AUT Journal of Mechanical Engineering

AUT J. Mech. Eng., 3(2) (2019) 139-148
DOI: 10.22060/ajme.2019.14364.5723

Numerical Investigation of Step Depth Effects on Hydrodynamic Performance of Planing Hull Using Dynamic Mesh and Two Degree of Freedom Model

R. Tork Chooran, R. Shafaghat*, R. Yoosefi

Department of Mechanical Engineering, Babol Noshiravani University of Technology, Babol, Iran

ABSTRACT: At low speeds, planing hull performs like a displacement one and buoyancy force has the most influence on it, but, when it reaches to enough speed, hydrodynamic lift force equilibrates 50–90 percent of its weight. Planing hull researchers have introduced different methods in order to achieve the highest speed. A desirable planing hull has low weight-to-power ratio and good maneuverability. Several ways have been applied to reduce drag and one of the best strategies is to use step that leads to less wetted surface and more lift power. This work addresses the numerical study of step height effect on hydrodynamic performance of planing hull. A specified form of a monohull was changed to the step one while important geometric parameters such as Deadrise angle, width and length were equal in both of them. In order to simulate hull movements, a comprehensive series of viscous computational fluid dynamics simulations considering free-surface and two degree of freedom motion of the hull (heave and pitch) have been performed by application of dynamic mesh. Results have been presented as contours and plots. According to the results, deeper steps provide greater levels of ventilation but, there is a limit in step depth increment because porpoising happens after a specific height.

Review History:

Received: 24 April 2018
Revised: 10 October 2018
Accepted: 27 December 2018
Available Online: 17 January 2019

Keywords:

High speed planing hull
Step
Two degrees of freedom
Trim angle
Volume of fluid method

1- Introduction

Shipbuilding industry have been improved same as other related industries and as an example, design of high speed hulls with various shapes and applications can be mentioned. During the last decades, attentions to planing hulls have increased significantly. A suitable design is important for this hull because of its great impact on weight and cost. Speed is the most important point for high speed hulls alongside good stability, so researchers tried to optimize these factors. At low speeds, planing hull performs like a displacement one and buoyancy force has the most influence on it, but, when it reaches to enough speed, hydrodynamic lift force equilibrates 50–90 percent of its weight. Specific geometric characteristics of planing hull make it possible to achieve high speeds, but that's the time when it runs into more Drag-Lift ratio. Using stepped planing hulls with transverse discontinuity at the bottom of the hull may be a solution to this problem [1].

Fig. 1 displays the wetted surface of the stepped hull in planing mode. For a stepped hull, flow separation occurs at step location and then reattach at aftbody. This phenomenon reduces the wetted area and may result in a Drag-Lift ratio reduction [2].

The first scientific study on the stepped planing hull was conducted by Baker [4], and since then, a huge number of racing hulls have used this design. Sottorf [5] performed the first comprehensive lab work which attracted much attention and was followed later by Shoemaker [6] and Sedov [7]. Study on stepped planing hull and parametric variations on it by John Dawson et al. [8] was one of the first researches on the effects of step ,however, the most important study on the performance of stepped planing hulls was performed by Savitsky [9]. He and Brown [10] then published some semi-empirical equations for Hydrodynamic Evaluation of a planing hull.

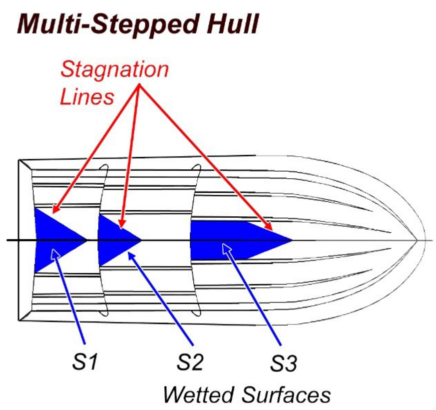


Fig. 1. Stepped hull [3]

In recent years, significant work has been done in this area to, Due to the importance of stepped planing hulls application. Clement et al. [11] studied Performance of Stepless and Stepped Planing Hulls. Makasyeyev [12] numerically studied the cavity flow on bottom of a stepped planing hull. Taunton et al. [13] examined the effects of trim, draft and resistance on stability for three different profiles. Matveev [14] modeled a stepped planing hulls with open and pressurized air cavities. Shin et al. [15] studied resistance characteristics of a stepped planing hull in a towing tank.

Later, David Svahn [16] studied Savitsky's equations [9] exactly and developed a model to predict the characterization of stepped planing hulls. To provide a complete and accurate

Corresponding author, E-mail: rshafaghat@nit.ac.ir

Review of planing hull mode, Savitsky and Morabito [17] presented a mathematical model for planing hulls. Garland and Maki [18] evaluated the performance of stepped planing hulls for non-linear flow in two-dimensional. Matveev [19] has examined the effect of step geometry on wave profiles on the free surface. He used an experimental method in this study to compare His analysis with a two-dimensional numerical analysis. Afshin Loni et al. [20] reviewed the details of stepped planing hull and used a mathematical model to investigate the effect of different variables on monohulls which has been introduced by Savitsky [9] in the past.

Numerical methods, such as those based on Computational Fluid Dynamics (CFD) simulations, can be used nowadays to calculate the hydrodynamic performance of a stepped hull with good accuracy [21]. Jones and Clark [22] used commercial software FLUENT to simulate the flow around a body DTMB5415. They used Finite Volume Method (FVM) to simulate free surface, total resistance, Waveforms and velocity field. Total resistance had a 3.8 percent error and the simulation of velocity field was carried out with 10% error. It was concluded that FLUENT is able to simulate waves, wake, free surface, Hydrodynamic forces and velocity field. Seif et al. [23] compared a monohull and a catamaran boat by using NUMERALS code written in Sharif University of Technology in finite volume method. They studied the friction and pressure drag forces and drag on the hull and compared the results. Ghassemi et al. [24] used a CFD method for performance prediction of stepped planing hulls. Bakhtiari et al. [25] numerically studied a stepped planing hull in calm water.

Vafaee sefat et al. [26] used finite volume method to calculate the forces on a planing hull with variable Deadrise angle, and then he compared the experimental results with their work. Yousefi and Shafaghat [27] compared several hydrodynamic analysis techniques for the planing hulls and chose a FVM based numerical method to study the drag reduction effect of tunnels [28], which were introduced at the bottom section of a planing mono-hull; a resistance reduction of ~14% was reported at 60 knot. Kazemi Moghadam and Shafaghat [29] numerically simulated the forward motion for a series of tunneled planing hulls with different tunnel aperture using the FLUENT software; their result showed that the small tunnel aperture could achieve more drag reduction at high Froude number. Ghaseemi et al. [30] applied a numerical method to investigate the influence of step on hydrodynamic characteristics of a modern high-speed chinned planing hull for speed range from 4.05 to 12.05. De Marco et al. [21] conducted experimental results of towing tank tests in calm water on a single-step hull model and the same flow conditions were analyzed via Reynolds Averaged Navier-Stokes (RANS) and Large Eddy Simulations (LES), with different moving mesh techniques.

There are many variables that must be considered when discussing a step design. These variables include step shape, step depth, longitudinal location of the step, and method of ventilation. Step depth may, in fact, be the most important variable, provided that the step is located in a reasonable location longitudinally [31].

In this work, a series of CFD tests considering free-surface. Although there are some studies done that used numerical methods to analyze flows on stepped hulls but it's rare to observe step height effect by CFD codes and they just simulate a stepped hull which was tested in a towing tank. Novelty points of this work are using 2 Degree Of Freedom (DOF) motion of hull, using dynamic mesh and investigating the effect of step height on it drag reduction performance. Navier-stokes equations were solved using Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm. For turbulence modeling *k-ε* Re-Normalisation Group (RNG) model was used. To acquire final convergent value of trim angle, sinkage and Drag force, the monitored data were allowed to reach a steady state.

2- Governing Equations

2- 1- The equation of conservation of mass

Considering the mass balance for a fluid element, Net rate mass flow input of the fluid element must be equal to the rate of increase in the mass of the fluid element, Assuming incompressible flow,

$$div\ U = 0 \tag{1}$$

2- 2- The equation of conservation of momentum

Newton's Second Law states that rate of change of momentum of a fluid particle is equal to the force exerted on the particle. Considering a fluid element, the equations of motion (Navier-Stokes equations) can be written as follows.

$$\begin{aligned}
\frac{\partial(\rho u)}{\partial t} + div(\rho u U) &= -\frac{\partial P}{\partial x} + div(\mu.grad(u)) \\
\frac{\partial(\rho v)}{\partial t} + div(\rho v U) &= -\frac{\partial P}{\partial y} + div(\mu.grad(v)) \\
\frac{\partial(\rho w)}{\partial t} + div(\rho w U) &= -\frac{\partial P}{\partial z} + div(\mu.grad(w)) - \rho g
\end{aligned} \tag{2}$$

2- 3- Equations of motion of rigid body

Modeling the hull requires to solve the equations of motion of rigid body during the time. To search for rigid body motion, two coordinate system was used. *XYZ* coordinate system is an inertial system and connected to the ground. *XYZ* coordinate is a free system with desired speed and acceleration. According to the principle of linear momentum and the angular momentum of float with "*m*" mass, Linear velocity *V* (*u*, *v*, *w*) angular velocity *Ω* (*p*, *q*, *r*),

$$\begin{aligned}
\frac{dT}{dt} &= F \\
\frac{dK}{dt} &= M
\end{aligned} \tag{3}$$

Eq. (3) can be written as,

$$\begin{aligned}
\frac{d\breve{T}}{dt} + \grave{U} \times T &= F \\
\frac{d\breve{K}}{dt} + \grave{U} \times K + V \times T &= M
\end{aligned} \tag{4}$$

Extending the above equations and rewrite in dynamic coordinate system,

Table 1. Using FLUENT software settings

Row	Subject	Preferred method	
1	Solver	Unsteady, Segregated, implicit	
2	Turbulence model	K-ε RNG	
3	Two-phase method	VOF	
4	Discretization method	Volume ratio	Modified HRIC
		Momentum	Second Order Upwind
		Pressure	Presto
		Turbulent Kinetic Energy	First order upwind
		Turbulent Dissipation Rate	First order upwind

$$\begin{aligned} &m(\dot{u}+qw-rv)=X \\ &m(\dot{v}+ru-pw)=Y \\ &m(\dot{w}+pv-qu)=z \\ &I_x\dot{p}+(I_z-I_y)qr=L \\ &I_y\dot{q}+(I_x-I_z)pr=M \\ &I_z\dot{r}+(I_y-I_x)qp=N \end{aligned} \tag{5}$$

All freedom of movement were ignored except for heave and pitch due to their importance and calm water assumption.

2- 4- Free surface equation

Free surface modeling was important because of free surface deformation and waves fracture. Free surface modeling based on surface capture and Volume Of Fluid[1] (VOF) was the best method. In this case, a transport equation was resolved to calculate the volume of a two-phase fluid. Volume fraction transport equation (Eq. (6)), was obtained by the equation of continuity. It was considered that two phases were incompressible. After solving the transport equation, density and viscosity of the fluid were calculated from Eq. (7).

$$\frac{\partial \alpha}{\partial t}+\nabla.(\alpha u)=0 \tag{6}$$

$$\begin{aligned} &\rho_{eff}=\alpha.\rho_1+(1-\alpha).\rho_2 \\ &\nu_{eff}=\alpha.\nu_1+(1-\alpha).\nu_2 \end{aligned} \tag{7}$$

α Is a number between zero and one and states the percentage of each fluid in the calculation domain.

2- 5- Turbulence modeling

Turbulence was modeled using k-ε RNG two equation model. Using this model in multiphase flow, increases the stability of the solution. In k-ε RNG model the turbulent field was expressed in terms of two variables, the kinetic energy of turbulent flow κ and Viscose dissipation rate of turbulent kinetic energy ε, which were obtained from the differential transport Eqs. (8) and (9).

$$\frac{\partial(\rho k)}{\partial t}+\nabla\cdot(\rho U k)=\nabla\cdot\left(\left(\mu+\frac{\mu_t}{\sigma_k}\right)\nabla k\right)+P_k-\rho\varepsilon \tag{8}$$

1 volumetric approach

$$\begin{aligned} &\frac{\partial(\rho\varepsilon)}{\partial t}+\nabla\cdot(\rho U\varepsilon)=\nabla\cdot\left(\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right) \\ &+\frac{\varepsilon}{k}\left(C_{\varepsilon1}(P_k)-C_{\varepsilon2}\rho\varepsilon\right) \end{aligned} \tag{9}$$

In this case to obtain the value of variables at different times, Unsteady Analysis should be used. Time step was chosen based on the Courant number formula which depends on mesh size, time step size and velocity. For open channel flow it is offered to consider courant number equal to 0.25. For example for velocity of 3.2 m/s time step size was 0.0001 s.

3- Geometry and Boundary Conditions

The main aspects and details of the base hull are provided in the Table 2. Step was vertical and its longitudinal location was at the 1/3 of hull length from the transom. To create step on the bottom of hull, paper of Garland [31] was studied and we used a similar algorithm for step creation. For the analysis, two sizes of step height were selected and we made it dimensionless to the hull length. It was as h/L=0.005 and h/L=0.01.

Table 2. Hull Main dimensions without step

Subject	Unit	Original dimensions	Model dimension
Hull total length	meter	10	1
Total beam	meter	2.52	0.252
Total height	meter	1	0.1
Vessel weight at full load	kN	68	6.8
Center of gravity length	meter	2	0.2

Inlet was consisted of water and air inlet which should be defined. So, the inlet face was divided into two parts which upper part was the air inlet and the lower part was water inlet. Both inlets were set as velocity inlet and velocity was equal to the hull's velocity. Pressure outlet boundary condition was used for the outlet region of flow. To solve the flow, dynamic mesh and two degrees of freedom were used. So hull was completely modeled. Right and left sides and up and down boundaries were far enough from the hull and flow effects at these boundaries on the hull was insignificant so we used the symmetry boundary condition. For hull wall, non-slip wall boundary condition was used. Details on hull boundary conditions are listed in Table 3.

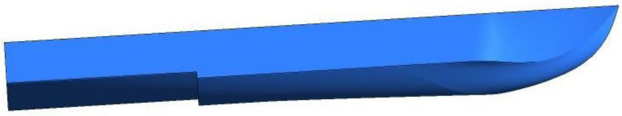


Fig. 2. View of a stepped hull

Table 3. Boundary conditions used

Boundary type	Boundary condition type
Air inlet	Velocity inlet
Water inlet	Velocity inlet
Outlet	Pressure outlet
Hull	Wall
Side boundaries	Symmetry

4- Computational Domain and Meshing

We used Solidworks and Gambit to produce hull geometry and the grid. Due to the complex geometry, unstructured mesh with tetrahedral elements was used around the body. If the same elements were used for entire domain, the mesh size and Computational time would increase. So to reduce the mesh size in farfield domains, the structured mesh with hexahedral cubic elements was used. Figs. 3 and 4 show shots of meshing on the hull.

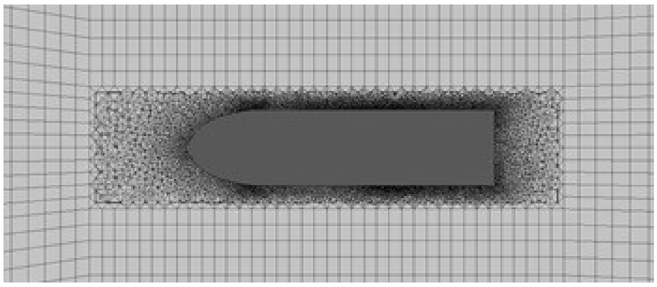
Mesh size effect is impressive in the results. The coarse mesh would cause error in analysis. So, the mesh size should be small as possible. But increasing number of meshes cause much more calculations and thus the calculation time. The mesh size should be such that in case of increasing, results does not change a lot.

We used 3 to 15 mm and Tetrahedral (Tet) mesh around the body as seen in Fig. 4, and 20 to 40 mm mesh and Hexahedral (Hex) mesh for regions far from the body as seen in Fig. 3. Flow solution around the hull was applied with two degrees of freedom and dynamic mesh to update hull draft numerical solution. Mesh quality was important in midsection which was with Tet mesh. By movement of hull in each iteration in this region, grid was modified to reach a steady state. Total number of elements at this problem was 1143627. To make sure that results are independent of mesh size, another case had been studied. It contained 1735461 elements. Comparing both lift and drag forces on the body showed an error about 2/94 percent.

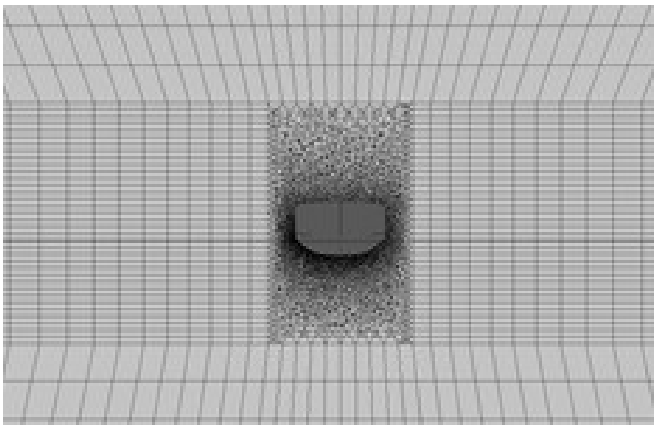
Table 4. Drag changes in stepped hull for increasing in mesh

Step	Mesh size	Drag (kN)
1	655430	13.3392
2	935467	11.9801
3	1143647	10.2983
4	1735461	10.0354

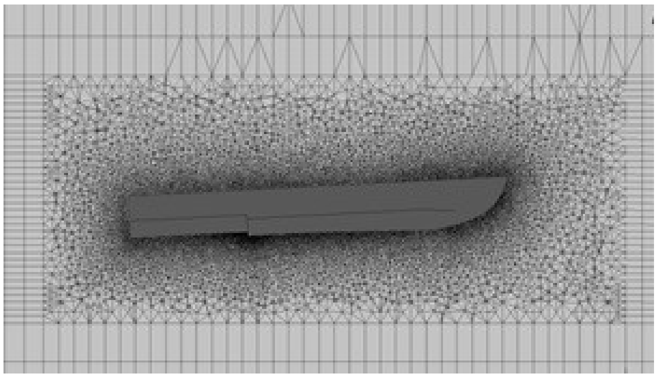
In this study, flow domain is three-dimensional, transient and in two phases. Trim angle and draft changes were monitored to ensure that results were time independent. Solution was continued untill there was insignificant changes in results. The results of the trim angle stability by increasing the computational time is presented in the figure below.



(a)



(b)



(c)

Fig. 3. Shots of meshing on the hull a) Top view, b) front view, and c) Side view

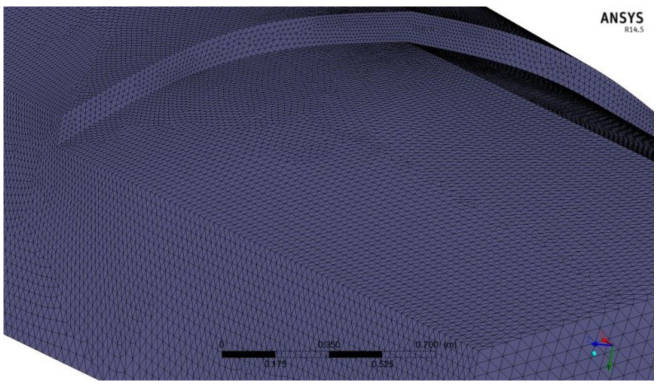


Fig. 4. Very fine mesh on step region

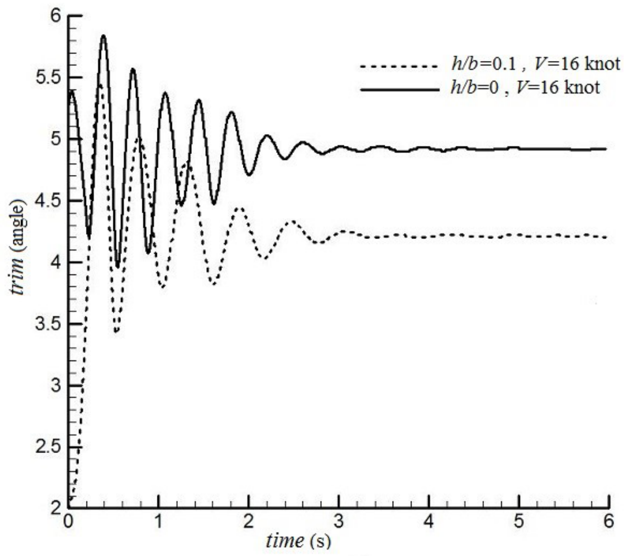


Fig. 5. Monitoring the solution convergence displaying trim angle during the time

Solution field and grid type are very important in hull dynamic analysis. Small boundaries cause error and large ones increase the cost and computational time. So to determine the solution field optimum size, for the front, back, up, down and side region of the vessel 3, 8, 1.5, 2.5, 2.5 times the overall length of the hull was selected as is shown in Fig. 6.

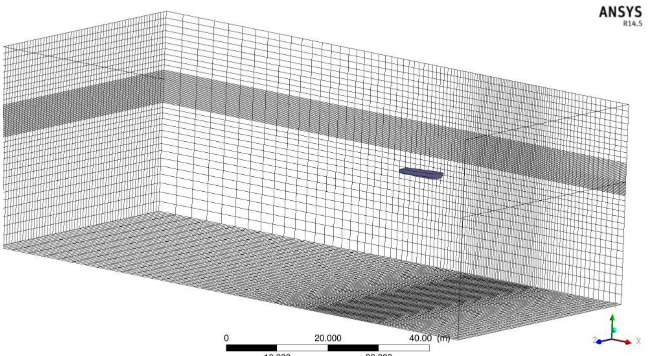


Fig. 6. Computational domain size of flow solution over the hull

5- The Result and Discussion

Froude (*Fr*) Number is important in the free surface problem. It is calculated as below.

$$Fr = \frac{V}{\sqrt{gl}} \tag{10}$$

So at this table velocity sections and their *Fr*. Number is presented.

Table 5. *Fr*. Number of Vessels at different speeds

Row	Velocity (knot)	*Fr*. Number
1	3.11	0.52
2	9.32	1.56
3	15.82	2.66

In this study effect of step height was studied using CFD method. In CFD methods which analyses turbulent flow, *YPlus* has a key role to ensure a correct answer. For k-ε RNG model, *YPlus* should be under 300. This is because over this limit CFD model cannot sense turbulent boundary layer correctly and drag results are not acceptable. In this study different grids were applied and a grid with suitable *YPlus* was selected. Fig. 7 shows *YPlus* in different speeds in this study.

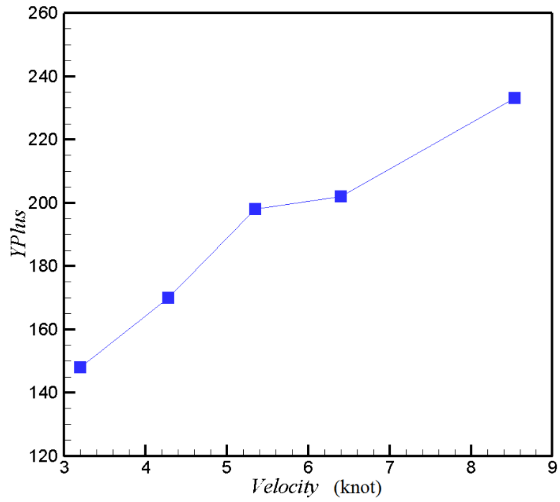


Fig. 7. *YPlus* vs. planing hull velocity

5- 1- Solution validation

Savitsky's semi-empirical method is a valid way to predict resistance and Hydrodynamic parameters. This method was used to validate the numerical solution. Fig. 8 shows results of drag for both numerical and Savitsky method. It indicated a good agreement between the present results and Savitsky method.

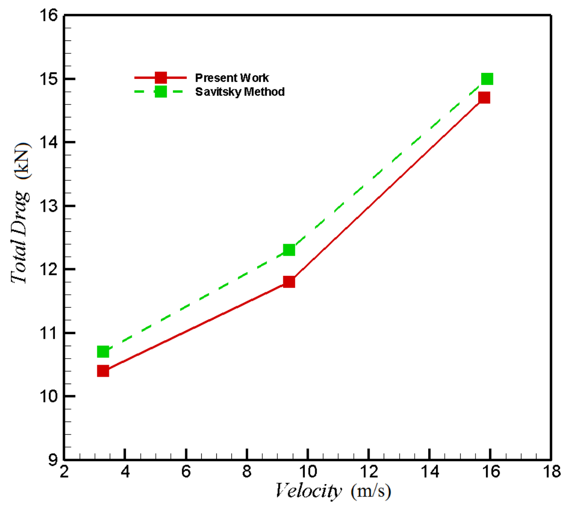


Fig. 8. Solution validation [10].

5- 2- The effects of adding step

To investigate effects of adding a step, a dimensionless ratio of step height to Kiel line length was used. So *h/L* for a normal vessel was zero. Reducing drag was the main purpose of

adding steps to a hull while it should not affect the stability. As the hull velocity increased, water lifted it toward the surface and draft decreased. When Hull comes out of water, it has less wetted area and reduced viscous drag. Maximum trim angle, minimum wetted surface and minimum viscous drag occurred in the beginning of planing mode. By increasing area facing fluid due to high trim angle, pressure drag increased. As the velocity increased in planing mode, trim angle reduced and wetted area and viscous drag incremented.
Fig. 9 compares total drag in original and stepped hulls. At low velocities and displacement mode, total drag was less in the original vessel, but in high velocity, total drag of the stepped hull was less. To continue drag reduction deeper step can be used. As the step gets deeper, at constant speed, the flow needs more length to reattach to the hull. It means that the ventilation length gets longer and reduces the wetted surface more. These events causes bigger viscous drag reduction because viscous drag is related to wetted surface. So it can be said that as the ventilation region gets longer, vessel has more drag reduction.
For a normal vessel, from displacement to planing mode, it was expected that increasing velocity would result in drag increment. While, results indicated that a stepped hull will have a drag reduction relatively. This drag made it easier to go to planing mode.

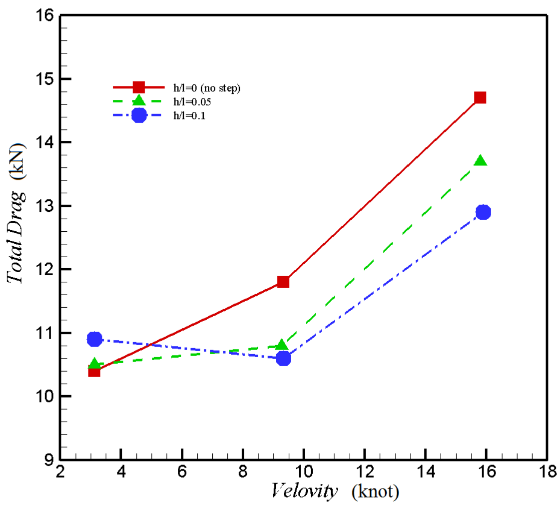


Fig. 9. The effect of adding a step in hull *total* drag

We can analyze drag reduction in another way. Increasing lift in stepped hull plays a key role in wetted area and drag decrease. Fig. 10 shows that as the velocity increased, draft decremented and vessel got out of the water, but this event was much more in a stepped hull.
Fig. 11 reveals a lift increment in stepped vessel. Adding step caused a low-pressure region aft of the step, but this would not last as we get far from a step. When water reattached to aft of the hull at transom, it made a dynamic pressure and thus an extra lift. This allowed the transom to come to water surface and caused a better performance by a draft reduction.
Fig. 11 also indicates that close to step, a low-pressure region was seen. Flow separation occurred in this region, thus wetted area aft of the vessel was reduced.
Fig. 12 shows dynamic pressure on hull. From point A, on hull bottom a pressure increment can be seen and by reaching point B, where step was there an intense pressure

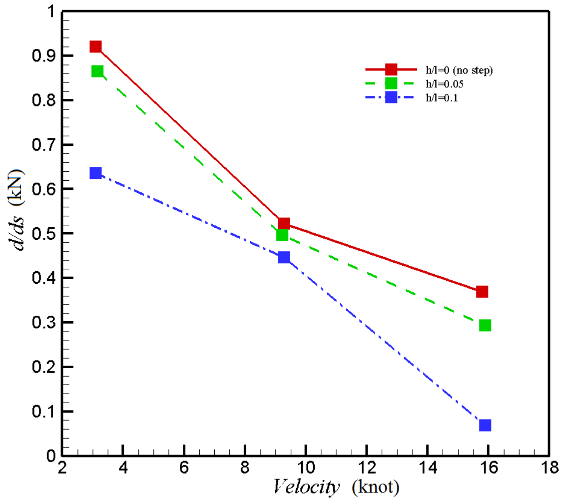


Fig. 10. Hull draft variations by creating step

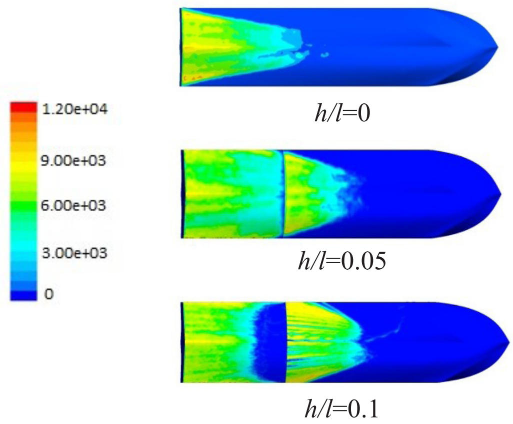


Fig. 11. The effects of adding steps on dynamic pressure planing hull at V=9.6 knots

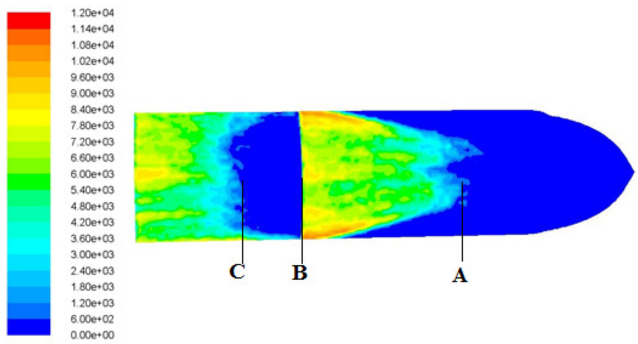


Fig. 12. Dynamic pressure changes on hull bottom in V=9.6 knots, Z=0 surface and h/L=0.15

reduction occurred due to water separation from hull bottom. At reattachment point C, to end of vessel, dynamic pressure increased.
Fig. 13 shows that as the velocity increased, wake aft of hull increased to, and vessel came to the water surface. This wake and trim angle would increase by adding step.
Observing trim angle change was an important parameter in hull performance analysis. As shown in Fig. 14 both original and stepped hull acted similar at trim angle variation. As

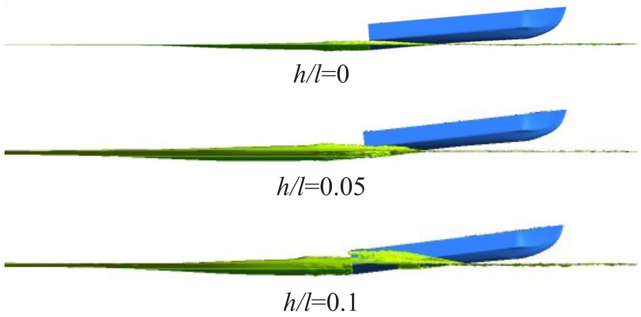


Fig. 13. Wake created aft of the hull, step height increment at *V*=9.6 knots

the velocity increased, there was a maximum point which represented the beginning of planing mode. When the hull shifted to planing mode, trim angle reduced. Adding step to the original vessel changed center of pressure length and increased trim angle compared to non-stepped mode.

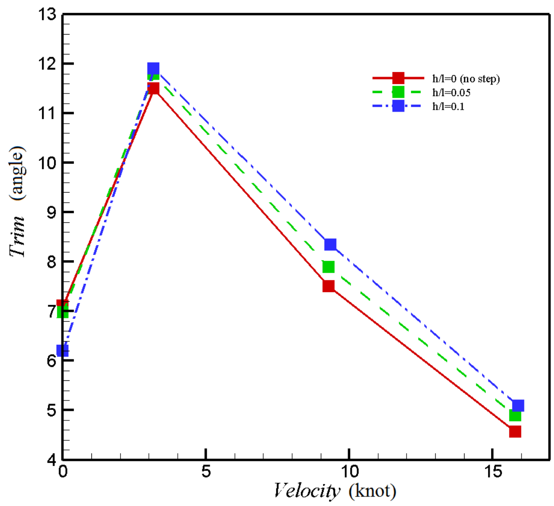


Fig. 14. Effect of adding step on trim angle

Fig. 14 indicates that in a constant step height, trim angle reduced as the velocity increased. This fact can be seen in Fig. 15 too.

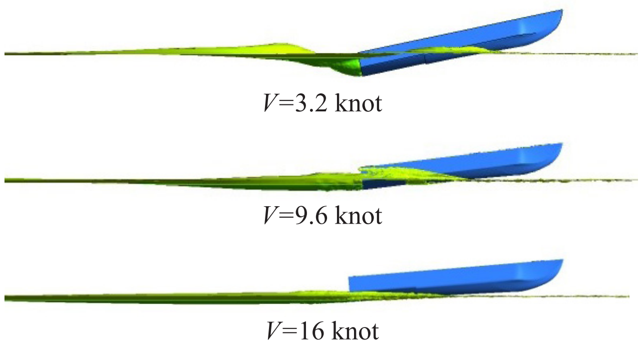


Fig. 15. Decrees in trim angle with a velocity increase for *h/L*=0.05

Presented results up to here showed that hull performance can be improved by deeper step. Investigations stated that deeper step can cause an inappropriate hull performance. Step is used to create lift while water reattaches to the hull transom, so if water do not reattach to hull transom, the ventilation region length is not suitable. To check this case, simulation and performance analysis had been done on *h/L*=0.125 and *h/L*=0.15 at velocity of 15.8 knots. As ventilation region length increased more than a specific value, water cannot attach to hull bottom. In other words, flow did not meet hull bottom and needs more trim angle for reattachment. Fig. 15 displays phase contour of hull bottom and indicates that step performance caused water not to reattach to hull bottom.

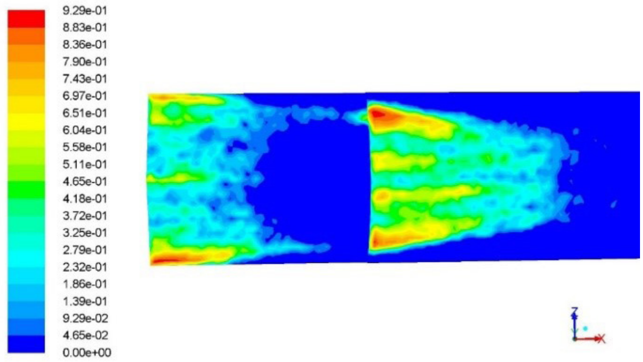


Fig. 16. A view of phase contour on hull bottom

In Fig. 16, most of water presence is at the first of step and vessel stern does not contact water. It is a deficiency that will cause the vessel not to perform correctly. Because although drag reduced, required lift was not generated and a stable condition was impossible. In this situation changing draft was a simple way to create needed lift. With changing draft and increasing trim angle, water could reattach to vessel stern. A situation that was undesirable in this unwanted changes is porpoising of the hull. So, according to presented results *h/L*=0.1 is optimum step depth.

6- Conclusion

In this paper a two phase flow was simulated to get drag force on a stepped planing hull by using ANSYS-FLUENT commercial software. The main purpose was to study effect of transverse step depth on its performance in drag reduction and hull stability. A base hull was chosen and transverse step was created on it. The longitudinal location of transverse step was extremely important, and due to previous works and simulations, it was selected equal to 1/3 of hull total length from the transom. To solve the flow field, a 2-DOF model and dynamic mesh was used in different hull velocities. After a period of time, hydrodynamic parameters such as trim angle and sinkage, reached to a steady state. As pointed in results, selecting a proper height for step is an effective strategy in drag reduction and increasing hull stability. Comparing drag force on vessels indicated that creating step and increasing its height reduced drag. Observing free surface counters showed that in high speeds, sinkage decreases and hull comes to water surface which was reasonable, because in planing mode the hull is completely out of water. At low speeds, drag force on a stepped planing hull was much more than a monohull, while in planing mode, it had less resistance compared to conventional hulls. Stepped planing hulls manage to increase trim angle to make a lower drag-lift ratio possible at high speeds while keeping away from porpoising instability, but by increasing step height more than a specific value, the hydrodynamic performance of the vessel got into trouble and resulted in porposing and hull instability.

Nomenclature

h	step height
L	vessel length
eff	effective
I	moment of inertia
div	divergence
grad	gradient
Fr	Froude Number
U	Velocity vector, m/s
u	*x* direction component of velocity, m/s
v	*y* direction component of velocity, m/s
w	*z* direction component of velocity, m/s
P	Static Pressure, Pa
m	mass, kg
g	gravitational acceleration, m/s^2
Ω	Vector of angular velocity, rad/s
p	*z*-component of angular velocity, rad/s
q	*r*-component of angular velocity, rad/s
r	*θ*-component of angular velocity, rad/s
Greek Symbols	
ρ	Density, kg/m^3
μ	Viscosity, Pa.s
κ	Kinetic energy of turbulent flow
ε	Viscose dissipation rate of turbulent kinetic energy
α	factor of a phase

References

[1] P. Lotfi, M. Ashrafizaadeh, R.K. Esfahan, Numerical investigation of a stepped planing hull in calm water, *Ocean engineering*, 94 (2015) 103-110.

[2] O.M. Faltinsen, *Hydrodynamics of high-speed marine vehicles*, Cambridge university press, 2005.

[3] http://www.aeromarineresearch.com/steps.html.

[4] L.R. Baker, Boat, in, *Google Patents*, 1916.

[5] W. Sottorf, *Experiments with planing surfaces*, (1934).

[6] J.M. Shoemaker, *Tank tests of flat and v-bottom planning surfaces*, (1934).

[7] L. Sedov, *Scale effect and optimum relations for sea surface planning*, (1947).

[8] J.R. Dawson, *Tank tests of two models of flying-boat hulls to determine the effect of ventilating the step*, (1937).

[9] D. Savitsky, Hydrodynamic design of planing hulls, *Marine Technology*, 1(1) (1964).

[10] D. Savitsky, P.W. Brown, Procedures for hydrodynamic evaluation of planing hulls in smooth and rough water, *Marine Technology*, 13(4) (1976) 381-400.

[11] E.P. Clement, J.D. Pope, Stepless and Stepped Planing Hulls-Graphs for Performance Prediction and Design, DAVID TAYLOR MODEL BASIN WASHINGTON DC, 1961.

[12] M.V. Makasyeyev, *Numerical modeling of cavity flow on bottom of a stepped planing hull*, (2009).

[13] D. Taunton, D. Hudson, R. Shenoi, 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.

[14] K.I. Matveev, Two-dimensional modeling of stepped planing hulls with open and pressurized air cavities, *International Journal of Naval Architecture and Ocean Engineering*, 4(2) (2012) 162-171.

[15] J.-I. Shin, J.-M. Yang, H.-W. Park, J.-S. Kim, H.-C. Kim, A Study on the Resistance Characteristics of a Stepped Planing Hull Using a High-Speed Towing System, *Journal of the Society of Naval Architects of Korea*, 42(4) (2005) 341-349.

[16] D. Svahn, *Performance prediction of hulls with transverse steps*, A Report of Masters Thesis, The Royal Institute of Technology, KTH, Centre for Naval Architecture, (2009).

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

[18] W.R. Garland, K.J. Maki, A numerical study of a two-dimensional stepped planing surface, *Journal of Ship Production and Design*, 28(2) (2012) 60-72.

[19] K.I. Matveev, T.J. Burnett, A.E. Ockfen, Study of air-ventilated cavity under model hull on water surface, *Ocean Engineering*, 36(12) (2009) 930-940.

[20] A. Loni, P. Ghadimi, H. Nowruzi, A. Dashtimanesh, Developing a computer program for mathematical investigation of stepped planing hull characteristics, *International Journal of Physical Research*, 1(2) (2013) 34-47.

[21] A. De Marco, S. Mancini, S. Miranda, R. Scognamiglio, L. Vitiello, Experimental and numerical hydrodynamic analysis of a stepped planing hull, *Applied Ocean Research*, 64 (2017) 135-154.

[22] D. Jones, D. Clarke, FLUENT Code simulation of flow around a naval hull: the DTMB 5415, DEFENSE SCIENCE AND TECHNOLOGY ORGANIZATION VICTORIA (AUSTRALIA) MARITIME, 2010.

[23] M.S. Seif, E. Jahanbakhsh, R. Panahi, M.H. Karimi, A unified computational method for simulating dynamic behavior of planing vessels, *China Ocean Engineering*, 23(2) (2009) 1-10.

[24] H. Ghassemi, M. Kamarlouei, S.T.G. Veysi, A hydrodynamic methodology and CFD analysis for performance prediction of stepped planing hulls, *Polish Maritime Research*, 22(2) (2015) 23-31.

[25] M. Bakhtiari, S. Veysi, H. Ghassemi, Numerical modeling of the stepped planing hull in calm water, *International Journal of Engineering-Transactions B: Applications*, 29(2) (2016) 236.

[26] A. Mehrizi, M.T. Dakhrabadi, A.V. Sefat, M. Seif, Hydrodynamic Optimization of Hull Form of High Speed Planing Craft by Multi Objective Genetic Algorithm in Calm Water.

[27] R. Yousefi, R. Shafaghat, M. Shakeri, Hydrodynamic

analysis techniques for high-speed planing hulls, *Applied ocean research*, 42 (2013) 105-113.

[28] R. Yousefi, R. Shafaghat, M. Shakeri, High-speed planing hull drag reduction using tunnels, *Ocean engineering*, 84 (2014) 54-60.

[29] H.K. Moghadam, R. Shafaghat, R. Yousefi, Numerical investigation of the tunnel aperture on drag reduction in a high-speed tunneled planing hull, *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 37(6) (2015) 1719-1730.

[30] S.T.G. Veysi, M. Bakhtiari, H. Ghassemi, M. Ghiasi, Toward numerical modeling of the stepped and non-stepped planing hull, *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 37(6) (2015) 1635-1645.

[31] W.R. Garland, Stepped planing hull investigation, *SNAME Trans.*, (2010) 1-11.

Please cite this article using:

R. Tork Chooran, R. Shafaghat, R. Yoosefi, Numerical Investigation of Step Depth Effects on Hydrodynamic Performance of Planing Hull Using Dynamic Mesh and Two Degree of Freedom Model, *AUT J. Mech. Eng.*, 3(2) (2019) 139-148.

DOI: 10.22060/ajme.2019.14364.5723