

Article

Interaction Effect between Hull and Accommodation on Wind Drag Acting on a Container Ship

Ngo Van He ^{1,*}, Ngo Van Hien ^{1,*} , Van-Thuan Truong ^{1,*}  and Ngoc-Tam Bui ^{2,3,*} ¹ School of Transportation Engineering, Hanoi University of Science and Technology, Hanoi 10000, Vietnam² School of Mechanical Engineering, Hanoi University of Science and Technology, Hanoi 10000, Vietnam³ College of Systems Engineering and Science, Shibaura Institute of Technology, Tokyo 135-8548, Japan

* Correspondence: he.ngovan@hust.edu.vn (N.V.H.); hien.ngovan@hust.edu.vn (N.V.H.);

thuan.truongvan@hust.edu.vn (V.-T.T.); tambn@shibaura-it.ac.jp (N.-T.B.)

Received: 3 October 2020; Accepted: 9 November 2020; Published: 16 November 2020



Abstract: In this paper, we present our research on applying the commercial Computational Fluid Dynamics (CFD) code to investigate interaction effect between hull and accommodation on wind drag acting above the water hull surface of a full scale 1200 TEU container ship. With this purpose, aerodynamic performances and wind drag acting on the ship hull with and without accommodations have been computed. Analyzing the obtained CFD results, the interaction effect between hull and accommodation on aerodynamic performances and wind drag acting on the ship have been found. Various new accommodation shapes have been proposed for the original ship to reduce the interaction effect on wind drag. A drastic reduction in the interaction effect between hull and accommodation on wind drag acting on the ship has been achieved and the obtained results have been shown in this paper.

Keywords: CFD; interaction effect; wind drag; container ship; aerodynamic

1. Introduction

Nowadays, research on reducing wind resistance acting on ship hull to save fuel consumption and improve economic efficiency is still an attractive topic in the field of marine transportation. This is especially true for container ships with a large windward area, due to their high above-water superstructure and multi-external forms, resulting from the position of containers on the deck, where wind drag accounts for a large percentage of the total resistance. Therefore, researchers around the world are of the opinion that research on reducing wind drag acting on container ships as well as the ships with high superstructure and or large windward areas is more and more important.

It goes without saying that in recent years, a large number of studies on the reduction in wind drag acting on ships' hulls as well as on optimal aerodynamic hull forms, the effect of side covers and domes on wind drag, the interaction effect between hull and accommodation, the effect of strong wind on resistance, the safety of the ship in strong wind, and so on, have been published. A comprehensive review is given as follows:

There are many studies on applying commercial Computational Fluid Dynamics (CFD) to solve the aerodynamic performances of the ships. The most important point of the research is that: the CFD has been a popular and useful tool to solve ships' aerodynamic problems with fairly good accuracy. Using CFD modelling, various types of modified hull shapes with reduced wind drag acting on the ship have been developed [1–7].

Janssen, W.D. et al. (2017) presented a study on using both the commercial CFD code and wind tunnel model test to study aerodynamic performances and wind drag acting on a container ship hull. The 3D steady Reynolds Averaged Navier Stokes (RANS) CFD simulation method was used to solve

the problem. The author concluded that CFD results were in good agreement with the experimental ones and that the CFD could be used to develop a new hull shape with reduced wind drag acting on the ship. The average absolute difference of wind drag obtained in CFD simulation and tunnel test ranged from 37.9% for a box shaped representation of the ship to only 5.9% for a more detailed model. Modelling the spaces in between container stacks decreased the average total wind load considerably. The average absolute difference of total wind load was 10.4%. Using a slender ship hull instead of a blunt ship one decreased total wind load by 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 shown by comparing the CFD results obtained in the narrow domain with those of the wider domain [1].

Andersen, I.M.V. (2013) presented a study that investigates the influence of the container configuration on the deck of a 900TEU container ship on wind drag by wind tunnel test with scale model of 1:450. The author concluded that the results serve as an indication that the magnitude of wind force acting on a large container ship depends on the container configuration on deck rather than the purpose of assessing the full scale wind resistance of a given container ship. For the reduced longitudinal force in relative wind, it was advantageous to make the container configuration as smooth as possible and streamlining could reduce the force in the head wind. However, streamlining of the configuration on the aft deck was a trade-off as it increased the yaw moment compared to full load on the aft deck. The high container stacks in the configuration appeared to increase the longitudinal force more than the low or empty container bays in the configuration. The transverse force of concern in beam wind depends largely on the side area of the ship, and for a fully loaded ship it could be reduced by reducing the stacking height of the outermost stacks. The yaw moment in relative wind could be reduced by achieving full load on the aft of the ship, and it is possible to reduce the resistance through a combination of drift and increased rudder angle. Moreover, the author gave a general recommendation that the external form of the ship should be made as smooth as possible. In addition, it is important that the center of gravity of the side area was as far from the aft as possible [2].

Fujiwara, T. et al. (2009) presented an experimental investigation on wind force for a container ship with various external forms due to different positions of containers on the deck, in order to study aerodynamic specifications. Based on the obtained experimental results, an estimation method for wind load acting on the container ship has been proposed. In the tunnel experiment, a scale model of 1.5 m long, a mean wind velocity of 25 m/s and a Reynolds number of about 2.4×10^6 have been selected. Under these conditions, the flow field was turbulent and the drag coefficients were independent of the Reynolds number. The authors have shown that a good agreement between their proposed estimation method the tunnel experimental test has been obtained. They also showed that the new method is important for the calculation of speed as well as other characteristics in the operation stage of container ships [3].

In a study presented by Kim, Y. et al. (2015), several design concepts and devices on the superstructure of a container ship have been suggested and tested in a wind tunnel to estimate the wind drag reduction. The authors have also used CFD with the RANS simulation method to estimate wind drag acting on the ship. The results show that the gap protectors between container stacks and visors in front of upper deck have been found to be the most effective means for reducing wind drag acting on the ship. The CFD results agreed with the experimental measurements in the wind tunnel, and the wind drag acting on the modified ships could be reduced by up to 56% in the wind direction angle from 0 to 50 degrees [4]. Other researchers [5–7] also presented results on using CFD and experimental tests to develop a modified hull shape with reduced wind drag acting on a container ship. The authors have proposed modified hull shapes with attached side covers, a center wall, a “T” center wall and a dome at the bow deck of the container ship to reduce wind drag. By using side covers and a center wall, the container ship could reduce wind drag by up to 40% in the head wind direction. A dome at the bow of the ship could reduce the total wind drag acting on the container ship by up to 30% at wind direction angles of less than 30 degrees.

Other available studies focused on the aerodynamic performances and wind drag acting on different types of ships and offshore using both the wind tunnel measurement and CFD simulation. Jun, S. et al. (2020) investigated the resistance performance of ships, using the air resistance correction method [8]. Wnęk, A.D. et al. (2011, 2015) focused on the wind load acting on a LNG carrier with a specific geometrical hull shape [9,10]. Saydam, A.Z. et al. (2018) conducted an evaluation of wind load acting on ships by CFD analysis [11]. He, N.V. et al. (2013, 2016, 2019) presented research on the interaction effect between hull and accommodation on aerodynamic performances and wind drag acting on a wood chip carrier hull [12–14]. Sugata, K. et al. (2010) studied the reduction in wind drag acting on the hull of a Non Ballast water Ship (NBS) [15], and so on [16–19].

The mesh of computed domain effects on the CFD results of a ship has been reported in many previous published works. Viola, I.M. et al. (2009) tested two different mesh numbers of 1 million and 6.5 million tetrahedral elements. The obtained CFD results, i.e., the wind forces, have been compared with those of the experimental ones. It has been observed that the wind forces computed by the turbulent viscous model Realizable $k-\varepsilon$ was in good agreement with the experimental results and the differences between computed results in the two mesh numbers were lower than 5% [20]. Wnęk, A.D. et al. (2011, 2015) used two models of a floating LNG platform and an LNG carrier in the 1:400 scale to conduct an experimental tunnel test and CFD computation to investigate the wind forces acting on the models. The turbulent viscous model $k-\omega$ SST and three different mesh generation techniques, which were CFD hexa ($y^+ = 5$), CFD tetra (no prism) and CFD tetra y^+ ($y^+ = 0.1$), were used for CFD computation. The CFD results of CFD tetra ($y^+ = 0.1$) showed the best agreement with the experimental data obtained in the wind tunnel test and the tetrahedral mesh was shown to be of good quality for the meshing of complex geometric shapes [9,10]. Saydam, A.Z. et al. (2017) used the mesh numbers from 1 million to 5 million elements of unstructured tetrahedral mesh to test CFD results independent of mesh number for computed wind forces acting on the ships. It has been shown that the CFD results were in good agreement with the experimental data obtained in wind tunnel test, and mesh independence of wind forces was up to more than 2 million elements [11]. Watanabe, I. et al. (2016) and Trieu, N.V. et al. (2017a, 2017b) used unstructured tetrahedral mesh with three mesh numbers of 2.2 million, 2.6 million and 3.8 million elements, with y^+ of less than 25, to test CFD results independent of mesh. They showed that the CFD results given were in good agreement with the experimental data, and about 2% of the wind drag coefficient was different from the mesh number [5,7,21]. In our previous paper [12–14], we used the same unstructured tetrahedral mesh to investigate the wind drag acting on a wood chip carrier. It has been shown that the CFD results were in good agreement with those of the experimental data obtained in the towing tank test conducted at Osaka Prefecture University, Japan.

In this paper, a commercial CFD code ANSYS-Fluent has been used to investigate the interaction effect between the hull and accommodation of a container ship on wind drag. By applying the CFD, aerodynamic performances and wind drag acting on a 1200 TEU container ship have been computed for two different cases, namely the hull with and without accommodation on its deck, to determine the interaction effect between the hull and accommodation of the ship. From the obtained results, several new hull shapes and frontal accommodation shapes, with reduced interaction effects between hull and accommodation on wind drag acting on the ship, have been proposed in this paper.

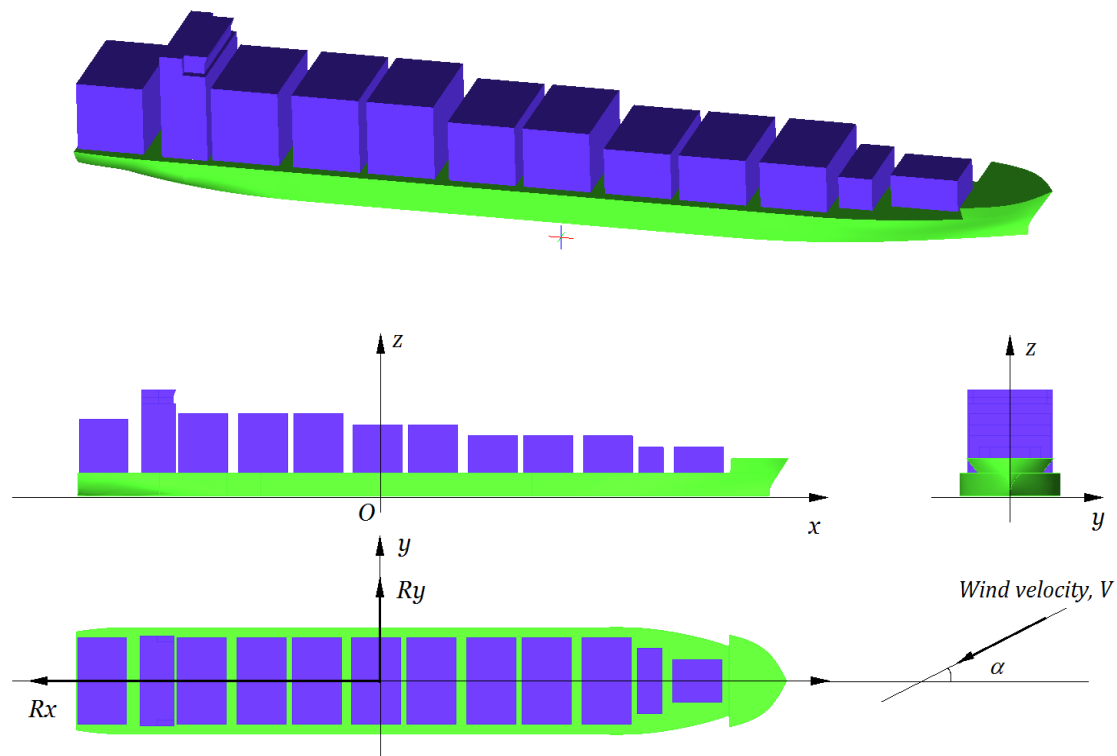
2. Original Model and Computed Domain

2.1. Model Ship Used for Computation

The full scale 1200 TEU container ship, which is designed with an accommodation located at the aft, has been used as the reference model for computation. The principal particulars of the ship are given in Table 1, and Figure 1 shows the designed hull form of the original model.

Table 1. Principal dimensions of the 1200TEU 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
$A_{F(x)}$	Frontal projected area	541.29	m^2
$A_{F(y)}$	Lateral projected area	2483.23	m^2
C_b	Block coefficient	0.68	-
α	Wind attack angle	0	degree
V	Wind velocity	12–18	knot
Re	Reynolds number	$(5.8\text{--}8.7) \times 10^7$	-

**Figure 1.** Model of the ship and coordinate system.

2.2. Computational Domain and Boundary Conditions

In this research, all steps for the designed simulation domain, meshing and setting of the boundary conditions have been done as per the user guide published by the International Towing Tank Conference (ITTC) manual for using CFD, [22–26] and our previous publications [12–14]. In this study, the computational domain has been designed with $6.5 L$, $2.2 L$ and $0.75 L$ instead of 1200 m length, 450 m breadth and 150 m height. Mesh of the computed domain has been used as unstructured mesh with a triple prism layer and y^+ has been taken to be less than 5. The popular turbulent viscous model $k\text{--}\epsilon$ for unsteady flow has been used. The velocity inlet is given at the inlet as the wind velocity, the pressure outlet is given at the outlet of the computed domain. The bottom, top and sides are given at the walls. To meet realistic case pressure, the outlet makes the open air condition possible, the limited dimension of the computed domain must be designed to be large enough. However, the dimensions of the computed domain has an effect on the mesh number and also an effect on running time. If the top and side of the domain is far enough, the effects on the CFD result become small. Therefore, we must design an appropriate domain and boundary condition. In computation, the time size step is of 0.005 s and

6000 time steps. All cases have been done by an Intel Xeon Gold 6138 2.0 GHz computer, with 3.7 GHz Turbo, 20 C with 64 GB of RAM. Figure 2 shows computed domain and mesh used for simulation.

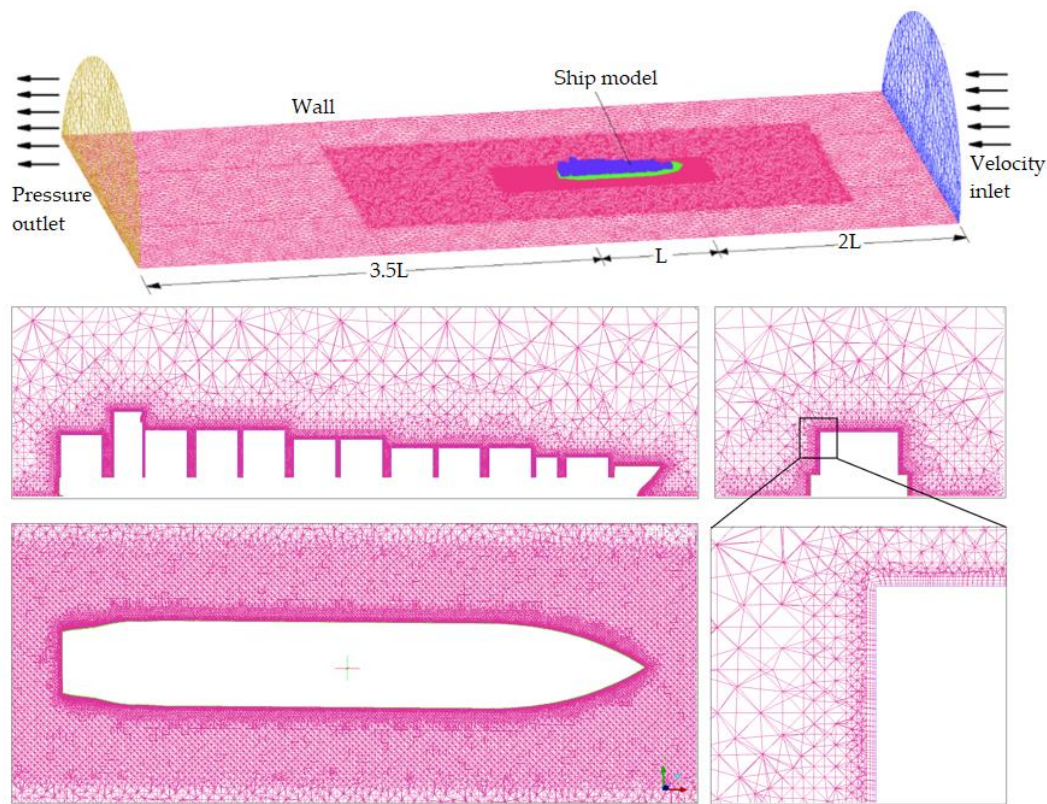


Figure 2. Computational domain and mesh.

2.3. Covegence of Mesh Using for Computation

In this research, the effect of mesh numbers generated in the computed domain on aerodynamic performance and wind drag acting on the original container ship has been investigated. The computed domain is shown in Figure 2, where twelve different mesh numbers have been used. The mesh numbers are from 0.05 to 21.52 million elements and value of y^+ is taken from 1.08 to 502.49, respectively. The meshes have been used to compute the aerodynamic performances of the ship under the same conditions. Analyzing the obtained CFD results, the effect of mesh numbers on aerodynamic performances and wind drag as well as the state of independence of mesh numbers have been found. The computation was done in head wind condition at a wind velocity of 14 knot instead of a Reynolds number of 6.73×10^7 . Table 2 shows a detailed mesh of the computed domain.

Table 2. The detailed meshes used in computation.

No.	y^+	Total Elements ($\times 10^6$)	Minimum Face Area, m^2 ($\times 10^{-2}$)	Maximum Face Area, m^2 ($\times 10^2$)	Minimum Volume, m^3 ($\times 10^{-3}$)	Maximum Volume, m^3 ($\times 10^3$)
1	502.49	0.0575	64.3907	3.1801	414.689	1.3602
2	358.92	0.1504	29.3068	2.3441	117.371	0.9033
3	215.35	0.2267	2.3832	2.0662	6.66706	0.8561
4	143.57	0.3881	2.9348	4.6849	4.32068	2.8079
5	71.78	1.0761	0.2197	3.6252	0.08926	1.8030
6	50.25	1.6178	0.5163	2.6564	0.49978	0.9891
7	35.89	2.4881	0.9272	3.0937	0.99869	2.1242
8	21.54	3.4288	1.1114	1.4511	0.81512	0.6278

Table 2. Cont.

No.	y+	Total Elements ($\times 10^6$)	Minimum Face Area, m^2 ($\times 10^{-2}$)	Maximum Face Area, m^2 ($\times 10^2$)	Minimum Volume, m^3 ($\times 10^{-3}$)	Maximum Volume, m^3 ($\times 10^3$)
9	14.36	5.4566	0.0548	4.8017	0.01296	2.7449
10	7.18	8.9854	0.0121	1.0236	0.00326	0.6128
11	3.59	12.3683	0.0092	1.2763	0.00163	0.7629
12	1.08	21.5228	0.0016	2.0125	0.00032	0.6293

Tables 3 and 4 show detailed results of wind drag, and the interaction effect of the mesh number on wind drag acting on the ship. Figure 3 shows the curve of the effect of the mesh number on wind drag, as well as the state of the mesh number becoming independent of the wind drag acting on the ship obtained in the computation.

Table 3. Wind drag acting on the original container ship in the different mesh numbers.

Total Elements ($\times 10^6$)	Wind Drag, R_x (N)			Wind Drag Coefficient, C_d		
	Acc	Hull	Total	Acc	Hull	Total
0.0575	13,464.3	3081.8	16,546.1	0.7829	0.1792	0.9621
0.1504	13,181.5	2910.2	16,091.7	0.7665	0.1692	0.9357
0.2267	13,350.4	2180.2	15,530.6	0.7763	0.1268	0.9031
0.3881	13,328.6	2025.4	15,353.9	0.7750	0.1178	0.8928
1.0761	13,245.9	1917.1	15,163.0	0.7702	0.1115	0.8817
1.6178	13,210.3	1880.2	15,090.5	0.7682	0.1093	0.8775
2.4881	13,115.6	1878.3	14,993.9	0.7627	0.1092	0.8719
3.4288	13,144.7	1837.1	14,981.7	0.7644	0.1068	0.8712
5.4566	13,142.3	1842.1	14,984.4	0.7642	0.1071	0.8713
8.9854	13,040.1	1798.4	14,838.5	0.7583	0.1046	0.8628
12.3683	13,001.2	1792.8	14,794.0	0.7560	0.1043	0.8603
21.5228	13,012.0	1778.1	14,790.1	0.7566	0.1034	0.8600

Table 4. Effect of mesh number on wind drag acting on the ship, at $Re = 6.73 \times 10^7$, in head wind.

Total Elements ($\times 10^6$)	Different of Wind Drag, ΔR_x , %			Different of Wind Drag Coefficient, ΔC_d , %		
	Acc	Hull	Total	Acc	Hull	Total
0.0575	3.36	42.30	10.61	3.36	42.30	10.61
0.1504	1.29	38.90	8.09	1.29	38.90	8.09
0.2267	2.53	18.44	4.77	2.53	18.44	4.77
0.3881	2.37	12.21	3.67	2.37	12.21	3.67
1.0761	1.77	7.25	2.46	1.77	7.25	2.46
1.6178	1.50	5.43	1.99	1.50	5.43	1.99
2.4881	0.79	5.34	1.36	0.79	5.34	1.36
3.4288	1.01	3.21	1.28	1.01	3.21	1.28
5.4566	0.99	3.48	1.30	0.99	3.48	1.30
8.9854	0.22	1.13	0.33	0.22	1.13	0.33
12.3683	0.08	0.82	0.03	0.08	0.82	0.03
21.5228	0.00	0.00	0.00	0.00	0.00	0.00

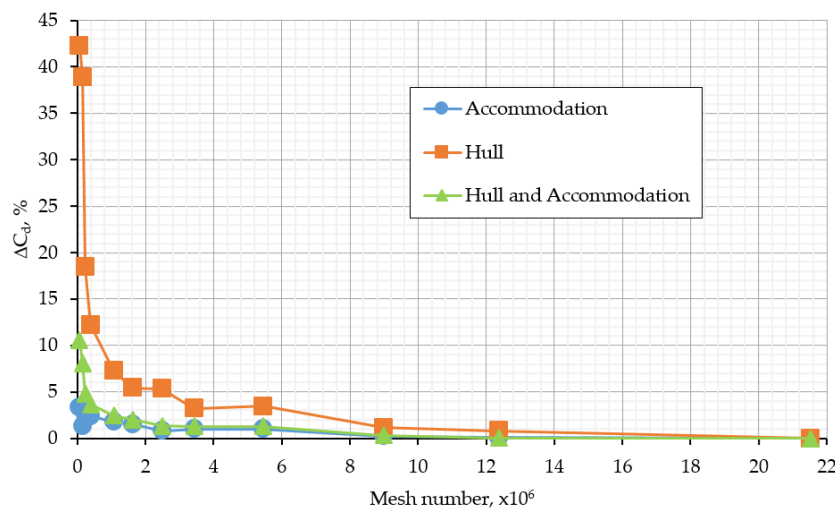


Figure 3. Effect of mesh number on wind drag acting on the ship in head wind, at $Re = 6.73 \times 10^7$.

Tables 3 and 4 clearly show wind drag acting on the original ship in the different computed mesh numbers, where the wind drag acting on the ship is defined by the following equation [27,28].

$$C_d = \frac{R_x}{0.5\rho A_{F(x)} V^2} \quad (1)$$

where:

C_d is the wind drag coefficient.

R_x is the wind drag acting on the hull, N.

$A_{F(x)}$ is the frontal projected area, m^2 .

V is the ship velocity, m/s.

The results given in the Tables 3 and 4 show that the wind drag acting on the ship decreases slowly with increasing mesh number, although the wind drag is slightly different when the $y+$ is less than 50 and retains the same value when the $y+$ is less than 7. The effect of mesh number on wind drag acting on the ship decreases with the increasing of the mesh number. The difference of wind drag is around 5% when the $y+$ is less than 50 and the same value when $y+$ is less than 7. When the $y+$ increases to over 50 but continues to be less than 500, the difference in wind drag increases by up to 42%, as shown. This is important in studying aerodynamic performances using the CFD tool. Apart from paying attention to calculating the wind drag acting on the ship, it is very useful to get a clear pattern of pressure distribution around the hull to understand how the flow has an effect on the hull, as well as factors affecting the wind drag, and so on.

Figure 3 shows the effect of mesh number on wind drag acting on the ship as well as the mesh convergence curve obtained in computation. From the results shown, it can be seen that when the mesh number increases to over two million instead of a $y+$ of less than 50, the effect of the mesh number on total wind drag acting on the ship drastically reduces and becomes zero when the $y+$ is less than 7, as shown. The obtained result is useful in the application of CFD to investigate aerodynamic performances of the ship.

3. Interaction Effect between Hull and Accommodation

In this section, aerodynamic performances of the ship have been computed in two different cases, namely the hull with and without accommodation on its deck. Analyzing the obtained CFD results for the two cases, the interaction effect on aerodynamic performances of the hull and accommodation in

head wind has been obtained. Figure 4 shows the pressure distribution around the ship in the two computed cases at a Reynolds number of 6.73×10^7 in head wind.

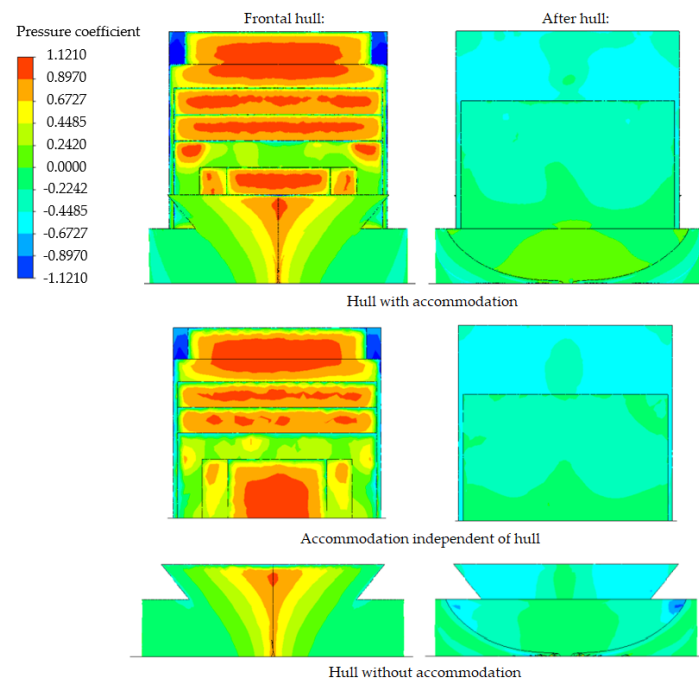


Figure 4. Pressure coefficient distribution over the hull surface of the model in the two computed cases at a Reynolds number of 6.73×10^7 , in head wind.

Analyzing the results as shown in Figure 4, the red and yellow regions show high pressure and the blue region shows lower pressure acting on the model. Hence, the effect of hull and accommodation on pressure distribution over the hull surface of the ship could be evaluated. The interaction effect between hull and accommodation may also be determined by comparison of the wind drag acting on the hull and accommodation in both of the computed cases, as shown.

In this research, interaction effect between hull and accommodation on wind drag acting on the ship has been determined by the following Equation (2) [12–14]:

$$\Delta C_d, \% = \frac{C_d(\text{Hull with Acc}) - C_d(\text{Independent Hull and Acc})}{C_d(\text{Independent Hull and Acc})} 100\% \quad (2)$$

where:

$\Delta C_d, \%$ is the interaction effect between hull and accommodation.

C_d (Hull with Acc) is wind drag coefficient acting on the hull with accommodation.

C_d (Independent Hull and Acc) is total wind drag coefficient acting on hull without accommodation and independent accommodation.

Tables 5–7 show, in detail, the wind drag acting on the hull, the accommodation, and the interaction effect between the hull and accommodation on the wind drag of the ship in head wind.

Analyzing the results shown in the Tables above, the wind drag acting on hull part and accommodation part in the case of the hull with accommodation is less than that of the case of accommodation independent of hull, by up to 42%. The total wind drag acting on the ship in the case of hull with accommodation is also less than that of the independent hull by up to 10%. For the wind drag acting on the hull part and the accommodation part, the interaction effect is about 42% for the hull part and 4% for the accommodation part. The interaction effect between the hull and accommodation on wind drag is around 10%, as shown.

Table 5. Wind drag acting on the hull with accommodation.

$R_e \times 10^7$	Wind Drag, R_x (N)			Wind Drag Coefficients, C_d		
	Acc	Hull	Total	Acc	Hull	Total
5.77	9906.00	1438.69	11,344.69	0.7844	0.1139	0.8983
6.73	13,021.11	1793.28	14,814.39	0.7572	0.1043	0.8614
7.69	17,125.29	2417.25	19,542.54	0.7622	0.1076	0.8697
8.65	22,089.69	3052.63	25,142.32	0.7766	0.1073	0.8839

Table 6. Wind drag acting on an independent hull and independent accommodation.

$R_e \times 10^7$	Wind Drag, R_x (N)			Wind Drag Coefficients, C_d		
	Acc	Hull	Total	Acc	Hull	Total
5.77	10,321.21	2329.01	12,650.22	0.8173	0.1844	1.0017
6.73	13,128.19	3100.13	16,228.31	0.7638	0.1804	0.9442
7.69	17,162.62	4012.14	21,174.76	0.7643	0.1787	0.9429
8.65	22,398.21	5051.24	27,449.45	0.7879	0.1777	0.9656

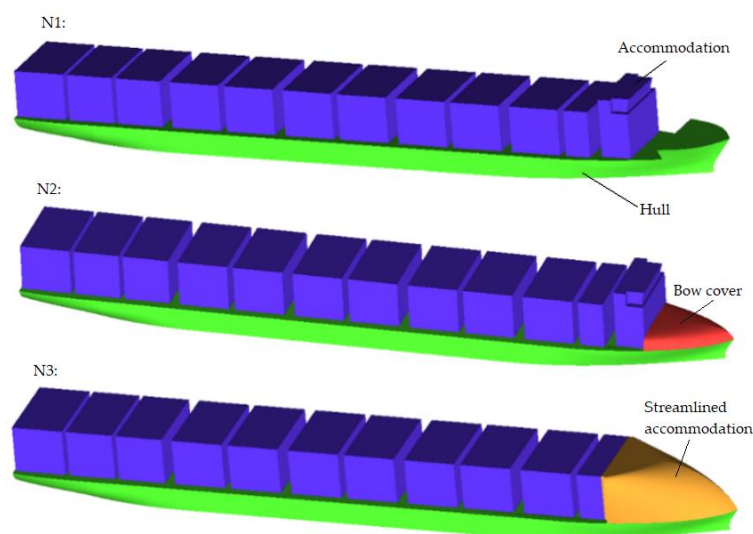
Table 7. Interaction effect between hull and accommodation on wind drag.

$R_e \times 10^7$	Different Wind Drag, ΔR_x , %			Difference, ΔC_d , %		
	Acc	Hull	Total	Acc	Hull	Total
5.77	4.02	38.23	10.32	4.02	38.23	10.32
6.73	0.82	42.15	8.71	0.87	42.19	8.77
7.69	0.22	39.75	7.71	0.28	39.79	7.76
8.65	1.38	39.57	8.41	1.44	39.60	8.46

4. Reduced Interaction Effect on Wind Drag Acting on the Container Ship

4.1. Proposed Accommodation Shape for the Ship

The original container ship has been designed with an accommodation located at the aft of the ship. In this section, three new hull forms with an accommodation located at the frontal part of ship have been proposed. Three models with frontal accommodation, named N1, N2 and N3, have been used for computation to determine the interaction effect on aerodynamic performances of the proposed streamlined hull shapes. The dimensions of the models are the same ones as those of the original ship shown in Table 1. The computation for all ships have also been done under the same conditions as shown in the Section 2. Figure 5 shows the proposed new models N1, N2 and N3.

**Figure 5.** Newly proposed shapes for the ship with frontal accommodation N1, N2 and N3.

In the proposed models, the N1 has the same accommodation shape of the original ship but is located at frontal hull, the N2 has a streamlined bow cover, and the N3 has a streamlined accommodation shape and it is located at frontal hull. Computation has been done for all the models under the same conditions to investigate the aerodynamic performances of the ships.

4.2. Reduced Interaction Effect of the New Hull Shapes on Pressure Distribution

In this section, the effect of the streamlined hull shapes on aerodynamic performances of the proposed container ships have been investigated by the CFD. Figures 6–11 show pressure distribution around and over the hull surface of the ships. Analyzing the obtained CFD results of pressure distribution around and over hull surfaces of the ships, the reasons for the increasing interaction effect between hull and accommodation have been clearly found.

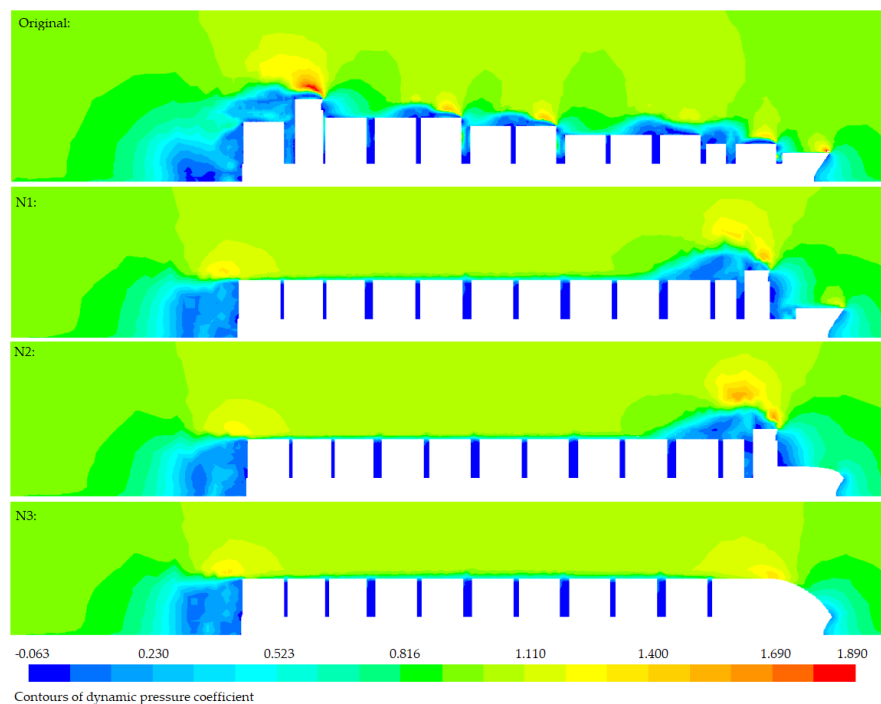


Figure 6. Dynamic pressure coefficient distribution around the ships at the center plane ($y/L = 0$) in head wind, at $Re = 6.73 \times 10^7$.

In the above results, the blue region shows low dynamic pressure and, consequently, high static pressure acting on the ship. The results, as shown in Figure 6, clearly show reduced separation flow at the lower dynamic pressure region (blue color region) around the streamlined hull shape N3. Figure 7 shows dynamic pressure distribution around the ships at several horizontal planes of the computed domain.

The results, as shown in Figure 7, clearly show the effects of the accommodation shapes on the dynamic pressure distribution around the ships. The separation flow regions at the low dynamic pressure regions (blue color regions) at the aft of the accommodation N2 and N3 have been drastically reduced, as shown. Figure 8 shows the dynamic pressure distribution around several cross sections of the computed domain of the ships.

The results obtained in the above figures clearly show differences of dynamic pressure distribution around the ships. A drastic reduction in low dynamic pressure region (blue color) could be seen around the hull shapes of N2 and N3, as shown. These results clearly show the reduced interaction effect on ship aerodynamic performance of streamlined accommodation shapes which are located at

frontal hull of the ship. There is a reason why a suitable frontal accommodation should be installed to reduce wind drag acting on the ship hull.

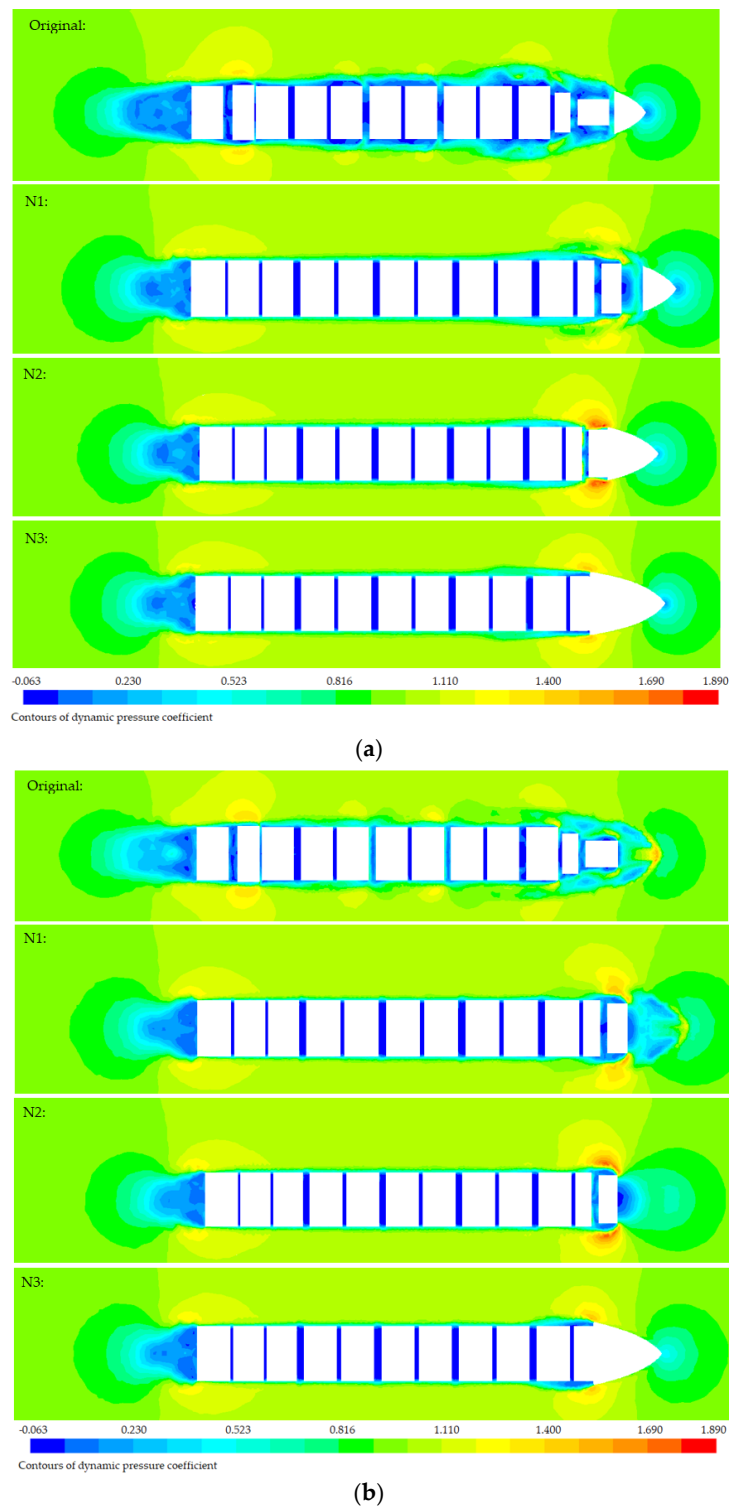


Figure 7. Dynamic pressure coefficient distribution around the ships at the horizontal plane in head wind, at $Re = 6.73 \times 10^7$. (a) $z/L = 0.08$ and (b) $z/L = 0.11$.

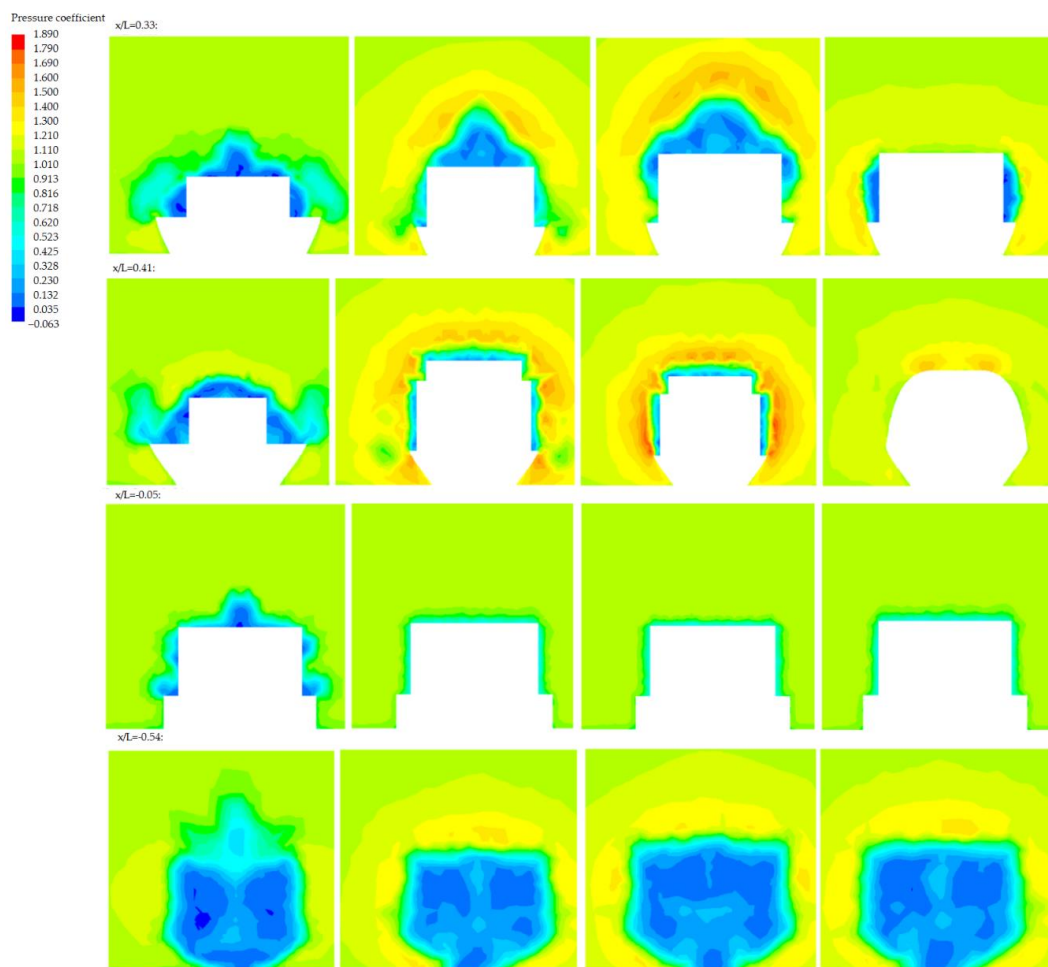


Figure 8. Dynamic pressure coefficient distribution around the ships at various cross sections of the computed domain in head wind, at $Re = 6.73 \times 10^7$.

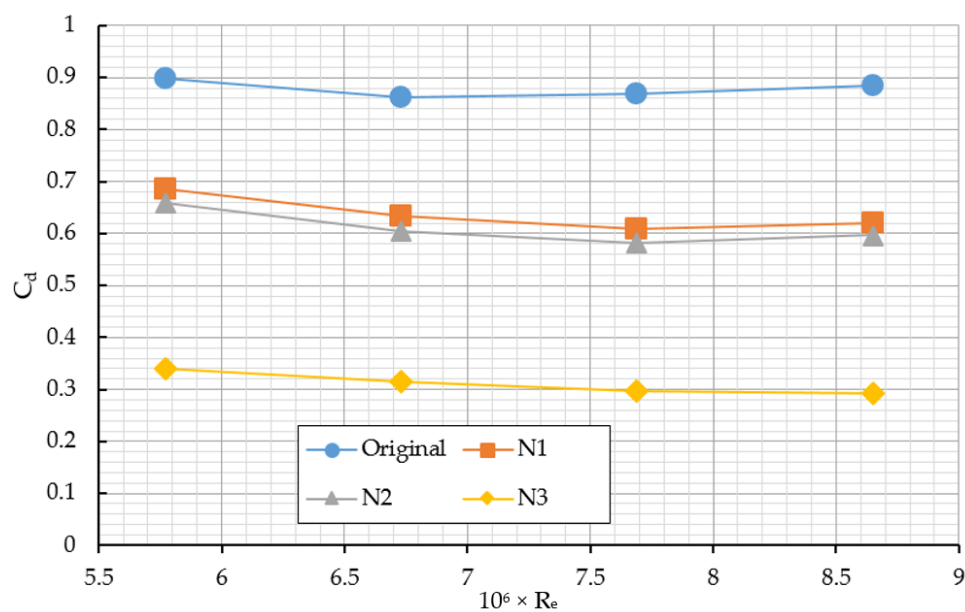


Figure 9. Wind drag acting on the ships with different frontal accommodation shapes in head wind.

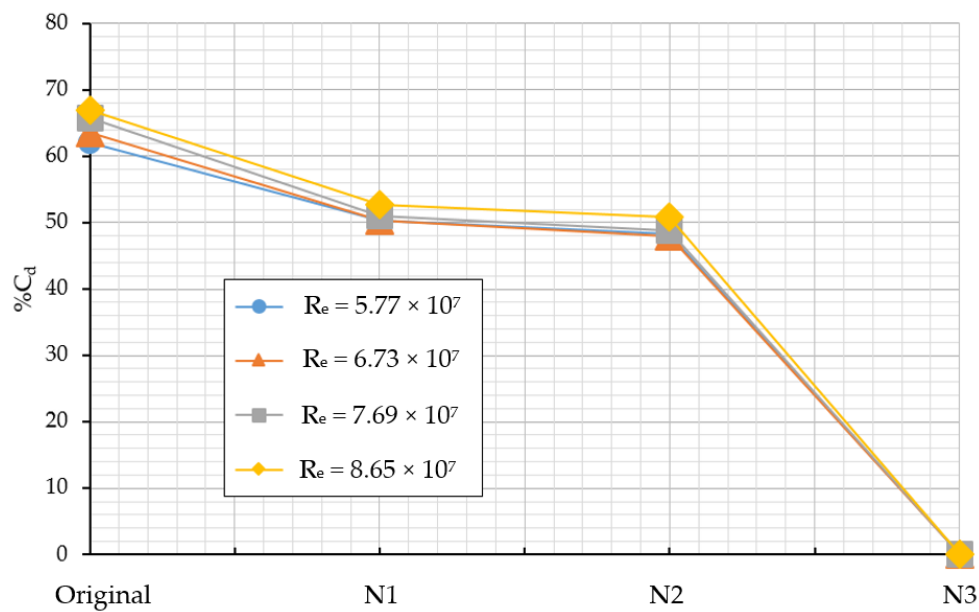


Figure 10. Different wind drag coefficients acting on the ships due to the developed frontal streamlined accommodations.

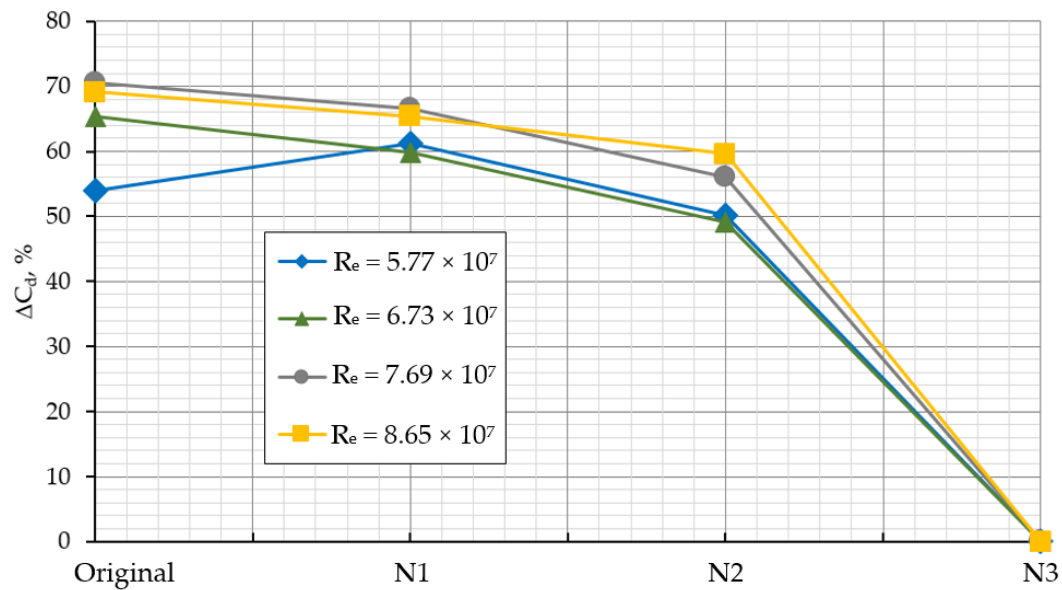


Figure 11. Comparison of interaction effect on wind drag acting on the ship with different frontal accommodation shapes.

4.3. Interaction Effect between Hull and Accommodation on Wind Drag

In this section, wind drag acting on hull of the ship with and without accommodation has been investigated. By comparison, the obtained results of wind drag acting on hull of the ship in two different computed cases, namely hull with and without accommodation, interaction effect between hull and accommodation on wind drag acting on hull, have been found. Moreover, by comparison with the obtained results of wind drag acting on the ship with different accommodation shapes on the deck, the effect of accommodation shapes on wind drag has also been found. Figure 9 shows wind drag acting on the ships with different accommodation shapes on their decks in head wind. A detailed comparison of wind drag acting on the ships is shown in Table 8.

Table 8. Wind drag acting on hull with different frontal accommodation shapes.

$R_e \times 10^7$	Wind Drag, R_x (N)			Wind Drag Coefficients, C_d		
	N1	N2	N3	N1	N2	N3
5.77	8667.38	8326.23	4298.26	0.6863	0.6593	0.3404
6.73	10,881.60	10,387.23	5406.22	0.6328	0.6040	0.3144
7.69	13,678.13	13,093.17	6692.28	0.6087	0.5827	0.2978
8.65	17,611.17	16,966.34	8328.29	0.6191	0.5965	0.2928

The results, as shown in Figure 9, show wind drag coefficients acting on the ship with different frontal accommodation shapes. The results show drastically reduced wind drag acting on the ship by developed frontal accommodations N2 and N3. Table 9 and Figure 10 show detailed differences of wind drag acting on the ship in comparison with those of the original one.

Table 9. Effect of accommodation shape on wind drag acting on the hull of the ship.

$R_e \times 10^7$	Different Wind Drag, R_x , %			Difference, C_d , %		
	N1	N2	N3	N1	N2	N3
5.77	23.60	26.61	62.11	23.60	26.61	62.11
6.73	26.55	29.88	63.51	26.55	29.88	63.51
7.69	30.01	33.00	65.76	30.01	33.00	65.76
8.65	29.95	32.52	66.88	29.95	32.52	66.88

The results given in Table 9 and Figure 10 show drastically reduced wind drag acting on the ships; a reduction of up to 66.88% of the total wind drag has been obtained by developed frontal accommodation shape N3. The frontal box shape accommodation N1 and N2 are also smaller wind drag models, with a reduction of up to 33% of wind drag, as shown. The results of wind drag acting on the ships are in good agreement with the obtained pattern of pressure distribution around and over the hull surface of the ship.

Analyzing the pressure distribution around the ships is a useful tool for finding the reason behind the effects on the wind drag acting on the ship hull. Moreover, the results of the interaction effect between the hull and accommodation on wind drag are highly important from the marine container transportation point of view. Tables 10–12 show the results of the interaction effect between hull and accommodation on wind drag acting on the ship with the different frontal accommodation shapes.

Table 10. Wind drag acting on the hull with accommodation.

$R_e \times 10^7$	Wind Drag, R_x (N)			Wind Drag Coefficients, C_d		
	Acc	Hull	Total	Acc	Hull	Total
Model N1:						
5.77	7524.57	1142.81	8667.38	0.5958	0.0905	0.6863
6.73	9398.81	1482.79	10,881.60	0.5465	0.0862	0.6328
7.69	12,049.96	1628.17	13,678.13	0.5363	0.0725	0.6087
8.65	15,623.23	1987.94	17,611.17	0.5492	0.0699	0.6191
Model N2:						
5.77	6796.24	1529.99	8326.23	0.5382	0.1212	0.6593
6.73	8376.86	2010.37	10,387.23	0.4871	0.1169	0.6040
7.69	10,695.20	2397.97	13,093.17	0.4760	0.1067	0.5827
8.65	13,869.29	3097.05	16,966.34	0.4876	0.1089	0.5965
Model N3:						
5.77	3271.08	1027.18	4298.26	0.2590	0.0813	0.3404
6.73	4083.03	1323.19	5406.22	0.2374	0.0769	0.3144
7.69	5106.26	1586.02	6692.28	0.2273	0.0706	0.2978
8.65	6575.66	1752.63	8328.29	0.2312	0.0616	0.2928

Table 11. Wind drag acting on independent hull and accommodation.

$Re \times 10^7$	Wind Drag, R_x (N)			Wind Drag Coefficients, C_d		
	Acc	Hull	Total	Acc	Hull	Total
Model N1:						
5.77	7162.61	2329.01	9491.62	0.5672	0.1844	0.7516
6.73	9011.22	3100.13	12,111.35	0.5240	0.1803	0.7043
7.69	10,987.28	4012.14	14,999.42	0.4890	0.1786	0.6675
8.65	14,406.12	5051.24	19,457.36	0.5064	0.1776	0.6840
Model N2:						
5.77	7264.60	2109.13	9373.73	0.5752	0.1670	0.7423
6.73	8905.23	3019.18	11,924.41	0.5178	0.1756	0.6934
7.69	10,985.50	3827.16	14,812.66	0.4889	0.1703	0.6592
8.65	14,063.60	5016.12	19,079.72	0.4944	0.1763	0.6707
Model N3:						
5.77	3432.35	2109.13	5541.48	0.2718	0.1670	0.4388
6.73	4220.22	3019.18	7239.40	0.2454	0.1756	0.4210
7.69	5263.25	3827.16	9090.41	0.2342	0.1703	0.4046
8.65	6456.65	5016.12	11,472.77	0.2270	0.1763	0.4033

Table 12. Interaction effect between hull and accommodation on wind drag.

$Re \times 10^7$	Wind Drag, ΔR_x , %			Difference, ΔC_d , %		
	Acc	Hull	Total	Acc	Hull	Total
Model N1:						
5.77	5.05	50.93	8.68	5.05	50.93	8.68
6.73	4.30	52.17	10.15	4.30	52.17	10.15
7.69	9.67	59.42	8.81	9.67	59.42	8.81
8.65	8.45	60.64	9.49	8.45	60.64	9.49
Model N2:						
5.77	6.45	27.46	11.17	6.45	27.46	11.17
6.73	5.93	33.41	12.89	5.93	33.41	12.89
7.69	2.64	37.34	11.61	2.64	37.34	11.61
8.65	1.38	38.26	11.08	1.38	38.26	11.08
Model N3:						
5.77	4.70	51.30	22.43	4.70	51.30	22.43
6.73	3.25	56.17	25.32	3.25	56.17	25.32
7.69	2.98	58.56	26.38	2.98	58.56	26.38
8.65	1.84	65.06	27.41	1.84	65.06	27.41

The results shown in Tables 10–12 clearly show wind drag acting on the ship with the different accommodation shapes in two computed cases, namely the hull with and without accommodation in head wind. The interaction effect between hull and accommodation on wind drag acting on the ship is drastically reduced due to the developed frontal accommodation shapes, as shown. The interaction effect on wind drag acting on the accommodation part is less than 10% and up to 65% of the hull part. The interaction effect on the total wind drag is about 10% of the frontal box shape accommodations N1 and N2, and up to 27% for the streamlined accommodation N3. Figure 11 shows a comparison of the interaction effects on wind drag acting on the ship with different frontal accommodation shapes.

The results show a drastically reduced interaction effect between hull and accommodation on wind drag of the model N3. The interaction effect between hull and accommodation of the original ship is higher than that of the model N3, by up to 70%, as shown. The obtained results are very useful to develop a new accommodation shape with reduced interaction effect and wind drag acting on the ship hull as little as possible.

For a streamlined accommodation shape like the proposed model N3, the interaction effect between hull and accommodation on wind drag has been drastically reduced, as shown. Most researchers are

of the opinion that the streamlined accommodation is the best shape to reduce wind drag acting on the ship when in operation. However, the huge streamlined accommodation shape is difficult to be made and will also require a very high cost. Therefore, an accommodation shape which is simply made of only flat plates but still capable of producing a reduced wind drag could be a good choice for the ship building industry.

5. Conclusions

In this paper, aerodynamic performances of a container ship in head wind have been simulated by CFD. From the obtained results, the interaction effect between hull and accommodation has been investigated. Furthermore, the obtained results in this paper have shown that the newly proposed frontal accommodation shapes can be used to considerably reduce wind drag acting on the container ships, leading to more economic fuel consumption in marine transportation. The main conclusions are summarized as follows:

- The effect of mesh number on aerodynamic performances of the reference container ship has been determined. The obtained results show that the effect of mesh number on wind drag acting on the container ship is about 10% when the value of y^+ is less than 50 and the same value when y^+ is less than 7.
- The CFD results for the two computed cases, namely the original ship hull with and without accommodation, have been obtained. The effect of the hull on the wind drag is as high as 42%, while the interaction effect between hull and accommodation of the container ship on the total wind drag has been evaluated and is only about 10%.
- The newly proposed frontal accommodation shapes' effect on aerodynamic performance as well as wind drag acting on hull of the ship. It has been shown that the developed frontal accommodation shapes affect both the pressure distribution around the ship hull and the separation regions around the accommodation. Hence, a suitable shape of the frontal accommodation could be installed to greatly reduce the total wind drag.
- The effect of accommodation shapes on wind drag acting on hull of the ship has been investigated. Using the proposed frontal accommodation shapes, up to 30% of the total wind drag in head wind (N1), and up to 66.88% of the total wind drag acting on the hull at a Reynolds number of 8.65×10^7 in head wind (N3), could be reduced.
- The interaction effect between the hull and accommodation of the ship, with different accommodation shapes, has been investigated. Drastically reduced interaction effects on wind drag acting on the hull, using the accommodation shape N2 (up to 49%) and using the accommodation shape N3 (up to 70%), have been shown.

Author Contributions: Conceptualization, N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien), V.-T.T. and N.-T.B.; methodology, N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien), V.-T.T. and N.-T.B.; software, N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien) and V.-T.T.; validation, N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien), V.-T.T. and N.-T.B.; writing—original draft preparation, N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien) and V.-T.T.; writing—review and editing N.V.H. (Ngo Van He), N.V.H. (Ngo Van Hien), V.-T.T. and N.-T.B.; supervision, N.V.H. (Ngo Van Hien); project administration, N.V.H. (Ngo Van He). All authors have read and agreed to the published version of the manuscript.

Funding: Vietnam National Foundation for Science and Technology Development (NAFOSTED).

Acknowledgments: This research is funded by Vietnam National Foundation for Science and Technology Development (NAFOSTED) under the grant number 107.03-2019.302. The authors would like to warmly express their thanks for the support.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Janssen, W.; Blocken, B.; van Wijhe, H. CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications. *J. Wind Eng. Ind. Aerodyn.* **2017**, *166*, 106–116. [[CrossRef](#)]

2. Andersen, I.M.V. Wind loads on post-panamax container ship. *Ocean Eng.* **2013**, *58*, 115–134. [[CrossRef](#)]
3. Fujiwara, T.; Tsukada, Y.; Kitamura, F.; Sawada, H. Experimental Investigation and Estimation on Wind Forces for a Container Ship. In Proceedings of the Nineteenth International Offshore and Polar Engineering Conference, Osaka, Japan, 21–26 July 2009; pp. 555–562.
4. Kim, Y.; Kim, K.-S.; Jeong, S.-W.; Jeong, S.-G.; Van, S.-H.; Kim, Y.-C.; Kim, J. Design and performance evaluation of superstructure modification for air drag reduction of a container ship. In Proceedings of the Twenty-Fifth International Ocean and Polar Engineering Conference, Kona, HI, USA, 21–26 June 2015.
5. Van Nguyen, T.; Kinugawa, A. Development of Practical Gap Covers to Reduce Air Resistance Acting on Deck Containers of a Ship. In Proceedings of the 23th conference Japan Society of Naval Architects and Ocean Engineers, Hiroshima, Japan, 2017; pp. 211–216.
6. Van Nguyen, T. Vortex Control in Gap Flow by Small Appendages to Reduce Air Resistance Acting on Deck Containers of a Ship. In Proceedings of the 24th Conference Japan Society of Naval Architects and Ocean Engineers, Tokyo, Japan, 26 May 2017; pp. 335–338.
7. Watanabe, I.; Van Nguyen, T.; Miyake, S.; Shimizu, N.; Ikeda, Y. A study on reduction of air resistance acting on a large container ship. *Proc. APHydro* **2016**, *2016*, 321–330.
8. Seok, J.; Park, J.-C. Comparative Study of Air Resistance with and without a Superstructure on a Container Ship Using Numerical Simulation. *J. Mar. Sci. Eng.* **2020**, *8*, 267. [[CrossRef](#)]
9. Wnek, A.; Soares, C.G. Numerical Analysis of the Shadow Effect of an LNG Floating Platform on an LNG Carrier under Wind Conditions. In *Sustainable Maritime Transportation and Exploitation of Sea Resources*; Taylor & Francis Group: London, UK, 2012.
10. Wnek, A.; Soares, C.G. CFD assessment of the wind loads on an LNG carrier and floating platform models. *Ocean Eng.* **2015**, *97*, 30–36. [[CrossRef](#)]
11. Saydam, A.Z.; Taylan, M. Evaluation of wind loads on ships by CFD analysis. *Ocean Eng.* **2018**, *158*, 54–63. [[CrossRef](#)]
12. He, N.V.; Ikeda, Y. A Study on Interaction Effects between Hull and Accommodation on air Resistance of a Ship. In Proceedings of the Conference Japan Society of Naval Architects and Ocean Engineers, Fukuoka, Japan, 27–28 May 2013; pp. 281–284.
13. Van He, N.; Mizutani, K.; Ikeda, Y. Reducing air resistance acting on a ship by using interaction effects between the hull and accommodation. *Ocean Eng.* **2016**, *111*, 414–423. [[CrossRef](#)]
14. Van He, N.; Mizutani, K.; Ikeda, Y. Effects of side guards on aerodynamic performances of the wood chip carrier. *Ocean Eng.* **2019**, *187*, 106217. [[CrossRef](#)]
15. Sugata, K.; Iwamoto, Y.; Ikeda, Y.; Nihei, Y. Reduction of Wind Force Acting on Non Ballast Ship. In Proceedings of the 5th Asia Pacific Workshop on Marine Hydrodynamics-APHydro2010, Osaka, Japan, 1–4 July 2010; pp. 1–4.
16. Du, L.; Lin, Z.; Jiang, Y.; Li, P.; Dong, Y. Numerical Investigation on the Scale Effect of a Stepped Planing Hull. *J. Mar. Sci. Eng.* **2019**, *7*, 392. [[CrossRef](#)]
17. Schmitt, P.; Windt, C.; Davidson, J.; Ringwood, J.V.; Whittaker, T. The efficient application of an impulse source wavemaker to cfd simulations. *J. Mar. Sci. Eng.* **2019**, *7*, 71. [[CrossRef](#)]
18. Zalesny, V.; Agoshkov, V.; Aps, R.; Shutyaev, V.; Zayachkovskiy, A.; Goerlandt, F.; Kujala, P. Numerical modeling of marine circulation, pollution assessment and optimal ship routes. *J. Mar. Sci. Eng.* **2017**, *5*, 27. [[CrossRef](#)]
19. Feng, D.; Ye, B.; Zhang, Z.; Wang, X. Numerical Simulation of the Ship Resistance of KCS in Different Water Depths for Model-Scale and Full-Scale. *J. Mar. Sci. Eng.* **2020**, *8*, 745. [[CrossRef](#)]
20. Viola, I.M. Downwind sail aerodynamics: A CFD investigation with high grid resolution. *Ocean Eng.* **2009**, *36*, 974–984. [[CrossRef](#)]
21. Nguyen, T.; Shimizu, N.; Kinugawa, A.; Tai, Y.; Ikeda, Y. Numerical studies on air resistance reduction methods for a large container ship with fully loaded deck-containers in oblique winds. In Proceedings of the VII International Conference on Computational Methods in Marine Engineering, MARINE, Gothenburg, Sweden, 13–15 May 2019; pp. 1040–1051.
22. Ittc, R. *Procedures and Guidelines: Practical Guidelines for Ship CFD Applications*, 7.5; ITTC: London, ON, Canada, 2011; pp. 1–18.
23. Celik, I.B.; Ghia, U.; Roache, P.J.; Freitas, C.J. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *J. Fluids Eng. Trans. Asme* **2008**, 1–130.

24. Collie, S.; Gerritsen, M.; Jackson, P. *A Review of Turbulence Modeling for Use in Sail Flow Analysis*; Report; University of Auckland's Faculty of Engineering: Auckland, New Zealand, 2001.
25. Xia, M.; Jiang, L. Application of an unstructured grid-based water quality model to Chesapeake Bay and its adjacent coastal ocean. *J. Mar. Sci. Eng.* **2016**, *4*, 52. [[CrossRef](#)]
26. Windt, C.; Davidson, J.; Schmitt, P.; Ringwood, J.V. On the assessment of numerical wave makers in CFD simulations. *J. Mar. Sci. Eng.* **2019**, *7*, 47. [[CrossRef](#)]
27. Bertram, V. *Practical Ship Hydrodynamics*; Elsevier: Amsterdam, The Netherlands, 2011.
28. Murdock, J.W. *Fundamental Fluid Mechanics for the Practicing Engineer*; CRC Press: Boca Raton, FL, USA, 1993.

Publisher's Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



© 2020 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>).