An Overview of Stepped Hull Performance Evaluation: Sea Trial Data vs Full-Scale CFD Simulation

Luigi VITIELLO[a,1] Simone MANCINI[b] Rasul NIAZMAND BILANDI[c] Vincenzo NAPPO[d]

[a] *Università di Napoli "Federico II", Department of Industrial Engineering, Italy*

[b] *Force Technology / Danish Maritime Institute, Kgs. Lyngby, Denmark*

[c] *Persian Gulf University, Būshehr, Iran*

[d] *MV Marine Srl*

Abstract. It is well known that the dynamic of the stepped hull in real scale is rather complex and it's not easy to predict that using empirical or mathematical approaches, and by the numerical and experimental way as well. Moreover, there is a huge lack in the literature of data related to sea trials of the stepped hull. Furthermore, the reliability of full-scale CFD simulations is not widely proven and validated especially for high speed and planing hull. For these several reasons, in this paper, the authors are focused on the comparison of the results carried out from model experimental tests performed in the model basin, full-scale CFD simulations, and sea trial tests. The performed simulations in full-scale have been compared to the extrapolated experimental tests and the sea-trial results. Moreover, the dynamic trim angle and the dynamic wetted surface have been taken into account to assess the reliability of the full-scale simulation performed. The stepped hull considered is a Mito 31 outboard Rigid Inflatable Boat (RIB) built by MV Marine Srl Company.

Keywords. CFD, full-scale simulation, sea trial data, stepped hull, planing hull, residuary resistance, friction resistance.

1. Introduction

Planing hull with steps on the bottom represents an improvement of the hydrodynamic behavior of these types of crafts. The step on the bottom of the planing hull cause flow separation from the steps and create an air cavity. Flow separation and air cavity form due to steps cause a decrease in the wetted surface, a decrease in drag to lift ratio, and uniform pressure distribution on the bottom of the stepped planing hull (Savitsky and Morabito [1], Niazmand Bilandi et al. [2]). The height and the longitudinal position of the step represent a non-trivial issue on the hydrodynamic of stepped planing hull (Niazmand Bilandi et al. [3]).

There are several experimental, numerical, and mathematical studies that are led to consider the hydrodynamic behavior of stepped planing hulls. The most important studies on the performance of the planing hull were performed by Clement and Blount

[1] Luigi Vitiello, Università degli Studi di Napoli *"Federico II"*; Napoli, Via Claudio 21, Italy, luigi.vitiello@unina.it.

[4], Savitsky [5], and Savitsky and Brown [6]. In particular, all formulations (drag and lift forces) were for planing hull without step. Creating steps on the planing hull has caused the bottom to be divided into several bodies. The combination of these body forces must provide for the vertical equilibrium of the stepped planing hull (Savitsky and Morabito [1], Dashtimanesh et al. [7], Niazmand Bilandi et al. [8]). After presenting the Savitsky model, researchers such as Svahn [9], Savitsky and Morabito [1], and Niazmand Bilandi et al. [8] were trying to enhance the Savitsky model into stepped planing hull. Indeed, Svahn [9] developed a mathematical model for performance prediction of one stepped hull and uses Savitsky and Morabito's [1] formulas for separated flow behind the step. Then, Niazmand Bilandi et al. [8&10] developed a mathematical model for one and two-stepped planing hull by using 2D+t approach and linear wake theory. Niazmand Bilandi et al. [11] shown that the 2D+t approach, numerical study, and experimental data are widespread and credible for a stepped planing hull. The Marine CFD Group of Eurisco Consulting Srls also brought its contribution to this study.

For the 3D Computational Fluid Dynamics (CFD) method, the full Navier-Stokes equation is solved for the flow in a fluid domain. Also, numerical methods, such as those based on CFD simulations, can be used to calculate the water entry problem for boat sections (Niazmand Bilandi et al. [12]), roll motion (Mancini et al. [13]), Self-propulsion using virtual disk (De Luca et al. [14], Roshan et al. [15]), hydrodynamic performance of a stepped hull (De Marco et al. [16], Cucinotta et al. [17], Mancini et al. [18]) and design and optimization of stepped planing hull (Di Caterino et al. [19]). All of these researches reached that the stepped hull is very efficient hydrodynamically in alternative another type of vessel more efficient is the hydrofoil as reported in Giallanza et al. [20].

In the present paper, the stepped planing hull with two steps was investigated through experiment (with 1:10 scale) and CFD (full scale) and sea trial. Operating conditions were considered in calm water in a range of Froude Numbers (*Fr*) from 0.5 to 3.0. The rest of the paper is organized as follows. The experimental tests are presented in Section 2. The numerical model is presented and discussed in Section 3. Section 4 presents the main results of the current paper, including measurements and computations. A summary of the current research is presented in Section 5.

2. Experimental and sea trial tests

The hull considered for the present study is the Rigid Inflatable Boat (RIB) MITO 31 built by the MV Marine Srl shipyard in Italy. Details of the hull are available in Table 1 and a sketch of the boat is shown in Figure 1. This hull was tested in the Towing Tank and after the sea-trials were performed and recorded as shown in Miranda and Vitiello [21].

The experimental tests were conducted at the Towing Tank of the Naval Section of the Department of Industrial Engineering of the Università Degli Studi di Napoli "Federico II". The main dimensions of the basin are 137.5 m length, 9 m width, and 4.25 m depth. The towing carriage can reach a maximum continuous speed of 10 m/s with a maximum acceleration of 1 m/s^2. The tests were performed following the Froude methodology with the ITTC'57 [22] friction line. The scale model was defined considering the maximum ship and carriage velocity.

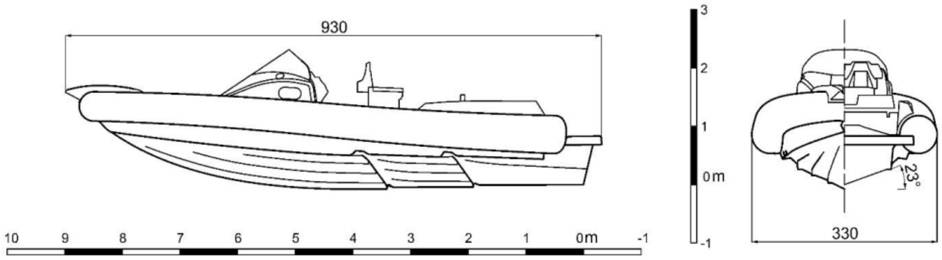

Figure 1. RIB Mito 31 sketch – side and front/back view.

About the experimental tests, a non-standard procedure, called "Down Thrust" (DT), has been adopted to carry out these tests.

This procedure has the towed point located in the hull, in particular in a point obtained through the intersection between the keel line and the engine thrust direction translated in the lower area of the engine bracket. The reason for the choice of the DT measurement solution is due to the high sensitivity of the hull model to the externally applied forces, i.e., instrumentation weight. The DT method releases the tested model from the instrumentation constrain and weight, and promotes higher accuracy in measurements of resistance, sinkage, and trim. Today this solution is the only able to reproduce the real system of forces exerted by the outboard engines. More details about this method are available in Miranda and Vitiello [21] and De Marco et al. [16].

About the sea trial tests, all the details about the procedures followed are available in Miranda and Vitiello [21].

Table 1. Geometrical details of RIB Mito 31.

	Unit	Value
L_{OA} (Length overall)	[m]	9.35
L_{WL} (Length waterline)	[m]	7.55
B_{MAX} (Breadth max)	[m]	3.35
T_M (Draft amidship)	[m]	2.40
Deadrise angle at transom	[°]	23.0
Displacement	[t]	3.13

3. Numerical tests and procedures

3.1. Theoretical Background

The full-scale simulations were carried out using the Unsteady Reynolds Averaged Navier-Stokes (URANS) approach. The equation for incompressible flow along with the continuity equation is given below:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \mathrm{U_i}}{\partial \mathrm{t}}+\frac{\partial\left(\mathrm{U}_i \mathrm{U}_j\right)}{\partial \mathrm{x_j}}=-\frac{1}{\rho}\frac{\partial \mathrm{P}}{\partial \mathrm{x_i}}+\frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial U_i}{\partial \mathrm{x_j}}+\frac{\partial U_j}{\partial \mathrm{x}_i}\right)\right]-\frac{\partial \overline{\mathrm{u_i'u_j'}}}{\partial \mathrm{x_j}} \tag{2}$$

where U_i is the mean velocity in i^{th} direction of the cartesian coordinate; ρ is the density, P is the mean pressure, $u'_i u'_j$ is Reynolds stress and v is the kinematic viscosity. In the present study, the *k-ω SST* turbulence model is used for modeling Reynolds stress (τ_{ij}). Detailed information about the turbulence model can be found in Wilcox [23, 24].

3.2. Numerical setup

The URANS simulations were conducted through the commercial CFD code Siemens PLM Star CCM+. A Semi- Implicit Method for Pressure-Linked Equations (SIMPLE) to conjugate pressure and velocity field has been used to find the field of all hydrodynamic unknown quantities, and an Algebraic Multi-Grid (AMG) solver was used to accelerate the convergence of the solution. A segregated flow solver approach has been used for all simulations. The free surface has been modeled with the two-phase VOF approach with a High-Resolution Interface Capturing (HRIC) scheme based on the Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM).

The wall treatment approach utilized for all simulations is the *All Wall y+*. This is a hybrid approach that emulates the *high y+* wall treatment for coarse meshes (for $y+>30$), and the *low y+* wall treatment for fine meshes (for $y+\approx 1$). Furthermore, this approach gives a reasonable answer for meshes of intermediate resolution (for $y+$ in the buffer layer), as depicted in Siemens PLM Star-CCM+ v 2019.1 User's Guide [25].

The URANS simulations were carried out using the Overset/Chimera grid to follow the hull motions. The linear interpolation method has been applied to establish the connectivity between the background and the overset region. More details about this approach are available, for instance, in De Luca et al. [14], and Begovic et al. [26].

The boundary conditions applied and the computational domain dimensions are shown in Figure 2 and these dimensions comply with the ITTC guidelines [27]. Furthermore, the time step size is determined through the formula suggested by the ITTC guidelines [27]. The setup adopted for all the simulations is substantially similar to the simulation setup for the overset grid case exposed in De Marco et al. [16].

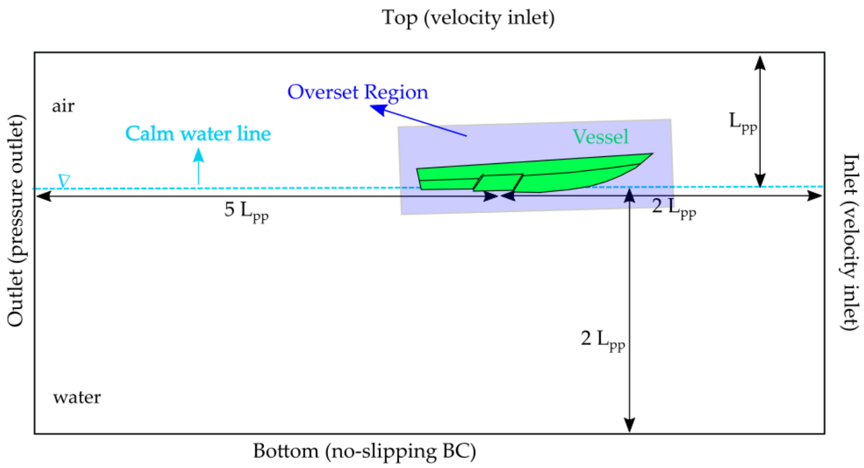

Figure 2. Computational domain with applied boundary conditions.

4. Results and discussions

The sea trial and the extrapolated towing tank results are compared against full-scale CFD simulation analyzing the following three parameters: total resistance (Figure 3), dynamic trim angle (Figure 4), and dynamic wetted surface (Figure 5). The resistance values of the sea trials have been derived from the delivered power through the propeller efficiency and propulsive coefficients (η_0, η_R, η_H) estimated with the self-propulsion tests carried out for the MITO 31 hull and available in Miranda and Vitiello [21]. The equation used is the following.

$$R_T = P_D / (V \eta_r \eta_0 \eta_H) \tag{3}$$

where V is the speed of the boat. In this evaluation, it is assumed that there is no loss in the shaft, which is a reasonable assumption since there is a negligible mechanical connection in the typical outboard engine for small high-speed boats.

The extrapolated towing tank results have the same trend of the sea-trials but overestimating the resistance by approximately 25–30% with an average of 27.5% compared to the sea trial results. Considering that towing tank tests are conducted at a much lower Reynolds number and extrapolated, hence is it important to remember that the extrapolation of the towing tank results is performed using the standard ITTC'57 procedure [22].

Full-scale CFD simulations have a trend intermediate between the extrapolated towing tank results and sea trials. Moreover, it can be seen in Figure 5 that the CFD overestimates the sea trial resistance by approximately 7–30% with an average of 17.3% and at Fr = 2.33 is extremely close to the extrapolated towing tank resistance value.

The CFD data are provided with the uncertainty bars derived from a previous verification and validation analysis on a simulation campaign with a similar stepped hull (De Marco et al. [16]).

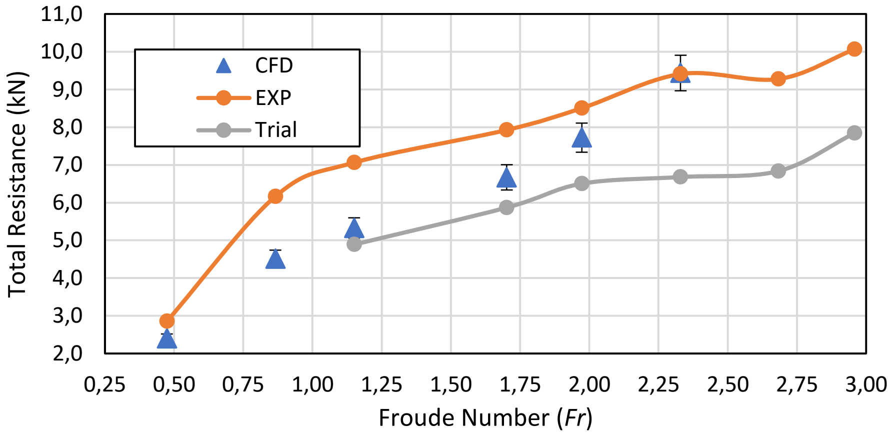

Figure 3. Total resistance comparison between experimental data, CFD (with uncertainty bar), and sea trial.

The dynamic trim angle, differently from the hull resistance, show, for all the different approaches, more similar trends, as clearly visible in Figure 4. It is observed that the results of the full-scale CFD are more close to the extrapolated full-scale towing tank values but underestimate the dynamic trim angle peak. Instead, the extrapolated full-scale towing tank results underestimate the sea trial trim angle of an average of 17.7%.

Hence, based on these comparisons, it's possible to observe the reliability of CFD full-scale simulations to predict the stepped hull performance well than the extrapolated full-scale towing tank results. This behavior was already detected for a displacement hull in a recent study (Mikkelsen et al. [28]).

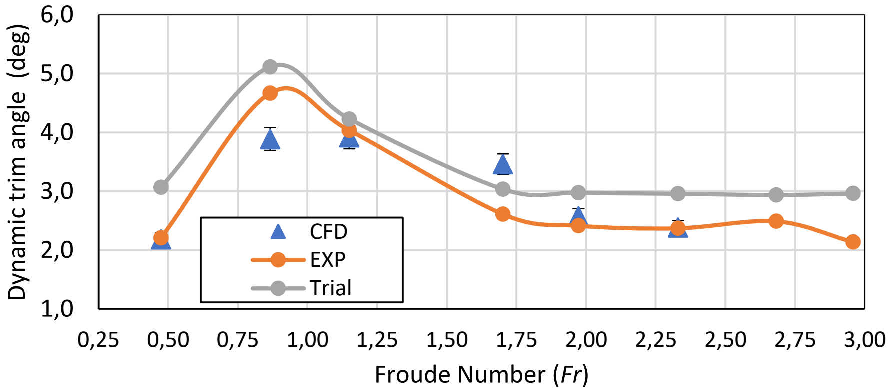

Figure 4. Dynamic trim angle comparison between experimental data, CFD (with uncertainty bar), and sea trial.

About the dynamic wetted surface, only the data from CFD and towing tank tests are available. The comparison in Figure 5 show that the CFD full-scale wetted surface is consistent with the full-scaled towing tank values, the main discrepancy is detected only for the lowest and highest *Fr* values. The overprediction is of an average of 11.0% with a minimum of 1.8% and a maximum of 24.5%. This comparison errors could appear huge values but is noteworthy to mention that is largely in line with similar analysis, for instance, De Luca et al. [14].

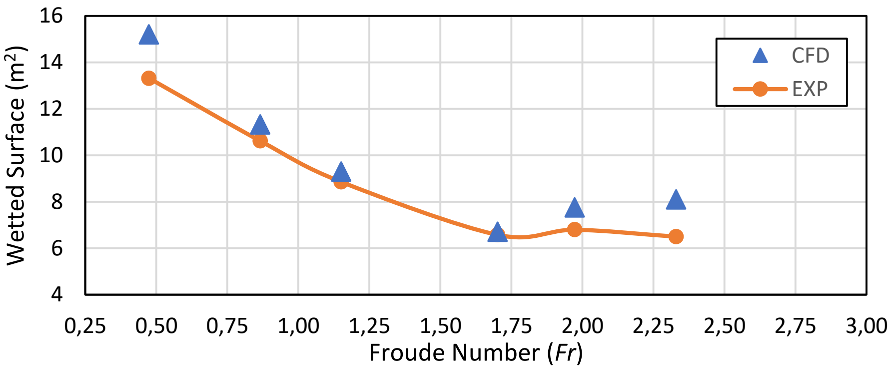

Figure 5. Dynamic wetted surface for full-scale CFD and full-scaled towing tank.

Based on these comparisons it's possible to observe that the full-scale simulations appear to be reliable especially observing the good agreement predicting the dynamic trim angle and the wetted surface. Anyway, the discrepancy detected for the resistance among the three different sets of data need a deeper investigation, that is a part of the further on-going investigation.

5. Conclusions

Full-scale resistance CFD simulations for the double-stepped MITO 31 RIB hull have been performed and compared against extrapolated full scale towing tank results and sea trials. The results of the full-scale CFD simulations showed an overestimation of the resistance by approximately 17.3% on average compared to the sea trial results. The experimental towing tank test results have been extrapolated to ship scale using the ITTC standard procedure. It was found that this prediction overestimated the hull resistance with 27.5% on average compared to the sea trial results. Differently from the resistance for the trim angle, the sea trials test gives a result slightly higher than the towing tank test and the CFD full-scale results are substantially close to the towing tank data. Anyway, all the results for the dynamic trim angle show the same trends. Similarly to the dynamic trim angle, the dynamic wetted surface predicted by full-scale CFD simulations shows a good agreement with the towing tank scaled wetted surface.

Based on these comparisons, the CFD full-scale simulations seems to be a reliable tool for the prediction of the stepped hull performance. Anyway, the full-scale simulations in general, and especially for the high-speed craft, are far to be a mature tool and needs more investigations and extended campaign of verification and validation: Unfortunately, it is not so easy to find available and reliable data set of sea trials for the high-speed craft.

References

[1] Savitsky, D., Morabito, M., 2010, Surface wave contours associated with the forebody wake of stepped planing hulls. Marine Technology, 47, pp. 1-16.

[2] Niazmand Bilandi, R., Dashtimanesh, A., and Tavakoli, S., 2020, Hydrodynamic Study of Heeled Double-Stepped Planing Hulls Using CFD and 2D+t Method. Ocean Engineering, 196, 106813.

[3] Niazmand Bilandi, R., Mancini S., Dashtimanesh, A., Tavakoli, S., De Carlini, M., 2019, A Numerical and Analytical Way for Double-Stepped Planing Hull in Regular Wave. MARINE 2019 VIII International Conference on Computational Methods in Marine Engineering (2019), Goteborg, Sweden.
[4] Clement, E. P., and Blount, D, 1963, Resistance Tests of a Systematic Series of Planing Hull Forms. SNAME Transactions, 71 pp. 491-579.
[5] Savitsky, D., Hydrodynamic design of planing hulls. Marine Technology 1964; 1 pp. 71-95.
[6] Savitsky, D., and Brown, PW, 1976, Procedures for Hydrodynamic Evaluation of Planing Hulls in Smooth and Rough Water," Marine Technology, Vol. 13, No. 4, pp. 381-400
[7] Dashtimanesh A, Tavakoli S, Sahoo P., 2018, A Simplified Method to Calculate Trim and Resistance of a Two-Stepped Planing Hull." Ships and Offshore Structures, 12(sup1), S317-S329.
[8] Niazmand Bilandi, R., Dashtimanesh, A., and Tavakoli, S., 2019, Development of a 2D+ T Theory for Performance Prediction of Double-Stepped Planing Hulls in Calm Water. Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment, 233(3), pp. 886-904.
[9] Svahn, D, 2009, Performance Prediction of Hulls with Transverse Steps (M.Sc. thesis), KTH Centre for Naval Architecture, Stockholm, Sweden.
[10] Niazmand Bilandi, R, Mancini, S, Vitiello, L, Miranda, S, De Carlini, 2018, A Validation of Symmetric 2D + T Model Based on Single-Stepped Planing Hull Towing Tank Tests, Journal of Marine Science and Engineering, 6, 136.
[11] Niazmand Bilandi, R., Vitiello, L., Mancini S., Nappo, V., Tavakoli, S., Roshan, F., Dashtimanesh, A., 2020, Calm-water performance of a boat with two swept steps at high-speeds: Laboratory measurements and mathematical modeling, Procedia Manufacturing 42, 467-474.
[12] Niazmand Bilandi, R., Jamei, S., Roshan, F., and Azizi, M., 2018. Numerical Simulation of Vertical Water Impact of Asymmetric Wedges by Using a Finite Volume Method Combined with a Volume-Of-Fluid Technique. Ocean Engineering, 160, pp. 119-131.
[13] Mancini, S.; Begovic, E.; Day, A.H.; Incecik, 2018, A. Verification and Validation of numerical modelling of DTMB 5415 roll decay. Ocean Eng., 162, 209–223.
[14] De Luca, F., Mancini, S., Miranda, S., Pensa, C., 2016, An Extended Verification and Validation Study of CFD Simulations for Planing Hulls, Journal of Ship Research, Vol. 60, No. 2, pp. 101–118.
[15] Roshan, F; Dashtimanesh, A; Tavakoli, S; Niazmand Bilandi, R; Abyn, H; 2020, Hull–propeller interaction for planing boats: a numerical study, Ships and Offshore Structures, DOI: 10.1080/17445302.2020.1790295.
[16] De Marco, A., Mancini, S., Miranda, S., et al., 2017, Experimental and numerical hydrodynamic analysis of a stepped planing hull", Applied Ocean Research, 64, 135-154.
[17] Cucinotta, F., Guglielmino, E., Sfravara, F., & Strasser, C., 2018, Numerical and Experimental Investigation of a Planing Air Cavity Ship and its Air Layer Evolution. Ocean Engineering, 152, 130-144.
[18] Mancini, S., De Luca, F., Ramolini, A., Towards CFD Guidelines for Planing Hull Simulations Based on the Naples Systematic Series. MARINE 2017 VII International Conference on Computational Methods in Marine Engineering, Nantes, France.
[19] Di Caterino, F, Niazmand Bilandi, R, Mancini, S, Dashtimanesh, A, De Carlini, M, 2018, A Numerical Way for a Stepped Planing Hull Design and Optimization. In Proceedings of the 19th International Conference on Ship & Maritime Research, Trieste, June 20–22, Italy
[20] Giallanza, A., Marannano, G., Morace, F., Ruggiero, V.,2019, Numerical and experimental analysis of a high innovative hydrofoil. International Journal on Interactive Design and Manufacturing (IJIDeM), vol 14, 43–57
[21] Miranda, S., Vitiello, L., 2014, Propulsive performance Analysis of a Stepped Hull by Model Test Results and Sea Trial Data, High-Speed Marine Vehicles Symposium, Naples, ISBN 9788890611216.
[22] ITTC, Recommended Procedures and Guidelines, 2002, 7.5-02-02-02.
[23] Wilcox, D. .C, Turbulence Modeling for CFD, 1998, DCW industries La Canada, CA.
[24] Wilcox, D. C., Formulation of the k-w Turbulence Model Revisited, AIAA Journal, 2008, 46(11), pp. 2823–2838.
[25] Siemens PLM Software, Star-CCM+ User's Guide 2020.
[26] Begovic, E., Bertorello, C., Bove, A., & De Luca, F., 2019, Experimental Study on Hydrodynamic Performance of SWATH Vessels in Calm Water and in Head Waves. Applied Ocean Research, 85, 88-106.
[27] ITTC, "Recommended Procedures and Guidelines 7.5-02-02-01," 2011.
[28] Mikkelsen, H. L., Steffensen, M., Ciortan, C., Walther, J. H., 2019. Ship Scale Validation of CFD Model of Self-Propelled Ship. MARINE 2019 VIII International Conference on Computational Methods in Marine Engineering (2019), Goteborg, Sweden.