International Journal of Engineering Applied Sciences and Technology, 2022
Vol. 6, Issue 11, ISSN No. 2455-2143, Pages 1-4
Published Online March 2022 in IJEAST (http://www.ijeast.com)

A STUDY ON THE AIR RESISTANCE OF AN LNG TANKER BY EMPIRICAL AND CFD METHOD

Nguyen Manh Chien
Department of Shipbuilding
Vietnam Maritime University, Hai Phong, Viet Nam

***Abstract*— Air resistance has a significant effazqect on ships with large main deck structures such as container ships, liquefied natural gas (LNG) tanker, and large cruise ships. This paper presents a method to determine the air resistance for LNG tankers model by doing the experimental test in the wind tunnel. In addition, the author also calculates the air resistance by computational fluid (CFD) and compares the calculated results with experimental results.**

Keywords— **LNG tanker, CFD, wind tunnel, experimental test, resistance**

I. INTRODUCTION

Nowadays, new built ships tend to increase in size, the wind load acting on the ship plays an increasingly important role when the ships maneuver in high winds or sail through limited space such as canals. In March 2021, the container ship Ever Given ran aground across the Suez Canal. One of the reasons is believed to be due to the influence of high winds that make the ship lose its maneuverability. However, wind resistance has not been mentioned much in the theory, and the results of testing in wind tunnels are also very limited. With the development of computer systems with high computational speed, the CFD method is increasingly being applied in solving problems related to hydrodynamics and aerodynamics. Author W.D. Janssena et al [1] presented the results of calculating the wind resistance for container ships by CFD method and compared with the results of the in-duct model testing. However, almost no author has tested and compared the calculation results for liquefied petroleum gas (LNG) tankers - this is also one of the ships with a large surface area and this type of ship is significantly affected by air resistance. This paper presents the procedure and results of calculating the wind resistance acting on two popular models of LNG ships by two methods: wind tunnel test (experimental test) and CFD method. The CFD calculations is done by Star CCM + software

II. MODEL TEST AND CALCULATION

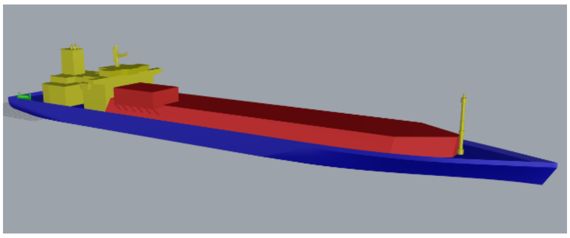
First, the 3D model was built on Rhinoceros software, based on the basic parameters of the LNG Norman Lady (IMO 7320344). However, the authors have 3D built two types of architecture on the main deck of the ship: the first is a spherical cargo tank, the second is a box. The basic parameters of the Norman Lady ship are shown in *Table 1*. The model was built in 3D and manufactured with a length of 1m, at the scale of 1/251 compared to the real ship.

Table -1 Basic dimension of MV. Norman Lady

Dimension		Unit	
Length overall	*LOA*	[m]	249.555
Length between perpendicular	L_{PP}	[m]	237.0
Breadth	*B*	[m]	23.0
Draft	*T*	[m]	10.641
Deadweight	*DWT*	[t]	50,764
Tank capacity		[m^3]	87,603
Speed	*V*	[knots]	17.5

Two models are created in Rhinoceros, which are shown in *Figures 1 and 2*. The scale factor is 1/251, leading to the overall length of each model is 1 meter. For the wind resistance test, we only need to model the shape of the veseel which are above the water. In addition, only main parts of superstructure are modelled, ignoring the small details, insignificantly affecting the air resistance.

After 3D modeling, the actual model is built using wood and plastic (3D printing) and then polished *(Figures 3 and 4)*



International Journal of Engineering Applied Sciences and Technology, 2022
Vol. 6, Issue 11, ISSN No. 2455-2143, Pages 1-4
Published Online March 2022 in IJEAST (http://www.ijeast.com)

Fig. 1. Ship model with rectangular tank in Rhinoceros

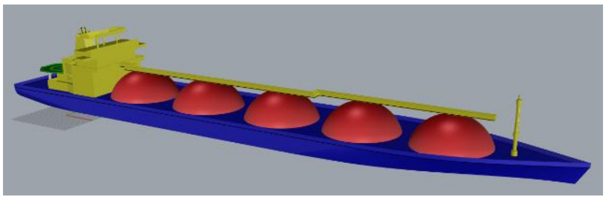


Fig. 2. Ship model with sphere tank in Rhinoceros

Fig. 3. Real ship model with rectangular tank for doing wind tunnel test

Fig. 4. Real ship model with sphere tank for doing wind tunnel test

III. TEST PROCEDURES IN WIND TUNNEL

The objective of the test is to determine the coefficient of air resistance. The model is fixed at the bottom of the wind tunnel and is rotated 10 degrees each time. For each 10 degree, force and torque are measured

The test was conducted with the two models mentioned above with rotation from 0 to 360 degrees, a total of 72 cases. The coordinate system when conducting the test is shown in Figure 5 below.

Here, there are 3 quantities that need to be considered, which are drag force in the x direction (Fx), drift force in the y direction (Fy) and torque around the z axis (Mz). These quantities are converted to dimensionless coefficients through the formula below

Drag coefficient:

$$C_A = \frac{R_A}{\frac{1}{2}\rho_A V_A^2 P_A}$$

Torque coefficient

$$C_{MA} = \frac{M_A}{\frac{1}{2}\rho_A V_A^2 P_A L}$$

In which: R_A: Drag force (along X, Y axis)

M_A: Torque (around Z axis)

ρ_A : air density, $\rho_A = 1.2$ kg/m^3

V_A: wind velocity $V_A = 23.6$ m/s

P_A: lateral surface area.

$P_{A1} = 0.054627$ (m^2) – for rectangular tank

$P_{A2} = 0.0607687$ (m^2) – for spherical tank

L = 0.994 (m) – model length

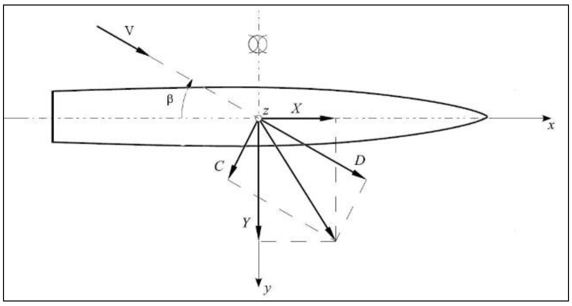


Fig. 5. Coordinate of the test

IV. CFD CALCULATION OF AIR RESISTANCE BY CFD METHOD

The CFD calculation is performed in Star CCM+ software. The calculation domain is set to the same size of the wind tunnel. The calculation domain is shown in Figure 6.

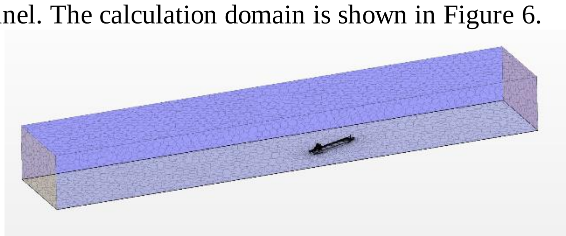


Fig. 6. Calculation domain

Here, polyhedral mesh is used instead of hexaheral mesh because using this mesh gives better results. The number of generated grid cells is about 550,000 grid cells for both types of models. Boundary condition is as follows: the wind velocity in the inlet is set as for the experiment, the ship model is set to the boundary condition as “wall boundary”, the turbulence model is k-w SST. In this setup, the boundary layer is solved directly through the “Low y+” setting. Because the number of calculations is relatively large: 72 cases, the author used the automatic calculation method through macros in Star CCM+. For each rotation angle, the calculation process is carried out

International Journal of Engineering Applied Sciences and Technology, 2022
Vol. 6, Issue 11, ISSN No. 2455-2143, Pages 1-4
Published Online March 2022 in IJEAST (http://www.ijeast.com)

for 1000 iteration, then the model will be rotated 10 degrees, re-grid and proceed to calculate and record the results

V. Result and Discussion

The CFD calculation result and experimental result are shown in Figure 7 and Figure 8 below

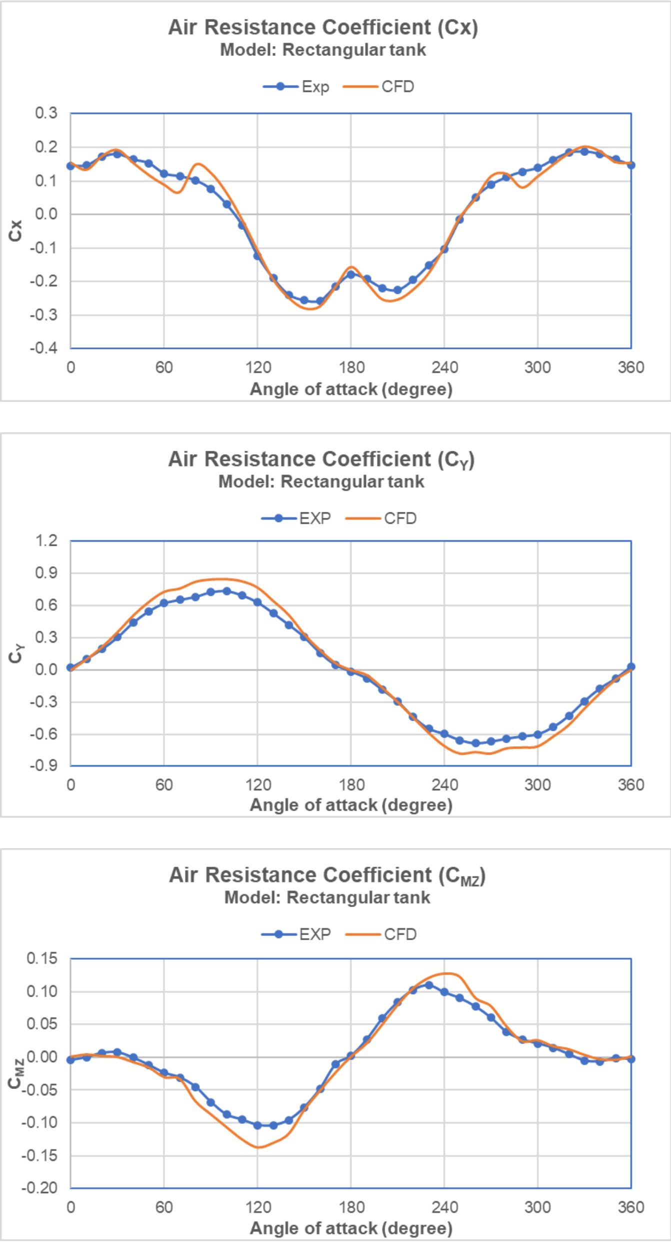


Fig. 7. Air resistance cofficient – rectangular tank

Fig. 8. Air resistance cofficient – spherical tank

Comparing the drag coefficient of the two models, it can be seen that the spherical tank has the torque coefficient (C_{MZ} about the Z axis) about 30% smaller than that of the rectangulara tank. In addition, the two drag coefficients Cx and Cy of these two types of tanks are quite similar. Large torque will especially affect the ship's maneuverability when the ship enters narrow spaces such as canals and wharf areas. Therefore, one may conclude spherical tank has an advantage over the rectangular tank.

The CFD calculation results give us quite accurate results compared to the test results, especially for the drag coefficient C_Y (this is the largest drag coefficient among the three coefficients C_X, C_Y and C_{MZ}). The trend of the graph shows that the CFD results are reliable. However, the CFD method gives relatively large results at some angles close to 90 degrees, 270 degrees. This can be explained by calculating a large number of cases (72 cases) and using macros to calculate automatically; some details were not treated well. Calculation results will be improved if the number of grid cells is increased in some areas such as the wake area behind the model.

VI. ACKNOWNLEDGEMENTS

The authors show the gratitude to the Vietnam Maritime University for their support to finish this paper.

VII. REFERENCE

[1] W.D. Janssen, B. Blocken, H.J. van Wijhe, CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications, Journal of Wind Engineering and Industrial Aerodynamics, Volume 166, 2017, Pages 106-116, ISSN 0167-6105,

[2] Ouchi, K, et. al. (2014). "A Study on Air Drag Reduction on the Large Container Ship in the Sea," International conference Design & Operation of Container Ships, London, pp 107-114.

[3] Kim, Y, et. al. (2015). "Design and Performance Evaluation of Superstructure Modification for Air Drag Reduction of a Container Ship," Proc 25th International Ocean and Polar Engineering Conference, Hawaii, Vol 4, pp 894-901.

[4] Bertram, V.,(2011) Practical ship hydrodynamics, Elsevier.

[5] Artjushkov, L., (1968) Wall effect correction for shallow water model tests. NE Coast Institution of Engineers and Shipbuilders.

[6] Geerts, S., Verwerft, B., Vantorre, M., and Van Rompuy, F., (2010) Improving the efficiency of small inland vessels. Proc., 7th European Inland Waterway Navigation Conf., Budapest Univ. of Technology and Economics, Budapest, Hungary.,

[7] Prakash, S. and B. Chandra, (2013) Numerical estimation of shallow water resistance of a river-sea ship using CFD. International journal of computer applications, **71**(5).

[8] Pacuraru, F. and L. Domnisoru (2017). Numerical investigation of shallow water effect on a barge ship resistance. in IOP Conference Series: Materials Science and Engineering. IOP Publishing.

[9] Patel, P.K. and M. Premchand (2015), Numerical investigation of the influence of water depth on ship resistance. International Journal of Computer Applications, **116**(17).

[10] Tezdogan, T., A. Incecik, and O. Turan (2016), A numerical investigation of the squat and resistance of ships advancing through a canal using CFD. Journal of Marine Science and Technology, **21**(1): p. 86-101.

[11] Molland, A.F., S.R. Turnock, and D.A. Hudson (2017), Ship resistance and propulsion. : Cambridge university press.

[12] Karpov (1946), A., Calculation of ship resistance in restricted waters. TRUDY GII. T. IV, Vol. 2 (in Russian).

[13] Perali, P., T. Lloyd, and G. Vaz (2016). Comparison of uRANS and BEM-BEM for propeller pressure pulse prediction: E779A propeller in a cavitation tunnel. in Proceedings of the 19th Numerical Towing Tank Symposium

[14] Brizzolara, S., D. Villa, and S. Gaggero (2008). A systematic comparison between RANS and panel methods for propeller analysis. in Proc. Of 8th International Conference on Hydrodynamics, Nantes, France

[15] Tu, Tran Ngoc; Chien, Nguyen Manh (2018), Comparison Of Different Approaches For Calculation Of Propeller Open Water Characteristic Using RANSE Method, Naval Engineers Journal, Volume 130, Number 1, pp. 105-111(7)