American Journal of Mechanical Engineering, 2018, Vol. 6, No. 3, 93-97
Available online at http://pubs.sciepub.com/ajme/6/3/2

DOI:10.12691/ajme-6-3-2

Numerical Hydrodynamic Results of the Two Stepped Planing Hull

Hadi Nourghassemi, Hassan Ghassemi*, Hamidreza Taghva

Department of Maritime Engineering Amirkabir University of Technology, Tehran, Iran
*Corresponding author: gasemi@aut.ac.ir

Received August 28, 2018; Revised October 18, 2018; Accepted November 02, 2018

Abstract This paper presents the hydrodynamic results of the two stepped planing hull by numerical software solver. Time-averaged Navier-Stokes equations are coupled with the standard k-ε turbulence model, and volume of fluid equations are solved to simulate transient turbulent free surface flow surrounding the hull by ANSYS-CFX. In order to predict the motion of the vessel, equations of two degrees of freedom for rigid body are coupled with governing equations of fluid flow. In order to validate the numerical model, the obtained numerical results are compared with the available experimental data. The numerical results obtained for drag, dynamic trim, rising of centre of gravity (CG) and the pressure distribution on the body at different speeds and different heights of the applied steps are presented and discussed.

Keywords: *pressure, drag, stepped planing hull*

Cite This Article: Hadi Nourghassemi, Hassan Ghassemi, and Hamidreza Taghva, "Numerical Hydrodynamic Results of the Two Stepped Planing Hull." *American Journal of Mechanical Engineering*, vol. 6, no. 3 (2018): 93-97. doi: 10.12691/ajme-6-3-2.

1. Introduction

In recent decades, much effort is made by researchers to reduce drag and increase speed of planing hulls by changing the geometrical parameters of the hull. Various forms have been used so far, such as Chine, Strake, Pad, Spray rail, Tunnel and Step. Among these methods, applying transverse step on the hull is known as the most effective. It is common to create holes on the outer walls of the step for air suction. Generally, it is expected to increase the speed of vessel by 10 to 15 per cent by applying stepping method [1]. The main idea of applying transverse step on the hull is to reduce the level of contact of the hull with water on two or more small planing surfaces instead of one large surface. Since the force is distributed on several surfaces throughout the hull, the longitudinal stability increases. However, despite the benefits of the transverse steps, there is also the risk of vessel capsizing if the airway is closed by the coming waves. If aeration is stopped, reverse flow occurs behind the transverse step, which leads to an excessive increase in drag. As a result, the speed suddenly decreases and the vessel may capsize. To avoid this problem, air is often sucked through the duct above the water or is supplied through pipes on deck surface. So far, most hydrodynamic studies using a variety of methods on planing hulls have been focused on the bodies without transverse steps. The process of some of these studies is provided in Table 1. Recently, according to the demands as well as the achievement of higher efficiencies, researchers have been more motivated by the use of transverse stepped hulls, and more effort is being made both numerically and experimentally to study and investigate this topic. A negligible number of studies can be found on transverse stepped hulls during the last 15 years. In Table 2, some of these studies are presented. In this paper, the effects of transverse step height on hydrodynamic performance of the planing hull in investigated by FVM. For doing this, ANSYS-CFX is used. Transient and turbulent free surface flow around the solid hull is modelled by the RANS equations along with the turbulence model standard k-ε coupled with VOF equations. To predict motion of the vessel, the equations of the two degrees of freedom of the rigid body are coupled with the equations governing the fluid flow. In this paper, the planing hull has two transversal steps. In six different modes, with variable height of transverse steps, and in three different displacements, the results of the numerical investigation obtained for drag, dynamical trim and pressure distribution on the surface of the hull are compared with the experimental data.

2. Governing Equations

In order to predict the hydrodynamic behaviour of planing hull in calm water, the craft is considered as a rigid body with two degrees of freedom, and transient simulation is carried out. Therefore, governing equations of the turbulent flow of free surface around the vessel should be coupled with equations of motion of rigid body with two degrees of freedom. The governing equations of fluid flow include the continuity and Navier-Stokes equations.

By applying Reynolds averaging, RANS equations will be obtained:

$$\frac{\partial(\rho)}{\partial t}+\frac{\partial(\rho u_j)}{\partial x_j}=0 \quad (1)$$

$$\frac{\partial(\rho u_j)}{\partial t}+\frac{\partial(\rho u_i u_j)}{\partial x_j}=-\frac{\partial P}{\partial x_i}+\frac{\partial}{x_j}\left(\tau_{ij}-\rho\overline{u_i' u_j'}\right)+g_i \quad (2)$$

$\rho\overline{u_i' u_j'}$ represents Reynolds stresses. Based on turbulent viscosity theory that provides relationship between Reynolds stress terms and velocity gradients, equation (2) will be as follows:

$$\frac{\partial\rho u_i}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_i u_j) =-\frac{\partial P}{\partial x_i}+\frac{\partial}{x_j}\left[\mu_{eff}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+g_i \quad (3)$$

where μ_{eff} is the effective viscosity defined as:

$$\mu_{eff}=\mu+\mu_t. \quad (4)$$

Two-equation *k-ε* model is used to model turbulence flow, in which *k* represents turbulent kinetic energy of flow, and *ε* is dissipation rate of energy. In this model, eddy viscosity is related to viscous kinetic energy and dissipation rate, defined by:

$$\mu_t=c_\mu\rho\frac{k^2}{\varepsilon} \quad (5)$$

where C_μ is constant and k, ε are determined by solving the following transport equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j k)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+p_k-\rho\varepsilon \quad (6)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j\varepsilon) =\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+\frac{\varepsilon}{k}(C_{\varepsilon1}p_k-C_{\varepsilon2}\rho\varepsilon) \quad (7)$$

where $C_{\varepsilon1}, C_{\varepsilon2}$ *and* σ_k are constant values, and P_k is turbulence generation due to viscous forces.

Table 1. Studies and experiments on planing hulls without step

No.	Researchers	Topic	Year
1	Dawson et al.	Investigation of the effect of step and parametric changes on planing hulls [2]	1964
2	Savitsky	Modeling of planing hulls on prismatic hulls with Deadrise angle and obtaining regression relationships based on model test for calculation of hydrodynamic forces [3]	1964
3	Savitsky et al.	Investigation of the effect of water spray on the stern and its effect on the drag of planing hulls [4]	2007
4	Brizola et al.	Study of planing surfaces at constant mode by computational fluid dynamics and comparing the results with experimental data [5]	2007
5	Ghassemi et al.	Development of a computer code based on the boundary element method for analysis of planing and non-planing hulls and presentation of these codes in the hydrodynamic analysis of the vessels [6-9]	2007, 2008 and 2010
6	Akerman et al.	Analysis of Friedman's planing hulls by numerical finite element method in six degrees of freedom [10]	2012
7	Yumin et al.	Investigating the hydrodynamic performance of a planing hull by computational fluid dynamics based on the RANS equations in six degrees of freedom [11]	2012
8	Ghassabzadeh et al.	Calculation of the forces acting on multi-hull tunnel vessel in a steady state using computational fluid dynamics in two degrees of freedom [12]	2014
9	Mansouri et al	Interceptor design for optimum trim control and minimum resistance of planing boats [13]	2017

Table 2. Studies and experiments on stepped planing hulls

No.	Researchers	Topic	Year
1	Tanten et al.	Investigating the effect of different parameters on trim, resistance and draft for stability of vessels by considering the effect of step [14]	2004
2	Matoyo et al.	Numerical and experimental studies on obstruction on the bottom of vessel [15]	2006
3	Savan	Development of a model to predict the performance of planing hulls with transverse steps [16]	2009
4	Savitsky and Morabita	Presentation of mathematical model for determining the profile of the stern of prismatic planing hulls by performing extended model test [17]	2010
5	Townton et al.	Experimental study of a new series of hard chine planing hulls with and without steps [18]	2010
6	Matoyo	Modeling steady flow around the stepped planing hull using hydrodynamic point sources [19]	2012
7	Garland and Maki	Studying the performance of stepped planing hull using numerical simulation of nonlinear flow under a two-dimensional object [20]	2012
8	Ghassabzadeh & Ghassemi	Numerical Hydrodynamic of Multihull Tunnel Vessel [21]	2013
9	Veisi et al.	Simulation of a planing hull with hard chain with and without steps in calm water using numerical methods [22]	2014
10	Lee et al.	The systematic variation of step configuration and displacement for a double-step planing craft [23]	2014
11	Bakhtiari et al.	Numerical modeling of stepped planing hulls [24]	2014
12	Nourghassemi et al.	Investigation of the effects of forward stepping angle on the hydrodynamic performance of planing hull [25]	2017

2.1. Computational Domain and Boundary Conditions

The present study is conducted on a model of stepped planing hull, which is known as Lee model. This model was tested in 2014 by Lee et al., and the results are available in ref. [23]. The model has been tested in six different step heights (0.009525 and 0.00635, 0.003175) with specifications that are given in Table 3. Besides, all of the models were tested and investigated at displacements of 43.09, 55.38, and 47.62 kg. Geometrical parameters of the models are provided in Table 4. Besides, a two-dimensional view of the model is shown in Figure 1.

The flow around the vessel can be assumed to be symmetric with respect to central plane. Therefore, calculations are reduced to only half of the solution domain. The distance between boundaries of this vessel and the body are chosen in such a way that it is possible to apply boundary conditions that are consistent with reality. Solution domain and boundary conditions are shown in Figure 2.

Table 3. Transverse step heights of the models

Model	Height of 1st step (m)	Height of 2nd step (m)
1	0	0
2	0.003175	0.003175
3	0.00635	0.003175
4	0.009525	0.003175
5	0.003175	0.00635
6	0.00635	0.00635
7	0.003175	0.009525

Table 4. Characteristics of the model

Parameter	Value
Length (m)	2
Width (m)	0.4572
Deadrise angle (degree)	15
Distance between the first step and stern (m)	0.508
Distance between the second step and stern (m)	1.016

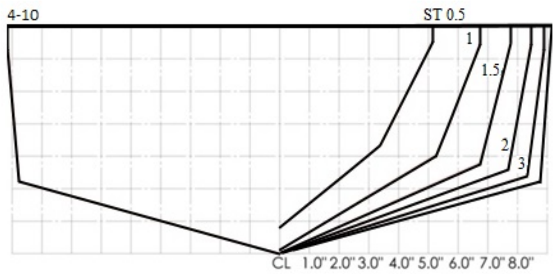

Figure 1. Body plan of the hull

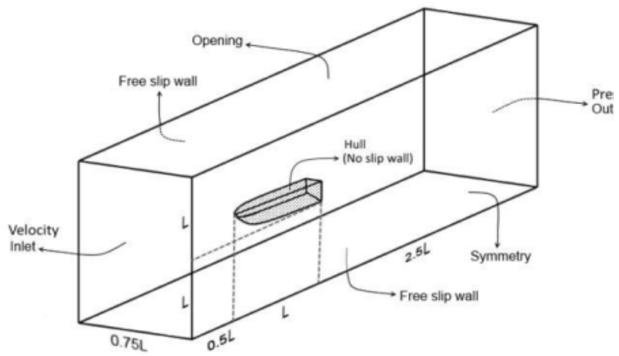

Figure 2. Solution domain and boundary conditions

3. Results

In this study, we used the use of k-ε turbulence model. The number of inflation layers within the boundary layer is considered equal to 22. The dimensionless thickness of the first layer on the surface is $Y^+ = 50$. In order to better approximate curves and sharp corners, the cell size is considered sufficiently small on the body surfaces. Cell sizes in the two regions in flow where high gradients are expected are considered smaller. The first region is around the free surface, where it is required to solve its shape, and the second region is a part of the first region near the body, where separation of the flow from the chine, step and water spray occurs. The computational grid generated around the body is shown in Figure 3. In order to check the grid independency, calculations are performed in several grids with different numbers of cells. For example, the variation of drag force to displacement ratio based on number of cells Froude number of 1.85 is illustrated in Figure 4. In this section, in order to validate the present numerical model, the numerical results obtained for different heights of steps are presented and compared with experimental data. The vessel model studied in this paper was tested in 2014 by Lee et al. for different heights of steps. The presented experimental results include the raise of center of mass, dynamic trim, and drag force. Comparison of the drag, trim and raise of CG is presented in Figure 5, Figure 6 and Figure 7, respectively. These results show relatively good agreement between numerical and experimental data. Figure 8 shows the pressure contour acted on the hull for different models at Fn=2.13. It can be observed that the model 1 that has the highest drag, achieves the lowest level of pressure peak. In contrast, the model 4 has the lowest drag and highest level of pressure peak.

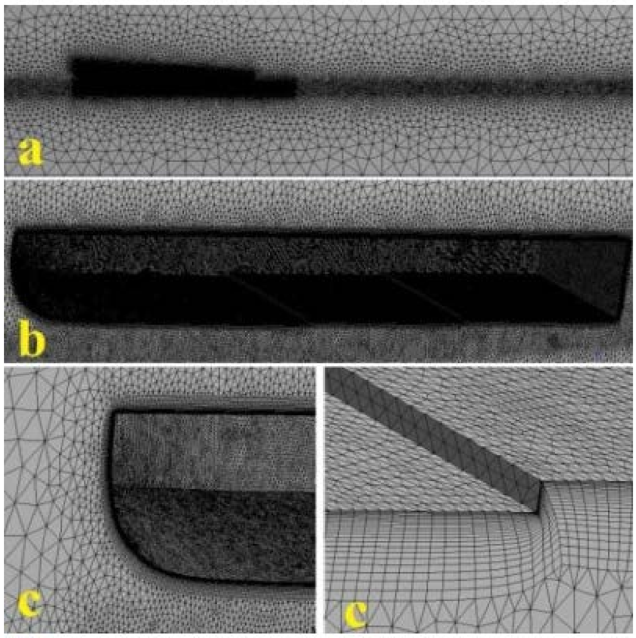

Figure 3. View of the generated grid, a), close view b) hawse region, c) step region

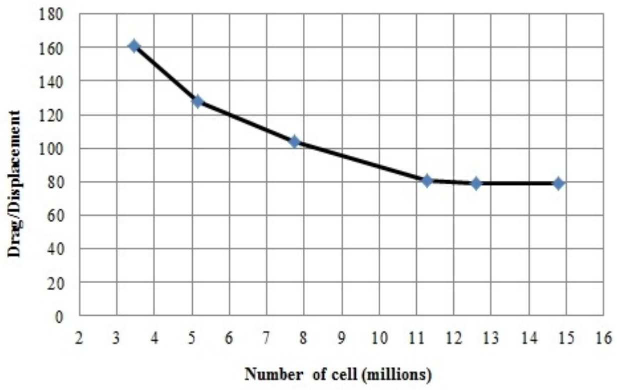

Figure 4. Ratio of drag force to displacement (Fn=1.85)

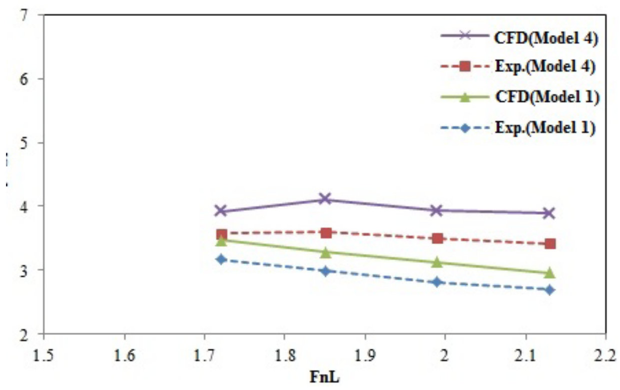

Figure 6. Comparison of dynamic trim [deg.] against Fn

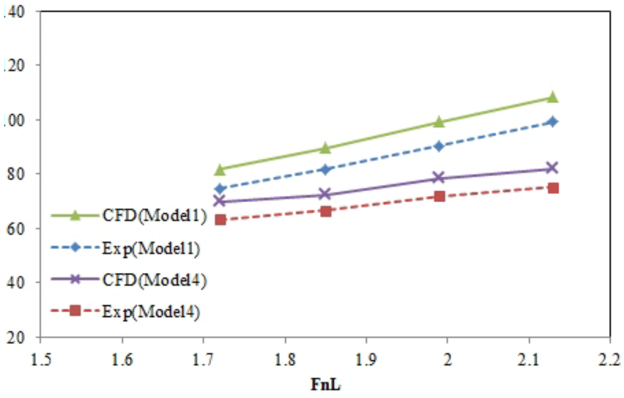

Figure 5. Comparison of drag [N] against Fn

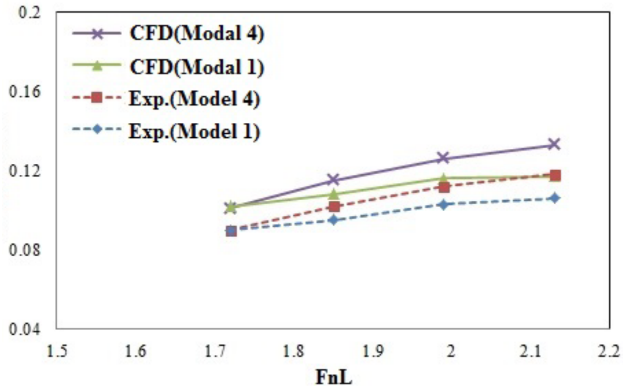

Figure 7. Comparison of raise of CG against Fn

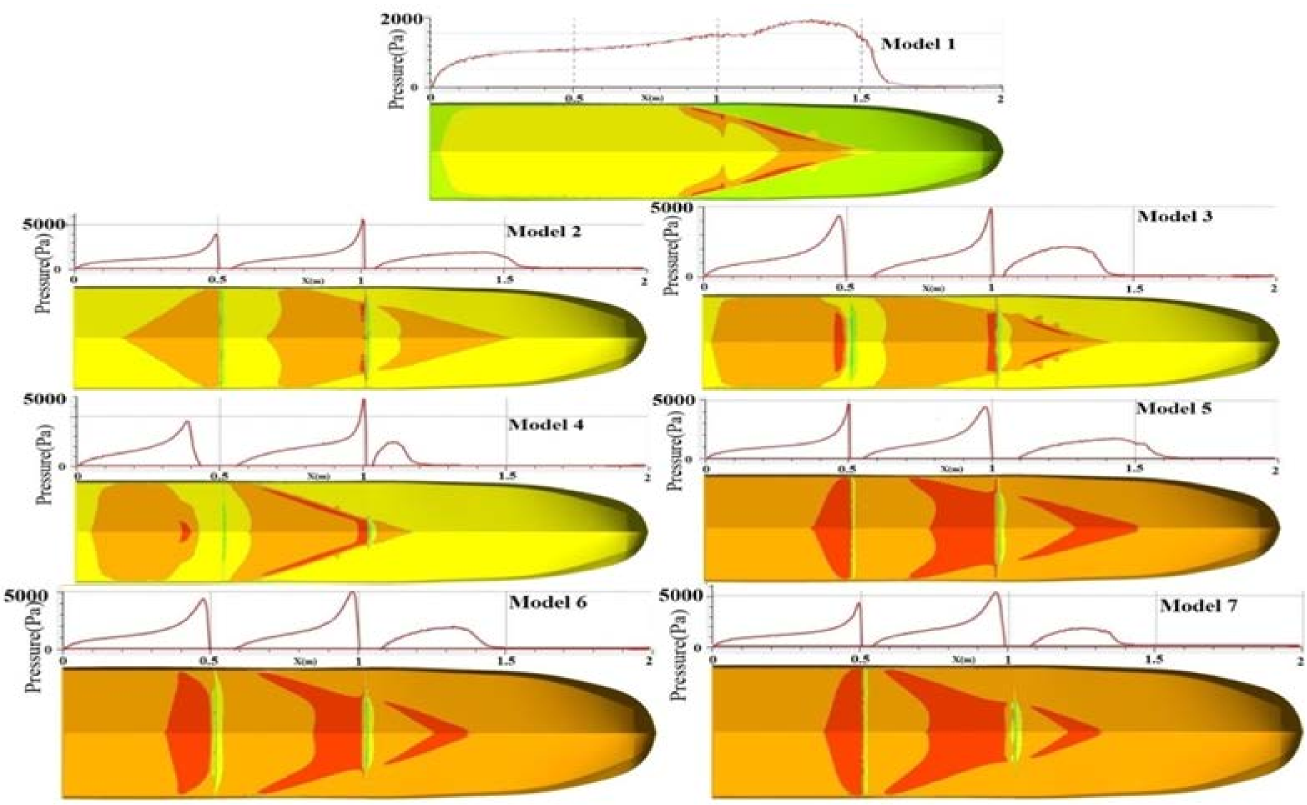

Figure 8. Pressure contour acted on the hull for different models (Fn=2.13)

4. Conclusion

In this paper, the hydrodynamic effect of planing hull is numerically calculated. The hull has two transverse stepped with different heights. The solver software is ANSYS-CFD with the standard k-ε turbulence model. The numerical results are revealed the drag, trim, rise of CG and pressure distributions. Some of the results are compared with experimental data and found relatively good agreement between them. It is found that the effect may reduce drag and pressure distribution acted on the hull makes better ride and moderate performance.

References

[1] "Stepped Hull", www.navaldesign.co.za.

[2] Effect of varying depth of step, angle of after body kell, length of after body chine, and Gross load; John R. Dawson, Robert Mckann, and Elizabeth S. Hay; Lagley Memorial Aeronautical Laboratory Langley Field, VA; July 1946.

[3] Savitsky, D., Hydrodynamic design of planing hulls, Marine Technology 1(1), 1964, 71-95.

[4] Savitsky D, DeLorme MF, Datla R., Inclusion of Whisker Spray Drag in Performance Prediction Method for High-Speed Planing Hulls, Maine Technology, Vol. 44, No. 1, 2007, pp. 35-56.

[5] Brizzolara S, Serra F., Accuracy of CFD codes in the prediction of planing surfaces hydrodynamic characteristics. In: 2nd international conference on marine research and transportation, Naples, 2007.

[6] Ghassemi H, Ghiasi M., A combined method for the hydrodynamic characteristics of planing crafts. Ocean Eng 35(3-4): 2007, 310-322.

[7] Ghassemi H, Yumin S., Determining the hydrodynamic forces on a planing hull in steady motion. J Mar Sci Appl 7: 2008, 147-156.

[8] Kohansal AR, Ghassemi H, Ghiasi M., Hydrodynamic characteristics of high speed planing hulls, including trim effects, Turkish. J Eng Environ Sci 34: 2010, 1-16.

[9] Kohansal AR, Ghassemi H., A numerical modeling of hydrodynamic characteristics of various planing hull forms. Ocean Eng 37(5-6): 2010, 498-510.

[10] Akkerman I, Dunaway J, Kvandal J, Spinks J, Bazilevs Y (2012) Toward free-surface modeling of planing hulls: simulation of the Fridsma hull using ALE-VMS. Comput Mech 50(6):719-727.

[11] Yumin S, Qington C, Hailong S, Wei L., Numerical simulation of a planing hull at high speed. J Mar Sci Appl 11(2): 2012, 178-183.

[12] Ghassabzadeh M, Ghassemi H., Determining of the hydrodynamic forces on the multi-hull tunnel vessel in steady motion. Brazilian Soc Mech Sci Eng 36(4): 2014, 1-12.

[13] Mansoori M., Fernandes A.C., Ghassemi H., Interceptor design for optimum trim control and minimum resistance of planing boats, Applied Ocean Research, 69, 2017, 100-115.

[14] Taunton, D. J., Hudson, D. A., & Shenoi, R. A. Characteristics of a series of high speed hard chine planing hulls-part 1: performance in calm water. International Journal of Small Craft Technology, 152, 2010, 55-75.

[15] Matveev, K. I., Three-dimensional wave patterns in long air cavities on a horizontal plane. Ocean Engineering, 34(13), 2007, 1882-1891.

[16] Svahn D (2009) Performance prediction of hulls with transverse step. Master thesis, Marina System Center for Naval Architecture, KTH, Stockholm Taunton et al.

[17] Savitsky D, Morabito M., Surface wave contours associated with the forebody wake of stepped planing hulls. Mar Technol 47(1), 2010, 1-16.

[18] Taunton D, Hudson D, Shenoi R., Characteristics of a series of high speed hard chine planing hulls—Part 1: performance in calm water. Int J Small Craft Technol 152: 2010, 55-75.

[19] Matveev, K. I., Two-dimensional modeling of stepped planing hulls with open and pressurized air cavities, International Journal of Naval Architecture and Ocean Engineering, Vol 4(2), 2012, pp162-171

[20] Garland WR, Maki KJA, Numerical study of a two-dimensional stepped planing surface. J Ship Prod Des 28(2): 2012, 60-72.

[21] Ghassabzadeh M. and Ghassemi H., Numerical Hydrodynamic of Multihull Tunnel Vessel, Open Journal of Fluid Dynamics, Vol. 3 No. 3, 2013, pp. 198-204.

[22] Veysi S T G · Bakhtiari M, Ghassemi H, Ghiasi M. Toward numerical modeling of the stepped and non-stepped planing hull, Brazilian Soc Mech Sci Eng. 2014,

[23] Lee, E., Pavkov, M., & McCue-Weil, L., The systematic variation of step configuration and displacement for a double-step planing craft, Journal of Ship Production and Design, 30(2), 2014, 89-97.

[24] Bakhtiari M, Veysi S.T.G, Ghassemi H., Numerical modeling of the stepped planing hull in calm water. International Journal of Engineering; Transactions B: Applications Vol. 29, No. 2, 2016, 236-245.

[25] Nourghassemi, H., Bakhtiari, M., & Ghassemi, H. Numerical study of step forward swept angle effects on the hydrodynamic performance of a planing hull, Scientific Journals of the Maritime University of Szczecin, (51), 2017, 35-42.