Vietnam Journal of Marine Science and Technology; Vol. 20, No. 3; 2020: 343–353
DOI: https://doi.org/10.15625/1859-3097/20/3/15249
http://www.vjs.ac.vn/index.php/jmst

Effect of mesh on CFD aerodynamic performances of a container ship

Ngo Van He*, Le Thi Thai

Hanoi University of Science and Technology, 10000, Hanoi, Vietnam
*E-mail: he.ngovan@hust.edu.vn

Received: 12 April 2020; Accepted: 30 June 2020

Abstract
In this paper, a commercial CFD code, ANSYS-Fluent has been used to investigate the effect of mesh number generated in the computed domain on the CFD aerodynamic performances of a container ship. A full-scale model of the 1200TEU container ship has been chosen as a reference model in the computation. Five different mesh numbers for the same dimension domain have been used and the CFD aerodynamic performances of the above water surface hull of the ship have been shown. The obtained CFD results show a remarkable effect of mesh number on aerodynamic performances of the ship and the mesh convergence has been found. The study is an evidence to prove that the mesh number has affected the CFD results in general and the accuracy of the CFD aerodynamic performances in particular.

Keywords: CFD, mesh, container ship, aerodynamic performance, hull.

Citation: Ngo Van He, Le Thi Thai, 2020. Effect of mesh on CFD aerodynamic performances of a container ship. *Vietnam Journal of Marine Science and Technology*, *20*(3), 343–353.

INTRODUCTION

Nowadays, research on applied computational fluid dynamics (CFD) to solve the technical problem is too popular. In ship research field, the CFD is a useful tool to improve ship performances and to develop new hull forms. It goes without saying that using CFD to investigate the aerodynamic performances of a ship is as important as experimental model test in a towing tank. Also, for a CFD code to compute the ship performances as well as the aerodynamic performances of the ship, the CFD manual or user guideline for using CFD published by the International Towing Tank Conference (ITTC) [1] must be followed step by step. Therefore, prior using CFD to solve any technical problem it is necessary to get needed experience available in the related up-to-date research area. The following is an overview of the most important previous studies in the field of ship aerodynamics:

Most of the published studies are related to researches on using commercial CFD codes and/or tunnel model test to solve the aerodynamic performances problem of ships in general or container ships in particular. The authors have used 3D steady Reynolds-averaged Navier-Stokes (RANS) to calculate wind loads acting on the above water hull surface of the ships. In those researches, the authors have concluded that validation of the CFD results with measurement results obtained in tunnel test is of very importance. It was shown that a close agreement between the CFD simulation results of a fairly detailed container ship and experiment results was about 5.9%. The larger deviations were found for the configurations with more simplified geometry from 6.9% up to 37.9%. Modeling the spaces in between container stacks decreased the average total wind load on the ship up to 10.4%. The slender ship hull instead of the blunt ship hull decreased the total wind load up to 5.9%. Taking into account wind tunnel blockage following the approach of the engineering science data showed an underestimation of up to 17.5% for the lateral wind load, as evidenced by comparing the CFD results in the narrow domain with those in wider domain [2]. Other authors presented studies on using CFD and experimental test to develop new modified hull shape with the reduced wind drag acting on a container ship. The authors proposed modified hull shape with an attached side cover, a center wall, a T center wall and a dome at the bow deck of the container ship. By using the side covers and the center wall, the container ship could reduce wind drag up to 40% of the total wind drag acting on ship at wind direction of zero degree. A dome at the bow ship could reduce up to 30% of the total wind drag acting on the container ship at the wind direction angle less than 30 degrees [3, 4]. Other papers also presented the studies on using RANS simulations and experiment towing test to design new concepts and devices on the superstructure of a container ship to reduce wind drag acting on the ship. Gap protectors between container stacks and visors in front of upper deck were found to be the most effective means for reducing wind drag acting on the ships. The authors concluded that CFD results agreed well with the experimental measurements and the wind drag acting on the modified ship could reduce up to 56% in the wind direction angle from zero to 50 degrees [5]. Other authors presented results of wind loads on a post-Panamax container ship. By using model test in wind tunnel, the wind forces acting on the ship have been investigated. The authors showed that a mere experimental approach provided directly applicable results for container ship operators and benchmark for development of new computational methods [6, 7]. Others published the numerical analysis of the wind forces acting on a LNG carrier model performed with CFD and experiment in wind tunnel. The results were represented in the form of coefficients of the wind force components for various angles of wind attack. The authors have compared CFD results with the different types and resolutions of the meshes in their simulations. Two empirical methods and additional experimental measurements of a similar LNG carrier have been compared. A reasonable agreement of the results has been shown in the research [8].

Others researchers presented the results on aerodynamic performances of the carrier ship such as the research on the reduced interaction effect between hull and accommodation on wind drag acting on hull of the ship. The authors have proposed a new hull form with different positions of accommodation and accommodation shapes on deck to reduce interaction effects between hull and accommodation. By using CFD simulation and experimental test in towing tank, drastically reduced wind drag acting on the ship had been found. The total wind drag acting on hull could reduce up to 60% of the total wind drag acting on hull [9–11]. Other researches on effects of the side guards on aerodynamics performance of a wood chip carrier were presented. By using CFD and experimental test in towing tank, the authors developed the side guards for the wood chip carrier. The CFD results clearly showed the effects of the side guards on aerodynamic performance of the ship and wind drag acting on hull drastically reduced up to 50% of the total wind drag [12]. Other authors presented researches on aerodynamic performances of a high speed ship, a passenger ship and other types of ships [11, 13].

In this paper, to have a better understanding in using a commercial CFD code to compute the aerodynamic performances and wind drag acting on a container ship, effect of mesh number and convergence of meshes on aerodynamic performances are investigated. By using the commercial CFD code ANSYS-Fluent, the aerodynamic performances of the above water surface hull of the container ship will be computed in the different mesh numbers.

MEHODOLOGY

Ship model used for computation

In this research, a full scale 1200TEU container ship has been used for computation. The aerodynamic performances and wind drag acting on the above water surface hull part of the ship at the wind direction angle of zero degree have been computed in five different mesh numbers. figure 1 shows the full scale above water surface hull part of the container ship used in the computation. The principal particulars of the ship are shown in the table 1.

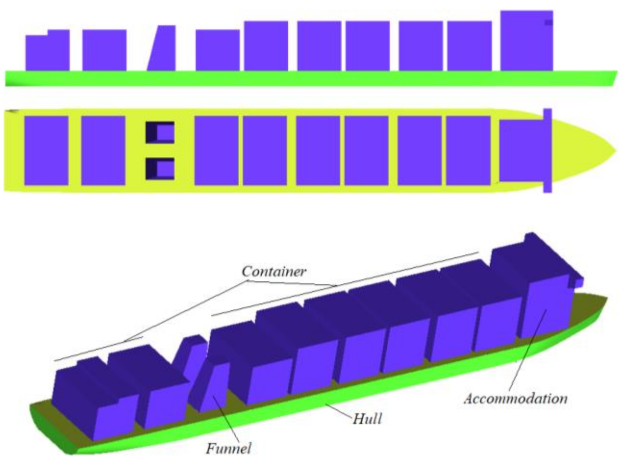

Figure 1. The above water surface hull part of the 1200TEU container ship

Table 1. Principal particulars of the container ship

Name	Description	Value	Unit
L	Length	176.20	m
B	Breadth	24.90	m
H	Depth	13.70	m
d	Draft	8.30	m
S_x	Frontal projected area of ship	423.95	m^2
C_b	Block coefficient	0.68	-
α	Wind attack angle	0	degree
R_n	Reynolds number	6.7×10^7	-

Computed domain and mesh

In this section, to investigate the effects of the mesh number on aerodynamic performances and mesh convergence, the computed domain is meshed in the five different mesh numbers. Figure 2 shows the designed computation domain. In CFD, the computed domain has affected CFD results, therefore, the same must be designed following the user guide for applied CFD in ship hydrodynamics or CFD manual published by the ITTC or CFD manufacturer [1]. Moreover, the researcher's experiment is of very importance in using CFD to solve the same problems on aerodynamic performances [9, 10, 13–16]. Figure 2 shows the limited dimension of the computed domain. The detailed mesh in computed domain with different mesh numbers is shows in figure 3. The detailed mesh generated in the computed domain is shown in table 2.

All meshes have been generated with the quality following the user guide for applied CFD in ship hydrodynamics [1, 14]. In this research, conditional boundary has been proposed appropriately based on the author's experience in using CFD and the available references [1, 9, 11, 13–15]. For computation, the turbulent viscous model k-ε has been used, the velocity inlet is set up for the inlet, the pressure outlet is set up for the outlet and the non-slip wall is used for the model [14, 17]. In this research, the ship is simulated in the condition at its service speed of 14 knots and wind direction of zero degree. After setting up the boundary conditions for the problems, all the cases with the different mesh numbers have been computed by the CFD to investigate the aerodynamic performances of the ship. Table 3 shows the computed conditions adopted for the problem.

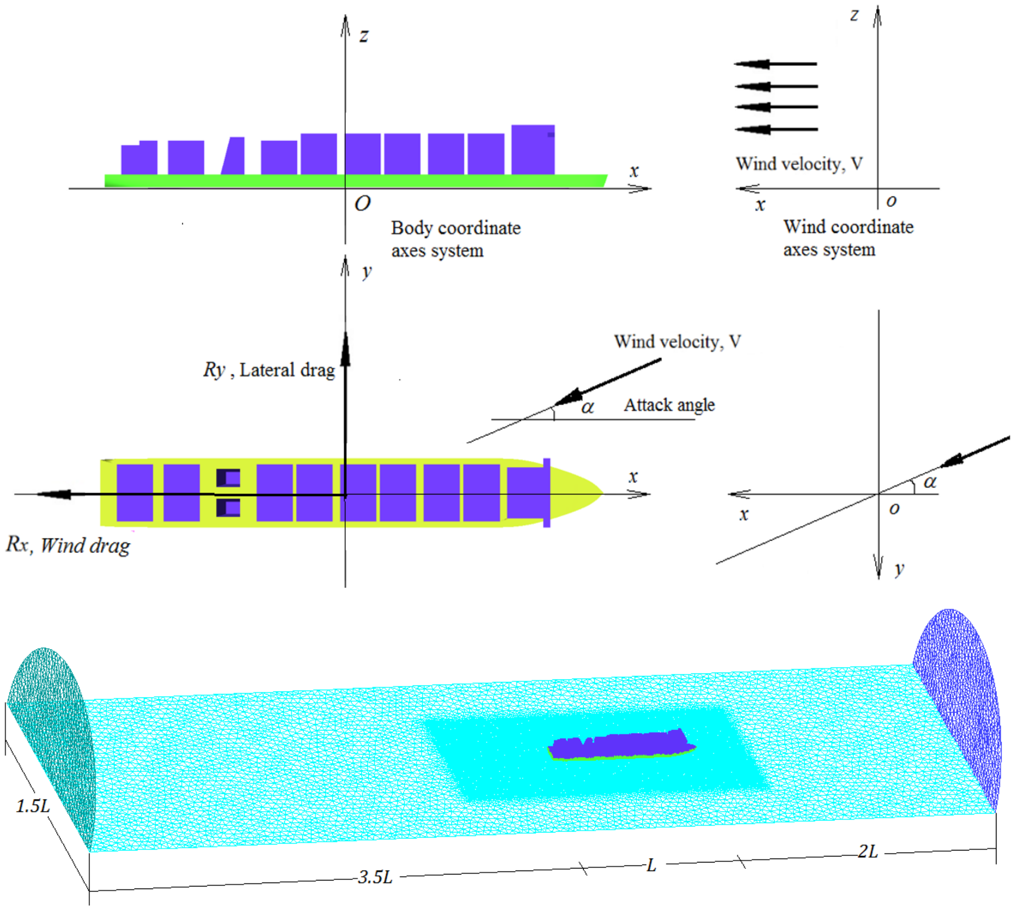

Figure 2. Computed domain and coordinate system

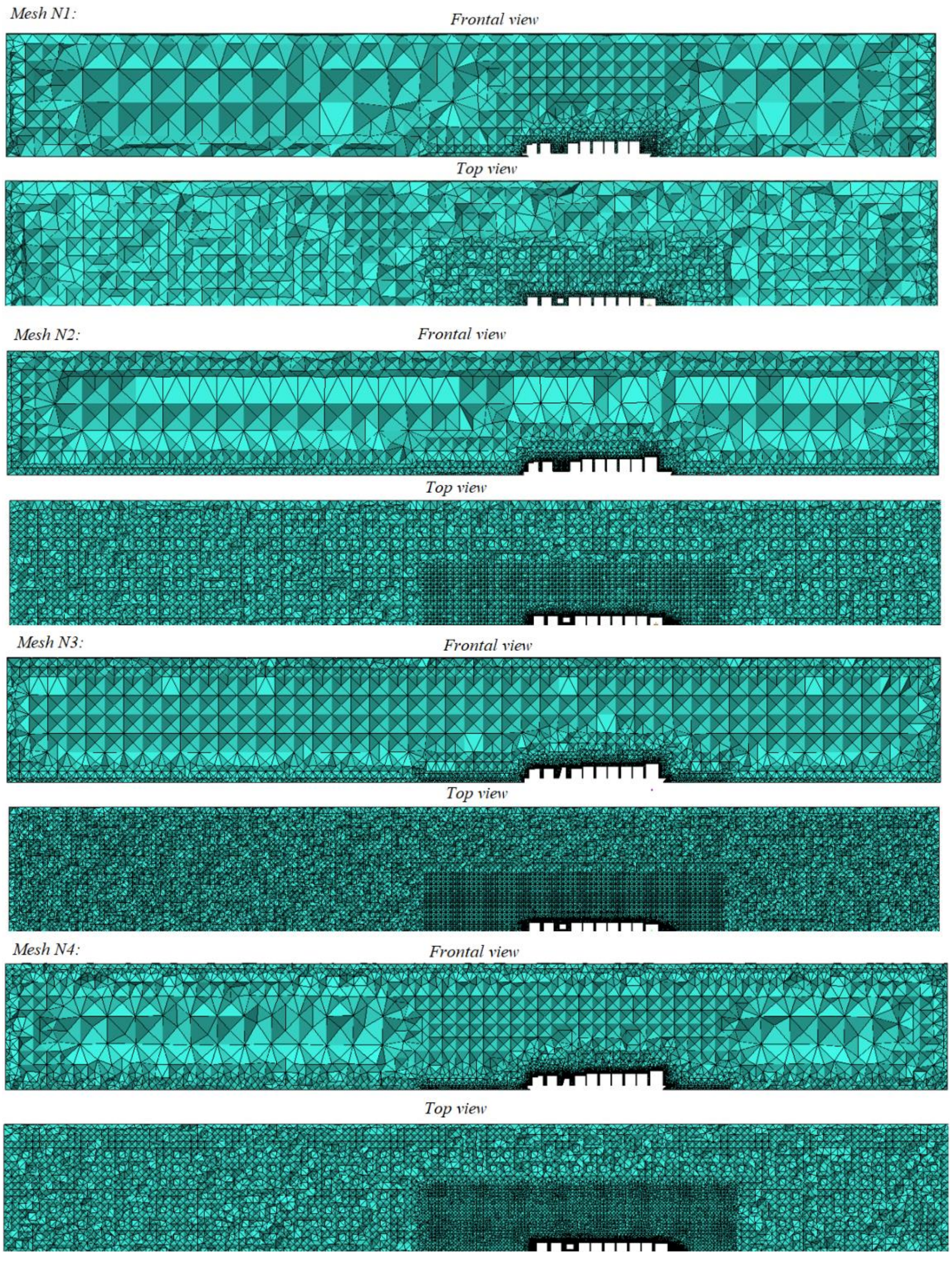

Figure 3. Mesh of the computed domain in the different mesh numbers

Table 2. The detailed mesh generated in computed domain

Name	Total elements	Minimum volume (m^3)	Maximum volume (m^3)	Minimum face area (m^2)	Maximum face area (m^2)
Mesh N1	126771	6.341e – 2	9.982e + 3	1.891e – 1	1.019e + 3
Mesh N2	434074	2.033e – 2	2.053e + 3	8.802e – 2	3.607e + 2
Mesh N3	1288325	1.425e – 3	1.599e + 3	1.875e – 2	2.966e + 2
Mesh N4	2178540	1.067e – 5	3.533e + 3	4.326e – 4	4.753e + 2
Mesh N5	3500900	2.214e – 6	1.178e + 3	1.171e – 4	2.350e + 2

Table 3. Computed condition setup for the problems

Name	Value	Unit
Turbulent viscous model	k-ε	-
Velocity inlet, V_∞	7.20	m/s
Pressure outlet, p_{out}	1.025×10^5	N/m^2
Air density, ρ	1.225	kg/m^3
Kinetic viscosity, υ	1.789×10^{-5}	kg/m s

RESULTS AND DISCUSSION

Effects of mesh number on cfd results

In this section, the CFD results of aerodynamics performances of the ship in computation with the different mesh numbers are shown. From results of comparison among cases with the different mesh numbers, effects of mesh number on aerodynamic performances of the ship are clear. Figures 4–6 show the pressure distribution around and over hull surface of the ship. Clear effects of mesh number on the results can be seen in these figures.

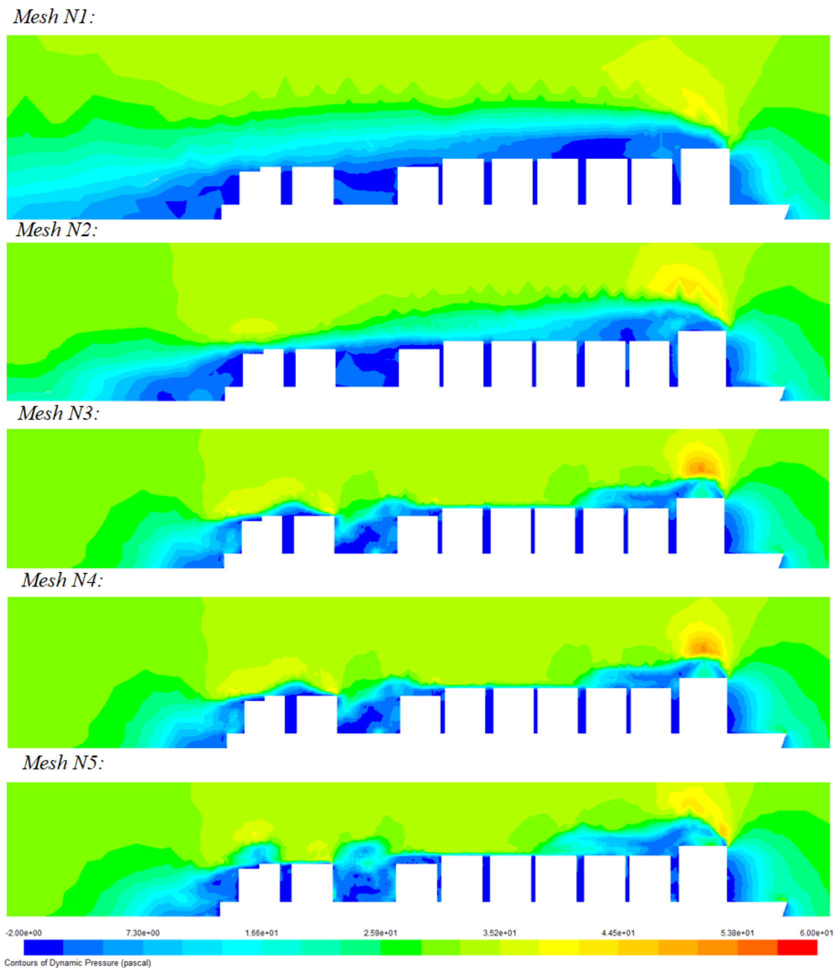

Figure 4. Dynamic pressure distribution around ship at central vertical plane of computed domain

The results as shown in the figures show the clear effects of mesh numbers on dynamic pressure distribution around hull in the computed domain. For the cases using meshes N1 and N2, a larger and longer separation area (blue color) can be seen at the back hull of the ship. At the regions around funnel and at the gap of the containers on deck, clear separation can be seen in the results of the meshes N3, N4 and N5. And, for the results of the meshes N3, N4 and N5 a slight difference in pressure distribution can be seen. From the results, clear effects of mesh numbers on pressure distribution at the frontal hull of the ship can also be seen. The high pressure area (red and yellow colors) around the frontal hull is clearly seen in the results of meshes N3, N4 and N5.

Figure 6 shows pressure distribution over a haft of frontal hull surface of the ship in the different mesh numbers. Figure 7 shows results of pressure distribution over the hull surface of the ship in the different meshes. Effects of mesh number on pressure distribution over hull surface of the ship can be seen clearly in the results.

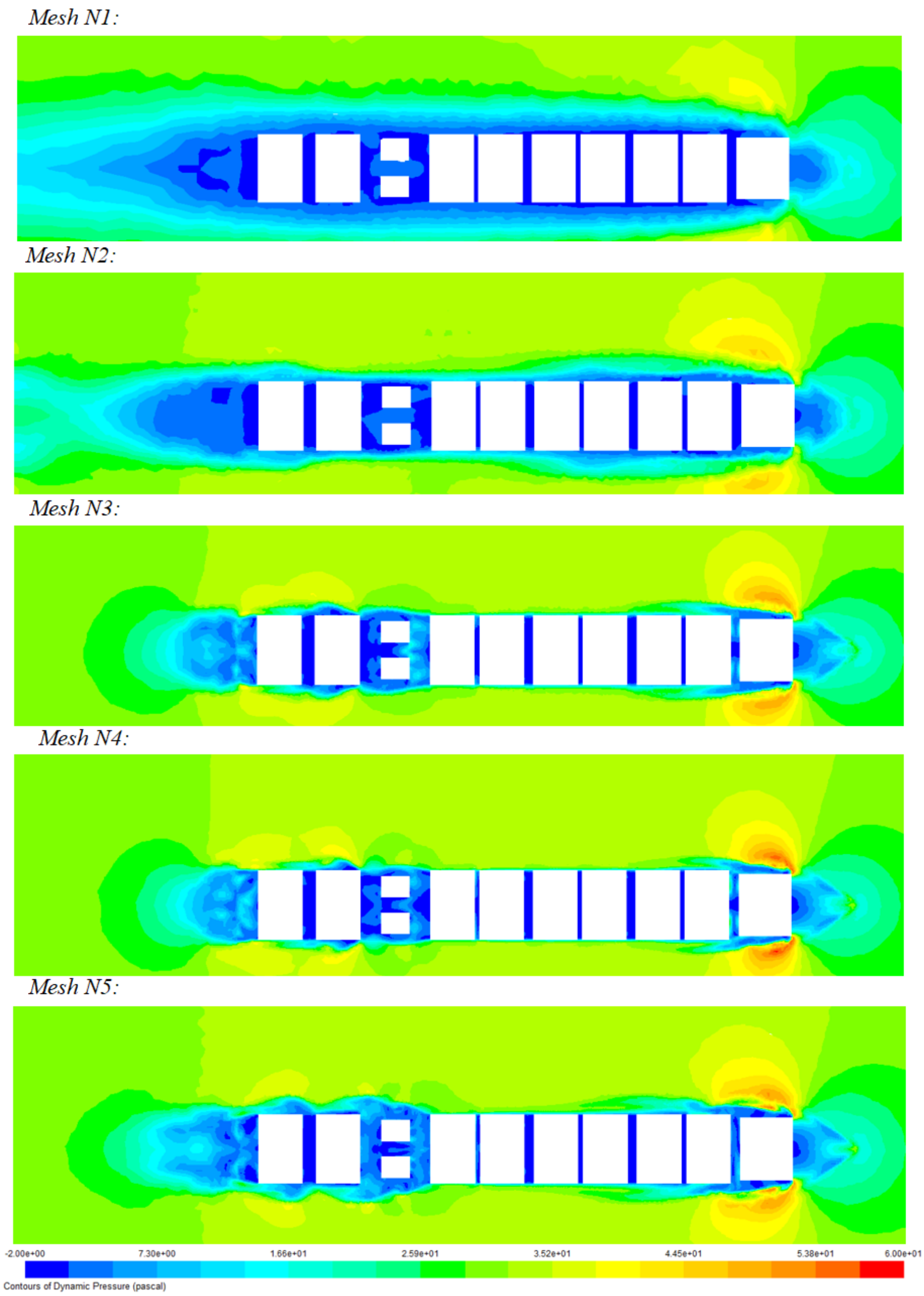

Figure 5. Dynamic pressure distribution around ship at horizontal plane of computed domain

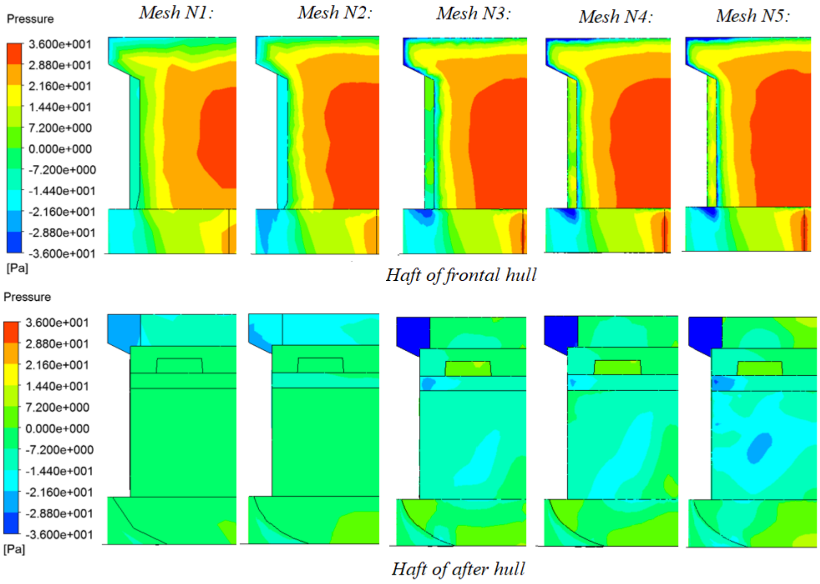

Figure 6. Pressure distribution over a haft of hull surface of the ship in the different meshes

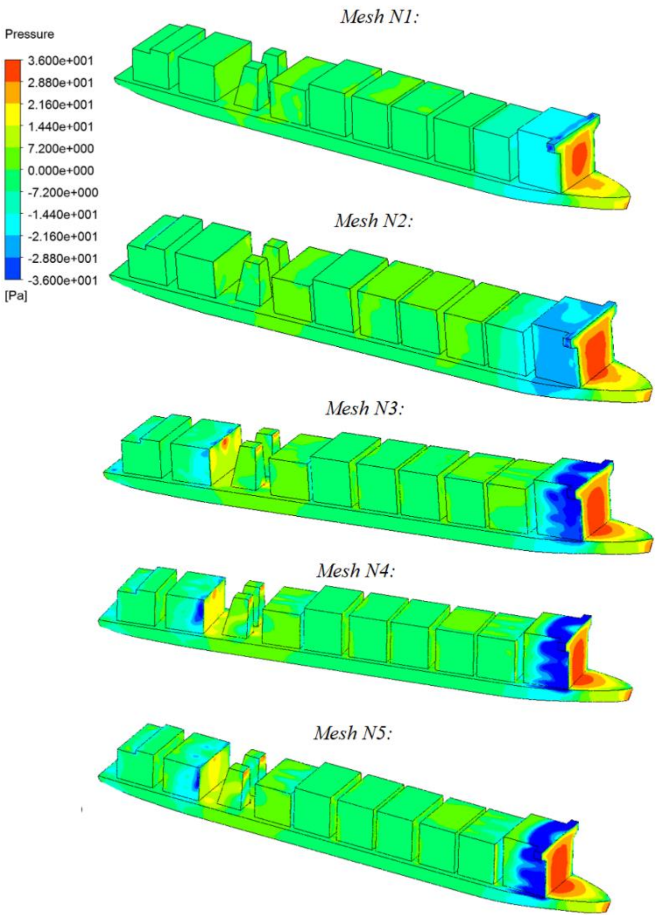

Figure 7. Pressure distribution over hull surface of the ship in the different meshes

Effects of mesh number on wind drag

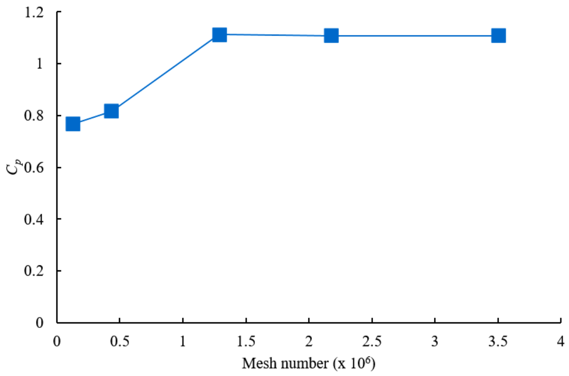

Figure 8. Viscous pressure wind drag coefficient (C_P) of the ship at different mesh numbers

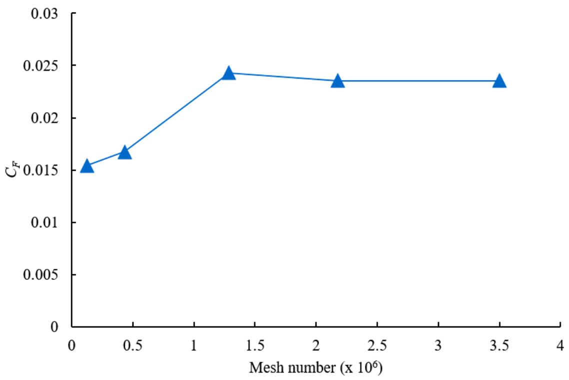

Figure 9. Viscous friction wind drag coefficient (C_F) of the ship at different mesh numbers

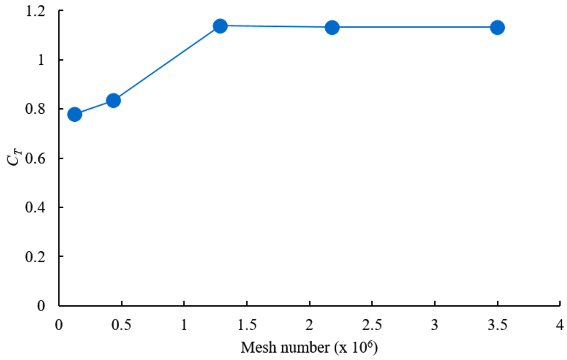

Figure 10. Total wind drag coefficient (C_T) of the ship at different mesh numbers

In this section, effects of mesh number on wind drag acting on the ship and the two viscous wind drag components acting on the hull will be computed by the CFD. Figures 8–10 show wind drag acting on the ship in the different mesh numbers.

The results presented in the figures 8–10 show that all drag components such as viscous pressure wind drag, viscous friction wind drag and total wind drag acting on the ship have the same form. When the mesh number is more than 1.2 million (Mesh N3), the wind drag coefficient does not change. From the results, we can see that effect of mesh number on wind drag acting on the ship reduces with the increasing mesh number. It comes to zero when the mesh number generated is larger enough. The detailed wind drag acting on the ship in the different mesh numbers is shown in table 4.

Table 4. Wind drag acting on the hull at different mesh numbers

Mesh number ($\times 10^6$)		Wind drag, R_x (N)			Coefficients, C_x		
		R_P	R_F	R_T	C_P	C_F	C_T
Mesh N1	0.127	10303.2	207.5	10510.7	0.7654	0.0154	0.7808
Mesh N2	0.434	11001.7	225.3	11227.1	0.8173	0.0167	0.8340
Mesh N3	1.288	14996.8	326.7	15323.5	1.1141	0.0243	1.1383
Mesh N4	2.179	14912.8	316.7	15229.5	1.1078	0.0235	1.1314
Mesh N5	3.501	14914.4	316.6	15231.0	1.1079	0.0235	1.1315

In the results, the wind drag components acting on the ship are defined by following equation [14].

$$C_x = \frac{R_x}{0.5\rho S V^2} \qquad (1)$$

Where: C_x is the wind drag coefficient; R_x is the wind drag acting on the hull, N; S is the frontal projected area, m^2; V is the velocity, m/s.

The total wind drag coefficient is defined by the following equation:

$$C_T = C_P + C_F \quad (2)$$

Where: C_P and C_F are the viscous pressure wind drag and viscous friction wind drag of the ship, respectively.

In the results as shown in the table 3, the wind drag acting on the ship increases with the increasing mesh number. However, the wind drag stays the same when the mesh number is more than 1.2 million (Mesh N3). Therefore, the effects of mesh number on wind drag acting on the ship and mesh convergence in computed aerodynamic performances of the ship decrease when increasing mesh number. Figure 11 shows mesh convergence curve in the computation of the aerodynamic performances of the container ship.

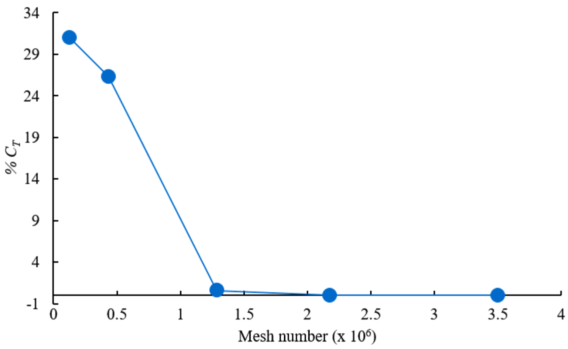

Figure 11. The mesh convergence curve on computation of aerodynamic performances of the ship

Figure 11 shows the mesh convergence curve in computation of the aerodynamic performances of the container ship. From the results, we can see that when mesh number increases up to 1.2 million, the effect of mesh number on wind drag acting on the ship hull drastically reduces and comes to zero. The obtained result is very useful in applied CFD computation of the aerodynamic performances and wind drag acting on the container ship.

CONCLUSIONS

In this research, the aerodynamic performances and wind drag acting on hull of the 1200TEU container ship have been investigated by the CFD. The obtained effects of mesh number on aerodynamic performances such as the pressure distribution around the ship hull and wind drag acting on the ship hull have been clearly found. The following are conclusive remarks of the paper:

1) By applying the CFD, the aerodynamic performances and wind drag acting on the 1200TEU container ship have been investigated. The obtained results of this study may be useful to design and calculate optimal aerodynamic performances for the container ship or any other types of ships having large above water surface hull form.

2) The obtained CFD results clearly show how the computing conditions affect the CFD results. Moreover, the obtained results are also important for the ship owner to find the way to reduce wind drag acting on the ship in marine transportation.

3) From the results, it can be seen that the effects of mesh numbers decrease when mesh number increases. For the full scale model 1200TEU container ship, the effect of mesh number decreases and comes to zero when the mesh numbers increase over 1.2 million.

Acknowledgements: This research is funded by Vietnam National Foundation for Science and Technology Development (NAFOSTED) under grant number 107.03-2019.302. The authors would like to thank for all the supports.

REFERENCES

[1] ITTC, R., 2011. procedures and guidelines: practical guidelines for ship CFD applications, 7.5.

[2] Janssen, W. D., Blocken, B., and van Wijhe, H., 2017. CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications. *Journal of Wind Engineering and Industrial Aerodynamics*, *166*, 106–116. https://doi.org/10.1016/j.jweia.2017.03.015.

[3] Watanabe, I., Van Nguyen, T., Miyake, S., Shimizu, N., and Ikeda, Y., 2016. A study on reduction of air resistance acting on a large container ship. *Proceeding of APHydro 2016*, 321–330.

[4] Nguyen, T. V., Shimizu, N., Kinugawa, A., Tai, Y., and Ikeda, Y., 2017.

Numerical studies on air resistance reduction methods for a large container ship with fully loaded deck-containers in oblique winds. In *Proceeding of VII International Conference on Computational Methods in Marine Engineering*, *MARINE* (pp. 1040–1051).

[5] Kim, Y., Kim, K. S., Jeong, S. W., Jeong, S. G., Van, S. H., Kim, Y. C., and Kim, J., 2015. Design and performance evaluation of superstructure modification for air drag reduction of a container ship. In *The Twenty-fifth International Ocean and Polar Engineering Conference*. *International Society of Offshore and Polar Engineers*.

[6] Andersen, I. M. V., 2013. Wind loads on post-panamax container ship. *Ocean engineering*, *58*, 115–134. https://doi.org/10.1016/j.oceaneng.2012.10.008.

[7] Ngo Van He, B. D. T., 2018. Effect of Accommodation and container on air resistance acting on hull of the container ship. In *The first international conference on fluid machinery and automation systems*. pp. 437–440.

[8] Wnęk, A. D., and Soares, C. G., 2015. CFD assessment of the wind loads on an LNG carrier and floating platform models. *Ocean Engineering*, *97*, 30–36. https://doi.org/10.1016/j.oceaneng.2015.01.004.

[9] He, N. V., and Ikeda, Y., 2013. A study on interaction effects between hull and accommodation on air resistance of a ship. In: *Proceedings of the 16th Japan Society of Naval Architects and Ocean Engineering*, (16), 281–284.

[10] Van He, N., Mizutani, K., and Ikeda, Y., 2016. Reducing air resistance acting on a ship by using interaction effects between the hull and accommodation. *Ocean Engineering*, *111*, 414–423. https://doi.org/10.1016/j.oceaneng.2015.11.023.

[11] Kha T. N, Tuan N. M, He N. V, 2018. Effect of an accommodation shape on aerodynamic performance and reduced air resistance acting on a cargo river ship. *Journal of Science and Technology, Thai Nguyen University*, *189*(13), 217–222.

[12] Van He, N., Mizutani, K., and Ikeda, Y., 2019. Effects of side guards on aerodynamic performances of the wood chip carrier. *Ocean Engineering*, *187*, 106217. https://doi.org/10.1016/j.oceaneng.2019.106217.

[13] Toan, N. C., and Van He, N., 2018. Effect of hull and accommodation shape on aerodynamic performances of a small ship. *Vietnam Journal of Marine Science and Technology*, *18*(4), 413–421. https://doi.org/10.15625/1859-3097/18/4/13292.

[14] Bertram, V., 2011. Practical ship hydrodynamics. *Elsevier*.

[15] Saydam, A. Z., and Taylan, M., 2018. Evaluation of wind loads on ships by CFD analysis. *Ocean Engineering*, *158*, 54–63. https://doi.org/10.1016/j.oceaneng.2018.03.071.

[16] He N. V, Loi L. N, Quang L., 2015. A study on improving economy efficiency of container ship by reducing resistance acting on hull. *The Transport Journal*, *56*, 217–219.

[17] Pope, S. B., 2001. Turbulent flows. *IOP Publishing*.