Analysis of the double steps position effect on planing hull performances

UDC 629.5.015.2:629.52
Original scientific paper

Summary

Along with developing high-speed craft technology, the planing hull is growing with modifications for better performance. One such technology is stepped hull, both single and double. Planing hull with steps allows the boat to run at a relatively low drag-lift ratio with lower frictional resistance due to reduced wetted area. In this study, the hull was modified with variations in the position of the double steps, which aimed to determine the effect of the first and second step positions on the total resistance, dynamic trim, and dynamic sinkage generated by computational fluid dynamics (CFD). Based on the analysis results, variations in the position of the stepped can change the hull performance. Shortening the distance between the two steps and moving both rearwards toward the transom can lower the total resistance. The dynamic trim and dynamic sinkage decreased as the position of the two steps was shifted further forward. An equation created in a non-dimensional form relates the positions of two steps to the desired results of total resistance, dynamic trim, and dynamic sinkage, namely: 
\[
\left\{ (x_1-x_2)/L + (x_1x_2)/(LB) \right\} \times Fr, \text{ where } x_1 \text{ is distance the first step from transom, } x_2 \text{ is the distance of the second step, } L \text{ is the length of the boat, } B \text{ is the beam of the boat, and } Fr \text{ is the volume Froude number.}
\]

Key words: double steps position; planing hull performances; step hull; boat resistance; Computational Fluid Dynamics (CFD)

1. Introduction

Efforts to decrease emissions on high-speed marine vehicles (HSMVs) are necessary due to their high energy consumption [1], often fueled by fuel oil. These emissions reduction measures are a major focus in the fight against climate change and global warming [2,3], as HSMVs pose a particular challenge. On-board methods to decrease emissions [4,5] include optimizing the hull shape [6–9], utilizing weather routing [10,11], preventing hull roughness caused biofouling [12–21] or coating [22–27], exploring more eco-friendly fuels [28–32], optimizing propellers [33,34], using advanced coating [35–41] and using the energy saving
devices [42–45]. Efforts to improve energy efficiency in high-speed marine vehicles must also be considered.

One type of advanced high-speed marine vehicle, especially high-speed craft, experiences planing phenomena. The planing phenomenon occurs because, in addition to the boat's hull moving very fast at a Length Froude number \((Fr_L)\) above 1.2 [46], the hull is specially designed like a prism so that the hydrodynamic flow causes a significant lift force. The hydrodynamic force in the direction along the hull length becomes the ship's resistance \((R_p)\), while the one in the vertical direction becomes the lift force. The hydrodynamic lift force and the hydrostatic force support the boat's weight. What is different from the displacement hull type is that on the planing hull, the value of the hydrodynamic force in the vertical direction is dominant compared with the hydrostatic force [47]. This description is the basis for categorizing hull types, where in the planing hull, the hydrodynamic force is relatively high and has an impact as a lift force. In contrast, the lift force is minimal in the displacement-type hull, where the hydrostatic force is dominant. With this lifting force, the boat's hull is lifted into its equilibrium position. The equilibrium position occurs due to the imbalance in the force on the vertical axis and the moment on the transverse axis of the craft, which will cause sinkage and trim, respectively [48]. As a result of lifting the hull, several values decreased, such as volume displacement, wetted hull area, and wetted length. Reducing these values can reduce the boat's resistance, both frictional resistance and residual resistance. The dominant residual resistance is pressure and wave making resistance.

Planing hull also has several issues and several ways to overcome the issues. Planing hulls can experience dynamic instabilities or operate with a high trim angle in the vertical plane due to the center of pressure being far from the transom. To prevent this, the vessel can be equipped with devices that actively or proactively control the trim angle, or modifications can be made to the hull. Applying a trim control device on a high-speed craft, such as such as interceptor [49–58], trim tab [59–61], the combination of interceptor and trim tab [50,58] [41, 45], and stern foil [62–66], can reduce drag and trim angle. Modifying the hull shape can also reduce the boat's drag, thus improving its performance, such as with a step hull [67] and tunneled hull [68], as examples.

The effect of step hull modification on the performance of planing hull has been widely studied. The stepped hull is a modification of the shape of the hull, in which transverse steps are placed at the bottom, giving the hull the appearance of having two bodies: the forebody and afterbody [69]. Studies using towing tank experiments to analyze the effects of both single step [67,70–77] and double steps [67,69,78–81] have been widely carried out. Taunton et al. [67] conducted experimental tests to study the effect of using a step hull on the performance of high-speed planing-type vessels. In this experimental test using three variations of the model with the same hull, the name of each model is C, C1, C2, where each model has zero, one, and double step hull. According to the tests by Taunton et al. [67], at \( Fr_L \) around exceed 3.5, the hulls using steps have a lower overall resistance when traveling at the same speed than a hull without using the steps. Savitsky and Morabito [69] conducted an experimental analysis of the longitudinal surface shape profile behind prismatic hulls, including stepped hulls. To provide a better understanding of the effect of implementing the double step in a planing hull, other methods are also being carried out, such as a simplified method [79], the 2D+T method [78], the morphing mesh method [80], the potential flow method [76], the open and pressurized air cavities method [76], CFD with fixed mesh method [74], and CFD with dynamic mesh method [73,75]. Based on the results of previous research, flow separation occurs in the step area which then allows air to enter and makes the area not wet, where this phenomenon can reduce the wetted area and can result in a decrease in the frictional resistance [47,67,69,74,76]. The parameters of the stepped hull each have a unique effect on the resulting boat performance, as discussed by Vitiello et al. [70]. According to Vitiello et al. [70], more steps are needed when
the boat's beam is narrower and its speed is lower. The step's height may impact the boat's trim angle. The longitudinal position of the steps affects where the lift force is located, affecting how the boat is trimmed. However, the detailed study of variations in the position of the first and second steps has not been fully studied.

This study will discuss the performance of the effects of modifications in the double-steps position on planing hull based on the findings of the research literature mentioned above. Modifications were made by shifting the position of the step forward and backward, both the first and second steps. The boat performance analyzed is the boat resistance, dynamic trim, and dynamic sinkage obtained using CFD simulations with an unsteady Reynolds-Averaged Navier-Stokes (uRANS) and overset (dynamic) mesh method. The findings of this study are expected to complete the picture of how the position of the double-steps affects the planing boat's performance.

2. Materials and methods

2.1 Hull model preparation

In this study, the research object used was a hull model that a towing tank had experimentally tested by Taunton et al. [67]. The chosen model was the C2 model which has a double step. In this research, the front step was referred to as the first step, and the back step was referred to as the second step. Table 1 describes the model's main particulars, and Fig. 1 and Fig. 2 show the model's generalized representation.

<table>
<thead>
<tr>
<th>No</th>
<th>Parameter</th>
<th>Symbol</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Length overall</td>
<td>L</td>
<td>2.00</td>
<td>m</td>
</tr>
<tr>
<td>2</td>
<td>Beam</td>
<td>B</td>
<td>0.46</td>
<td>m</td>
</tr>
<tr>
<td>3</td>
<td>Draught</td>
<td>T</td>
<td>0.09</td>
<td>m</td>
</tr>
<tr>
<td>4</td>
<td>Displaced weight</td>
<td>Δ</td>
<td>243.40</td>
<td>N</td>
</tr>
<tr>
<td>5</td>
<td>Length overall/ Cube root of the displaced volume</td>
<td>( L/V^{1/3} )</td>
<td>6.86</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>Length overall/ Beam</td>
<td>( L/B )</td>
<td>4.35</td>
<td>-</td>
</tr>
<tr>
<td>7</td>
<td>Deadrise angle</td>
<td>( β )</td>
<td>22.50</td>
<td>degree</td>
</tr>
<tr>
<td>8</td>
<td>Longitudinal center of gravity</td>
<td>( L_{CG} )</td>
<td>0.33</td>
<td>%</td>
</tr>
<tr>
<td>9</td>
<td>Keel center of gravity/ Draught</td>
<td>( V_{CG}/T )</td>
<td>1.10</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>The first step position</td>
<td>( x_1/L )</td>
<td>0.185</td>
<td>-</td>
</tr>
<tr>
<td>11</td>
<td>The second step position</td>
<td>( x_2/L )</td>
<td>0.310</td>
<td>-</td>
</tr>
<tr>
<td>12</td>
<td>Steps height/ Length overall</td>
<td>( H_s/L )</td>
<td>( 5\times10^{-3} )</td>
<td>-</td>
</tr>
</tbody>
</table>
2.2 Variation of the double-step position

The variation of the model is to shift the position of the first and second steps. The two steps are varied based on the $x/L$ value measured from the transom. The length of these variations is presented in Table 2 and illustrated in Fig. 3. The shifting of these steps impacts changes in the displacement ($\Delta$) value. Therefore, the error difference value of displacement ($\varepsilon \Delta \%$) needs to be explained using Equation (1) and the value is presented in Table 2 as well:

$$\varepsilon \Delta \% = \frac{\Delta C_{2.1} - \Delta C_{2}}{\Delta C_{2}} \times 100\%$$  \hspace{1cm} (1)

Fig. 1 Lines plan of the C2 model from Taunton et al. [67]

Fig. 2 The 3D view of the C2 model

Fig. 3 Illustration of variations in the position of the hull’s steps
ed air fluid functions rely on the volume fraction-

\[ \frac{\partial (\rho \bar{U}_i)}{\partial x_i} = 0 \]  

(2)

\[ \frac{\partial (\rho \bar{U}_i)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \bar{U}_i \bar{U}_j + \rho \bar{U}_i u'_j) = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j} \]  

(3)

\[ \bar{\tau}_{ij} = \mu \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \]  

(4)

This simulation employed the VOF (volume of fluid) method to depict the impact of a free surface on the computational model. The VOF technique was appropriate for modeling multiple distinct flow phases. The water and air fluid functions rely on the volume fraction attribute in Equation (5), where \( V \) represents the designated calculation area, \( V_1 \) corresponds to the volume of fluid 1, and \( V_2 \) represents the volume of fluid 2. Each grid is assumed to have a volume fraction value of either 1 or 0 to differentiate between the air and water fluids [84]:

\[ \alpha(\hat{x}, t) = \begin{cases} 
A_1, \hat{x} \in V_1 \\
A_2, \hat{x} \in V_2 
\end{cases} \]  

(5)

The continuity equations for volume fraction describe the conservation of mass for each component in a mixture. For two-phase flow, the continuity equation for each phase can be written as in Equations (6) and (8). Where: \( \alpha_1 \) and \( \alpha_2 \) are the volume fractions of the first and second fluids, respectively; \( \rho_1 \) and \( \rho_2 \) are the densities of the first and second fluids, respectively; \( \bar{U} \) is the boat’s speed; and \( \nabla \) is the divergence operator. The VOF \( C_{ijk} \) function is
an integral of $\alpha(\vec{x}, t)$ on each grid cell in each volume cell as described in Equation (8) which became Equation (9). Where: $\mathcal{C} = 1$ for the grid that defined the fluid; $\mathcal{C} = 0$, indicating that the grid comprised a mixture of water and air phases under the air phase and when $0 < \mathcal{C} < 1$:

$$\frac{\partial (\alpha_1 \rho_1)}{\partial t} + \vec{U} \cdot \nabla (\alpha_1 \rho_1) = 0$$  
(6)

$$\frac{\partial (\alpha_2 \rho_2)}{\partial t} + \vec{U} \cdot \nabla (\alpha_2 \rho_2) = 0$$  
(7)

$$\mathcal{C}_{ijk} = \frac{1}{\Delta V_{ijk}} \int \alpha(\vec{x}, t) dV$$  
(8)

$$\frac{\partial \mathcal{C}}{\partial t} + \vec{U} \cdot \nabla \mathcal{C} = 0$$  
(9)

The realizable $k$--$\varepsilon$ (epsilon) turbulence model with a standard wall function, which relates the Reynolds stress to the average flow property, was used to approximate the system of Equations (2) and (3). This turbulence model has two equations representing the turbulence kinetic energy transport $k$ and the turbulence dissipation rate $\varepsilon$ [85].

A time-step determination needs to be considered in simulations with an unsteady flow. The time step is defined as the period interval for each iteration calculation. The time step value is related to the Courant-Frederich-Lewis (CFL) number to ensure the stability of numerical calculations. Determining the time-step value in this CFD simulation follows the procedure from ITTC [86], where it is a function of the length and speed of the craft (see Equation (10)). The faster the boat’s speed, the smaller the time-step value used. Determining the value of this time step is also very important to capture dynamic phenomena because this simulation uses a dynamic mesh, where the hull model can trim and heave dynamically:

$$\Delta t = 0.005-0.01 \frac{L}{V_s}$$  
(10)

2.3.2 Domain and boundary condition settings

The creation of a computing domain is described in this subsection. The domain settings are domain size and boundary conditions. The size of the domain formed is depicted in Fig. 4, where the size is based on the length of the simulated boat model and is based on work done by Lotfi et al. [74]. The setting of the boundary conditions is described in Fig. 4. The simulation uses a multi-fluid in the presence of a water surface as a free surface. So that the inlet and outlet are arranged so that there are two fluids, namely air, and water.

The type of boundary conditions must be set correctly for the numerical simulation to run appropriately. The inlet, top, bottom, and side boundary conditions are conditioned as velocity inlets. The inlet section is used as a passage for water and air to enter at a predetermined speed to simulate a speeding boat. In the outlet section, boundary conditions are defined as pressure outlets with a field function of hydrostatic pressure. In addition, a symmetry boundary condition is applied as a Symmetry Plane in the middle plane and a no-slip condition on the hull model surface. This simulation uses a multiphase model of water and air to predict wave patterns on the free surface. The $k$--$\varepsilon$ model is used to consider turbulent flow effects. The simulation is completed on only half of the hull to reduce computational costs.
2.3.3 Mesh generation

The settings for preparing the mesh are described in this subsection. Mesh refinement focuses on objects and water surfaces to shorten computation time while getting accurate results. The objects were refined on the hull and the area around the hull. The refinement is carried out using the anisotropic mesh method, which aims to focus the mesh on the $x$, $y$, or $z$-axis ordinate. The refinement results obtained three variations in the number of elements, namely coarse, medium, and fine arrangement, each producing a number of elements of 1.3 M, 2.9 M, and 5.9 M, respectively. Fig. 5 shows the arrangement of the medium arrangement mesh with many elements of 2.9 M.

The dynamic mesh was used to simulate boat motion using dynamic fluid-body interaction (DFBI) by combining two degrees of freedom: free heave and trim. The study utilized a rigid body's motion and an overset grid system to depict the movement of the boat in
the fluid domain. The translational (sinkage) and rotational (trim) motions at the center of mass of the boat model are simulated based on Equations (11) and (12), respectively [51,76]. The equations involve several variables: $M$, which represents the net moment acting on the boat model for $y$-axis rotation; $I$, the moment of inertia for $y$-axis rotation; $\omega$, the angular velocity of the boat for $y$-axis rotation; $m$, the mass of the boat; $F$, the net force acting on the surface of the boat for $z$-axis translation; and $U$, the speed of the boat. The forces and moments acting on the boat were obtained from fluid pressure and shear forces on the surface of the boat:

\[
l \frac{d\omega}{dt} = \sum M
\]

\[
m \frac{dU}{dt} = \sum F
\]

The dynamic mesh simulation method was used for these simulations, namely overset mesh method. The Overset mesh method has been proven to be more efficient in handling complex movements. This technique enables the use of multiple overlapping meshes to handle the complex motion of planing hull boats. Moreover, the Overset mesh method can provide higher accuracy in simulations since the moving mesh can adjust to the motion of the planing craft [87], minimizing numerical errors. Some examples of works that use the overset mesh method in planing craft simulation as done by De Marco et al. [73], Di Caterino et al. [75], Hosseini et al. [87], Samuel et al. [51] and more. The simulation results using the dynamic mesh method are closer to the results from the experiment than the static mesh method [88]. In the overset mesh method, the domain modeling is divided into two parts: the background geometry part as a donor and the overset geometry part as an acceptor. It should be noted that in the overset mesh method, the mesh density of the donor and acceptor must be of the same dimension, or there is no significant difference. Differences in dimensions that are too large can cause data transfer errors, so the simulation cannot be continued [89].

Determining the $y^+$ is very important in CFD simulations using the turbulence model [90]. For turbulence models using the wall function method, the recommended $y^+$ is above 30 to avoid the buffer area, and it must be right in the log law area of the boundary layer structure. In this study, $y^+$ was used with a value of 50. The $y^+$ was between 45 – 60 to get accurate simulation results [91]. According to ITTC, the calculation of the $y^+$ value is obtained through Equation (13) [86], where: $y$ is the thickness of the first layer that must be adjusted in making the mesh arrangement, $L$ is the length of the hull, $Re$ is the value of the Reynolds number, and $C_f$ is the skin friction coefficient of the object that can use Empirical formula from ITTC’57 [92]. Therefore, to get the desired $y^+$ value, the first mesh distance to the wall must be adjusted:

\[
\frac{y}{L} = \frac{y^+}{Re \sqrt{C_f/2}}
\]

The way to set this distance is to apply an inflation layer mesh. Fig. 5 shows the implementation of the inflation layer mesh in this numerical model.

3. Result and discussion

3.1 Verification study

To evaluate the potential inaccuracies in both space and time of the simulations, convergence studies were conducted. In order to estimate the numerical uncertainties, the Grid Convergence Index (GCI) method based on Generalized Richardson Extrapolation [93]. The GCI method involves calculating the ratio of the error between two different grid resolutions,
which provides valuable information about the rate at which the error decreases as the resolution increases. According to Celik et al. [94], the sequence of calculation for this method is as follows:

\[ p_a = \frac{1}{\ln(r_{21})} \ln \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) + q(p_a) \]  

(14)

\[ q(p_a) = \ln \left( \frac{r_{21}^{p_a} - s}{r_{32}^{p_a} - s} \right) \]  

(15)

\[ s = \text{sign} \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \]  

(16)

where, \( r_{21} \) and \( r_{32} \) are refinement factors given by \( r_{21} = \sqrt[3]{N_1/N_2} \) for a spatial convergence study of a 3D model. convergence study. \( N_i \) are the cell number. \( \varepsilon_{32} = \phi_3 - \phi_2 \), \( \varepsilon_{21} = \phi_2 - \phi_1 \), and \( \phi_i \) denotes the simulation result, i.e., \( R_T/\Delta \) in this study.

The extrapolated value is calculated by:

\[ \phi_{\text{ext}}^{21} = \frac{r_{21}^{p_a} \phi_1 - \phi_2}{r_{21}^{p_a} - 1} \]  

(17)

The approximate relative error, \( e_a^{21} \), is obtained by:

\[ e_a^{21} = \frac{\phi_1 - \phi_2}{\phi_1} \]  

(18)

The extrapolated relative error, \( e_{\text{ext}}^{21} \), is obtained by:

\[ e_{\text{ext}}^{21} = \frac{\phi_{\text{ext}}^{21} - \phi_1}{\phi_{\text{ext}}^{21}} \]  

(19)

Finally, the fine-grid convergence index is found by

\[ GC_{\text{fine}}^{21} = \frac{1.25e_a^{21}}{r_{21}^{p_a} - 1} \]  

(20)

The result of numerical uncertainty calculation was obtained as 1.25%, with the detailed calculation shown in Table 3.

Table 3 Parameters used for the calculation of the discretization error for the spatial convergence study, key variable: \( R_T/\Delta \) of C2 model simulation at \( Fr = 4.81 \) (8.13 m/s)

| \( N_1 \) (Coarse) | 1.3 \times 10^6 | \( \varepsilon_{32} \) | -0.017 |
| \( N_2 \) (Medium) | 2.9 \times 10^6 | \( \varepsilon_{21} \) | -0.007 |
| \( N_3 \) (Fine) | 5.9 \times 10^6 | \( s \) | 1 |
| \( r_{21} \) | 1.329 | \( e_a^{21} \) | 0.003 |
| \( r_{32} \) | 1.260 | \( q \) | 0.237 |
| \( \phi_1 \) | 2.117 | \( p_a \) | 3.948 |
| \( \phi_2 \) | 2.110 | \( \phi_{\text{ext}}^{21} \) | 2.138 |
| \( \phi_3 \) | 2.094 | \( e_{\text{ext}}^{21} \) | 0.99% |
| \( GC_{\text{fine}}^{21} \) | 1.25% |  |  |
3.2 Validation Study

A validation test was carried out by comparing the CFD simulation results with the results of experimental testing conducted by Taunton et al. [67]. Errors are calculated using Equation (21) to assess the precision of current CFD results. The term $\psi$ in this context refers to any of the three parameters that are being validated, namely $R_T/\Delta$ for the resistance, $\theta_V$ for dynamic trim, and $Z_V/\gamma^{(1/3)}$ for dynamic sinkage. Experimental and numerical data are indicated by the subscripts EXP and CFD, respectively. Equation (22) calculates RMSE (Root Mean Square Error), which measures the errors of the samples based on the speed variations:

$$E_\psi\% = \frac{\psi_{\text{CFD}} - \psi_{\text{EXP}}}{\psi_{\text{EXP}}} \times 100\%$$  \hspace{1cm} (21)

$$\text{RSME} = \sqrt{\frac{\sum_{i=1}^{N} E_i^2}{N}}$$  \hspace{1cm} (22)

Fig. 6 shows a sample of the convergence of the results. The results were generated using the C2 model and correspond to a Froude number of 4.81 (flow velocity of 8.13 m/s). The time histories plotted in the figure indicate that all data converges after 2 seconds, and the simulation is stopped at 5 seconds. As such, the results obtained were the average values of the real-time data collected between $t = 2$ seconds and $t = 5$ seconds.

![Graph showing convergence of results](image)

**Fig. 6** A sample of convergence of the resistance, trim, and sinkage results, showing how a CFD model calculates the equilibrium condition of the C2 model and correspond to $Fr = 4.81$ (8.13 m/s)
Fig. 7 Comparison of the resistance results obtained from the present CFD simulation with the experimental results from Taunton et al. [67]

Fig. 8 Comparison of the dynamic trim results obtained from the present CFD simulation with the experimental results from Taunton et al. [67]

Fig. 9 Comparison of the dynamic sinkage results obtained from the present CFD simulation with the experimental results from Taunton et al. [67]
The validation findings showed that, although not flawless, the outcomes are acceptable. Graphical comparisons of the CFD and experimental results are presented in Fig. 7, Fig. 8, and Fig. 9, with the corresponding RMSE values provided in Table 4. Fig. 7 reveals a slight difference in resistance results between the CFD simulation and the experiment, with an RMSE value of 7.46%. In Fig. 8, the dynamic trim results display a quite different, with an RMSE value of 20.33%. The dynamic sinkage results in Fig. 9 are also slightly different, but with an RMSE value of 11.8%. It should be noted that the validation of dynamic trim and sinkage results is challenging, as description by Lotfi et al. [74], Dashtimanesh et al. [80], and Hosseini et al. [87].

![Graph showing wetted surface area comparison]

**Fig. 10** Comparison of the wetted surface area results obtained from the present CFD simulation with the experimental results from Taunton et al. [67]

The validation results revealed a substantial error due to the assumption of the moment of inertia value input into the model. The moment of inertia value is based on the recommendation from ITTC [95]. From the validation analysis of the trim, it appears that the trim value in the CFD simulation is currently too high, and the lift (heave) value is lower than the experimental results. This indicates a higher value of wetted surface area than the experimental results, as seen in Fig. 10. Therefore, it is estimated that the frictional resistance value is too high in the experimental results. Nevertheless, the authors decided to continue the study and focus on the effects of variations in the double step position in the research results.
3.3 Uncertainty Analysis

To calculate the uncertainty value ($U_V$) in this analysis, the method from ITTC [96] was used. The $U_V$ value was calculated using the following Equation:

$$U_V^2 = U_D^2 + U_{SN}^2 \quad (23)$$

where $U_D$ represents the uncertainty of the experimental results, and $U_{SN}$ represents the numerical uncertainty. Based on Taunton et al. [67], the experimental uncertainty is ~10%. According to the section of Verification study, the value of $U_{SN}$ is 1.25%. Thus, using Equation (23), the uncertainty value for this analysis was $U_V = 10.08\%$. According to ITTC [96], validation is considered successful if $|E| < U_V$. The value of $|E|$ is the comparison value calculated in the section of Validation study, which was found to be 7.46%. Therefore, validation is achieved as $|E| < U_V$.

3.4 Total resistance results

Analysis of the CFD simulation for resistance results is described in this subsection. The graph in Fig. 11 illustrates the overall resistance results of all model modifications. Analysis of the effect of the position of the first step and the second step on the resistance results is described in Fig. 12.

![Fig. 11 Resistance results for all model variations](image)

Fig. 11 describes the resistance results for all model variations based on the $Fr_T$ value. All resistance results showed that the higher the $Fr_T$ or boat speed, the higher the resistance. Then, all changes in step position, whether the first or second step, showed a lower resistance than the original model, namely model C2. Model C2.4 had the lowest resistance value for speeds at $Fr_T$ below 4.7s, while model C2.2 had the lowest resistance at $Fr_T$ above 4.7s.

Several model variations showed different trends of the changes in the resistance based on the $Fr_T$ value. First, the comparison resistance results of models C2, C2.1, and C2.2, where the three models had a different first step position, and the second step was the same. It can be seen that C2.2 had lower boat resistance than C2.1 and C2 models. The trend of increasing resistance based on $Fr_T$ showed that models C2.1 and C2.2 were lower slopes than C2. These results indicated that changing the position of the first step will effectively reduce resistance at high speeds ($Fr_T$). Second, the comparison results of the models C2, C2.3, and C2.4, where
they had a different second step position, but the first step was the same. It can be seen that C2.3 had lower boat resistance than C2.4 and C2 models at $F_{r_\gamma}$ above 4.8s, but at $F_{r_\gamma}$ below 4.8s, the C2.4 was the lowest. The trend of increasing resistance based on $F_{r_\gamma}$ showed that models C2 and C2.3 were similar, while the trend from C2.4 was slightly different. These results indicated that changing the position of the second step will effectively reduce resistance at low speeds ($F_{r_\gamma}$).

Fig. 12 The graphs plot the resistance against the position of the hull steps: (a) first step only, (b) second step only, (c) the multiplication of the two steps, and (d) the difference between the two steps times the multiplication of the two steps.

Fig. 12 explains how the two steps' positions affected the outcomes of boat resistance. Fig. 12 (a) showed that the boat's resistance increased with the first step's forwardness. In contrast, if the position of the second step was getting more forward, the resistance became a bit smaller, as shown in Fig. 12 (b). The graph in Fig. 12 (c) showed that the effect of multiplying the position of the two steps was inconsistent based on the speed value, where at a lower speed ($F_{r_\gamma}$), the higher the multiplication of the two steps, the resistance decreases. Still, at high speed, the opposite occurred. This result corresponded to the results expressed by Vitiello et al. [70] and Najafi et al. [71], where shifting the step further forward will reduce the drag. Therefore, the authors tried to add the effect of the distance between the two steps, plotted in Fig. 12 (d). Based on the plot, the higher the value of the distance of the two steps multiplied by the multiplication of the two steps, the resistance was higher.
The author attempted to find the relationship between the position of the two steps, distance between the steps, and boat speed, which was then obtained and explained in Fig. 13. The curve shows the relationship between the three variations made. The distance between the two steps is represented by $x_1 - x_2$, and the position of the both steps is represented by the multiplication of $x_1$ and $x_2$. All parameters were made into nondimensional form. The regression results showed an acceptable value, with a determination coefficient of $R^2 = 0.5779$ and a slope of 0.059.

$$y = 0.1385e^{0.2599x}$$
$$R^2 = 0.5675$$

Fig. 13 Regression analysis of the relationship between the distance and the location of the double step, as well as the speed parameters to the total resistance result

3.5 Frictional and residual resistances results

This chapter discusses the resistance results in more detail, down to the components of resistance, such as frictional resistance, residual resistance, pressure resistance, and wave-making resistance. The difference between frictional resistance and the ratio of frictional resistance to residual resistance is shown in Fig. 14. Fig. 15 to Fig. 17 describe the distribution of the wall shear stress X contour on all models, taken at a velocity of 10.13 m/s ($Fr_v = 6.01$). Fig. 18 and Fig. 19 show the local value of wall shear stress X on the centerline section and $1/4B$ section for each model, taken at a velocity of 10.13 m/s ($Fr_v = 6.01$). Fig. 20 visualizes the flow separation that causes air to be trapped behind the step, decreasing frictional resistance and even reversing its direction (pushing the boat forward).

Based on the analysis results in Fig. 14 (a) and (b), it was found that the position of the steps and the distance between the two steps greatly affected the frictional resistance results. Model C2.2 produced the smallest frictional resistance among the other models at high speeds. As described in Fig. 12 (c) and (d), Model C2.2 had the furthest back step position compared to the other models. The distance between the first and second steps in Model C2.2 was also the closest compared to the other models. This was further reinforced by the trim value results of Model C2.2 being the highest, which will be explained in the subsection 3.5 Dynamic trim results. The dynamic sinkage value of Model C2.2 was also the highest, based on the dynamic sinkage results explained in the subsection 3.6 Dynamic sinkage result. This made the wetted length of C2.2 suspected to be shorter than the other models. This phenomenon was reinforced in the plot in Fig. 18 and Fig. 19, where it can be seen that the local wall shear stress value of Model C2.2 was the lowest at the front of both steps and the back of the steps. This was different
from Model C2 and C2.4, where they produced the high frictional resistance for the area in front of both steps and at the back of the steps. This phenomenon occurred in both the centerline and ¼ beam of the hull.

Fig. 14 Comparison of the results of frictional resistance of all models: (a) frictional resistance to displacement, and (b) frictional resistance to residual resistance

Fig. 15 Comparison of wall shear stress distribution for C2 and C2.2 models at 10.13 m/s ($Fr_T = 6.01$)

Fig. 16 Comparison of wall shear stress distribution for C2.2 and C2.4 models at 10.13 m/s ($Fr_T = 6.01$)

Fig. 17 Comparison of wall shear stress distribution for C2.1 and C2.3 models at 10.13 m/s ($Fr_T = 6.01$)
Analysis of the Double Steps Position
Effect on Planing Hull Performances

Andi Trimulyono, Muhammad Luqman Hakim,
Chairizal Ardhah, Syaiful Tambah Putra Ahmad,
Tuswan Tuswan, Ari Wibawa Budi Santosa

---

**Fig. 18** Comparison of local wall shear $X$ values of each model on the centerline

**Fig. 19** Comparison of local wall shear $X$ values of each model on the $\frac{1}{4} B$

**Fig. 20** Comparison of separation length results for all variations at 10.13 m/s ($Fr_Y = 6.01$)
Fig. 21 Volume fraction of water in the cross-section of Model C2 and C2.2 that taken before the first step, between the two steps, and after the second step ($Fr_\gamma = 6.01$)

Fr$_\gamma$ = 3.72±1.3%

Fr$_\gamma$ = 6.02±2.1%

Fig. 22 Comparison of volume fraction of water to determine the wetted surface area
Fig. 20 compares the separation length of all models from the plot of volume fraction results. It is then possible to see how an air cavity was created once the flow had passed through the steps. The result from model C2.2 shows that the air cavity appeared slightly bluer (higher) compared to model C2, indicating that the air cavity of model C2.2 is higher than that in model C2. This result was also evidenced by the negative pressure results shown in Fig. 24. With the negative pressure, the air cavities were created and will reduce the wall shear stress value because of the difference in the density of water and air. The wall shear stress contour was compared in Fig. 15, where model C2.2 had a slightly wider yellow color pattern (zero shear value) than model C2.

Fig. 21 shows the contour of the volume fraction of water in the cross-section of Model C2 and C2.2. The cross-sections are taken before the first step, between the two steps, and after the second step. It can be seen that in the cross-section before the first step, \( x/L = 0.402 \), both Model C2 and Model C2.2 do not have any air cavity, indicated by the dark blue contour on the bottom. Then, in the cross-section between the two steps, \( x/L = 0.247 \), Model C2.2 already shows an air cavity near the chine, but it is not yet visible in Model C2. Only in the cross-section after the second step, \( x/L = 0.092 \), both of them have air cavities on their bottoms, but Model C2.2 has more air cavities compared to Model C2. Therefore, due to the difference in the water cavity, the wetted surface area for each variation of step position became different. The differences in the wetted area contours are explained in Figure 22.

In Fig. 22, the differences in contour for the volume fraction of water values are explained, which indicate the differences in wetted surface area. The wetted surface area is also calculated including the spray that occurs. At low velocity, \( F_r = 3.72\pm1.3\% \), it can be observed that Model C2.4 has the smallest wetted surface area. Meanwhile, at high velocity, \( F_r = 6.02\pm2.1\% \), it can be seen that Model C2.2 has a smaller wetted surface area. It can also be observed that Model C2 has the highest wetted surface area both at low and high velocities. Therefore, it can be concluded that the position of these two steps greatly influences the wetted surface area values at each velocity.

In Fig. 23, a comparison of the wave-making elevation contours of the C2 and C2.2 models is presented. It can be observed that the angle of the Kelvin wave generated from the C2 model was slightly larger than that of the C2.2 model. This result indicated that the wave resistance of the C2 model was higher than that of the C2.2 model. High wave resistance occurred due to significant pressure contour differences on the boat’s hull. The pronounced pressure contours mean that there was a sudden change in pressure from very high to very low, which caused a high wave elevation. This extreme pressure difference led to the formation of high wave-making elevation contours. This phenomenon can be observed in Fig. 24 as the pressure coefficient distribution in the total and static pressure. Equation (24) describes the total pressure coefficient and Equation (25) describes the dynamic pressure coefficient, where: \( p \) is local pressure; \( \rho \) is density of fluids that consist of water and air; \( V \) is velocity of the boat. In Fig. 25 and Fig. 26, the local pressure curve showed that the C2 model had a pattern that went up and down, and then up again. This was different from the C2.2 model, where when the pressure value increased and then decreased, the decrease was not as extreme as in the C2 model. The distance between the steps also resulted in a significant difference in pressure contours, as exemplified by the C2 model, which had the longest distance between steps compared to the other models. Furthermore, the differences in wetted length, wetted surface area, and displacement due to different lift forces can also cause differences in wave-making resistance:

\[
c_{P_{(total)}} = \frac{p_{(total)}}{0.5\rho V^2}
\]  

(24)

59
\[ c_{p \, \text{dynamic}} = \frac{p_{\text{total}} - p_{\text{static}}}{0.5 \rho_i V^2} \]  

Fig. 23 Comparison of wave-making elevation distribution for C2 and C2.2 models at 10.13 m/s ($Fr_\gamma = 6.01$)

Fig. 24 Comparison of pressure coefficients ($c_p$) distribution: (a) total pressure and (b) dynamic pressure of all models at 10.13 m/s ($Fr_\gamma = 6.01$)
The performance of planing vessels, both conventional and with steps, can be analyzed based on the wake profile created at the stern of the hull [69]. The wake profile is illustrated in Fig. 27. By using several models and speeds, wake profile plots were obtained as shown in Fig. 28. On the horizontal axis, denoted as $x/B$, where $x$ is the distance compared to the hull beam as indicated in Fig. 27. On the vertical axis, denoted as $H_x/H_s$, where $H_x$ is the wake profile height at distance $x$ compared to the step height as shown in Fig. 27. From the results of these
plots, it can be observed that as the Froude number \( Fr \) or vessel speed increases, the wake profile becomes lower. This indicates that resistance increases with lower wake profile height, and vice versa for lower resistance where the wake profile height becomes higher.

From the wake profile plots at the same speed, it was observed that model C2.2 consistently has the highest wake profile, while model C2.4 consistently has the lowest wake profile. Further analysis on this inconsistency with the resistance results needs to be investigated in future research. The resistance results show that at \( Fr \) = 3.72, the order of resistance values is \( C2 > C2.2 > C2.4 \), and at \( Fr \) = 6.01, the order of resistance is \( C2 > C2.4 > C2.2 \).

![Fig. 28](image) The results of the comparison of the height of the wake profile of several models and the speed.

3.6 Dynamic trim results

This subsection describes the analysis of the dynamic trim results from CFD simulation. Overall, the graph of Fig. 29 organized the outcomes of the dynamic trim of all model modifications. Fig. 30 and Fig. 31 provides an analysis of how the positions of the first and second steps affect the results of the dynamic trim.

The outcomes of the dynamic trim for all model variations on the \( Fr \) value are shown in Fig. 29. All dynamic trim results demonstrated that the trim decreased as boat speed (or \( Fr \)) increased. Increased trim occurred when the first step was shifted backward, as shown in models C2.2 and C2.1 compared to model C2. According to models C2.4 and C2.3, moving the second step forward results in a lower trim value.

![Fig. 29](image) Dynamic trim results for all model variations
Fig. 30 The graphs plot the dynamic trim against the position of the hull steps: (a) first step only, (b) second step only, (c) the multiplication of the two steps, and (d) the difference between the two steps times the multiplication of the two steps.

Fig. 30 explains how the position of the two steps affected the outcomes of the dynamic trim. The dynamic trim decreased as the first step was moved farther ahead, as seen in Fig. 30 (a), but the dynamic trim decreases a bit as the second step was moved farther forward, as seen in Fig. 30 (b). The graph in Fig. 30 (c) demonstrated that the lower dynamic trim was produced due to the higher multiplying positions of the two steps. It will also be the same in the graph showing how the dynamic trim depicted in Fig. 30 (d) was affected by the distance between two steps multiplied by the multiplication of the two steps. This result was in line with what was found by Najafi et al. [71,72], where shifting the step hull position closer to the transom can increase the dynamic trim value. The trim values of these models can be observed from the distribution of wall pressure in Fig. 24 to Fig. 26. It can be seen that models C2.2 and C2.1 had a pressure distribution area that appeared to be more towards the rear than the other models. This event was the cause of the higher trim values of Model C2.2 and Model C2.1.

The author also attempted to find the relationship between the position of the two steps, distance between the steps, and boat speed parameters to the dynamic trim results, which was then obtained and explained in Fig. 31. The curve shows the relationship of the three variations that had been made. The distance between the two steps was represented by \( x_1 - x_2 \), and the position of the both steps was represented by the multiplication of \( x_1 \) and \( x_2 \). All parameters were made into nondimensional form. The regression results showed a satisfactory value, with a determination coefficient of \( R^2 = 0.9371 \) and a slope of -0.7032.
3.7 Dynamic sinkage results

The analysis of the findings of dynamic sinkage is covered in this subsection. The outcomes of the dynamic sinkage of all model variations were organized in the graph of Fig. 32. A breakdown of how the first and second step positions impact the dynamic sinkage outcomes is shown in Fig. 33 and Fig. 34.

Fig. 32 shows the results of the dynamic sinkage for all model variations on the $F_{r_Y}$ value. The sinkage increased as the boat's speed (or $F_{r_Y}$) increased, as shown by all dynamic sinkage data. As demonstrated in models C2.2 and C2.1 compared to model C2, increased sinkage happened when the first step was moved backward. When the second step was shifted forward, as in models C2.4 and C2.3, it produced dynamic sinkage values that were relatively unchanged from model C2.
Fig. 33 illustrates how the positioning of the two steps impacted the outcomes of the dynamic sinkage. Fig. 33 (a) shows how the dynamic sinkage decrease as the first step was shifted forward. Meanwhile, Fig. 33 (b) shows how the effect of shifting the second step forward did not really have an impact on the dynamic sinkage value. The graph in Fig. 33 (c) illustrates that the higher multiplication of the two-step positions resulted in lower dynamic sinkage. The graph illustrating the impact of the distance between two steps multiplied by the multiplication of the two steps will decrease the dynamic sinkage results, as shown in Fig. 33 (d). This outcome is consistent with what Najafi et al. [71,72] discovered: moving the stepped hull closer to the transom can raise the dynamic sinkage value.

The author also tried to establish a correlation between three parameters, namely the position and distance between two steps and the boat's speed, with the dynamic sinkage outcomes presented in Fig. 34. The plot illustrates the relationship between these three varied variables. The distance between the steps was denoted by $x_1 - x_2$, while the position of both steps was obtained by multiplying $x_1$ and $x_2$. To standardize the values, all parameters were transformed into nondimensional form. The regression analysis yielded a weak correlation, with a determination coefficient of $R^2 = 0.0109$ and a slope of 0.0109.
The dynamic trim was reduced when moving both the first and second steps further forward. The dynamic trim decreased as the distance between the first and second steps increased. The modified position of the steps had a relationship to the dynamic sinkage result. The relationship of these parameters to the results can be seen in Fig. 13, Fig. 31, and Fig. 34.

The position of the two steps clearly affected the boat's performance. The analysis results of the position of the steps on the resistance showed that shifting the first step further forward caused the resistance to increase. Meanwhile, moving the second step further caused the resistance to decrease slightly. Moving both steps (the first and second) further forward caused resistance to decrease at low speeds but increase at high speeds. The longer distance between the first and second steps caused the resistance to increase. According to the analysis of how the position of the steps affected the dynamic trim, moving the first step forward reduced the dynamic trim. However, the dynamic trim got a bit higher as the second step moved forward. The dynamic trim was reduced when moving both the first and second steps further forward. The dynamic trim decreased as the distance between the first and second steps became longer.

Because there are several weaknesses in the results of this study, it is necessary to improve the research to strengthen the findings. The first weakness lies in the validation test, which...
yielded unsatisfactory experimental results. The next weakness is the absence of a comprehensive measurement of the wake profile of the two steps, similar to what was conducted by Savitsky and Morabito [69]. To gain a complete global perspective, future investigations should consider more sophisticated modeling techniques, conduct extensive parametric analyses, and optimize studies with additional parameters. More parameters include step height ($H_s$), $L/B$ ratio, deadrise angle ($\beta$), more velocities or Froude number, and others.

ACKNOWLEDGMENTS

This research was supported by the Hydrodynamics laboratory of the Department of Naval Architecture, Faculty of Engineering, Universitas Diponegoro.

REFERENCES


https://doi.org/10.46754/jssm.2021.04.012


ITTC, 2014. ITTC – Recommended Procedures and Guidelines - Practical guidelines for ship CFD applications. 7.5-03-02 (Revision 01).


ITTC, 2017. ITTC – Recommended Procedures and Guidelines - Resistance and Propulsion Test and Performance Prediction with Skin Frictional Drag Reduction Techniques. 7.5-02-02-03.

Richardson, L.F., 1911. The approximate arithmetical solution by finite differences of physical problems


[95] ITTC, 2010. The Specialist Committee on High Speed Craft Final Report and Recommendations to the 26th ITTC. Rio de janeiro, Brazil.


Submitted: 18.01.2023. Andi Trimulyono
Muhammad Luqman Hakim*, mluqmanhak@lecturer.undip.ac.id

Accepted: 17.07.2023. Chairizal Ardhan
Syafiiul Tambah Putra Ahmad
Tuswan Tuswan
Ari Wibawa Budi Santosa

Department of Naval Architecture, Faculty of Engineering, Universitas Diponegoro, Semarang, Indonesia 50275