# Methodology for Improvement of the Hydrodynamic Efficiency of an Amazon School Boat Utilizing a CFD Tool

**Breno Farias da Silva** , **Fernando Costa da Cruz** ,
**Harlysson Wheiny Silva Maia** , **Toshi-Ichi Tachibana** ,
**Vitor Hugo Macedo Cardoso** **and Yuri Victor Remígio Guedes**

**Abstract** The Solimões—Amazonas waterway is mainly composed of the Solimões and Amazonas Rivers, along with several tributaries of both, forming a network of almost 7000 km in length. Due to this, the waterway modal stands out in the transportation of passengers and cargo in the Amazon region. In sight of this scenario, the National Fund for Development of Education implemented in the region the "School Boat" project, originating from the Caminhos da Escola (School Path) program, whose objective is to provide a faster, safer, and more efficient transportation of students from the riverside regions to the schools. However, the medium boat model was the subject of complaints from users about the long duration of the journey. In order to analyze the hydrodynamic characteristics of the medium boat, the full-scale sea trials performed showed high residuary resistance values. Hence, it was noted the generation of a large amplitude wave train and a high bow wave height compared with the boat's depth, implied a high fuel consumption for speeds that are incompatible with the design guidelines. These characteristics corroborate the low hydrodynamic efficiency of the hull. Based on this problem, the proposal of this work consists in the analysis of a numeric model for the medium school boat hull, based in CFD, correlated with the full-scale sea

B. F. da Silva · F. C. da Cruz · H. W. S. Maia (✉) · V. H. M. Cardoso
Y. V. R. Guedes
Federal University of Pará, Belém, PA, Brazil
e-mail: harlysson.maia@itec.ufpa.br

F. C. da Cruz
e-mail: fernando.cruz@itec.ufpa.br

V. H. M. Cardoso
e-mail: vitor.cardoso@itec.ufpa.br

T.-I. Tachibana
Polytechnic School of the University of São Paulo, São Paulo, SP, Brazil
e-mail: tatibana@usp.br

trials. In this scenario, the hull lines are modified and simulated, in comparison with the original model, in order to reduce the total resistance to advance of the vessel.

**Keywords** Hydrodynamic · Simulation · CFD

## 1 Introduction

This work was conceived with the objective of producing a consistent solution to the problem of hydrodynamic efficiency of the hull shape of a riverboat. The studied hull model has specific problems to reach the design speed (11 knots). Based on this, the main objective of this work is to increase the hydrodynamic efficiency of this hull within a range of operating speeds.

In general, the problem is related to the shape of the bow and the keel region. These regions hinder the flow of the incident fluid, the generation of hydrodynamic lift for entering into the planing regime, and still facilitate the generation of bow waves within the displacement regime. In this case, the elaboration of evaluation criteria to identify discontinuities in the hull is necessary, as the design velocity is not reached. Thus, the proposed approach is the application of tools in a computational platform, interconnected to the real-scale tests of the boat.

The speed of service and the wave train profile of the current hull form were determined by sea trials. Based on the geometric characteristics and initial conditions of the sea trial, it was possible to define the computational model of the hull geometry as well as the fluid domain, along with turbulence conditions and the phase model. Under these conditions two numerical hull models were created, one representing the current model and another with proposed geometric variations of the first.

## 2 School Transportation in the Amazon Region

The proposed hull model has a shape with displacement characteristics, emphasizing that the hull structure is composed entirely of naval steel with sufficient tensile strength to maintain the structural integrity with respect to the forces from the external environment of the vessel, as shown in Figs. 2 and 3.

Judging the inconsistencies with the hull shape criteria as well as the conditions to reach the design speed of the vessel, the need to remodel the hydrodynamic behavior of the hull becomes clear, keeping the design guidelines fixed and the main dimensions unchanged.

For the computational analysis method, it is necessary to validate the shape of the model through computational fluid dynamics (CFD). Table 1 gives the main characteristics of the vessel studied in this work.

**Table 1** Main characteristics of the school boat

| Characteristic | Dimension |
| --- | --- |
| Length Overall (LOA) | 7.80 m |
| Length between Perpendiculars (LPP) | 7.52 m |
| Molded breadth | 2.55 m |
| Molded depth | 1.40 m |
| Design draught | 0.67 m |
| Design speed | 20 km/h |
| Capacity | 20 passengers + 1 crewman |

*Source* National Fund for Development of Education (FNDE)

## 3  Literature Review

With the development of computer-aided design (CAD) and CFD tools, several processes for measuring and analyzing issues regarding the area of naval hydrodynamics can be studied with greater accuracy [2–4]. Theoretical models used as the basis for this work are presented below.

### 3.1  Experimental Method on Ship Hydrodynamics

According to Tachibana [4], from the assumption that the flow around the hull of a ship is three-dimensional, the process can be analyzed from the individualization of each resistance component. By this means, an adequacy is identified in the area of flows around hull forms, within the area of ship hydrodynamics.

Most of the "pure" resistance components and the existing intersections do not have satisfactory experimental means of verification, being subjected to imprecise numerical measurement and extrapolation processes, with a tradition in experimental hydrodynamics within the area of determination of resistance components.

### 3.2  Resistance to Forward Motion

The movement of a vessel through a fluid (water), at constant speed, generates two types of forces against the hull, normal and tangential. The force against the movement of the vessel is called resistance to forward motion [1].

The interaction between fluid and structure results in generation and destruction of waves, and this demanded energy composes the wave resistance. A schematic diagram with detailed components of the resistance to forward motion is presented in Fig. 1, based on Tachibana [4].
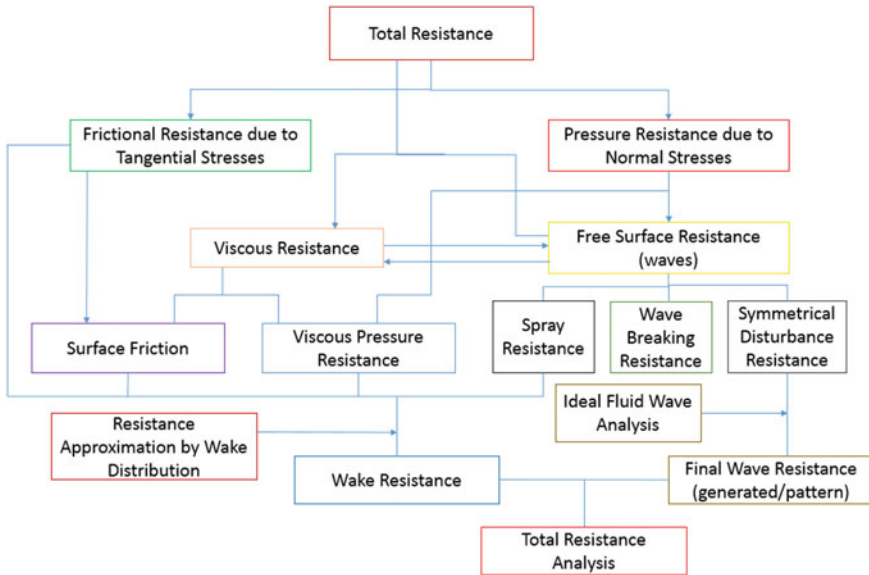
**Fig. 1** Schematic diagram of the components of the resistance to forward motion

Due to the no-slip condition of viscous fluids on solid surfaces, when the vessel moves through a fluid at rest, the particles close to the hull tend to adhere to the surface, acquiring the speed of the vessel. The integral of the friction components on the wetted surface of the hull results in the frictional resistance [1].

## 3.3 CFD Analysis of Vessels

According to Iervolino [5], the computational fluid dynamics is characterized by a numerical simulation of any physical or chemical process where a fluid flow happens. The application of boundary conditions on several finite element models can be recorded on the time and space domain, based on the behavior of several parameters of the flow itself in each of these elements.

Within this procedure, the separation of the model into three main stages is highlighted: preprocessing, solution, and postprocessing. In the preprocessing stage we define the flow model and the boundary conditions of the problem. In the solution step, the resolution procedures are used to determine the desired results. Finally, the postprocessing phase allows the analysis of the results with illustrative graphics.

# 4　Methodology

## 4.1　Computational Modeling of the Medium Boat Hull

The full-scale model of the medium boat used on the sea trials is illustrated in Fig. 2. The three-dimensional model was generated from the lines plan provided by the manufacturer.

The profile view of the medium school boat hull and the model with the proposed shape modifications are illustrated in Fig. 3.

The modified hull shape was generated based on the original hull, by removing the side keels at the bow and modifying the flat bottom area to a V-shaped keel, in order to attenuate the flow lines by the hull.

A view of the transverse sections of the bow, displaying the proposed shape modifications is illustrated in Fig. 4.

In Fig. 5, the three-dimensional hull models are illustrated, as exported from the 3D CAD software Rhinoceros 5.0.



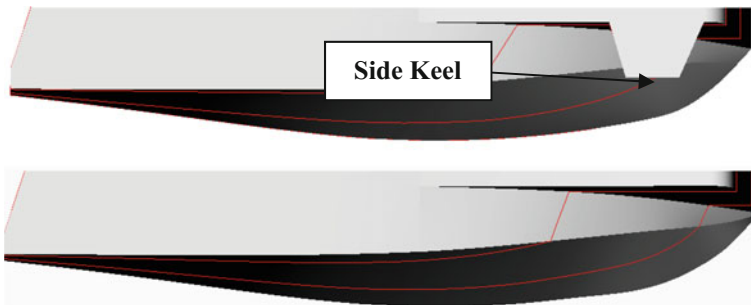**Fig. 2** Longitudinal view of the medium school boat



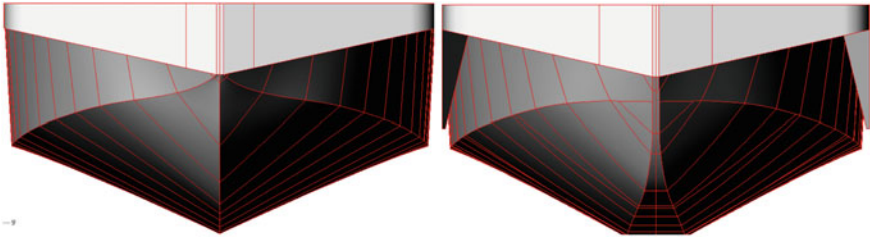**Fig. 3** Profile view of the unchanged hull with side keel (above) and the modified hull (below)

**Fig. 4** View of the transverse sections of the bow. Left: the modified hull shape, and right: the unchanged hull of the medium school boat
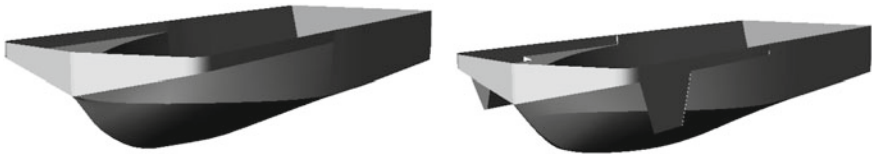


**Fig. 5** Three-dimensional hull model with the proposed modifications, with a V-shaped bottom keel on the left, and the unchanged medium school boat hull on the right

With the three-dimensional hull models modeled, the next phase is to define the fluid control volume that the water and air will occupy around the boat in the simulation. The dimensions of the fluid control volume are illustrated in Fig. 6, related to the overall length of the boat, defined as $L$.

For the criterion analysis, it is important to analyze the wave resistance generated in the case, using a resistance model that simulates the interaction of the hull discontinuities effects on the generation of the wave train, taking into account the vorticity formation in abrupt shape changes in the hull.

## 4.2 Physical Model Considerations

For the analysis of the wave formation, a transient analysis is used, as the hull geometry is complex and the calculation model used in the permanent analysis generates numerical inconsistencies that must be taken into account. The section that represents the hull is symmetrical in shape with the plane of symmetry, thus only the port side is simulated.

Into the case conditions of operation, the following physical parameters are defined: the gravitational acceleration: $g_z = -9.81\,\text{m/s}^2$, the density of operation: $\rho_{op} = 1.225\,\text{kg/m}^3$, the pressure of operation: $p_{op} = 101,325\,\text{Pa}$, and the reference position of pressure: $z_{ref} = L$, where $L$ is the overall length of the hull. For the multiphase model, the volume of fluid method is used, considering the flow along
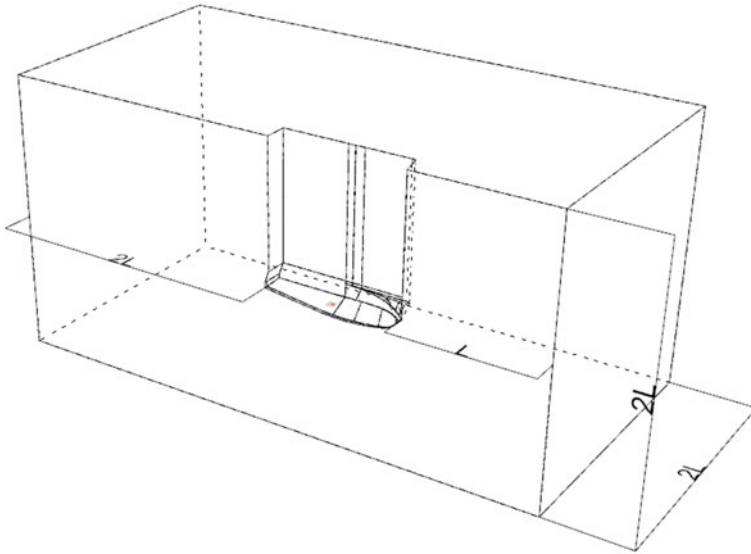
**Fig. 6** Dimensions of the fluid control volume surrounding the boat hull, related to the overall length of the boat, $L$. The control volume is a box with dimensions $4L \times 2L \times 2L$ (port)

the control volume an open channel flow type, and the implicit body force model for force delimitation.

The turbulence model used is the SST K-$\omega$ and the Flat method is used for the phase initiation.

## 5   Results

The definition of the parts of the control volume was adjusted to the model of calculation used by the software Ansys Fluent. The surfaces of the control volume were named according to the regions of analysis. The water inlet represents a cross-section of the control volume, namely the entrance of the fluid flow (air + water) for the analysis. The water outlet corresponds to a cross-section of the control volume where the fluid flow (air + water) exits the channel in the analysis. The symmetry section delimits a symmetry plane for half the control volume. The walls that surround the control volume are defined by a no-slip condition, thus the model has reliability to measure the degree of resistance.

On Fig. 7 it is possible to check the details of the mesh used for the computational simulation in Fluent.

The orthogonal quality of the mesh can be verified, highlighted in the legend. No refinement tool was used around the hull. The mesh was generated with a minimum element size on the order of 5 cm. A total of 452,461 nodes and a total number of 2,577,166 elements were generated.
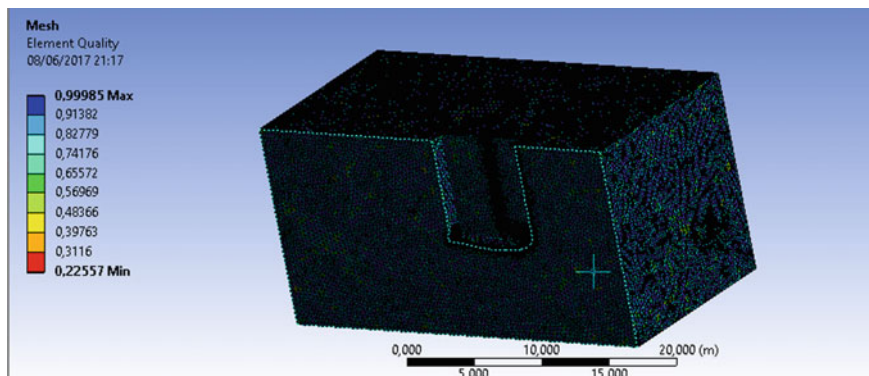
**Fig. 7** Indication of the mesh elements present in the fluid control volume around the hull, with the legend indicating the orthogonal quality of the mesh

According to the values indicated in Fig. 7, it can be verified that about 2,096,942 elements presented an orthogonal quality of over 0.75, and 1,510,942 elements presented an orthogonal quality of over 0.85. The quality of the mesh did not reach higher values because the model was based on tetrahedral elements with a random method for filling the control volume.

The angle of curvature used corresponded to 18°, with a minimum gap size of 1.5 cm and a maximum element size of 30 cm. A growth rate of 1.4 was used for the elements of the hull region and for the outermost regions of the control volume.

For the calculation of 1500 iterations of the computational model, a total of 30 iterations per timestep was defined, resulting in 50 timesteps for each velocity. The timestep duration was defined as 0.02 s. The restriction degree used for the residual convergence was on the order of $10^{-5}$. Figure 8 illustrates the longitudinal profile of the medium school boat with the free surface wave profile for a velocity of 20 km/h.

The wave elevation in the region of the side keel is illustrated in Fig. 9. The irregularities in the surface of the hull can be noticed due to the mesh quality used in this region.

Based on the results of the three analyses (sea trials and both numerical hull models), the methodology was validated, and based on the proposed hull shape modifications, a maximum reduction of the total resistance of about 41% for a velocity of 25 km/h was reached. Figure 10 illustrates the progression of the total resistance for the range of velocities simulated for both hulls, the medium school boat and the model with the proposed modifications.

Figure 11 illustrates the percentage reduction of resistance for each velocity in the simulated range, varying from 8 to 25 km/h.

The largest reduction rate was identified on the velocity of 25 km/h, as seen in Fig. 11. For this velocity, the corresponding value for the Froude number is equal to
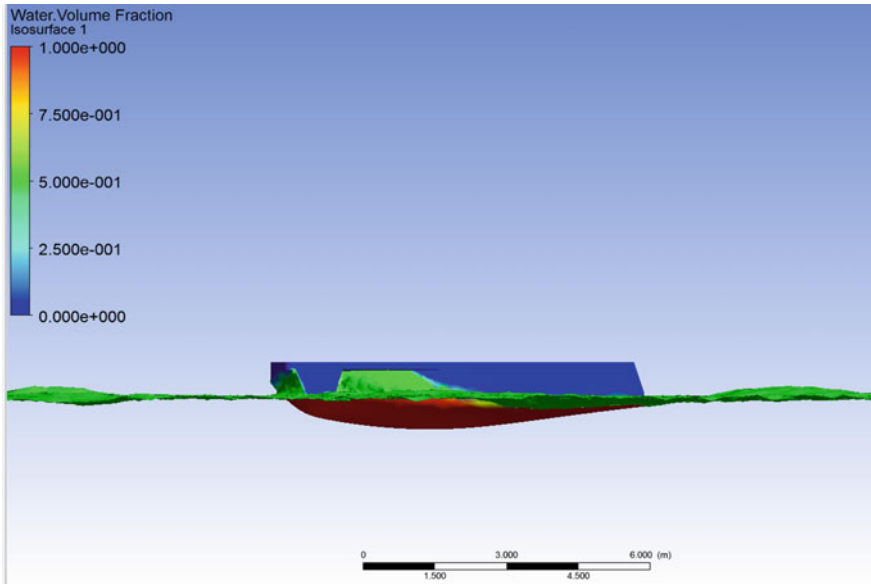
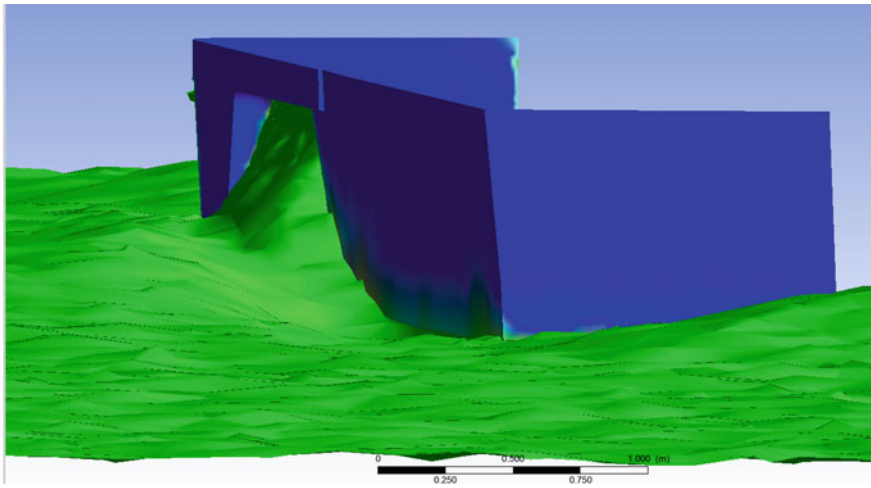**Fig. 8** Free surface wave profile at 20 km/h



**Fig. 9** Wave elevation at the bow region

0.794. Relating the Froude number value with the large resistance reduction achieved, a change on the planing mode of the proposed hull is identified. The absolute resistance values are presented in Table 2.
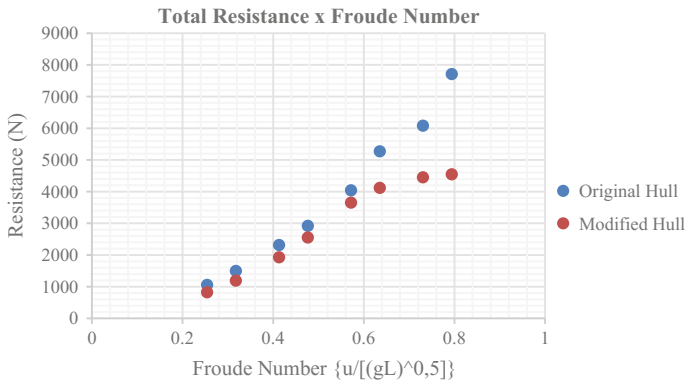
**Fig. 10** Total resistance values in relation to the Froude number for both hull models. Blue dots: Original hull of the school boat. Orange dots: Modified hull shape
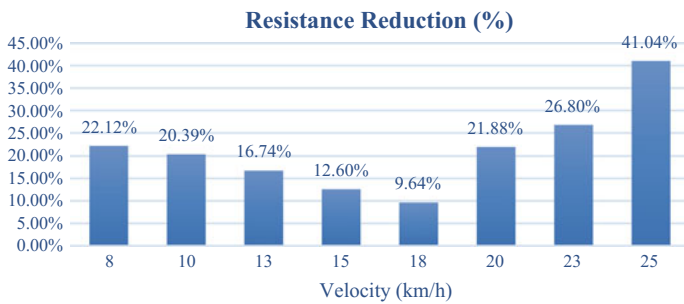


**Fig. 11** Percentage of resistance reduction for each velocity

**Table 2** Total resistance values obtained in CFD simulation

| Velocity (km/h) | Froude # ($V/\sqrt{gL}$) | Total resistance/modified hull | Total resistance/original hull |
|---|---|---|---|
| 8 | 0.254 | 822.38 N | 1055.94 N |
| 10 | 0.318 | 1190.79 N | 1495.69 N |
| 13 | 0.413 | 1926.57 N | 2313.86 N |
| 15 | 0.476 | 2551.10 N | 2919.01 N |
| 18 | 0.572 | 3651.21 N | 4040.74 N |
| 20 | 0.635 | 4117.62 N | 5271.12 N |
| 23 | 0.730 | 4450.65 N | 6079.81 N |
| 25 | 0.794 | 4545.24 N | 7709.15 N |

## 6 Final Considerations

With the results obtained in this work, a great capacity of variation of the total resistance of the school boat's original hull was identified for subtle changes in the hull shape. The calculated resistance for the model was based on the viscous pressure and wave resistance components of the vessel.

Therefore, the proposed objective of the work was achieved, with a large reduction of the total resistance to forward motion in the range of velocities analyzed. With the present study, a proposal for geometric modification after a hydrodynamic analysis resulted in a maximum reduction of 41.04%. For future studies, a parametric optimization can be done in order to propose further modifications on the bow frames of the boat, related to the wave resistance.

In this way, an even more significant reduction of the resistance can be achieved for the proposed speed range, in order to make the operation of the medium school boat economically feasible. Thus, students residing in more isolated areas of the Amazon basin, where the operation of the large school boat is impracticable, can be benefited with a quality school transport.

We would like also to thank the entities CAPES and FAPESPA for their contribution with research grants to the students, which is of great importance in directing academic work within the area of academic article publication.

## References

1. Bertram, V.: Pratical ship hydrodinamics, 1st edn. Butterworth-Heinemann, England (2000)
2. Coelho, D.F.: Evaluation of the hydrodynamic behaviour of a school boat using CFD, pp. 72. Federal University of Pará—UFPA, Belém-PA, Brazil (2016)
3. Graefe, A.V. et al.: Comparison of Aqwa, Gl Rankine, Moses, Octopus, Pdstrip and Wamit with model test results for cargo ships wave-induced motions in shallow water. In: ASME 2015 34th International Conference on Ocean, Offshore and Arctic and Artic Engineering OMAE 2015, Canada (2015)
4. Grinberg, M., Padovezi, C., Tachibana, T.: Utilization of model testing in reduced scale for definition of optimized ship shapes. In: XXII COPINAVAL—IPIN, pp. 28. Buenos Aires, Argentina (2017)
5. Iervolino, L.A.: Study of the resistance to forward motion of a planing vessel of 20 feet: computational approach based on CFD. Undergraduate Thesis, Federal University of Santa Catarina—UFSC, pp. 88. Joinville, SC, Brazil (2015)