ICMIEE18-185

Prediction of Resistance, Sinkage and Trim of a Bulk Carrier by Computational Fluid Dynamics Method

Md. Rakibul Hasan, Md. Abdur Rahim, Md. Sharier Islam, Md. Mashiur Rahaman[*]
Department of Naval Architecture & Marine Engineering, Bangladesh University of Engineering & Technology
DHAKA-1000, BANGLADESH

ABSTRACT

Prediction of resistance, sinkage and trim has always been a challenging task for the naval architects at the design stage to achieve optimum power requirement and fuel consumption for desired speed. Previously, experimental methods laid the foundation of ship design. Later, towing-tank experiments become more practical but long dated, expensive, limited on the availability of physical tanks. Flow characteristics differ significantly from full scale due to insufficient Reynolds similarity at model test. Now-a-days, the applications of computational fluid dynamics (CFD) are advancing rapidly in marine hydrodynamics fields. CFD solves Reynolds similarity problem by offering both model and full scale results with a great details of flow fields. In this paper, a commercial CFD code STARCCM+ is used to simulate and compute the calm water resistance, sinkage and trim of a bulk carrier. The simulation results were compared with experimental data which mark a wealthy harmony between two results.

Keywords: resistance, CFD, marine hydrodynamic, STARCCM+, bulk carrier,.

1. Introduction

Bulk carriers play an important role in international maritime transport and trade. A bulk carrier, bulk freighter, or colloquially, bulker is a merchant ship specially design to transport unpackaged bulk cargo such as grain, coal, ore and cement in its cargo holds. Bulk carriers are ranked as the second most common type of ship in the world, accounting for over 20 percent of the global merchant fleet. The statistics illustrated in Fig.1 represents the world's merchant fleet between January 1, 2008 and January 2017 [1] with a breakdown by type. Of the around 52,000 merchant ships trading internationally, some 11,000 ships were bulk carriers.

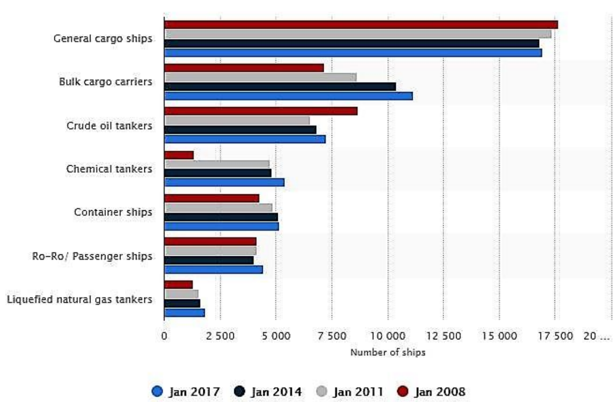


Fig.1 Number of merchant ships by type (2008-2017).

Resistance is one of the important factor related to ship. And prediction of resistance is must in order to obtain an economically profited design that will fulfill the mandatory regulations imposed by IMO [2] having an appropriate speed because if the speed gap is too large, the owner can refuse the ship. Empirical methods are the simplest and fastest among of three different ways of resistance prediction but can be applied only at the earliest design stage when principal particulars and hull coefficients have lack of accuracy. Towing tank experiment (2nd method) is not too much acceptable method when compared to the third method, numerical solutions which is also known as Computational Fluid Dynamics (CFD) described by Lars Larsson and Hoyte C. Raven (2010) [3]. According to Versteeg & Malalasekera (2007, p.1) [4], "Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation". Application of CFD is rapidly increasing in ship hydrodynamics. Main advantage of CFD analysis is its flexibility and ability to fulfill both Froude and Reynold similarities where towing tank method is unable to fulfill Reynold similarity. In this paper, CFD technique is used to predict the resistance, sinkage and trim of Japan Bulk Carrier (JBC) without Energy Saving Device (ESD). The JBC hull was selected as one of the test cases in the framework of the Tokyo 2015 Workshop on Numerical Ship Hydrodynamics (T2015) [5]. It is a Cape size bulk carrier designed by National Maritime Research Institute (NMRI), Yokohoma University, Ship Building Research Centre of Japan (SRC). The hull design and measurements were conducted

* Corresponding author. Tel.: +88- 01765251899
E-mail addresses: rhrahul45@gmail.com

with the support of ClassNK as part of the ClassNK joint R&D for Industry Program. Also, LDV measurement data in the wind tunnel are available from Technical University of Hamburg-Harburg. The simulations are performed through a finite volume based commercial code STARCCM+. Finally, the numerical results are compared with the experimental values and a good agreement is found.

2. Physical Phenomena

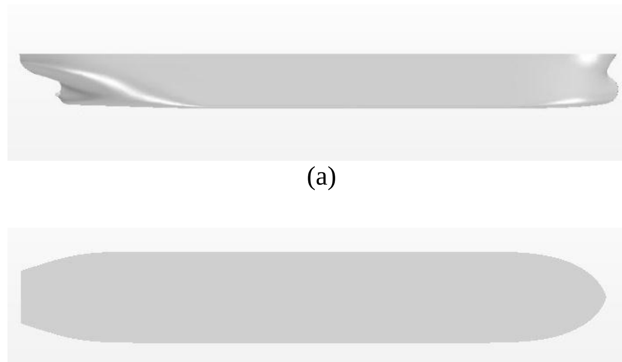
Three governing equations on which Computational Fluid Dynamics is based on are Continuity equation, Navier-Stokes equation, and Energy equation. The Bernoulli equation is derived from the momentum equation for inviscid flows, where viscus effects are negligible. Star-CCM+ uses the Volume of Fluid model exclusively for 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 S. Caretto [6]. STAR-CCM+ offers a wide variety of turbulence modelling options, including Spalart-Allmaras, k - ϵ, k - ω, Reynolds Stress Transport. In this thesis, the turbulent viscosity is calculated using the realizable k - ϵ turbulence model. The k - ϵ model is a two equations model using the turbulent kinetic energy k and the turbulent dissipation rate ϵ.

The interaction between the fluid and the ship is very important for the flow and the resistance on the hull. The two primary degrees of freedom (DOF) for ships in calm water are heave and pitch. The pitch and heave of the ship is dependent on the speed of the ship. At a certain speed the ship obtains a pitch and a heave through equilibrium of static and hydrodynamic forces acting on the ship. The equilibrium pitch and heave are called the dynamic pitch and heave. The fluid-body interaction is modelling using a dynamic fluid-body interaction (DFBI) model in STAR-CCM+. The DFBI model calculates the forces on the ship at certain intervals and translates and rotates a number of times the ship until the ship reaches an equilibrium.

It is important to model the boundary layer flow as it is used to calculate the frictional resistance on the ship. For the case studying in this paper, the Froude number are relatively low. The low Froude number causes the majority of the total resistance to be frictional resistance. A non-dimensional distance from a cell to the nearest wall is called the y^+ value. The y^+ value for the cells near the wall is called the wall y^+ value, and it is an indication of how well the boundary layer is discretized. The volume mesh consists of hexahedrons in a structured grid. The mesh is generated using the trimmer-mesh function in STAR-CCM+. Boundary layer mesh is used on the hull surface in order to make a good estimation of the shear stresses on the hull.

3. Geometry

Bulk carriers have an important role in international maritime transport. In this paper, we carried out a numerical study on a model of bulk carrier vessel. JAPAN Bulk Carrier (JBC) is a Cape size bulk carrier and the present numerical simulations are carried out for the JBC model. In Fig.2, the geometry of bulk carrier model is shown and the characteristics of the ship and that of the model are listed in Table 1.



(a)

(b)

Fig.2 Geometry of JBC hull. (a) Side view, (b) Top view.

Table 1 Ship and model principal particulars.

Main Particulars		Full Scale	Model Scale
Scale of model (λ)		40	
Length between perpendiculars	L_{PP} (m)	280.00	7.00
Length of waterline	L_{WL} (m)	285.00	7.125
Maximum beam of waterline	B_{WL} (m)	45.00	1.125
Depth	D (m)	25.00	0.625
Draft (mid)	T_M (m)	16.50	0.4125
Displacement volume of hull	∇ (m^3)	178369.9	2.7872
Wetted surface area of hull	S_0 (m^2)	19556.1	12.201
Block coefficient	C_B	0.858	0.858
Midship section coefficient	C_M	0.9981	0.9981
Moment of inertia	K_{xx}/B	0.4	0.4
Moment of inertia	K_{yy}/L_{PP} K_{zz}/L_{PP}	0.25	0.25

Conditions for resistance tests are calm water condition, without rudder, without propeller, without ESD, $FR_{Z\theta}$, $R_e = 7.46\times10^6$, $F_r = 0.142$, $U = 1.179$ ms^{-1}, $\rho = 998.2$ kgm^{-3}, $\nu = 1.107\times10^{-6}$ m^2/s and $g = 9.80$ ms^{-2} which are developed by the workshop organizing committee (NMRI, 2015).

The JBC hull is placed within a virtual towing tank as shown in Fig.3.

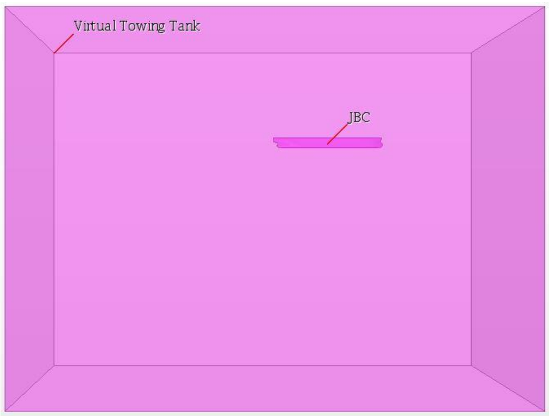
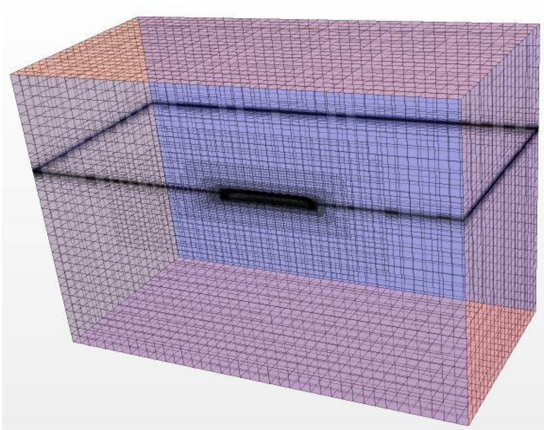


Fig.3 Computational Domain

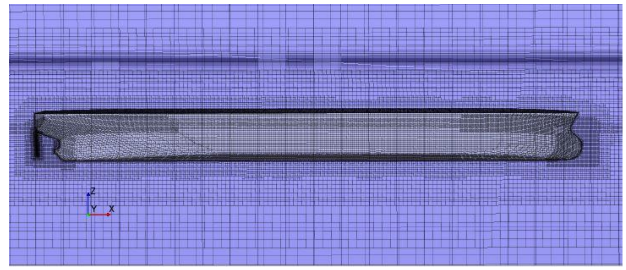
The coordinate origin is at aft perpendicular and still water surface. The still water surface is at 0.4125 m above the keel. Due to symmetry conditions, only half of the body is modeled.

4. Mesh Generation

Resistance analysis are generally performed using a trimmed volume mesh within prism layers on the wetted surface of the hull. Using trimmed cells means that the mesh is aligned with the undisturbed free surface. The background domain [7] extends to $-1.5Lpp < x < 5.0Lpp$, $-1.5Lpp < y < 1.5Lpp$, $-1.0Lpp < z < 0.5Lpp$, and the hull domain has a much smaller region with a range of $-0.15Lpp < x < 1.2Lpp$, $-0.13Lpp < y < 0.13Lpp$, $-0.2Lpp < z < 0.2Lpp$. Finite volume method requires grid cells in order to discretize the partial differential equations and approximate algebraic equations. The simulation was run for three volume mesh. All snapshots and monitor plots shown in this paper are for the case of Mesh 1. Details grid distributions are shown in Fig.4 (a & b) for Mesh 1.



(a) Grid distributions within the computational domain.



(b) Grid distributions around the hull.

Fig.4 Grid distributions for Mesh 1 (a & b).

5. Boundary Conditions

Before creating the volume mesh, it is mandatory to set the appropriate boundary types. The boundaries of the hull are defined as wall with no slip conditions. Following boundary conditions (Fig.5) are applied to the faces of the domain and the hull surface: -

Boundary Name	Boundary Type
Tank.Bottom	Velocity Inlet
Tank.Inlet	Velocity Inlet
Tank.Outlet	Pressure Outlet
Tank.Side	Symmetry Plane
Tank.Symmetry	Symmetry Plane
Tank.Top	Velocity Inlet

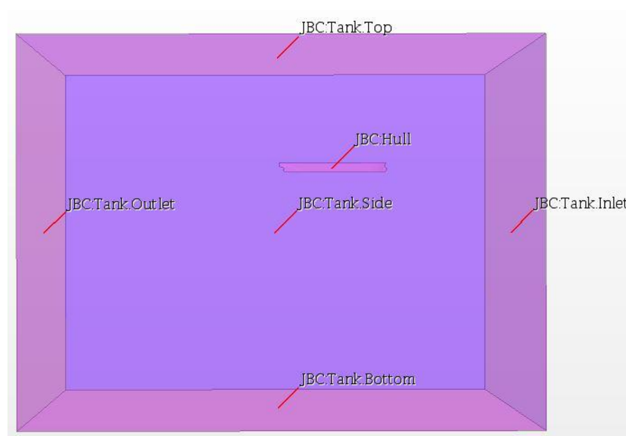


Fig.5 Computational domain with boundary conditions.

Two Eulerian phases (air and water) are created. The properties of water are liquid state, constant density and turbulent flow whereas air is in gaseous state.

6. Results and Discussion

In these simulations, the ship is allowed to move with two degrees of freedom to account for sinkage (Z Motion) and trim (Y Rotation). One of the most important factor is the setting of initial center of mass [3.507, 0.0, 0.313] m,m,m. A large deviation can be produced for inappropriate value of center of mass. The simulations have been carried out for around 2700 time steps with a time step of 0.04s, total physical time was 108+ seconds. Release time of one second was set which released the hull one second after the start of the simulation. It took more than three days (five cores computer) to complete the simulations. Fig.6 shows a close-up water surface around ship hull on the symmetry plane at the end of the simulation respectively. As the hull moves though the water, it produces a Kelvin wake pattern (Fig.7).

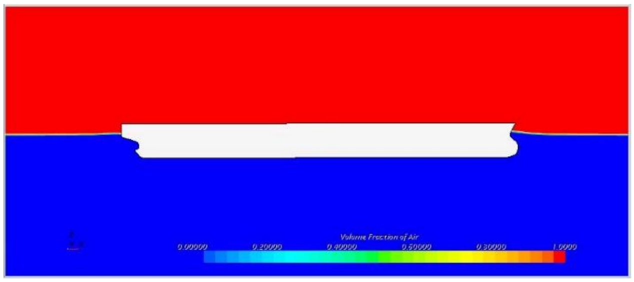


Fig.6 Free surface at the time, 108 s.

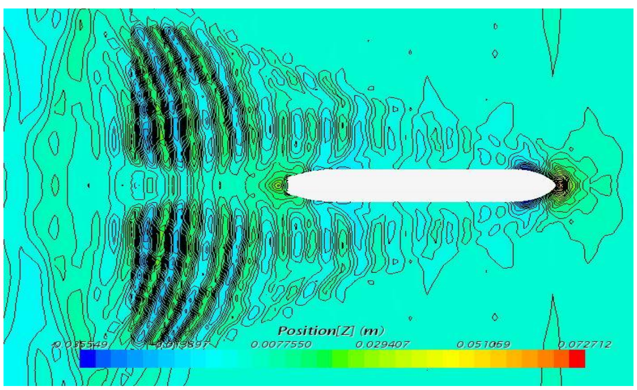


Fig.7 Kelvin wave pattern for speed of 1.179 ms^{-1} and a time step of 0.04s.

Forces obtained from the simulations have to be multiplied by 2 because only half of the geometry was modeled due to symmetry. The residuals up to 27000 iterations are shown in Fig.8.

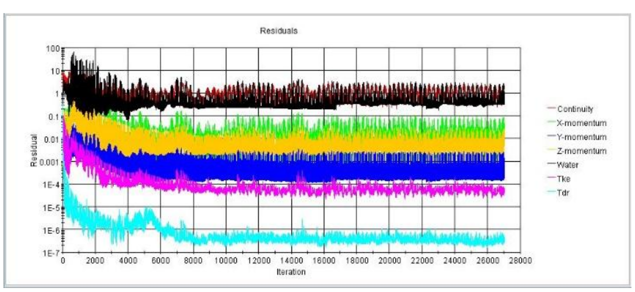


Fig.8 Convergence of residuals for high mesh resolution and a time step of 0.04s (Mesh 1).

A plot of total drag over time is shown in Fig.9 and shear and pressure drag is represented in Fig.10. From Figs. 8 - 10 it is indicated that the strong oscillations start converging almost 30 second later, which is level out after approximately 80-85s.

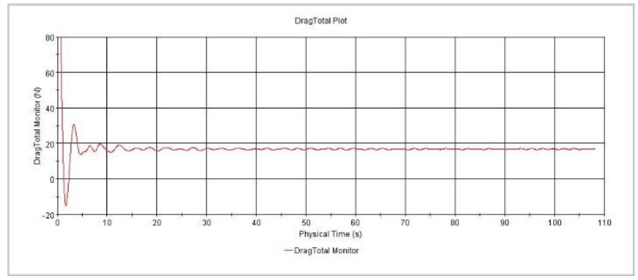


Fig.9 Convergence of the total drag.

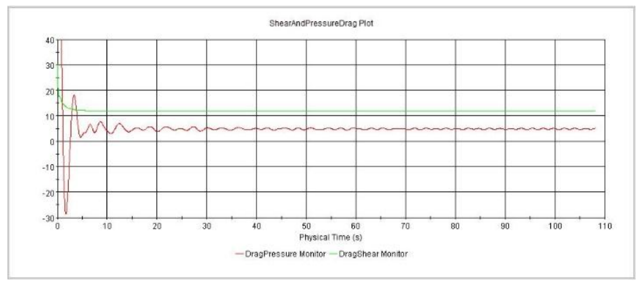


Fig.10 Convergence of the shear and pressure drag.

All Corresponding forces are obtained by averaging the numerical results from physical time 20s to 108s and the comparison with the experimental data are tabulated in Table 2. The calculated average number of oscillation of drag due to wave is 18.595. Since experimental values are not available for other parameters without total resistance, sinkage and trim, only simulation results for them are listed. It is expected that, the deviation decreases with increasing mesh refinement. As can be observed from the results, mesh 1 with the highest resolution predicts total resistance with highest accuracy. It is also the same for the prediction of sinkage.

Table 2 Comparison of CFD results and experimental results [resistance coefficient ($\times 10^3$)].

Mesh No	Parameter	EFD	CFD	Deviation*
M 1	C_T	4.289	4.3787	-2.09%
	C_F		3.078	
	C_P		1.198	
M 2	C_T	4.289	4.511	-5.18%
	C_F		2.805	
	C_P		1.121	
M 3	C_T	4.289	4.181	2.52%
	C_F		2.807	
	C_P		1.138	

But the trim results that are tabulated in Table 3 show relatively large deviation for Mesh 1 & 3. This exception may be due to the increase in grid spacing in z-axis. However, to better capture the trim, a good mesh refinement in z-axis may be required.

Table 3 Comparison of CFD results and experimental results [Sinkage (%L_{PP}), Trim (degree)].

Mesh No	Parameter	EFD	CFD	Deviation*
1	Sinkage	-0.086	-0.093	2.52%
	Trim	-0.18	-0.147	18.33%
2	Sinkage	-0.086	-0.0846	-5.18%
	Trim	-0.18	-0.183	-1.67%
3	Sinkage	-0.086	-0.096	11.63%
	Trim	-0.18	-0.165	8.33%

Note: a positive (+) sinkage value is defined upwards and a positive (+) trim value is defined bow up.

*Comparison error, E%D = (D-S)/D×100; where D is the EFD value and S is the simulation (CFD) value.

7. Conclusion

In present study, a commercial computational fluid dynamics method named STAR-CCM+ is used to predict the resistance, sinkage and trim for a bulk carrier model in calm water for F_r = 0.142. The numerical results are compared with the available experimental results. In general, the agreement with the experimental references throughout the investigated cases have been good. In case of high-resolution mesh, the deviation between numerical and experimental result for resistance and sinkage is less than 5 % but for trim the deviation is more than 10%. The possible reason may be due to the increase in grid spacing in z-axis. Also, there may be an issue with the VOF method which does not guarantee the conservation of energy near the free surface interface, with the minimum mesh resolution requirements then ensuring an accurate enough approximation to yield acceptable results. However, with the VOF method implemented in Star-CCM+, where both phases are solved for on a co-located grid, this criticism, which was of the original implementation of Hirt and Nichols [8] where the air phase was omitted from the solution, would appear to be less valid. Another possibility of occurring the deviation is that a minimum resolution is required to capture all the relevant spectral components making up the free surface wave system.

REFERENCES

[1] www.statista.com/statistics/264024/number-of merchant-ships-worldwide-by-type

[2] B. Barrass and D.R. Derrett. *Ship Stability for Masters and Mates (6th ed.).*

[3] Lars Larsson and Hoyte C. Raven. (2010). *Ship Resistance and Flow*. Jersey City, New Jersey: The Society of Naval Architects and Marine Engineers.

[4] Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics* (2nd ed.).

[5] NMRI. (2015). *Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics.* Retrieved from http://www.t2015.nmri.go.jp/

[6] Caretto, L., Gosman, A., Patankar, S., and Spalding, D. (1973). *Two calculation procedures for steady, three-dimensional flows with recirculation*.

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

[8] Hirt, C. W. and Nichols, B. D. (1981). *Volume of fluid (VOF) method for the dynamics of free boundaries*. Journal of computational physics, 39(1):201–225.