TRANSACTIONS B: Applications Vol. 29, No. 2, (February 2016) 236-245

International Journal of Engineering

Journal Homepage: www.ije.ir

Numerical Modeling of the Stepped Planing Hull in Calm Water

M. Bakhtiari, S. Veysi, H. Ghassemi*

Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran

PAPER INFO

Paper history:
Received 10 November 2015
Received in revised form 29 December 2015
Accepted 26 January 2016

Keywords:
Stepped Hull
Turbulent
Free Surface Flow
Volume of Fluid Model
Drag
Wake Profile

ABSTRACT

This article describes a 3D CFD (computational fluid dynamics) simulation implementation of the stepped planing hull in calm water. The turbulent free surface flow around the stepped planing hull is computed with a RANSE method, using the solver ANSYS-CFX. The turbulence model used is standard k–ε. In order to simulate the disturbed free surface, volume of fluid (VOF) model is implemented. The CFD model has been firstly validated using the available experimental data. The numerical results of drag, pressure distribution, wetted surface, water spray, wake profile and wave generated by the planing hull are presented and discussed at various speeds. Wake profiles calculated from present model are also compared with the ones calculated from Savitsky's emprical equations at different speeds.

doi: *10.5829/idosi.ije.2016.29.02b.13*

1. INTRODUCTION

Drag to lift ratio have been always the most important parameter in the design of a planing hull and several approaches have been studied to obtain the hull form so as to reduce the drag in order to increase the speed. Up to date, various configurations such as chine, step, strake, pad and tunnels have been set up to increase the speed and improve performance. Among these strategies, step is one of the most efficient approaches. As a result of using step at hull bottom, flow separates from the step, and the aft body will be partially ventilated. This diminishes the frictional drag by decreasing the wetted surface area while maintaining a high lift force as shown in Figure 1. Since the lift is now spread to several surfaces along the hull, the longitudinal stability becomes very large. Beside those advantages, there is somewhat a danger due to closing air by waves. When the air supply is lost, a reverse flow occurs behind the step causing an excessive increase in drag. The speed drops rapidly and craft will turn suddenly, and possibly even capsize. To avoid this problem, air is often sucked through openings well above the waterline, or it may be supplied through tubes in the deck level. Generally, two methods are defined for hydrodynamic analysis of the planing hulls. The first method is to use computation fluid dynamic (CFD) and the second is to apply an experimental method which needs model basin and other test facilities. Finite volume method (FVM) and boundary element method (BEM) are more popular numerical techniques for hydrodynamic analysis of the flow around planing hulls. BEM is based on potential solver for lifting bodies and planing hulls that the pressure force is dominant, while FVM solves the Navier-Stokes equations that takes much calculation time. Comprehensive analysis techniques for hydrodynamic simulations of high-speed planing hulls have been categorized by Yousefi et al. [1], as presented in Figure 2.

Until now, most studies have been focused on hydrodynamics of the simple mono-hull, means without chine and step. Savitsky [2] comprehensively contributed to the understanding and modeling of planing crafts. He developed regression formulas based on prismatic hull form model tests to estimate the hydrodynamic forces acting on planing crafts. Savitsky et al. [3] have investigated the effect of the whisker spray at the bow and its effect on the drag.

*Corresponding Author's Email: *gasemi@aut.ac.ir* (H. Ghassemi)

Please cite this article as: M. Bakhtiari, S. Veysi, H. Ghassemi, Numerical Modeling of the Stepped Planing Hull in Calm Water, International Journal of Engineering (IJE), TRANSACTIONS B: Applications Vol. 29, No. 2, (February 2016) 236-245

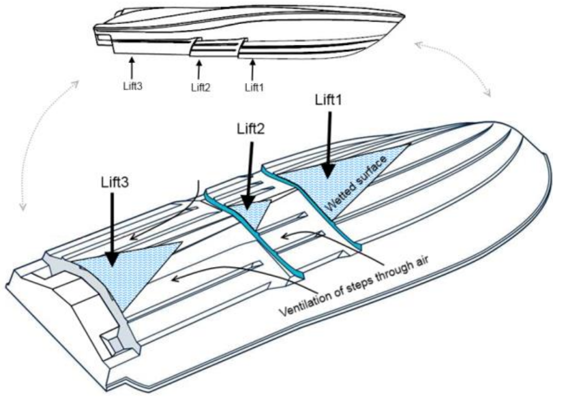

Figure 1. A schematic bottom view of a stepped planing hull

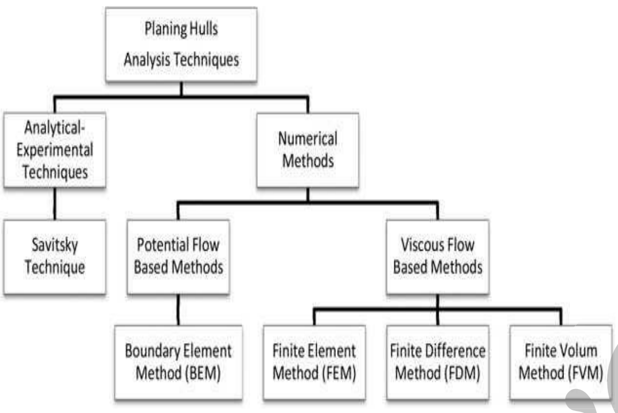

Figure 2. Various approaches for hydrodynamic analysis of planing hulls [1]

Savander et al. [4] applied the BEM problem to a planing plate and obtained relationships between potential perturbation and vortex distribution. They calculated the hydrodynamic pressure, lift and drag forces for the planing plate at different speeds. Ghassemi et al. have developed a computer code, based on BEM in conjunction with boundary layer for hydrodynamic analysis of planing and non-planing hulls [5-8]. One of the drawbacks of this code is that it is based on the potential solvers (BEM), although the main forces on the planing hulls and lifting bodies (hydrofoils and propellers) are due to the pressure forces. Concerning the effect of the viscous term, FVM is nowadays the most popular and powerful method that can be used. Seif et al. studied the effect of asymmetric water entry on the hydrodynamic impact by FVM method [9]. Many researchers employed FVM method for planing hull; see Capennetto [10], Richard et al. [11], Savander and Rhee [12], Kihara [13], Brizzolara and Serra [14], Fultz [15], Akkerman et al. [16], Yumin et al. [17].

Planing tunnel vessels are also especial type of the high speed vessels that is high lift-drag ratio and good seakeeping performance (Subramanian et al. [18], Ghassabzadeh and Ghassemi [19]). Nowadays, any work on this field is very attractive and owners encourage the researchers to work on this topic. Ghassabzadeh and Ghassemi [20] prepared software to generate various hull form of the planing tunnel vessel.

More recently, due to the market needs and in order to gain more efficiency by the stepped hull, marine researchers have rigorously pursued this topic and much effort has been devoted to explore it numerically and experimentally. It A small number of contributions can be found on the planing stepped hull during last 10 years (Savitsky et al. [21], Svahn [22], Taunton et al. [23], Grigoropoulos and Damala [24], Ghassemi et al. [25], Garland and Maki [26], Veysi et al. [27]).

In this study, hydrodynamic characteristics of a stepped planing hull are numerically investigated using CFD approach based on finite volume method employed by ANSYS-CFX code. For the case study, a C1-model stepped planing hull is selected. The geometry and experimental results of C1-model is available (Taunton et al.[23]). In order to calculate the turbulent free surface flow around the planing hull, the RANS equations with k-ε turbulent model is used in a coupled manner with VOF free surface model. The governing equations are solved in a computational domain discretized by a high quality unstructured mesh. The validity of CFD simulation model used is tested by a comparison between numerical and experimental drag at different speeds. The numerical results of drag, pressure distribution, wetted surface, water spray, wake profile and wave generated by the planing hull are presented and discussed at various speeds.

2. GOVERNING EQUATIONS

Using the Reynolds averaging approach, the Navier–Stokes equations governing the turbulent flow can be written as:

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

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

where u, P, and g are the velocity, pressure and gravitational acceleration, respectively. The term $-\rho\overline{u_i' u_j'}$ in Equation (2) represents the Reynolds stresses.

Reynolds stresses can be related to the mean velocity gradients and eddy (turbulent) viscosity on the basis of eddy viscosity Hypothesis as follows:

$$\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad (3)$$

In order to calculate the eddy viscosity, the *k–ε* model is used. In this model, eddy viscosity is related to the turbulence kinetic energy (*k*) and turbulence dissipation rate (*ε*) by the following formula:

$$\mu_t = c_\mu \rho \frac{k^2}{\varepsilon} \quad (4)$$

where C_μ is a constant. k and ε are calculated from the following transport equations:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j k)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+p_k-\rho\varepsilon \quad (5)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_j\varepsilon)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+\frac{\varepsilon}{k}(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon) \quad (6)$$

where $C_{\varepsilon 1}, C_{\varepsilon 2}$, σ_k and σ_ε are constant and P_K is the turbulence production due to viscous forces. In order to capture the air-water interface, the VOF (volume of fluid) model is implemented. The related transport equation is:

$$\frac{\partial \alpha}{\partial t}+\vec{\nabla}.(\alpha\vec{u})=0 \quad (7)$$

where, $0<\alpha<1$ is the volume fraction occupied by each phase inside each computational cell.

3. SOLUTION DOMAIN AND BOUNDARY CONDITIONS

A new stepped planing hull, named C1, was selected to study hydrodynamic characteristics. This model has been tested by Taunton et al. [23] at the University of Southampton and their experimental results are available for public use. The sketch of this model is illustrated in Figure 3, whereas the main dimensions of the models are presented in Table 1.

The flow around the ship hull can be assumed symmetrical with respect to center plane of the hull. This is a reasonable assumption that leads to significant reduction in computational costs, so the solution domain was reduced to a half of full domain. The domain boundaries were placed at a position far enough away from the hull so that the boundary conditions applied are compatible with real physics. Dimensions of solution domain around the planing hull and boundary conditions applied are shown in Figure 4.

Figure 3. Sketch of C1-model stepped planing hull hull

TABLE 1. Main dimensions of C1-model stepped planing hull

Parameter	Value
Length overall (m)	2
Beam (m)	0.46
Draft (m)	0.09
Displaced weight (N)	243.4
Deadrise angle (degree)	22.5
Distance of the step from transom (m)	0.62
Height of step (m)	0.02

Details of boundary conditions applied are as follows: a) A uniform velocity equivalent to ship speed at inlet boundary. b) A hydrostatic pressure distribution at outlet boundary. c) An opening condition at top boundary, where the air is permitted to flow through the boundary. d) A wall with free slip condition at Bottom and side boundaries, e) A wall with no slip condition at hull surface, and, f) The center plane of the hull was set to symmetry condition.

4. DOMAIN DISCRETIZATION

Because of the geometric complexity of the solution domain, especially on the hull surface, the whole domain was discretized by using an unstructured mesh of tetrahedral cells. A high-resolution inflation layer mesh with Yplus of about 50 was created in the boundary layer region of the hull in order to accurately capture the high wall-normal velocity gradients at this region.

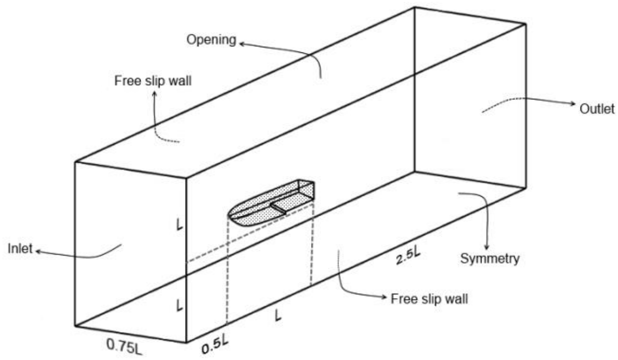

Figure 4. Solution domain and boundary conditions

This inflation mesh consists of about 20 inflation layers inside the boundary layer. Furthermore, the hull surface mesh was refined to the point that a good approximation of the hull surface curvatures and sharp corners is achieved. Some local volume mesh refinements were also applied to the zones that larger gradients of the solution variables are expected to occur, including two zones. The first is a zone around free surface where accurate capturing of air-water interface is needed and the second is a part of the first zone near the hull where water separation and reattachment and water spray occur. A detailed view of the mesh created around the hull is shown in Figure 5 along with the magnified views of inflation layer mesh at bow and step regions.

It is clear that one of the common causes of erroneous results in CFD is the mesh resolution so that reducing the size of the cells can increase the accuracy of numerical solution. Therefore, it is necessary to ensure that solution is independent of the mesh resolution. For this purpose, the numerical simulations were conducted in four computational meshes with different cell numbers for each speed. Figure 6 illustrates the mesh independence diagram for drag-displacement ratio at a speed of 10.13 m/s.

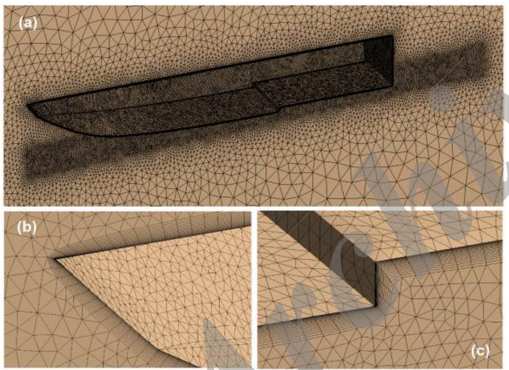

Figure 5. A view of the mesh around the hull, a) Near-field flow region, b) Bow region c) Step region

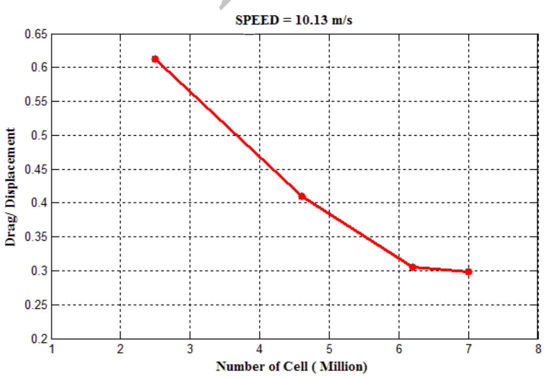

Figure 6. Mesh independence diagram at a speed of 10.13 m/s

5. RESULTS AND DISCUSSIONS

In this section, the computed numerical results for a C1-model stepped planing hull at the speeds from 4.05 to 12.05 m/s are presented. The draft and trim angle corresponding to each speed for steady state simulation is determined from experimental results measured by Taunton et al. [23] at the University of Southampton. The steady state solver was employed in ANSYS-CFX with high resolution scheme for Spatial Discretization. The volume fraction equation was solved in coupled manner. RMS residual level of 10^{-4} was considered as solution convergence criteria for all equations. Furthermore, the hull drag was also monitored to reach a stable value after satisfying RMS residual criteria.

The calculated Yplus distributions on hull bottom along the center line, at different speeds, are presented in Figure 7. From this figure, it can be seen that the Yplus values on wetted surface are in the range of 50 to 75 which have a good compatibility with required Yplus of k–ε model.

5. 1. Hull Drag Numerical results obtained for drag/displacement ratio are presented in Figure 8 and compared with experimental results at different speeds for validation purposes. From this figure, we can see that the drag values computed by present CFD model, with an average error of about 10%, have a good conformity with experimental results. This good agreement refers to the appropriate prediction of wetted hull surface, pressure distribution on hull and other parameters affecting the drag. These important hydrodynamic characteristics of stepped planing hull will be presented and discussed in the following sections.

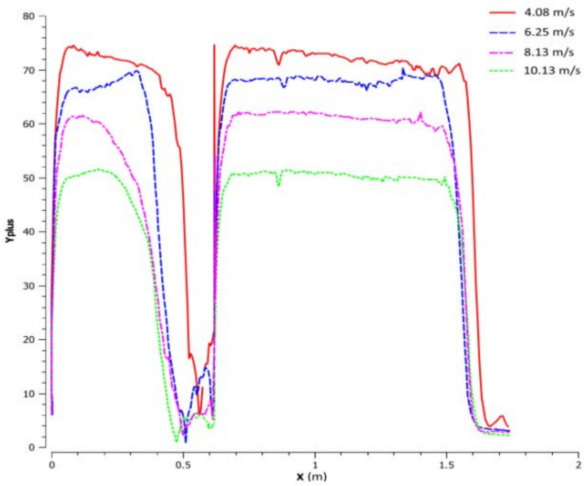

Figure 7. Yplus distribution on hull along center line at different speeds

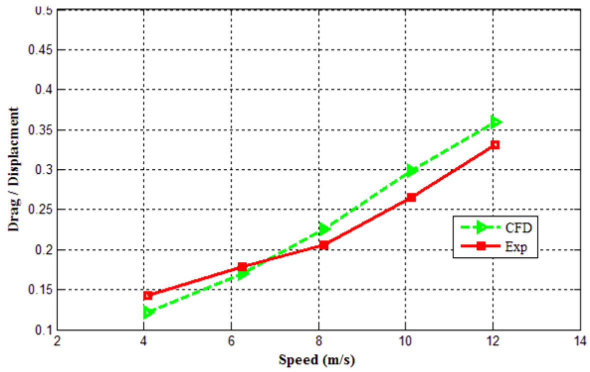

Figure 8. Comparison between numerical and experimental results of drag/displacement ratio

5. 2. Wetted Surface As mentioned before, the idea behind a stepped bottom is to reduce wetted surface area. This effect results in a significant decrease in drag compared with a non-stepped hull. The distribution of water volume fraction on hull, which is a good criterion for estimating wetted surface area, is shown in Figure 9.

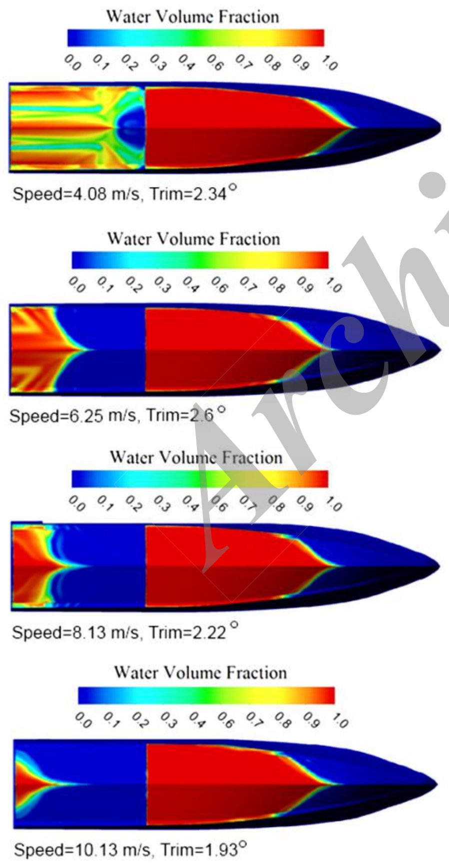

Figure 9. Distribution of water volume fraction on hull at different speeds

According to this figure, it seems that step has a significant effect in reducing surface in contact with water. This reduction, especially at higher speeds, is more significant.

5. 3. Pressure Distribution on Hull Pressure distributions on the hull surface at different speeds are shown in Figure 10. In the same figure, diagrams of pressure coefficient on the keel line and three transverse sections of hull are also presented. At all speeds, It can be seen that pressure on forbody develops from it's maximum value at stagnation line to it's minimum value at step where the water separation occurs. After the step, due to the flow separation, the relative pressure on aft body is negative and reaches its maximum value at stagnation line of aft body where the water reattaches the hull. Clearly, with increasing speed, pressure area on aft body decreases as a result of increased water separation. The coefficient pressure diagrams for three transverse sections of hull show that there is a low pressure region where the water separates from the chine.

5. 4. Free Surface and Water Spray Obviously, a more accurate simulation of disturbance induced to free surface by hull motion leads to more accurate estimation of wetted surface area, water separation and spray, wave pattern and wake profile. In order to appropriately capture the disturbed free surface, VOF model was implemented with an enough refined mesh near the water-air interface. Here, a surface with a constant value of VOF=0.5 is considered as free surface. Two views of free surface are illustrated in Figure 11 from front and bottom views at a speed of 8.13 m/s. Water separation from chine at aft and fore bodies are marked in this figure.

Additionally, to view the details of separation and water spray from the hull, VOF distribution at different cross sections of x=0.4, x=0.8 and x=1.2 are presented in Figure 12. Here, x is the distance from the transom.

5. 5. Wake Profile and Wave Pattern Savitsky and Morabito [21] introduced a series of empirical equations to estimate the wake profile aft of a prismatic stepped hull in centerline and ¼ beam buttock. These equations, which are presented for deadrise angles of 10, 20 and 30 degrees, are as follows:

For centerline :

$$H = 0.17\left[1.5 + 0.03 L_k \tau^{1.5}\right] Sin\left[\frac{\pi}{C_V}\left(\frac{X}{3}\right)^{1.5}\right] \tag{8}$$

For 1/4 beam buttock :

$$H = 0.17\left[0.75 + 0.03 L_k \tau^{1.5}\right] Sin\left[\frac{\pi}{C_V}\left(\frac{X}{3}\right)^{1.5}\right] \tag{9}$$

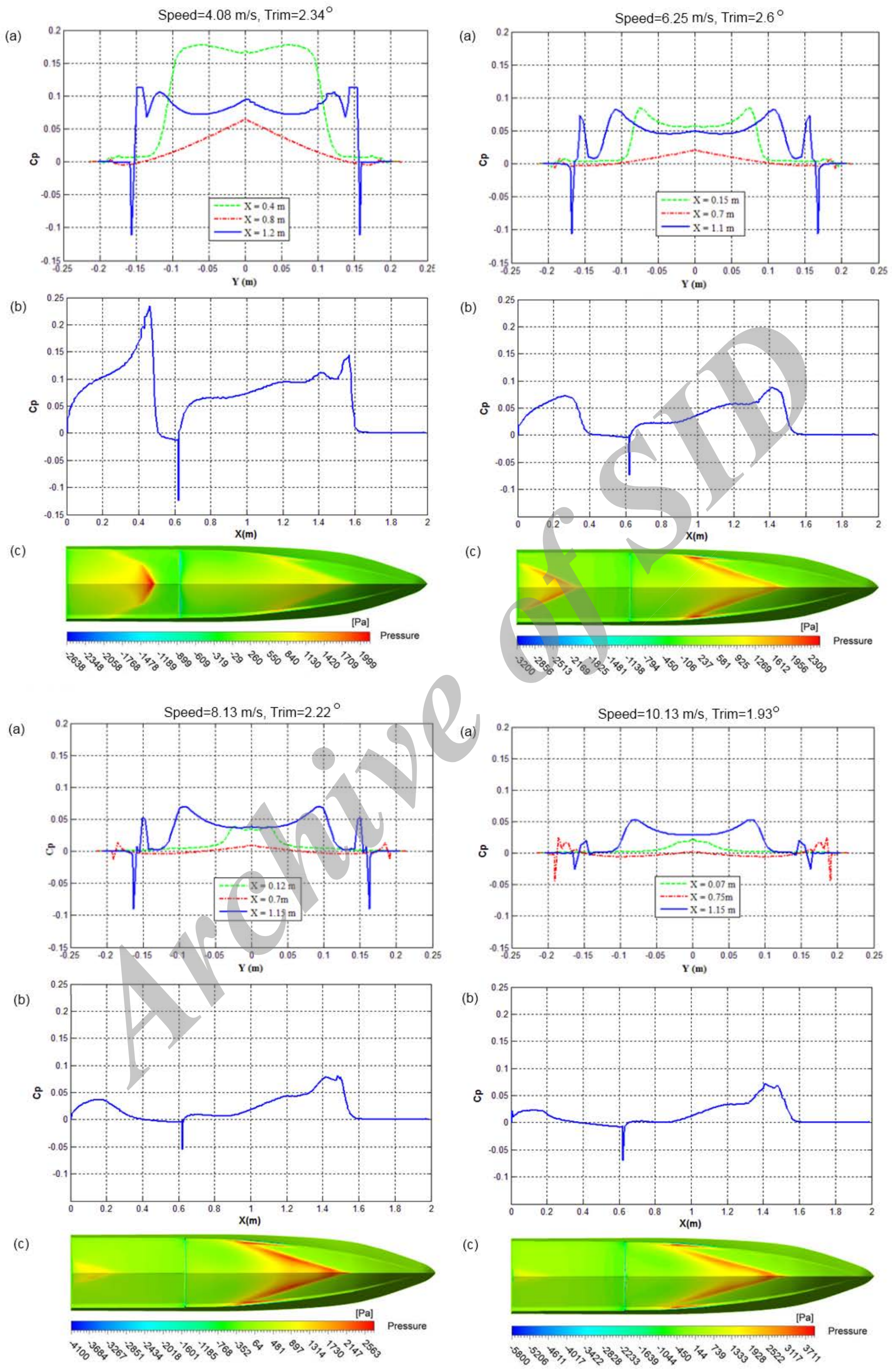

Figure 10. Pressure distributions, a) on transverse sections, b) on keel line, c) on hull surface

where H is the height of wake profile above extended keel or ¼ beam buttock, X is distance aft of transom and τ is trim angle (see Figure 13). Also, L_k is wetted keel length, and $C_V\left(=V_S/\sqrt{gB}\right)$ is Froude number based on craft's breadth, where V_S is craft's speed and B is breadth of the craft.

Figure 14 shows wake profile at centerline for all investigated speeds. Also, wave pattern around the hull are illustrated in Figure 15. As is clear from this figure, the wave angle relative to hull reduces with speed growth, while the height of stern wave decreases that this trend shows a good compatibility with actual physics. A comparison between wake profiles numerically calculated from present model and the ones calculated from Savitsky's empirical equations at different speeds are shown in Figure 16.

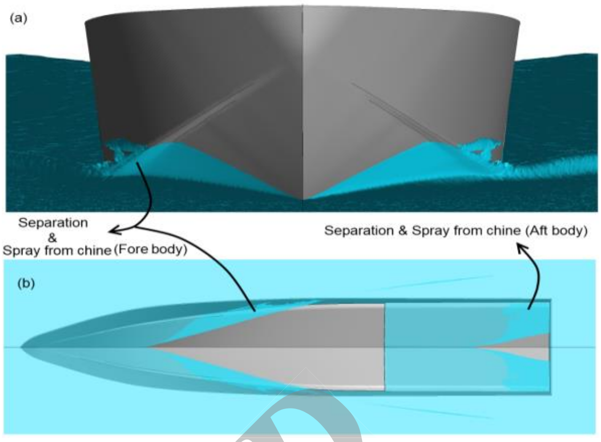

Figure 11. Water separation and spray from chine at fore and aft body, a) Front view, b) Bottom view

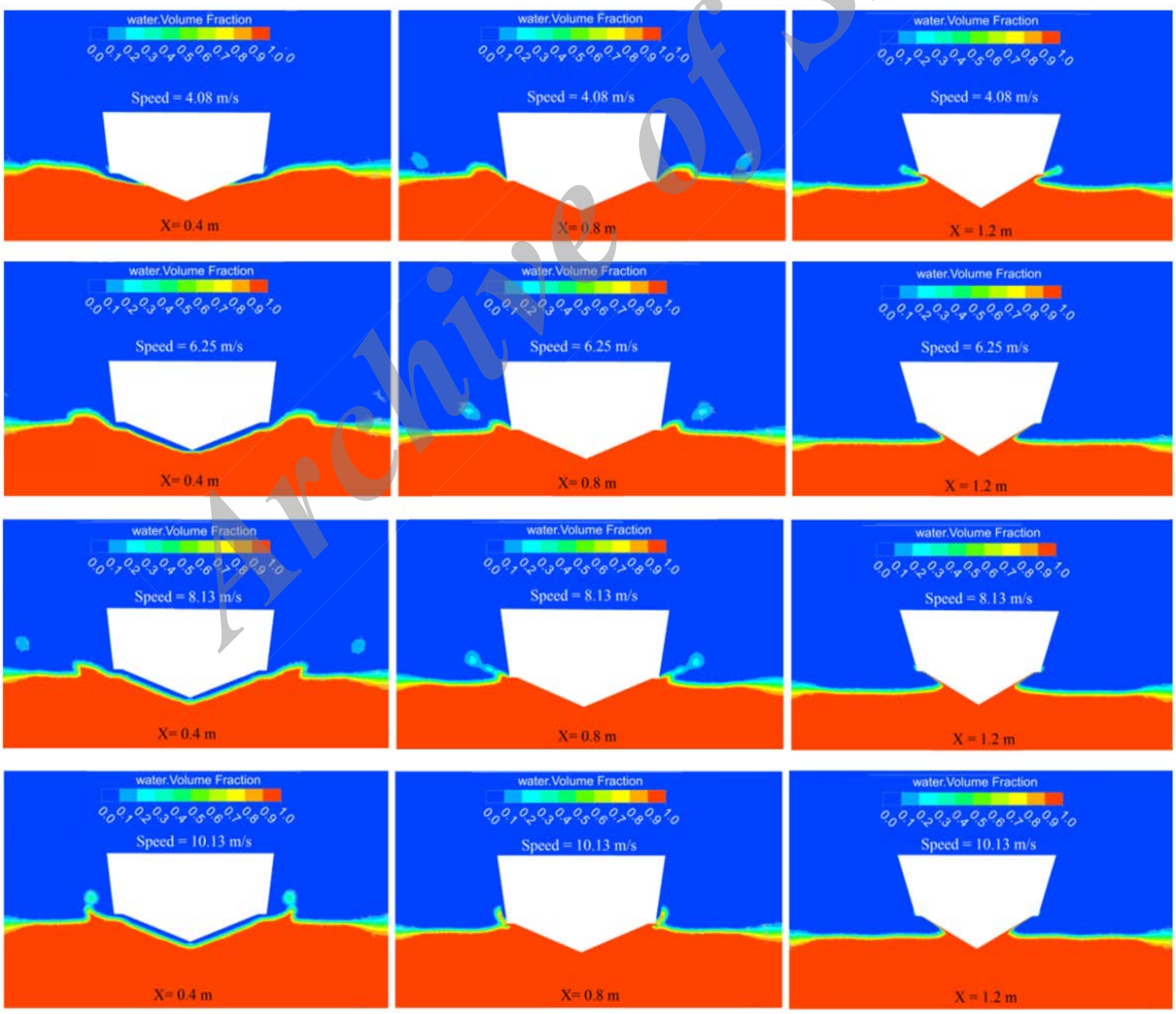

Figure 12. Water separation and spray at different cross sections of hull

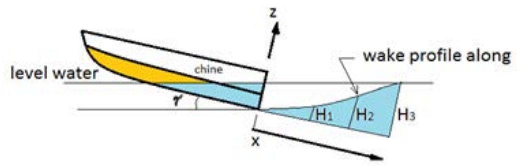

Figure 13. A schematic view of wake profile behind the hull

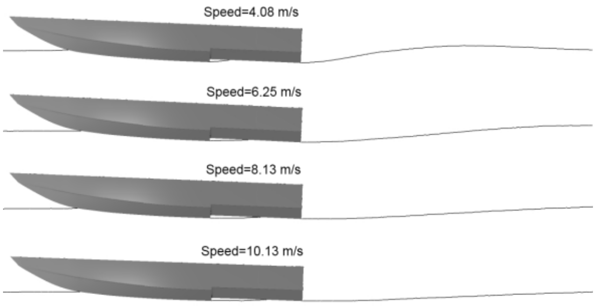

Figure 14. Wake profiles at centerline for different speeds

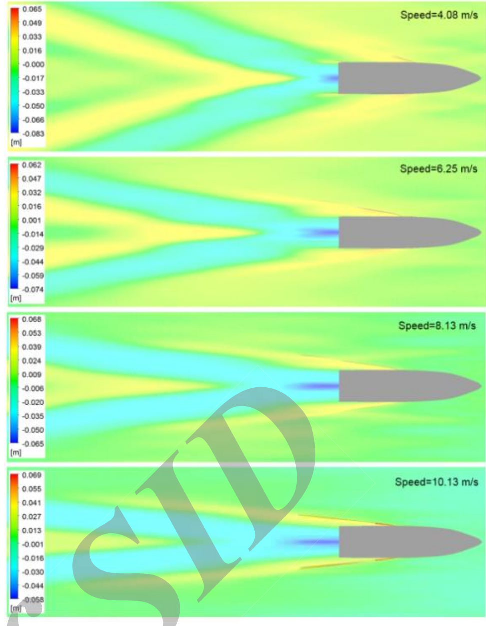

Figure 15. Contours of wave height around the stepped hull at different speeds

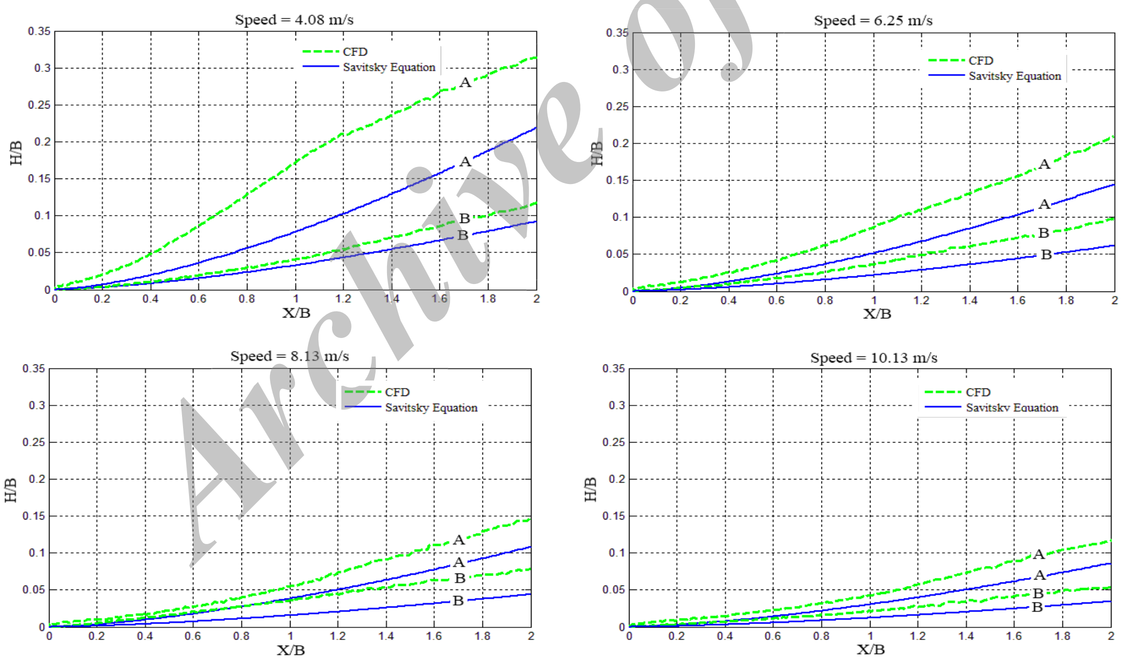

Figure 16. Comparison between numerical results and Savitsky's emprical equations for wake profile, A) Centerline, B) 1/4 beam buttock

6. CONCLUSIONS

CFD approach based on finite volume method was used to investigate hydrodynamic characteristics of a stepped planing hull at different speeds in calm water. RANS equations with standard k–ε turbulent model coupled with VOF multiphase model were utilized to simulate turbulent free surface flow around a C1-model stepped planing hull (Tested by Taunton et al. [23]). In order to validate numerical model employed, numerically calculated drags at different speeds were compared with available experimental data and, as a result of this comparison, a good compatibility was observed. So, the CFD model used in this study can be offered for proper estimation of drag. In addition, wake profile resulted from present CFD model was compared with empirical formulae and showed a good consistency. Numerical results for stepped hull showed that using step can significantly decrease total drag as a result of reduced wetted surface. This is our future plan to investigate the effect of step location and number of steps on hydrodynamic behaviour of stepped planing hull.

7. ACKNOWLEDGEMENTS

The work presented in this paper has been supported by the High Performance Computing Research Center (HPCRC) at Amirkabir University of Technology (AUT). Also, authors would like to thank the marine research center of AUT for grant-in-aid.

8. REFERENCES

1. Yousefi, R., Shafaghat, R. and Shakeri, M., "Hydrodynamic analysis techniques for high-speed planing hulls", ***Applied Ocean Research***, Vol. 42, (2013), 105-113.
2. Savitsky, D., "Hydrodynamic design of planing hulls", ***Marine Technology***, Vol. 1, No. 1, (1964), 71–95 .
3. Savitsky, D., DeLorme, M.F. and Datla, R., "Inclusion of whisker spray drag in performance prediction method for high-speed planing hulls", ***Marine Technology***, Vol. 44, No. 1, (2007), 35-56.
4. Savander, B.R., Scorpio, S.M. and Taylor, R.K., "Steady hydrodynamic analysis of planing surfaces", ***Journal of Ship Research***, Vol. 46, No. 4, (2002), 248-279.
5. Ghassemi, H. and Ghiasi, M., "A combined method for the hydrodynamic characteristics of planing crafts", ***Ocean Engineering***, Vol. 35, No. 3, (2008), 310-322.
6. Hassan, G. and Su, Y.-m., "Determining the hydrodynamic forces on a planing hull in steady motion", ***Journal of Marine Science and Application***, Vol. 7, No. 3, (2008), 147-156.
7. KOHANSAL, A., GHASSEMI, H. and GHIASI, M., "Hydrodynamic characteristics of high speed planing hulls, including trim effects", ***Turkish Journal of Engineering and Environmental Sciences***, Vol. 34, No. 3, (2011), 155-170.
8. Kohansal, A.R. and Ghassemi, H., "A numerical modeling of hydrodynamic characteristics of various planing hull forms", ***Ocean Engineering***, Vol. 37, No. 5, (2010), 498-510.
9. Seif, M., Mousavirad, S. and Sadat, H.S., "The effect of asymmetric water entry on the hydrodynamic impact", Vol. 17, No. 2, (2004), 205-212.
10. Caponnetto, M., "Practical cfd simulations for planing hulls", in Proc. of Second International Euro Conference on High Performance Marine Vehicles, Hamburg. (2001), 128-138.
11. Pemberton, R., Turnock, S. and Watson, S., "Free surface cfd simulations of the flow around a planing plate", In: *FAST2001*, Southampton, UK, (2001)
12. Savander, B. and Rhee, S., "Steady planing hydrodynamics: Comparison of numerical and experimental results", ***Fluent Users' Group Manchester***, (2003).
13. Kihara, H., "A computing method for the flow analysis around a prismatic planing-hull", in HIPER 06: 5th International Conference on High-performance Marine Vehicles, Australian Maritime College. (2006).
14. Brizzolara, S. and Serra, F., "Accuracy of cfd codes in the prediction of planing surfaces hydrodynamic characteristics", in 2nd International Conference on Marine Research and Transportation., (2007), 147-159.
15. Fultz, E.R., "Cfd analysis of a pentahulled, air entrapment, high speed planing [sic] planning vessel", Monterey, California. Naval Postgraduate School, (2008),
16. Akkerman, I., Dunaway, J., Kvandal, J., Spinks, J. and Bazilevs, Y., "Toward free-surface modeling of planing vessels: Simulation of the fridsma hull using ale-vms", ***Computational Mechanics***, Vol. 50, No. 6, (2012), 719-727.
17. Su, Y., Chen, Q., Shen, H. and Lu, W., "Numerical simulation of a planing vessel at high speed", ***Journal of Marine Science and Application***, Vol. 11, No. 2, (2012), 178-183.
18. Subramanian, V.A., Subramanyam, P. and Ali, N.S., "Pressure and drag influences due to tunnels in high-speed planing craft", ***International shipbuilding progress***, Vol. 54, No. 1, (2007), 25-44.
19. Ghassabzadeh, M. and Ghassemi, H., "Determining of the hydrodynamic forces on the multi-hull tunnel vessel in steady motion", ***Journal of the Brazilian Society of Mechanical Sciences and Engineering***, Vol. 36, No. 4, (2014), 697-708.
20. Ghassabzadeh, M. and Ghassemi, H., "An innovative method for parametric design of planing tunnel vessel hull form", ***Ocean Engineering***, Vol. 60, (2013), 14-27.
21. Savitsky, D. and Morabito, M., "Surface wave contours associated with the forebody wake of stepped planing hulls", ***Marine Technology***, Vol. 47, No. 1, (2010), 1-16.
22. Svahn, D., "Performance prediction of hulls with transverse steps", A Report of Masters Thesis, The Royal Institute of Technology, KTH, Centre for Naval Architecture, (2009).
23. Taunton, D., Hudson, D. and Shenoi, R., "Characteristics of a series of high speed hard chine planing hulls-part 1: Performance in calm water", ***International Journal of Small Craft Technology***, Vol. 152, (2010), 55-75.
24. Grigoropoulos, G.J. and Damala, D.P., "Dynamic performance of the ntua double-chine series hull forms in random waves", in 11th international conference on Fast Sea Transportation FAST. (2011).
25. Ghasemi, H., Mansouri, M. and Zaferanlouei, S., "Interceptor hydrodynamic analysis for handling trim control problems in the high-speed crafts", ***Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science***, Vol. 22, (2011) 247-269
26. Garland, W.R. and Maki, K.J., "A numerical study of a two-dimensional stepped planing surface", ***Journal of Ship Production and Design***, Vol. 28, No. 2, (2012), 60-72.
27. Veysi, S.T.G., Bakhtiari, M., Ghassemi, H. and Ghiasi, M., "Toward numerical modeling of the stepped and non-stepped planing hull", ***Journal of the Brazilian Society of Mechanical Sciences and Engineering***, Vol. 37, No. 6, (2015), 1635-1645.

Numerical Modeling of the Stepped Planing Hull in Calm Water

M. Bakhtiari, S. Veysi, H. Ghassemi

Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran

P A P E R I N F O

Paper history:
Received 10 November 2015
Received in revised form 29 December 2015
Accepted 26 January 2016

Keywords:
Stepped Hull
Turbulent
Free Surface Flow
Volume of Fluid Model
Drag
Wake Profile

چکیده

در این مقاله، کاربرد یک شبیه سازی دینامیک سیالات محاسباتی سه بعدی برای یک شناور پلنینگ استپ دار توصیف می‌شود. جریان مغشوش سطح آزاد اطراف شناور پلنینگ ا دار به روش RANS، با استفاده از حل‌گر ANSYS-CFX محاسبه می‌شود. مدل اغتشاشی به کار گرفته شده، مدل Standard k–ε است. به منظور شبیه سازی سطح آزاد مغشوش شده، از مدل حجم سیال (VOF) استفاده می‌شود. مدل CFD حاضر در ابتدا با استفاده از نتایج آزمایشگاهی اعتبار سنجی شده است. نتایج عددی به دست آمده برای درگ، توزیع فشار، سطح خیس، اسپری آب، پروفیل ویک و موج ایجاد شده توسط بدنه شناور در سرعت-های مختلف ارائه و مورد بحث قرار می گیرند. همچنین، پروفیل‌های ویک محاسبه شده توسط مدل حاضر، با پروفیل‌های محاسبه شده توسط معادلات تجربی Savitsky مقایسه می‌شوند.

***doi**: 10.5829/idosi.ije.2016.29.02b.13*