Scientia Iranica B (2014) 21(3), 600–608

Sharif University of Technology
Scientia Iranica
Transactions B: Mechanical Engineering
www.scientiairanica.com

Experimental and numerical studies on resistance of a catamaran vessel with non-parallel demihulls

A. Ebrahimi[a,*], M. Rad[b] and A. Hajilouy[b]

a. *Department of Marine Engineering, Chabahar Maritime University, Chabahar, Iran.*
b. *School of Mechanical Engineering, Sharif University of Technology, Tehran, Iran.*

Received 3 October 2012; received in revised form 16 July 2013; accepted 4 November 2013

KEYWORDS
Catamaran vessel;
Resistance;
Non-parallel demihulls;
Model experiments;
Numerical investigation.

Abstract. In common catamaran vessels, demihulls are parallel to each other. In this paper, the total resistance of a catamaran vessel with non-parallel demihulls is investigated experimentally and numerically. Experiments are carried out at different Separation Ratios (S.R.), that is the ratio of fore to aft separation of the catamaran demihulls; and also in two ratios of length to separation in amidships (L/S_m). The FLUENT solver, based on the Finite Volume Method (FVM), was used for numerical solution. Applying the VOF model, the free surface around the catamaran vessel and total resistance are calculated and compared with experimental results. Finally, the frictional resistance of the catamaran from the ITTC 1957 correlation line is calculated and compared with CFD frictional resistance. The results show that non-parallel demihulls cause total resistance to increase at Froude numbers below 0.8, and decrease at Froude Numbers over 0.8. In the numerical part, at low Froude numbers, numerical results have an error up to 10% relative to model test results, but error increases at high Froude numbers up to 25%.

1. Introduction

The demand for high speed multihull vessels has greatly increased during the last decades for both commercial and military purposes. In particular, catamaran configurations are very attractive because of their excellent performances with respect to speed, safety, resistance and transversal stability [1].

In recent decades, fast catamaran vessels have become a popular means of passenger transport throughout the world [2]. This may have several reasons, e.g. high transverse stability, large deck area and favorable high maneuverability. These vessels have higher frictional resistance compared with monohull vessels, mainly because of their larger wetted surface. However, their wave-making resistance at high Froude numbers is smaller in comparison with monohull vessels, because of their slender demihulls. Reducing the resistance of catamarans is a favorite field for researchers. Different methods for resistance reduction were applied to these vessels and they are in practice now. In the design or optimization of a vessel, the prediction of forces acting on the hull moving in water is very important. A catamaran vessel is composed of two demihulls having interaction. The resistance characteristics of catamaran vessels are more complicated compared with monohull ships. In addition to the resistance of each demihull, interferences between demihulls are effective in the total resistance of the catamaran. These interferences have two parts:

1. Body interference;
2. Waves interference.

These two interferences can increase or decrease the

*. *Corresponding author. Tel: +98 545 412 2181; Fax: +98 545 222 1015*
E-mail addresses: Ab_ebrahimi@cmu.ac.ir (A. Ebrahimi); Rad@sharif.edu (M. Rad); Hajilouy@sharif.edu (A. Hajilouy)

total resistance of the catamaran. In traditional catamarans, demihulls are parallel to each other. In this study, the resistance characteristics of a catamaran model with non-parallel demihulls were investigated experimentally and numerically. Finally, in each case, the total resistance of the catamaran model is compared with the resistance of common catamarans. In a catamaran vessel with non-parallel demihulls, the separation of demihulls is not constant from fore to aft of the vessel, unlike the traditional catamarans, and either decreases or increases along the catamaran. The numerical investigation of free surface flow around the catamaran vessel is performed by a commercial CFD code, and the numerical results are compared with experimental results.

According to the open literature, non-parallel demihulls have not been investigated and, therefore, we have used the results of common catamaran studies.

Insel [3] carried out experimental and theoretical investigations into the resistance components of a high speed displacement catamaran model series. Effects of hull separation, running trim, sinkage and other parameters have been studied during tests. Finally, resistance components of the catamaran were calculated by theoretical methods and empirical formula, and the results were compared with experimental results.

Steen et al. [2] estimated the resistance of high speed catamarans by empirical formula and then accomplished experiments at the MARINTEK towing tank. The wave-making resistance of the catamaran was also calculated by numerical methods, and a comparison between results was performed.

Salas et al. [4] carried out experiments on a high speed catamaran and calculated its resistance. Then, two CFD codes were used for calculation of catamaran resistance.

Thornhill et al. [5] used a finite volume code to simulate the flow around a planing vessel at steady speed through calm water using a 3D unstructured hybrid mesh. Force and moment data from the simulations were used in an iterative scheme to determine the dynamic equilibrium position of the model at selected speeds. Moraes et al. [6] used two methods for calculating the resistance of a catamaran. The first method was slender body and the other one was a CFD code (SHIPFLOW). Finally, the effect of water depth on the resistance of the catamaran was investigated.

2. Model tests

Model testing is an accurate and reliable method for measuring and investigating ship resistance. In this study, the model tests are accomplished in the towing tank at the Marine Engineering Laboratory (MEL) at Sharif University of Technology, and the towing tank principal dimensions are reported in Table 1.

Table 1. Principal dimensions of towing tank.

Length	23.5 m
Beam	2.50 m
Depth	1.50 m

Figure 1. The catamaran model.

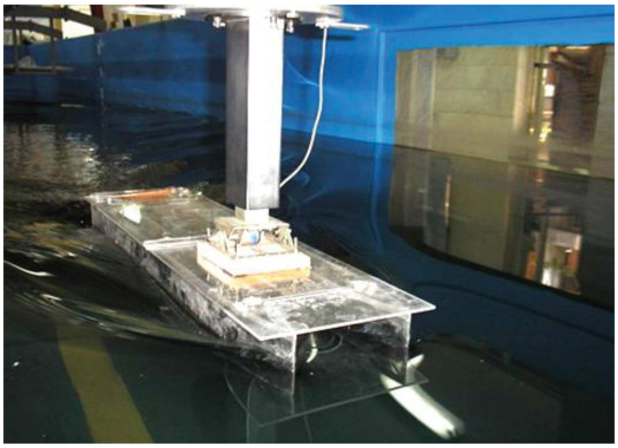

Figure 2. The model test setup in towing tank.

The catamaran model used in experiments is shown in Figure 1. The model was made of Plexiglas planes that are joined with glue. As we would like to study the effects of non-parallel demihulls on the resistance of a catamaran vessel in different states, demihulls were connected together by a simple mechanism. They were connected by a plate with six screws moving in slots that permit a change in separation ratio.

The carriage system of the towing tank was outfitted with accurate sensors that measured calm water resistance, transverse force and the running trim of the model. Because of symmetric demihulls, resultant side forces were practically zero in all cases. For better analysis, model experiments were carried out in free trim and sinkage, whilst movements in surge, sway, roll and yaw were restrained. In Figure 2, one of the model test setups in the towing tank is shown. The main dimensions of the model are reported in Table 2.

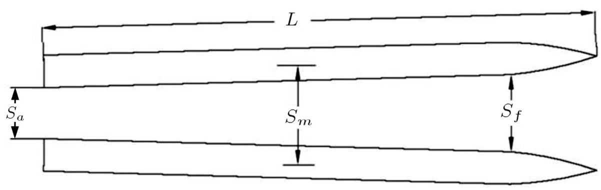

Figure 3. Top view of model and main parameters.

Table 2. Main dimensions of catamaran model.

Length overall	0.665 m
Beam of demihulls	0.040 m
Draft	0.030 m
Height	0.075 m

Table 3. States of model tests.

L/S_m	Separation Ratio (S.R.)				
5.6	0.6	0.73	1	1.4	3
7.5	0.4	0.5	1	2	3

The top view and main parameters of the model are shown in Figure 3. In this figure, parameters L, S_m, S_f and S_a, respectively, are length of model, separation in amidships, and separation in the fore and aft of the model. S_m and S_f are measured at points located 32 and 55 cm from aft of the model.

All model tests were conducted over a speed range of 0.5 to 2.5 m/s corresponding to a length Froude Number range of 0.2 to 1. At Froude numbers above 1, the spray of water from the model sides was so great and could affect test results.

According to the model speed, it is obvious that flow around the model is turbulent. Therefore, a turbulent simulator is not utilized in the model tests. Accordingly, in CFD modeling, we assume fully turbulent flow around the model.

Experiments are carried out at different S.R., that is ratio of fore to aft separation of the catamaran demihulls (S_f/S_a), and also two ratios of overall length to separation in amidships (L/S_m). The value of the separation ratios only depended on the geometry and shape of the model. Model test states are shown in Table 3. The S.R. = 1 means that demihulls are parallel, the same as in conventional catamaran vessels.

2.1. The model tests results

Figures 4 and 5 show the running trim of the model, in terms of Froude number, for different separation ratios at $L/S_m = 5.6$ and 7.5, respectively.

For $L/S_m = 5.6$ and 7.5, the trim of the model to Froude number of about 0.5 is approximately constant, but for greater Froude number, the trim increases. The largest trim angle for $L/S_m = 5.6$ is related to S.R. = 1.4 and 3, and for $L/S_m = 7.5$ is related to S.R. = 2 and 3. It seems that when S.R. is greater than 1, the pressure between demihulls increases. This result in an upward lift force at the fore of the model and, consequently, trim by aft. Therefore, when S.R. is greater than 1, the trim of model is greater than a state of S.R. is smaller than 1.

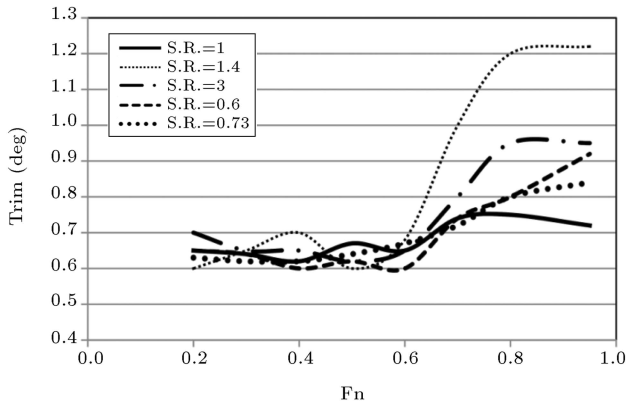

Figure 4. Running trim of model for $L/S_m = 5.6$.

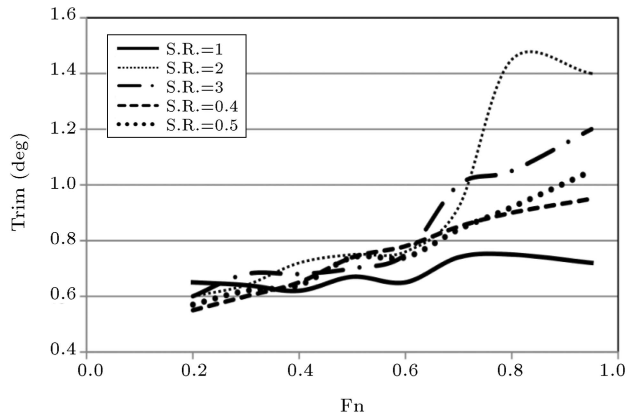

Figure 5. Running trim of model for $L/S_m = 7.5$.

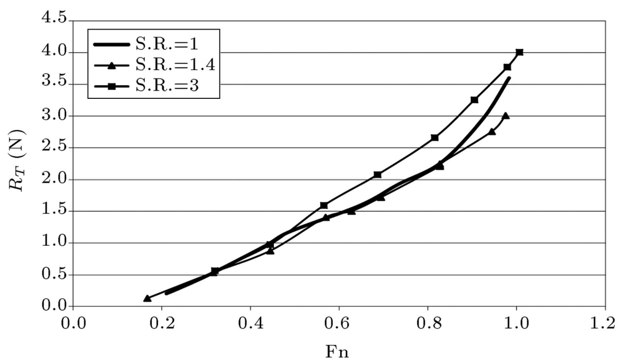

Figure 6a. Comparison of R_T results for S.R. = 1, 1.4 and 3 ($L/S_m = 5.6$).

The total resistance of model (R_T) at different Froude numbers for ratio of $L/S_m = 5.6$ and $L/S_m = 7.5$ is reported in Figures 6 and 7, respectively.

In case of $L/S_m = 5.6$, results show that for S.R. = 1.4, the total resistance increases in comparison by S.R. = 1 (parallel demihulls) at Fn < 0.8, and

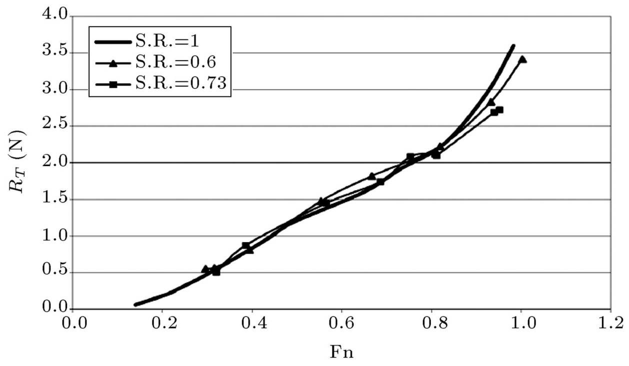

Figure 6b. Comparison of R_T results for S.R. = 1, 0.6 and 0.73 ($L/S_m = 5.6$).

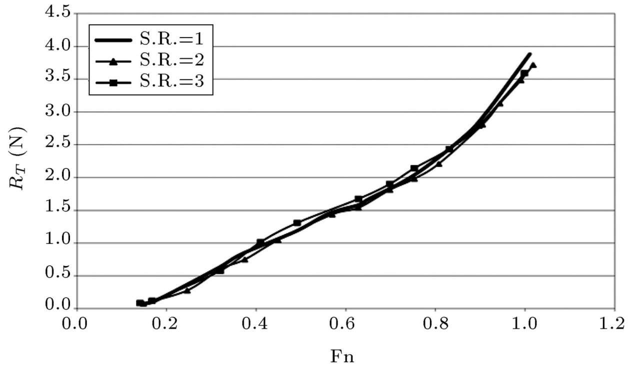

Figure 7a. Comparison of R_T results for S.R. = 1, 2 and 3 ($L/S_m = 7.5$).

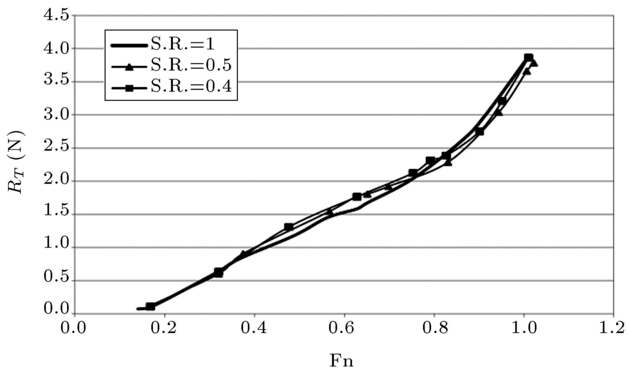

Figure 7b. Comparison of R_T results for S.R. = 1, 0.4 and 0.5 ($L/S_m = 7.5$).

decreases at Fn > 0.8. This status is observed for the case of $L/S_m = 7.5$, when S.R. = 2 and S.R. = 3. At Fn < 0.8, non-parallel demihulls increase the pressure resistance and total resistance of the model in comparison with parallel demihulls. At Fn > 0.8, a significant increase in the trim of the model by aft reduces wave making resistance, frictional resistance and, consequently, the total resistance of the model, which compensates the increase of resistance due to non-parallel demihulls. In the case of $L/S_m = 5.6$ and S.R. = 3, the total resistance increases relative to parallel demihulls at all Froude numbers. In this state,

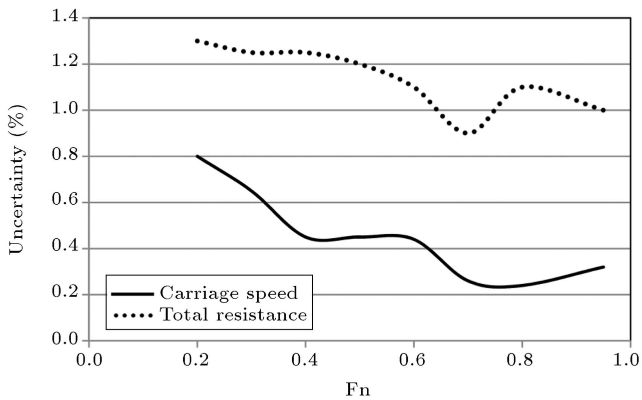

Figure 8. Uncertainties of experimental results for S.R. = 1 and $L/S_m = 7.5$.

the separation of demihulls in aft of the model is too small and causes large interference and magnification in waves generated by the demihulls. So, the total resistance of the model increases.

For all cases in which S.R. < 1 ($L/S_m = 5.6$ and $L/S_m = 7.5$), total resistance increases at Fn < 0.8 and decreases at Fn > 0.8 relative to parallel demihulls. This result is the same as S.R. > 1, but at Fn > 0.8, because of the smaller trim angle, the reduction in total resistance is not so significant.

2.2. *The uncertainty analysis*

For results of the model experiments, an uncertainty analysis was carried out according to ITTC procedures and guidelines [7]. These analyses were done for carriage speed and total resistance of the model.

For instance, uncertainty results for S.R. = 1 at ratio of $L/S_m = 5.6$ in terms of Froude number are shown in Figure 8. Results show that maximum uncertainty for total resistance occurs in Fn = 0.3, which is about 1.3%.

3. The numerical solution

In recent years, the numerical solution method and, especially, Computational Fluid Dynamics (CFD) have developed for studying the hydrodynamics of ships. Although the CFD method, in comparison with the model test, has less accuracy, we can use this method for detecting flow characteristics, e.g. drag and lift forces, pressure and velocity contours and waves. In this study, free-surface flow around a catamaran vessel is simulated using the FLUENT, Version 6.3.26, software. This commercial software solves Reynolds-Averaged Navier-Stokes (RANS) equations in the computational domain and finally calculates pressure and frictional resistance. For capturing free surface around the vessel, the Volume Of Fluid (VOF) method is used. The VOF method simulates the motion of two fluids by solving a single set of momentum equations and tracking the volume fractions of each of the fluids throughout the

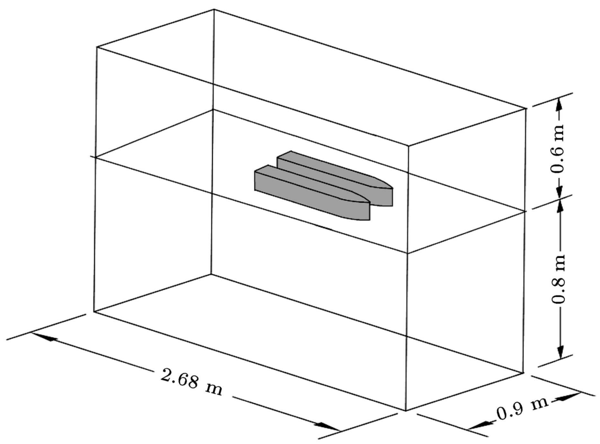

Figure 9. Dimensions of computational domain.

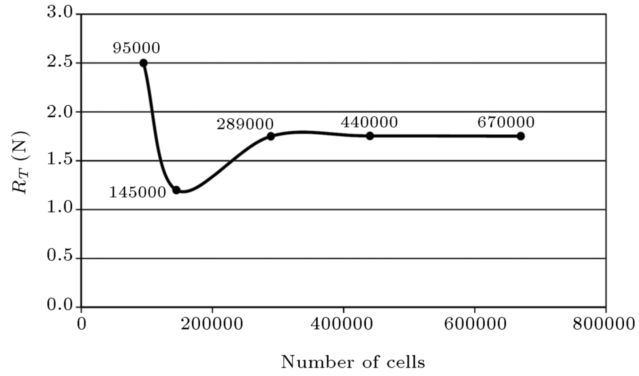

Figure 10. Mesh independency.

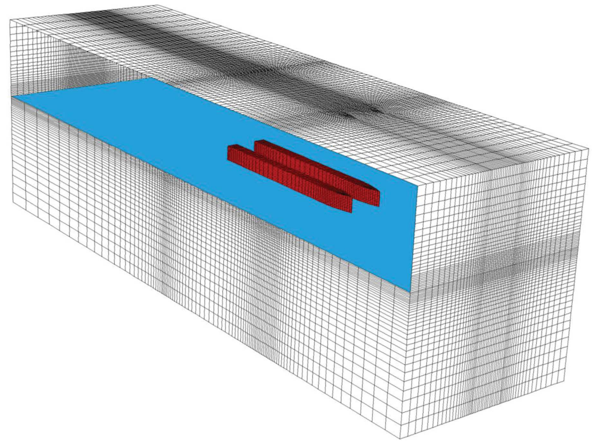

Figure 11. Details of grid.

domain. A volume fraction function is defined for each of the fluids in the simulation and is set to unity if the given fluid occupies a cell volume or is set to zero otherwise. When the interface between two fluids cuts through a computational cell, the value of the function for a particular fluid represents the fraction of the cell volume occupied by that fluid [8].

A time-dependent solution with time step of 0.002 seconds is used for the numerical solution. Monitoring of the resistance and free surface waves around the model are used to let us know when the problem has reached a stationary solution.

Because of the hardware limitations, CFD investigation is only done for S.R. = 1, 0.73 and 3 at ratio of $L/S_m = 5.6$, similar to that of model experiments. The computational domain and its dimensions are displayed in Figure 9. These dimensions are selected according to [4]. Dimensions of the numerical model are the same as the catamaran model used in the experiments. Therefore, numerical results can be compared with experimental results directly.

The domain was defined by a cube volume, from which the hull volume was cut off. The bottom of the domain to the initial water surface was occupied by water and the rest by air. The hull and flow field are symmetric about the center plane between demihulls. Therefore, one half of the computational domain was considered for numerical treatment and the symmetry boundary condition was applied at the center plane.

The GAMBIT pre-processing software was used for mesh generation. A structured mesh is used in the domain in which all cells are of the hexahedral type. At the stern and bow side of the model and near free surface, meshes have been refined in all cases. To check the grid independency of numerical results, we examined 5 grid sizes. By visual inspection of the wave pattern and detailed comparison of other results, such as pressure and velocity contours and resistance force, we selected the optimum mesh. For example, total resistance ($V = 2$ m/s, S.R. = 1) of the model due to different grid sizes is plotted in Figure 10. It seems, by increasing grid size over 132000 elements, variation in resistance is inconsiderable, although, using higher grid sizes, the free surface can be captured more accurately. The numbers of cells that are finally used for numerical solution at 3 states are given in Table 4. For instance, the details of the grid used for state of S.R. = 1 are displayed in Figure 11.

Table 4. Grid size of numerical solution.

States	Number of cells
S.R. = 1	289000
S.R. = 3	296000
S.R. = 0.73	291000

For turbulence modeling, standard, RNG and Realizable $k - \varepsilon$ models were used in the numerical solution. Results show that there is not much difference between the accuracy of these models, but computing time for the standard $k-\varepsilon$ model is less than for others. Accordingly, the standard $k - \varepsilon$ model was selected for numerical simulation. For the near wall treatment, Standard Wall Functions were used.

In numerical solution, the trim and sinkage of the model were kept fixed. Although it can put out errors in results, because of our limitations and the complexity of the 6-DOF modeling of the hull, we applied this strategy. Boundary conditions applied to the computational domain are velocity inlet for air and water inlet, outflow for air and water outlet, no-slip wall for the hull, moving wall for side walls, top and bottom of the domain and, finally, symmetry for the center plane of the demihulls.

A variety of pressure-velocity coupling schemes are available in FLUENT. The PISO scheme was selected in this study. For pressure and momentum spatial discretizations, body force weighted and second order upwind methods are selected, respectively.

The motion of the free surface flow is governed by gravitational force. Hence, the boundary conditions to be imposed must take into account gravity effects. For this purpose, the computational domain is modeled as an open channel flow, which is also consistent with the experimental setup.

3.1. The numerical results

Free surface and waves generated around the catamaran model are displayed in Figure 12. In these figures, contours show the height of the waves, and the wave systems of the bow and stern are obvious. Pressure contours on the surface of the model bottom for $V = 2$ m/s (for different S.R. values) are shown in Figure 13. As shown in the figure, the pressure on the bow is greater than the stern of the model in all S.R. states. These result to lift force which exerted in bow of model as shown in Figure 14.

The lift force applied to the model in terms of Froude number, which was calculated by FLUENT,

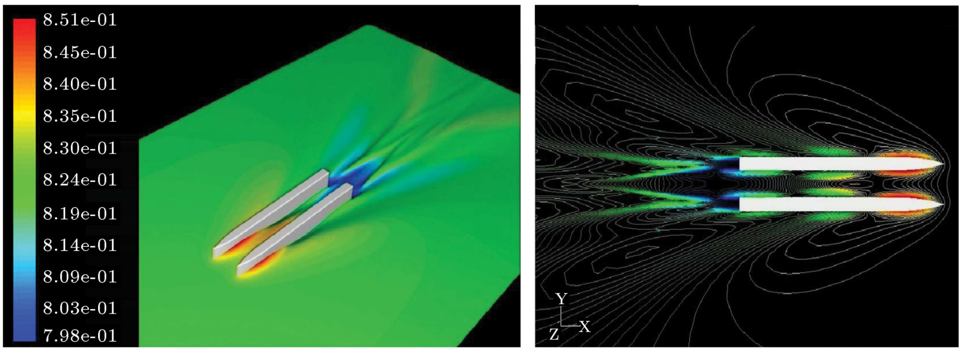

Figure 12a. Free surface around model for S.R. = 1.

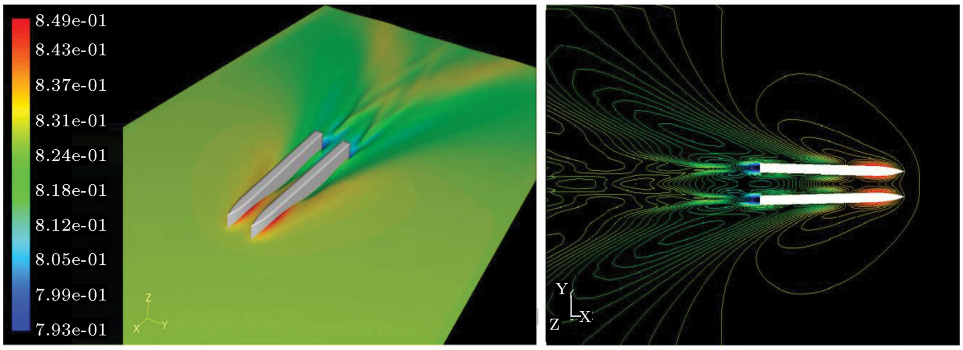

Figure 12b. Free surface around model for S.R. = 0.73.

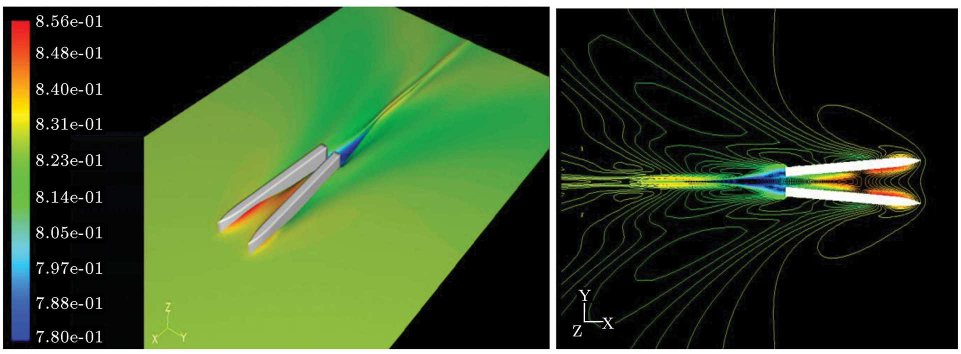

Figure 12c. Free surface around model for S.R. = 3.

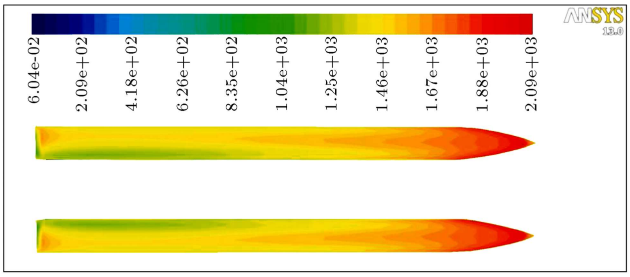

Figure 13a. Pressure contours on bottom of model for S.R. = 1.

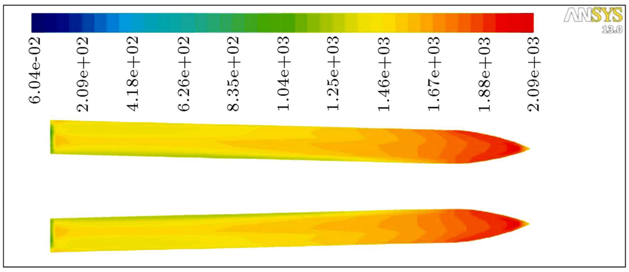

Figure 13b. Pressure contours on bottom of model for S.R. = 0.73.

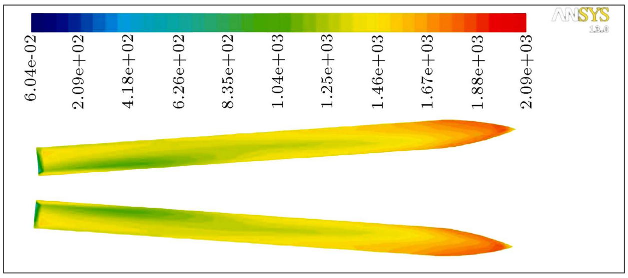

Figure 13c. Pressure contours on bottom of model for S.R. = 3.

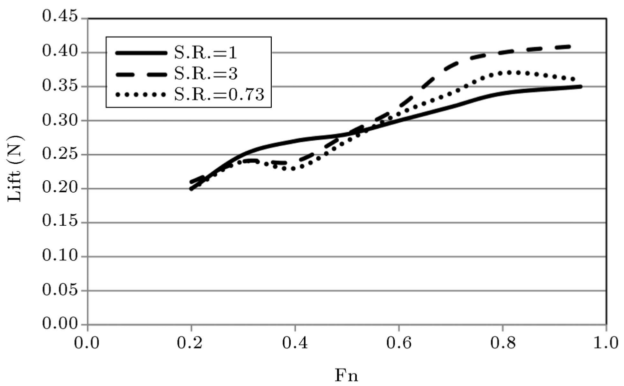

Figure 14. The lift force applied to model calculated from numerical calculation.

is shown in Figure 14. The results show that by increasing the speed of the model, the lift force increases, whereas, the lift acting forward of the amidship results in trim by the aft. The total resistance calculated by the CFD method for 3 states is shown in Figure 15 and compared with experimental results.

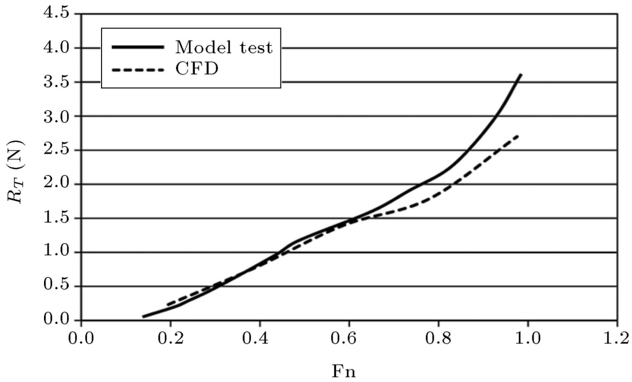

Figure 15a. Comparison of R_T from CFD and model test for S.R. = 1.

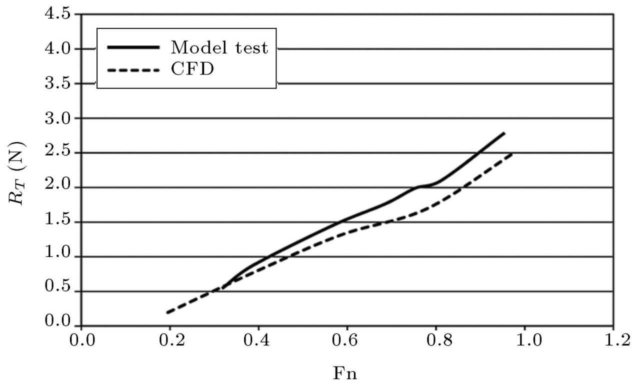

Figure 15b. Comparison of R_T from CFD and model test for S.R. = 0.73.

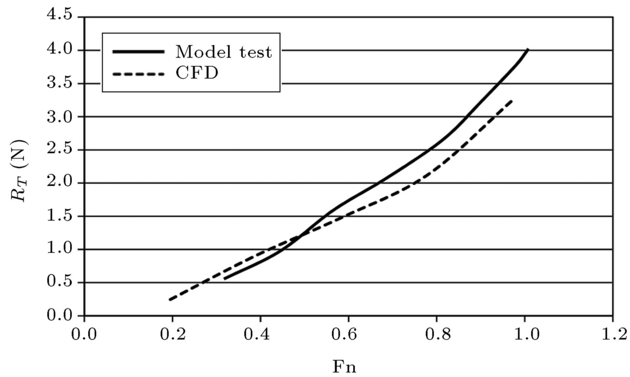

Figure 15c. Comparison of R_T from CFD and model test for S.R. = 3.

As shown in these figures, in all states, total resistance of the catamaran calculated from the CFD code at low Froude numbers up to 0.6 has, maximum, 10% difference from experimental results, but, after that, this reaches to 25%. This error can be due to the following reasons. The trim and sinkage of the model were kept fixed in CFD modeling, unlike the model test conditions. The other reason may be the weakness of this CFD code for calculating wave making resistance. However, at low Froude numbers, in which wave making resistance has smaller magnitude,

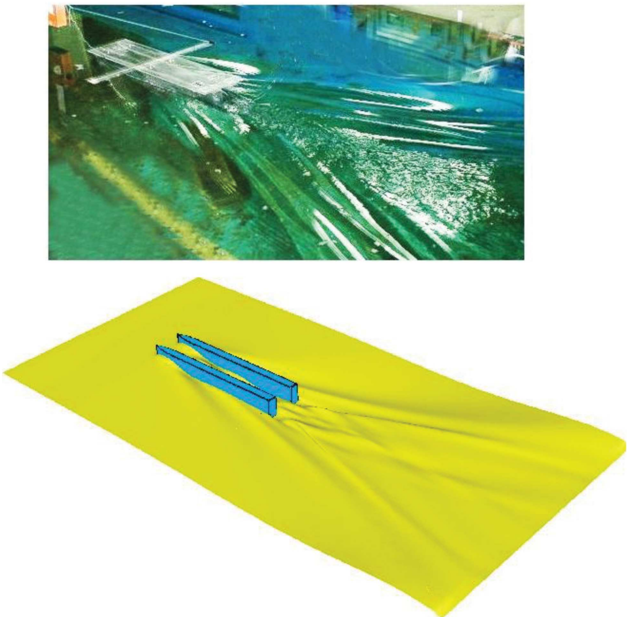

Figure 16. Comparison of free surface around model calculated by CFD and model test.

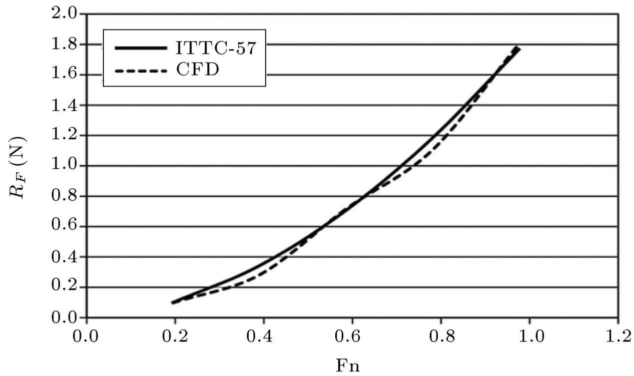

Figure 17. Comparison of frictional resistance calculated by CFD and ITTC 1957.

CFD results have less difference with experimental results.

In Figure 16, the free surface around the model in the towing tank and CFD modelling are compared. As shown, there is good agreement between experimental and CFD results.

4. ITTC 57 method

Another way to estimate the resistance of a ship is by empirical formulae. One of the most common methods in this way is the ITTC 57 method. In this study, we just investigate the frictional resistance of a catamaran model for comparison by CFD results. The frictional resistance of the ship (R_F) can be calculated from Eq. (1):

$$R_F = C_f \left(\frac{1}{2} \rho V^2 S_w \right), \tag{1}$$

in which C_f, V and S_w are frictional resistance coefficient, velocity, and wetted surface area of the model, respectively. The frictional resistance coefficient can be computed by the formula presented by ITTC-57, shown in Eq. (2):

$$C_f = \frac{0.075}{(\log \mathrm{Rn} - 2)^2}. \tag{2}$$

The coefficient of frictional resistance and, consequently, frictional resistance are calculated. In Figure 17, frictional resistance from the ITTC 1957 method is compared with frictional resistance from the CFD method. As shown in the figure, ITTC 1957 results for frictional resistance have good correlation with CFD results.

5. Conclusion

In this paper, the effects of non-parallel demihulls on the resistance characteristics of a catamaran vessel are investigated. The following conclusions can be drawn from the results.

For both states of S.R. $>$ 1 and S.R. $<$ 1, the total resistance of the vessel increases relative to parallel demihulls for Froude numbers below 0.8 and decreases above that. For high separation ratios, e.g. S.R. = 3 for L/Sm = 7.5, because of small aft separation, interference between waves causes total resistance to increases relative to parallel demihulls at all Froude numbers.

From the numerical solution, the following conclusions can be drawn.

In all states, numerical results have a difference of about 10% relative to the model test results at low Froude numbers. This increases for high Froude numbers up to 25%.

Differences between lab-based observations and model results are explained as follows.

CFD modeling assumptions are nearly relevant to the experimental setup, except the DOF of the catamaran model. In the experimental setup, the model is free in trim (changes in aft and fore draft) and heave (motion in z direction). But, in CFD modeling, these DOF are set to zero because of some reasons, i.e. hardware limitations, complexity of mesh, etc. By using UDF programming in FLUENT, and by using moving mesh, 6 DOF of the model can be modeled. This application needs advanced hardware, such as parallel processors.

Frictional resistance calculated from CFD has good agreement with ITTC 1957 results.

References

1. Zaghi, S., Broglia, R. and di Mascio, A. "Experimental and numerical investigations on fast catamarans interference effects", *9th International Conference on Hydrodynamics*, Shanghai, China (2010).

2. Steen, S., Rambech, H.J., Zhao, R. and Minsaas, K.J. "Resistance prediction of fast displacement catamarans", FAST'99, Seattle, USA (1999).

3. Insel, M. "An investigation into the resistance components of high speed displacement catamarans", PhD Thesis, University of Southampton, Department of Ship Science (1990).

4. Salas, M., Luco, R., Sahoo, P.K., Browne, N. and Lopez, M. "Experimental and CFD resistance calculation of a small fast catamaran", Institute of Naval and Maritime Sciences, Faculty of Engineering Sciences, University Austral of Chile, Valdivia, Chile (2004).

5. Thornhill, E., Bose, N., Veitch, B. and Liu, P. "Planing hull performance evaluation using a general purpose CFD code", *24th Symposium on Naval Hydrodynamics*, The National Academy of Sciences (2003).

6. Moraes, H.B., Vasconcellos, J.M. and Latorre, R.G. "Wave resistance for high speed catamarans", *Journal of Ocean Engineering*, **31** pp. 2253-2282 (2004).

7. ITTC Recommended Procedures and Guidelines, Testing and Extrapolation Methods, General Guidelines for Uncertainty Analysis in Resistance Towing Tank Tests (2008).

8. Jones, D.A. and Clarke, D.B., *Fluent Code Simulation of Flow Around a Naval Hull: the DTMB 5415*, Defense Science and Technology Organization (2010).

Biographies

Abouzar Ebrahimi was born in Iran, in 1982. He received a BS degree in Naval Architectural Engineering from the Persian Gulf University of Bushehr, Iran, in 2004, and an MS degree in Marine Engineering (hydrodynamics) from Sharif University of Technology (SUT), Tehran, Iran in 2008.

He is currently working in the Department of Marine Engineering at Chabahar Maritime University, as a Lecturer. His main research interests include marine hydrodynamics, high speed vessels, ship design and computational fluid dynamics.

He has published some scientific papers and a book in the field of marine engineering and has reviewed more than 100 papers in related conferences.

Manocher Rad was born in Iran, in 1942. He received a PhD degree from Imperial College at the University of London, UK, in the field of Mechanical Engineering, and is currently faculty member of the Mechanical Engineering Department at Sharif University of Technology (SUT), Tehran, Iran, and the Science and Research Branch of Azad University, Tehran, Iran.

He has published a large number of scientific papers and books in the field of aero-hydrodynamics and fluid mechanics. His research interests include fluid mechanics, aerodynamics, marine hydrodynamics and drag reduction.

Ali Hajilouy Benisi received a PhD degree from the Mechanical Engineering Department of Imperial College at the University of London, UK, in 1993. He has been faculty member of the Institute of Water and Energy and then the School of Mechanical Engineering at Sharif University of Technology (SUT) since 1978.

He served as Director of the Fluid Mechanics Laboratory from 1993–1995, Founder and Director of the Turbocharger Laboratory from 1993, Founder and Director of the Turbocharging Laboratory from 2000, Founder and Director of the Gas Turbine Laboratory from 2008, at the School of Mechanical Engineering of SUT. He has also served as Research Director of SUT from 1993-1995.

His research interests are experimental and theoretical investigations of turbochargers, turbocharging, and gas turbines.

He is author or co-author of over 90 conferences and journal papers, coordinator for compiling and publishing four volumes of research reports for SUT in 1991-1995, and translator of an English text book to Farsi (Persian). He has also supervised over 40 graduate theses, and conducted more than seven research funded projects.