted by the ITTC (International Towing Tank Conference) 1978 method (the revised form of ITTC in 1957 method) and also used as a recommended benchmark for calculation of ship resistances. The brilliant combination of mathematical proficiency with the practical experimentation accomplished by him in the towing tank experiment is yet practiced today [2, 5]. In the present study, an investigation has been carried out on resistance of the ship model using CFD (Computational Fluid Dynamics) method by STAR CCM$^+$ simulation software. Now, it is under the present situation of competitiveness between the numerical calculations by CFD and the model basin experimental results, where CFD opens a new field for the traditional Towing Tank experiments [4].

II. LITERATURE REVIEW

Study was conducted on ship hydrodynamics to find out the total resistance over a container vessel, using the CFD software, "Star CCM$^+$". Reynolds Averaged Navier Stokes (RANS) equation was used. Total resistance was calculated from CFD and from statistical and empirical method of Holtrop & Mennen at different Froude numbers. The results obtained from both the methods synchronized very intimately [9].

CFD analysis was carried out in another study to calculate the drag coefficient (C_D) from the flow of air over the ground vehicles. Ansys Fluent was used as the CFD software in the study to compare the simulation results with the actual data obtained from the wind tunnel experiment [10].

III. METHODOLOGY

Since, Jadavpur University has no arrangement of ship model basin i.e. a towing tank facility to carry out hydrodynamic tests with ship models. The existing towing tank in the NSTL [3] (Naval Science and Technological Laboratory) must be booked many months in advance [5]. Thus the main disadvantage of towing tank test is the towing tank availability and cost of making the test model. Hence, sample data were collected from an existing model testing report, performed in the Marine Hydrodynamic Laboratory

(MHL). Equipments & Software required for the experiment are:
(a) Towing Tank (120 m x 4m x 3m) and a 3.28 m. tanker scaled ship.
(b) MS Excel program for manual/ analytical calculations &
(c) CFD software Star CCM+ (Version 9.06) for computational simulation.

Fig. 1 High Speed Towing Tank [3]

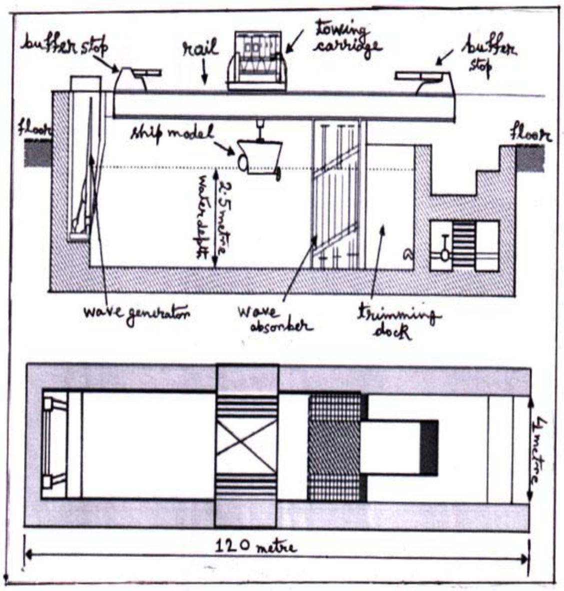

Fig. 2 Towing Tank Principle Dimensions

IV. RESULTS & DISCUSSIONS

A. Analytical Method from Model Test Results:

TABLE I
MAIN INPUT PARTICULARS FOR BOTH VESSEL AND MODEL (INPUT TABLE)

Designation	Symbol	Unit	Ship	Model
Length between Perpendiculars	L_{pp}	m	100.00	3.13
Length on Waterline	L_{wL}	m	101.66	3.18
Length overall submerged	Los	m	104.98	3.28
Breadth moulded on WL	B	m	17.50	0.55
Draught moulded on FP (Fwd. Perpendicular)	T_F	m	6.65	0.21
Draught moulded on AP (Aft Perpendicular)	T_A	m	6.65	0.21
Displacement volume moulded	V	m^3	8702.00	0.27
Displacement mass in seawater	V_1	m^3	8919.00	0.28
Wetted Surface Area of Bare Hull	S	m^2	2513.00	2.46
Wetted Surface with Appendages	S_1	m^2	2550.00	2.50
LCB (Longitudinal Center of Buoyancy) position aft of FP	FB	m	49.05	1.54
Block Coefficient	C_B	m	0.75	0.75
Midship Section Coefficient	C_M	m	0.995	0.995
Prismatic Coefficient	C_P	m	0.751	0.751
Length Breadth Ratio	L_{pp}/B	m	5.741	5.741
Breadth Draught Ratio	B/T	m	2.632	2.632

The suggested methods for the experimental evaluation are [12]:

$Vc = Vm + \Delta V$, where V_c = corrected speed and V_m = speed of model during testing.

ΔV = speed correction

Now, $\Delta V = Vm \times [1.1 \times (m_1) \times (Lm/b)^{3/4}]$,

where, $m_1 = A_m/A$

Am = maximum sectional area of model = $(Los_m \times B_m) = (3.28 \times 0.55)\ m^2$

A = sectional area of tank = $(120 \times 2.5)\ m^2$

L_m = length of model (length between perpendiculars of model) = L_{pp} = 3.13 m.

b = breadth of towing tank = 4.0 m.

TABLE III
DETERMINATION OF VARIABLES FOR PROHASKA PLOT

Test No.	V_m (m/s)	V_c (m/s)	Fr_m	Rn_m	C_{Fm}
1	0.728	0.7320	0.1321	2626870.90	3.8395×10^{-3}
2	0.819	0.8235	0.1486	2955229.76	3.7521×10^{-3}
3	0.910	0.9150	0.1651	3283588.62	3.6764×10^{-3}
4	1.001	1.0065	0.1816	3611947.48	3.6099×10^{-3}
5	1.092	1.0980	0.1982	3940306.35	3.5508×10^{-3}
6	1.183	1.1895	0.2147	4268665.21	3.4982×10^{-3}
7	1.274	1.2810	0.2312	4597024.07	3.4501×10^{-3}
8	1.364	1.3715	0.2475	4921794.31	3.4066×10^{-3}
9	1.453	1.4610	0.2637	5242975.93	3.3671×10^{-3}

V_m = Model speed (m/s)
V_c = Corrected speed (m/s)
Froude number = $Fr_m = V_m / (g \times Lm)^{0.5}$
Reynold's number = $Rn_m = (V_m \times Lm)/\nu_m$
L_m = 3.13 mtr. (from Table I)
g = $9.81 m/s^2$
C_{Fm} = Frictional resistance coff. for model = $0.075 \div (\log_{10} Rn_m - 2)^2$
ν_m = kinematic viscosity of fresh water in towing tank at 25° C for model = $(0.8674 \times 10^{-6})\ m^2/sec$ (obtained by interpolation).

Calculations for variables used in Table III are shown below [5, 11]:
Rn_m = Reynold's number for model corresponding to model speed (V_m) (from Table II)
ρ_m = density of fresh water corresponding to 25°C for model = 997 kg/m^3 and
S_m = Wetted surface area of bare hull for model = 2.46 m^2 (from Table I, input table).
The deep water form factor $(1 + k)$ is calculated using a simplified version of an empirical formula reproduced from Millward (1989);
$(1 + k) = 2.4806 \times C_B^{0.1526} \times (B/T)^{0.0533} \times (B/L)^{0.3856}$, where C_B = block coefficient = 0.75, B/T = breadth draught ratio = 2.632 and B/L = breadth moulded on waterline/ length between perpendiculars = 0.55 m/ 3.13 m (for model) (all values obtained from Table I). Suffix 's' indicates ship and 'm' indicates the model under testing.
$C_{Tm} = ((1 + k) \times C_{Fm})$;
at low Froude number, $C_{Tm} = (1.2979 \times C_{Fm})$

TABLE III
DATA FOR PLOTTING TOTAL RESISTANCE FOR MODEL VS. VELOCITY OF MODEL

Test No.	V_m (m/s)	C_{Fm}	C_{Tm}	R_{Tm} (kN)
1	0.728	3.8395×10^{-3}	5.3933×10^{-3}	3.5059
2	0.819	3.7521×10^{-3}	5.2799×10^{-3}	4.3439
3	0.910	3.6764×10^{-3}	5.1816×10^{-3}	5.2630
4	1.001	3.6099×10^{-3}	5.0953×10^{-3}	6.2622
5	1.092	3.5508×10^{-3}	5.0186×10^{-3}	7.3403
6	1.183	3.4982×10^{-3}	4.9503x10-3	8.4974
7	1.274	3.4501×10^{-3}	4.8879×10^{-3}	9.7308
8	1.364	3.4066×10^{-3}	4.8314×10^{-3}	11.0253
9	1.453	3.3671×10^{-3}	4.7802×10^{-3}	12.3784

Now, as the ship model moves through calm water, the model experiences force acting opposite to its direction of motion. This force is the water's resistance to the motion of the ship model in towing tank, which is referred to as **"Total Hull Resistance R_{Tm}"**. The total hull resistance increases as model speed increases, as shown in the Table 3 above. Total Resistance for Model, $R_{Tm} = \left(C_{Tm} \times \frac{1}{2} \times \rho_m \times V_m{}^2 \times S_m\right)$ kN.

B. Computational method using CFD software 'Star CCM+'; defining dimensions of the Towing Tank, Tank Domain and the Tested Ship Model:

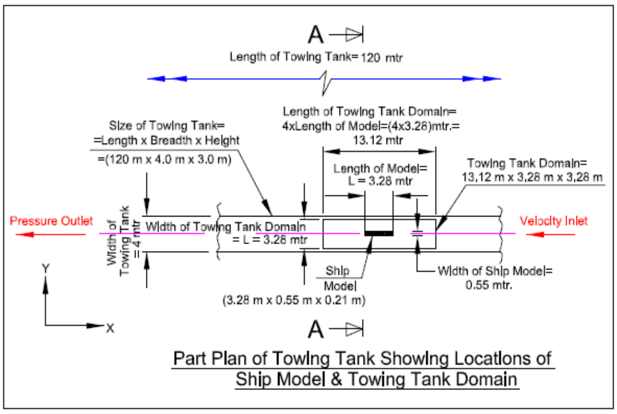

Fig. 3 Part plan of Towing Tank with Ship Model

TABLE IIIV
COMBINED TABLE TO DETERMINE DRAG FORCE

Test No. (C1)	V_m (m/s) (C2)	C_f (C3)	Pressure (kPa or kN/m^2) (C4)	(x . y) =(3.28 *0.55) (C5)	F_D (kN) =(C3* C4*C5) =(C6)
1	0.728	6.769E+59	3.183E-59	1.804	38.8685
2	0.819	8.281E+59	2.67E-59	1.804	39.8869
3	0.91	1.0118E+60	2.2457E-59	1.804	40.9905
4	1.001	1.2352E+60	1.8872E-59	1.804	42.0525
5	1.092	1.5074E+60	1.5847E-59	1.804	43.0935
6	1.183	1.8398E+60	1.3323E-59	1.804	44.2190
7	1.274	2.2468E+60	1.1177E-59	1.804	45.3029
8	1.364	2.7463E+60	9.396E-60	1.804	46.5508
9	1.453	3.3617E+60	7.885E-60	1.804	47.8186

F_D = Drag force (kN) = Skin friction coff.×pressure formed on model× wetted area of model.
C_f = Skin friction coff.
$(\boldsymbol{x} \times \boldsymbol{y})$ = wetted area in m^2, where x = length of the test-model = 3.28 mtr. & y = width of the test-model = 0.55 mtr.

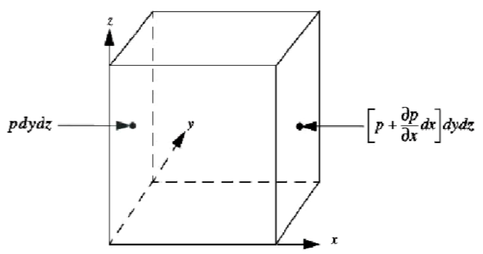

Fig. 4 Pressure acting on the Faces of a Fluid Element [7]

C. Comparison between output results obtained from Analytical and Computational methods (i.e. R_{Tm} (kN) & F_D (kN)) with the velocity of model Vm (m/s):

TABLE V
DATA SHOWING VELOCITY OF MODEL, DRAG FORCE AND TOTAL RESISTANCE FOR MODEL

Test No. (C1)	Vm (m/sec) (C2)	R_{Tm} (kN) (C4)	F_D (kN) (C3)
1	0.728	3.5059	38.87
2	0.819	4.3439	39.89
3	0.91	5.263	40.99
4	1.001	6.2622	42.05
5	1.092	7.3403	43.09
6	1.183	8.4974	44.22
7	1.274	9.7308	45.30
8	1.364	11.0253	46.55
9	1.453	12.3784	47.82

Numerical CFD simulations corresponding to the model tests were done for comparison with the analytical results. The assessment for model tests calculations is excellent. The Star CCM$^+$ results are validated against experimental data obtained from the towing tank experiment and has shown excellent accuracy [6].

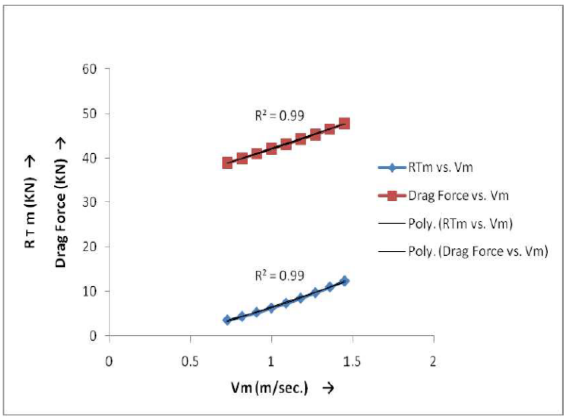

Fig. 5 Comparison between Analytical Results obtained by ITTC 1978 and Computational Results from CFD Analysis

In the above figure (Fig. 5), total model resistance (R_{Tm}) calculated from ITTC method was compared with the drag force obtained from the CFD method. The reason of the errors in the result may be due to the weakness of the CFD code for calculating the wave making resistance [7]. However, we could apparently reduce the errors by using the high speed computers [7] for numerical modeling and using finer grids or meshes. The results obtained by experimental method in towing tank are slightly lower than the computational results. Moreover, we considered the ship test model as a rectangular block and performed the computational simulation on it using the CFD software, which in turn increased the drag force drastically in the present study. But in the towing tank experiment, the forward cross section of the ship model or the bow part is reduced in area following a curved surface dropping the total resistance force to pull the ship model by the carriage.

V. CONCLUSION

From the experiment of resistance test of the ship model discussed in the present paper, we can finally conclude that the study is a successful one, in achieving its objectives to design and implement a resistance test for a ship model and thus to derive the equivalent full scale resistance for the entire ship.

The performance of the commercial software Star CCM^{+} has been evaluated in the study for the application in the shipbuilding industries as simulation software for design, analysis and feasibility purposes.

This study also elaborates the optimization of ship hydrodynamics [8] performances, which in turn reduce the consumption of ship power and thus to increase the ship operational behaviors.

Thus the comparative study can be implemented widely, in the field of ship hydrodynamics, for an investigational purpose.

ACKNOWLEDGMENT

The authors express their sincere gratitude to all professors and staffs of School of Water Resource Engineering and Fluvial Hydraulics Laboratory, Jadavpur Unversity for their extended support in carrying out the numerical tests (CFD) under the study. They are also thankful to their family members for their gentle support in affording time for completion of the paper.

REFERENCES

[1] Baar, J. J. M., "A Three Dimensional Linear Analysis of Steady Ship Motion in Deep Water", Ph.D. Thesis, Brunel University, U. K., p 6-15, Nov. 1986.

[2] (2015) The ITTC website. [Online]. Available: https://ittc.info/media/2021/75-02-02-02.pdf

[3] Technology Focus, "Naval Test Facilities", Bulletin of DRDO, ISSN: 0971-4413, Vol.22, No.1, Feb., 2014.

[4] Marco, A. D., Mancini, S., Miranda, S., Scognamiglio, R., and Vitiello, L., "Experimental and numerical hydrodynamic analysis of a stepped planning hull", Applied Ocean Research, Vol. 64, p 135-154, 2017.

[5] Smoker, B., "Escort Tug Performance Prediction: A CFD Method", Master of Applied Science Dissertation, Univ. of Victoria, 2012.

[6] Wieghardt, K., "Review of Contributions to the 12th ITTC on Viscous Resistance", Resistance Session, Univ. of Hamburg, p 86-87, 1969.

[7] Asen, P., "Analysis of the Flow around a Cruise Ferry Hull by the Means of Computational Fluid Dynamics", M. S. (Tech.) Thesis, School of Engineering, Aalto University, Espoo, p 8-17, 2014.

[8] Marzi, J., "Targeted Advanced Research for Global Efficiency of Transportation Shipping (TARGETS)", Report No. M 40 (36), Hamburg, 2014.

[9] Atreyapurapu, K., Tallapragada, B., and Voonna, K., "Simulation of a Free Surface Flow over a Container Vessel Using CFD", International Journal of Engineering Trends and Technology (IJETT), Vol. 18, No. 7, p 334-339, Dec. 2014.

[10] Khan, R. S., and Umale, S., "CFD Aerodynamic Analysis of Ahmed Body", International Journal of Engineering Trends and Technology (IJETT), Vol. 18, No. 7, p 301-308, Dec. 2014.

[11] (2015) The ITTC website. [Online]. Available: https://fenix.tecnico.ulisboa.pt/downloadFile/3779577315265/RP_Lecture6and7.pdf

[12] (2015) Ship Resistance and Propulsion.[Online]. Available: http://textofvideo.nptel.ac.in/114106026/lec10.pdf