Proceedings of 8th International Conference on Mechanical, Industrial and Energy Engineering 2024
02-04 January, 2025, Khulna, BANGLADESH
SciEn Conference Series: Engineering Vol. 3, 2025, pp 43-49 https://doi.org/10.38032/scse.2025.3.12

Comparative Study on Hydrodynamic Performance of Planing Hull based on Empirical and Numerical Methods

Syed Sadik Siddique[1,*], *Tausif Razzak*[1], *Md. Mashiur Rahaman*[2], *Laboni Afroz*[2]

[1,*]Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh
[1]Offshore and Marine Engineering Limited, Shahid Syed Nazrul Islam Sharani, Dhaka-1000, Bangladesh
[2]Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh

ABSTRACT

Predicting the resistance of a planing hull is a challenging task due to its complex hydrodynamic behavior. The present study evaluates the hydrodynamic performance of a high speed patrol boat using Savitsky's empirical method and an open-source RANS-based CFD (Computational Fluid Dynamics) solver, OpenFOAM 11. The key performance parameters include resistance, trim, and sinkage. The study begins with the validation of results obtained from CFD and empirical method against the experimental result of a deep V-type planing hull, known as the GPPH (Generic Prismatic Planing Hull), towed in calm water. The numerical results are also compared with the well-known Savitsky's 1964 method. The validation reveals that the CFD model, with an error of 0.95%, is more reliable than the empirical method, which has an error of 2.79%, as the CFD approach captures the dynamic behavior of the planing hull more effectively. A systematic verification study is performed to assess the numerical uncertainty based on grid size. Following the validation study, the same CFD model and empirical method were applied to a similar deep V-type patrol boat sailing in the waterways of Bangladesh. At higher speeds, a lower deviation between the empirical and numerical methods is observed. The dynamic pressure distribution on the hull and wave pattern were less affected due to a reduction in wave making resistance at a higher Froude number. Finally, the study extrapolates the resistance to the full scale, and the power requirements at five different speeds are obtained using both empirical and numerical methods. Savitsky's method and CFD method provide reliable resistance and power prediction for planing hull, with a deviation of 5% at higher Froude numbers.

Keywords: CFD, Savitsky's method, RANS, GPPH, OpenFOAM

1. Introduction

Patrol boats mainly fall under the category of planing hull. Determining resistance, effective power, and the running trim angle of the planing hull are, therefore, important in the early design stage. Numerous researchers have investigated the hydrodynamics of planing hull in an effort to predict these parameters. While experimental studies provide precise results, conducting model tests for small vessels like patrol boats is often impractical due to cost and scale constraints. Generally, empirical and semi-empirical methods, such as Savitsky's 1964 formula, have been practiced by the naval architects. However, the empirical methods have some limitations in capturing the full complexity of planing hull hydrodynamics. With the continual improvement of computational power and development of RANS-based CFD code in the previous few decades, numerical results show enhanced accuracy when comparing with experimental results. For displacement-type hulls, RANS-based CFD methods typically achieve less than 5% error compared to experimental results. However, it is difficult to accurately predict the resistance in the planing hull due to the higher trim angle and high mesh deformation that often leads the solver instability. Besides, it requires higher mesh resolution and computation time.

Savitsky [1] made a great effort on the hydrodynamics of planing hull and developed an empirical formula based on the regression of prismatic series hull model test and calculated hydrodynamic drag force on hull. Brizzolara and Serra (2007) [2] compared RANSE-based CFD prediction of drag, lift, and trim moments of planing hulls with a 20° deadrise angle against experimental results and semi-empirical models. The findings indicate CFD method has advantages in capturing complex flow phenomena like spray root regions and wetted lengths. Clement and Blount [3] conducted a systematic experiment on Series 62 hulls, which became a benchmark of planing hull resistance estimation. Chen et at. (2010) [4] investigated the hydrodynamics of planing crafts, considering detailed insights into wave pattern and pressure field at various speeds through numerical simulations. Brizzolara and Villa (2010) [5] performed a numerical simulation of a planing hull using the Reynolds-Averaged Navier-Stokes (RANS) method with 2 DOF (heave and pitch free motion) and compared the findings with the experimental measurements, which shows good agreement with experimental result. Lotfi et al. (2015) [6] predicted hydrodynamics of a planing hull using commercial ANSYS-CFX and Svahn (2009), but their results showed significant deviation from experimental

*Corresponding Author Email Address: syed.sadik.siddique@gmail.com

results. Mancini et al. (2017) [7] performed computational fluid dynamic study of Naples Systematic Series using two commercial software, Fine Marine and Star CCM +. Their results show that the Fine marine provide less error in case of higher speed while Star CCM + provides good agreement with experimental results at low speed. Avci and Barlas (2018) [8] predicted the resistance of planing crafts by substituting the air phase with the water phase under the hull to overcome the numerical ventilation problem (NVP). This improves the accuracy of resistance prediction with 2.86% error from experimental results for Froude number greater than 0.50. According to Fillippo and Sfravara (2018) [9], URANSe-based CFD simulations, are implemented in commercial software (Ansys Fluent), provide useful parameters for measuring the wetted surface area of the planing hull, which is hardly evaluated experimentally.

The numerical studies on patrol boats operating at high speed regions is still limited due to complexity like wave breaking. Most of the researchers rely on commercial or in-house CFD codes while predicting the hydrodynamic performances. The present study aims to implement open source solver OpenFOAM and Savitsky's 1964 method to evaluate the hydrodynamic performance parameter of planing hull, especially resistance, trim and sinkage at high speed regions. Initially, findings from both methods are validated against experimental data of the Generic Prismatic Planing Hull (GPPH). Following validation, the methods are applied to a high speed patrol boat, and a final comparison is made, concluding with power predictions. The numerical results show good agreement with experimental results. Although the deviation of empirical results is larger, Savitsky's method aligns well with CFD results in the high speed range. However, CFD analysis should be performed to get a complete scenario of the hydrodynamics of the planing hull.

2.Methodology

2.1 Geometry

A deep V-type patrol boat is selected for the present study. The patrol boat is designed for Coast Guard of Bangladesh. Before the beginning of the present study a similar type of prismatic hull named GPPH (Generic Prismatic Planing Hull) has been taken to validate the CFD code with experimental results. The experimental data of the hull is readily available and widely used by the researcher for validation of CFD code. The hull is a deep V-type planning hull with a hard chine and flat transom, which are common features of a power boat. Profile view and body plan of the two hulls are shown in Fig. 1. The principal particulars of the two hulls are mentioned in Table 1.

2.2 Savitsky's Method

Savitsky's [1] 1964 method was developed based on the regression analysis of the results obtained from model test of prismatic series hull. This method is specifically validated for prismatic hulls with triangular cross-sections and does not account for variations in deadrise angle along the hull's length. This method is simple and calculation can be performed manually or using a spreadsheet. In the present study, an Excel work sheet has been used for calculation. Forces acted on planning hull are shown in Fig.2.

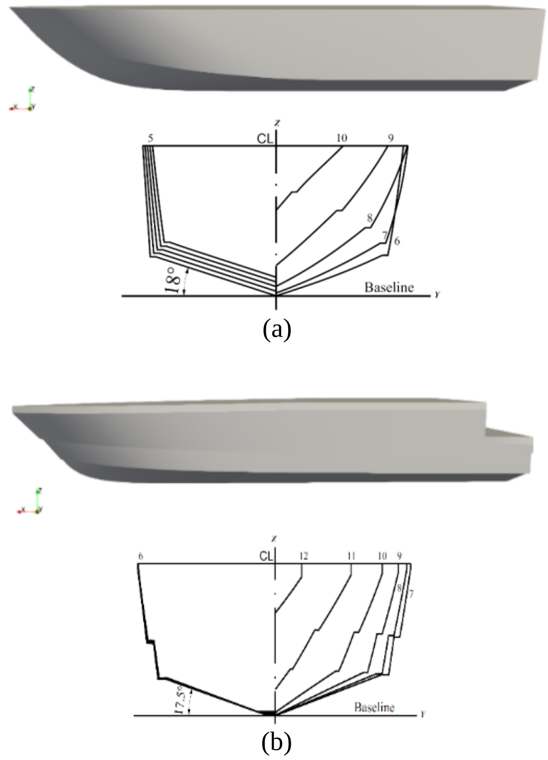


Fig.1: Profile view and body plan of (a) Generic Prismatic Planing Hull (b) Patrol boat.

Table 1 Principal particulars of GPPH and Patrol Boat

Principal Particulars	Symbol	GPPH	Patrol Boat	Unit
Length Between Perpendiculars	L_{BP}	2.414	1.05	[m]
Maximum Projected Chine Beam	B_{px}	0.627	0.3041	[m]
Deadrise Angle	β	17.50	17.50	[deg]
Displacement	∇	101.50	8.452	[kg]
Hydrostatic Draft at Transom above Baseline	T	0.1476	0.061	[m]
Hull type	--	Deep V-type	Deep V-type	--
Scale	--	1:1	1:10	--

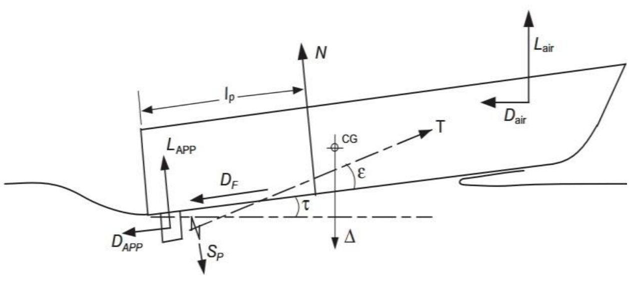


Fig.2 Forces acting on planing hull

The force (T) acting in the parallel direction of the keel and its related components are described in Savitsky's equation (1) [10]:

$$T = \Delta\, sin\tau + D_F \tag{1}$$

Where D_F is the frictional drag component computed from equation (2):

$$D_F = \frac{1}{2}\rho S V^2 C_F \tag{2}$$

The frictional coefficient is calculated from ITTC's 1957 formula:

$$C_F = \frac{0.075}{(Log_{10}Rn - 2)^2} \tag{3}$$

And wetted surface area $S = l_m b\, sec\beta$, where l_m , b and β define mean wetted length, mean wetted beam and deadrise angle, respectively. The value of l_m is calculated from an iterative method where, initially, a trim angle τ is guessed. For a flat surface with zero deadrise angle β the lift coefficient is given by,

$$C_{L0} = \tau^{1.1}\left[0.012\lambda^{0.5} + 0.0055\frac{\lambda^{2.5}}{C_v^2}\right] \tag{4}$$

Where τ is running trim angle in degree, λ and C_v are the coefficients expressed as $\lambda = \frac{l_m}{b}$, $C_v = \frac{V}{\sqrt{gB}}$.

For a surface with a deadrise angle β, the lift coefficient is given by

$$C_{L\beta} = \frac{\Delta}{0.5\rho b^2 V^2} \tag{5}$$

Here, Δ , ρ , b, V represents displacement in [N], density in kgm^{-3} , beam in [m] and speed in [ms^{-1}], respectively. Finally, a balance between two moments $N \times l_p$ and $\Delta \times lcg$ is done when the balance is obtained the solution converges. Here, N , l_p , Δ and lcg define lift force, location of the center of pressure, displacement and location of longitudinal center of gravity. N and l_p can be calculated by equation (6) and equation (7), respectively.

$$N = \Delta cos\tau \tag{6}$$

$$l_p = \left[0.75 - \frac{1}{5.21\left(\frac{C_v^2}{\lambda^2}\right) + 2.39}\right] \times l_m \tag{7}$$

2.3 Numerical method

An open source RANS based CFD solver OpenFOAM-11 has been used in this study. The governing equations are continuity equation (8) and Navier Stoke's equation (9) for incompressible flow.

$$\nabla . v = 0 \tag{8}$$

$$\rho\left(\frac{\partial u}{\partial t} + u.\nabla u\right) = -\nabla p + \mu \nabla^2 u + \rho g \tag{9}$$

The governing equations are discretized using FVM (Finite Volume Method). The RANS equation has been solved using an iterative PIMPLE algorithm, which combines the PISO and SIMPLE algorithms. Two equations SST k-omega turbulence model have been used for turbulence energy and dissipation rate. To capture the water-air interface, VOF (Volume of Fluid method) has been used. A rigid body motion solver has been applied to accommodate the motion of hull, which is defined inside the dynamicMeshDict. Three-dimensional view of the computational domain is shown in Fig. 3.

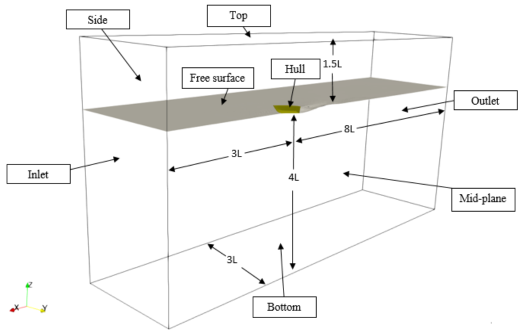
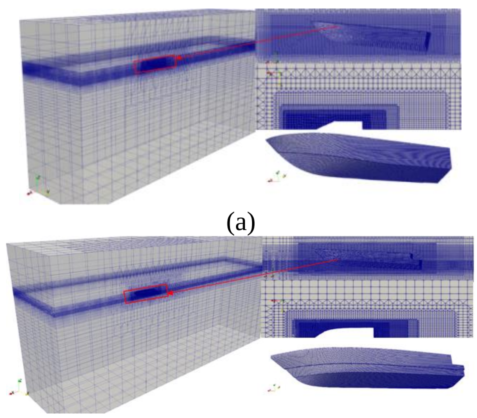


Fig.3 3D view of computational domain in x, y, z space

The boundary conditions used for numerical simulation are: (i) velocity boundary condition for inlet, outlet and hull are taken as fiexedValue, outletPhaseMeanVelocity and movingWallVelocity respectively (ii) pressure boundary condition for inlet, outlet and hull are taken as fixedFluxPressure, zeroGradiendt (no change in pressure with respect to time). For top velocity and pressure boundary conditions are pressureInletOutletVelocity and totalPressure. The bottom, side and midplane are considered as symmetryPlane (no interaction with flow).

2.4 Mesh Generation

All the simulations have been performed in half domain (half hull) to save computational time**.** To generate computational domain (shown in Fig.4) OpenFOAM's built in mesh generation utility blockMeshDict. The dimension of the domain is eight (8) ship length from aft-perpendicular to outlet, three (3) ship lengths from fore-perpendicular to inlet, three (3) ship length from centerline to side, four (4) ship length from waterline to bottom and 1.5 ship length from waterline to top of the domain.



(a)

(b)

Fig.4 General mesh assembly for (a) Generic Prismatic Planing Hull (b) Patrol Boat.

Multiple refinements have been done around the free surface with six topoSetDict. For proper caption of free-surface and motion mesh density is increased from bottom to upper chine of the hull. To accommodate within the computational domain, snappyhexMeshDict has been used. It is a difficult task to properly capture the free surface of planing hull due to heavy motion and speed. Mesh resolution near the free surface and hull is increased. However, finer mesh is a relative term. The finer the mesh, the better capture of free surface. Increasing cell numbers in the vertical direction will provide better results in the case of trim and sinkage prediction. In the present study, the fine mesh of 2.9 million and 2.8 million cells have been used for GPPH (Generic Prismatic Planing Hull) and patrol boat, respectively, according to available computational power. A desktop computer configured with an Intel core-i7 10700 (8 cores) processor with 16 GB of DDR4 RAM has been used for the present study.

3. Verification and Validation Study

3.1 Verification Study

For the verification study, the two most popular methods used to analyze uncertainties are the Grid Convergence Index (GCI) with Factor of Safety and Correction factor (C_i). The GCI with the Factor of Safety approach was described well by Celik et al. [11]. The GCI method estimates uncertainty from grid and time step errors using Richardson extrapolation with multiple solutions on refined grids. A lower GCI value indicates near grid independence and proximity to the true physical solution. Stern et al. [12] proposed a correction factor-based approach.

Table 2 Verification study of results

Parameter	Cell No. (million)	Resistance (N)	Trim (deg)	Sinkage (cm)
Mesh 1 (Fine)	2.9	162.5	6.15	7.57
Mesh 2 (Medium)	1.54	164.8	6.24	7.68
Mesh 3 (Coarse)	0.90	167.8	6.31	7.74
Grid Convergence Index $GCI_{(21)}$	--	0.06	0.06	0.02
Grid Convergence Index $GCI_{(32)}$	--	0.07	0.05	0.01
Corrected Uncertainty (F_s), $U_{(1C,Fs)}$	--	1.16%	1.28%	0.43%
Corrected Uncertainty (F_s), $U_{(2C,Fs)}$	--	1.49%	0.98%	0.23%
Corrected Uncertainty (C_i), $U_{(1C,Ci)}$	--	2.81%	3.48%	1.14%
Corrected Uncertainty (C_i), $U_{(2C,Ci)}$	--	3.62%	2.67%	0.62%

Correction factors approximate the impacts of higher-order terms in error making and offer an approach for quantifying the distance of solutions from the asymptotic range. For uncorrected solutions, the uncertainty estimation is derived from the absolute value of the corrected error estimate combined with the correction amount. In corrected solutions, the corrected error estimate, considering both sign and magnitude, establishes a numerical benchmark based on the absolute value of the correction amount. Three different meshes with a constant refinement ratio (1.25) have been used. Resistance, trim, and sinkage have been considered for the uncertainty estimation. In our analysis, all the predictions have been verified as tabulated in Table 2, considering a target uncertainty under 5%.

3.2 Validation Study

Validation of the numerical model along with Savitsky's method is done with the experimental result of GPPH (Generic Prismatic Planing Hull) at speed 5.56 ms^{-1} (10.80 knots), which corresponds to Froude number 1.14 provided in the report of Naval Surface Warfare Center Carderock Division (NSWCCD) in November 2015 [6]. Results obtained from numerical and Savitsky's method are verified with experimental results mentioned in Table 3.

Table 3 Validation with experimental result

Result	Resistance (N)	Trim (degree)	Sinkage (cm)
Experimental	161.00	5.81	7.45
CFD	162.52	6.15	7.57
Error (%)	0.944	5.85	1.59
Savitsky's Method	165.50	7	--
Error (%)	2.795	19.66	--

Numerical results show good agreement with experimental results, with minimum error (0.944 %) for resistance and maximum error for trim prediction (5.58 %). Savitsky's method shows good approximation with experimental results in resistance prediction but a significant error in trim prediction. However, sinkage cannot be compared to Savitsky's method, as the vertical center of gravity is considered fixed in position.

4. Result and Discussion

Initially, a validation study of the numerical and Savitsky's method has been performed. After the validation study, the same numerical model was applied to the patrol boat designed for the Bangladeshi Coast Guard. Finally, a comparison of results obtained from the CFD model and Savitsky's method was made for resistance, trim, sinkage, and required power at different speeds. Besides, this study has predicted the generated wave pattern and pressure distribution on the wetted surface area of the patrol boat.

4.1 Convergence

All the simulations are run at heave and pitch motion free condition up to 5 sec for GPPH and 3 seconds for Patrol boat with a time step (Δt) of $1x10^{-4}$ seconds to satisfy CFL (Courant–Friedrichs–Lewy) condition. Most of the simulations are converged after 3 seconds. The simulation convergence curve for GPPH and Patrol boat are shown in Fig.5 and Fig.6, respectively.

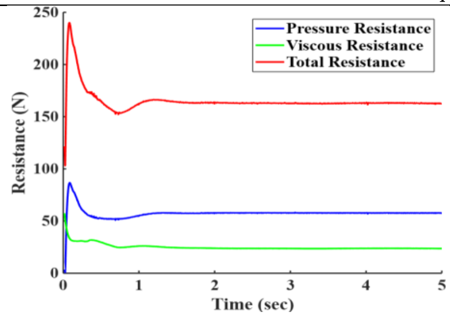


Fig.5 Simulation convergence curve for Generic Prismatic Planing Hull.

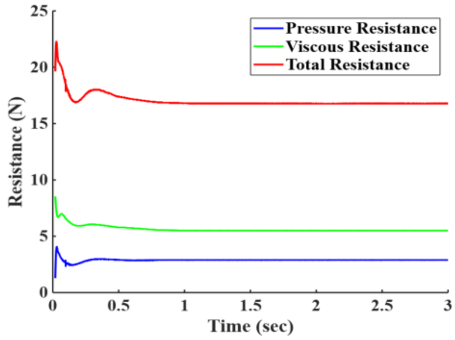


Fig.6 Simulation convergence curve for a Patrol boat.

4.2 Resistance, Trim, Sinkage

A comparison of numerical results was made with Savitsky's method at five different Froude numbers, respectively, at 1.115, 1.216, 1.317, 1.419 and 1.520, corresponding to design speeds of 22, 24, 26, 28 and 30 knots. For planing hull, the particular concern should be given to high speed region.

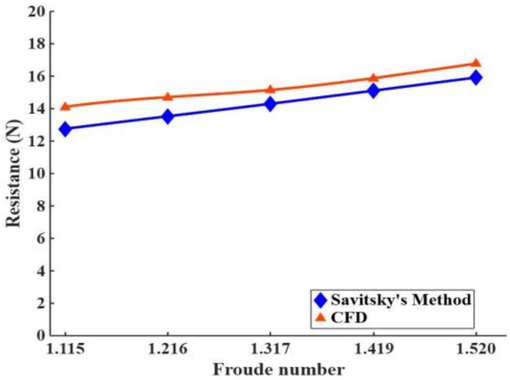


Fig.7 Resistance vs Froude number of Patrol boat.

The two resistance curves increase sharply with Froude number, as shown in Fig.7. This occurs because, at a higher Froude number, the hull rides over the water surface, wave making resistance become less significant. At this stage, the total resistance is primarily dominated by viscous drag, spray drag, and lift forces. The CFD method over-predicts the resistance than Savitsky's method across all the Froude numbers considered. The average deviation between the empirical and numerical results for total resistance does not exceed 8%. The maximum deviation, 10.57%, occurs at a Froude number of 1.115. It is worth noting that Savitsky's method was developed based on regression analysis of results obtained for high-speed planing conditions, making it more accurate in this regime. At higher Froude numbers, both methods demonstrate reasonable consistency in trend predictions. At higher Froude number, deviation of resistance decreases, which represents the inherent features of Savitsky's method.

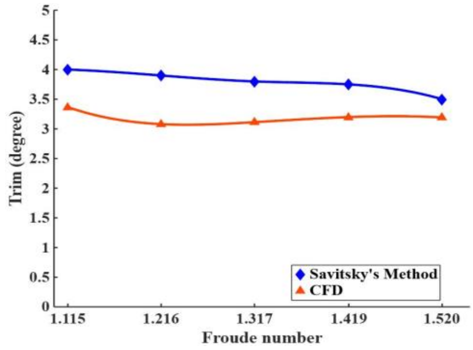


Fig.8 Trim angle vs Froude number of Patrol boat

Savitsky's method predicts the running trim angle through several iterations by balancing equilibrium of the moment of forces acting upward normally to the center of pressure and the weight of the hull acting downward. In CFD analysis, the hull is free to heave and pitch, requiring an initial trim angle to initiate the simulation. The initial trim angle is chosen based on the iterative result of Savitsky's method. At the end of the simulation, the hull is settled down at a final trim angle acting towards the LCG (longitudinal center of gravity) by balancing hydrodynamic forces and weight. The results show that Savitsky's method predicts the trim angle than CFD analysis, although the two approaches converge more closely at higher Froude numbers shown in Fig.8. The deviation between Savitsky and CFD result of trim increases from 8.25% to 21.57 within the Froude number range of 1.115 to 1.210. However, beyond a Froude number 1.317, those results become closer. With increasing Froude number (speed), the boat settles down at a small trim angle.

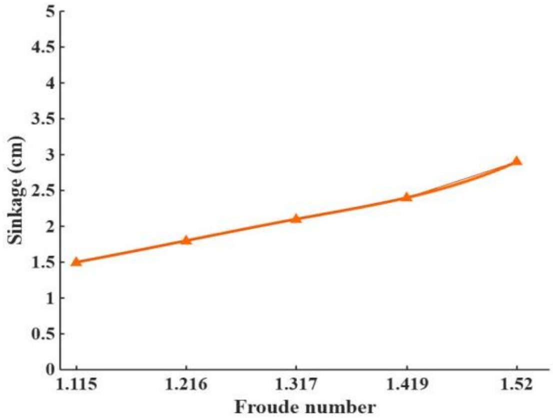


Fig.9 Sinkage vs Froude number of Patrol boat.

Fig.9 illustrates that as speed increases, lift force becomes dominant, resulting in reduced contact area with the water. Consequently, the boat rises, causing sinkage to increase at higher speeds. However, a direct comparison between numerical results and Savitsky's method for predicting sinkage is not feasible, as Savitsky's method assumes a fixed position for the vertical center of gravity (VGC) of the hull.

4.3 Power Prediction

Power is predicted by extrapolating the model results to full-scale using the traditional ITTC-1957 method. The power vs speed curve is shown in Fig.10. After exceeding the vessel speed of 26 knots, the deviation in power predictions between Savitsky's method and the CFD approach remains nearly constant.

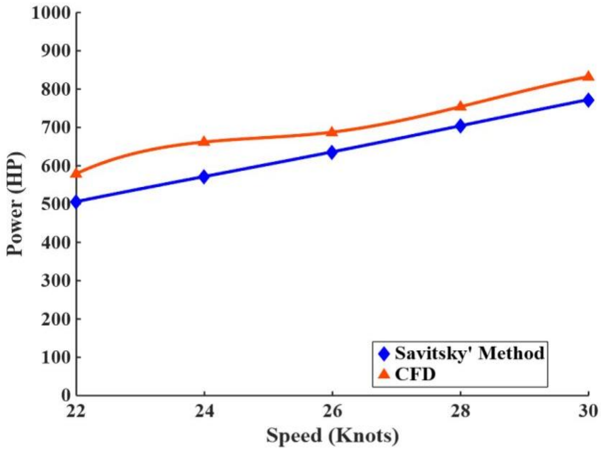


Fig.10 Power vs speed curve of Patrol boat.

The boat considered here is powered by 2 x 435 HP engines at designed speed of 30 knots. The power predicted by CFD analysis closely matches the installed power, whereas Savitsky's method predicts a significantly lower power requirement of 772.77 HP. This discrepancy highlights the limitations of the empirical method in fully capturing the dynamics of the planing regime. Since the present study mainly focused on planing regions, so the power vs speed curve tends to be flattened at higher speed indicating that less power is required to increase the speed. This suggests the boat operates more efficiently at higher speeds. For calculation of the required break power, QPC (quasi-propulsive coefficient) is considered 0.55, with shafting and gear loss taken as 4% each. Additionally, the Maximum Continuous Rating (MCR) is set at 85%. A weather and sea margin of 20% is also incorporated, considering the boat's high-speed operation in potentially rough conditions.

4.4 Flow Field Visualization

Planing hull generally operates in three regions: (i) displacement (Fr < 0.50), (ii) transition (0.50 < Fr < 0.85), and (iii) planing (Fr>0.85) region. In this paper, only planing region is studied. The wave pattern generated by the boat and hydrodynamic pressure distribution on the hull at five different Froude numbers are shown in Fig. 11 and Fig.12, respectively. The size and angle of the wake are strongly dominated by the type and speed of the hull. Planing hull surfs on the water surface. With increasing speed, there is less contact area with water due to strong hydrodynamic lift and less wave making resistance. So, the wake size and angle narrowed down at high speed, as shown in Fig. 11.

With increasing Froude number, the hull surfs over the water surface. With increasing lift force, the contact area with the water decreases. In higher speed regions, the resistance is mainly dominated by viscous drag and spray drag rather than wave making resistance. The hydrodynamic distribution of pressure on the impacts the hull is shown in Fig. 12.

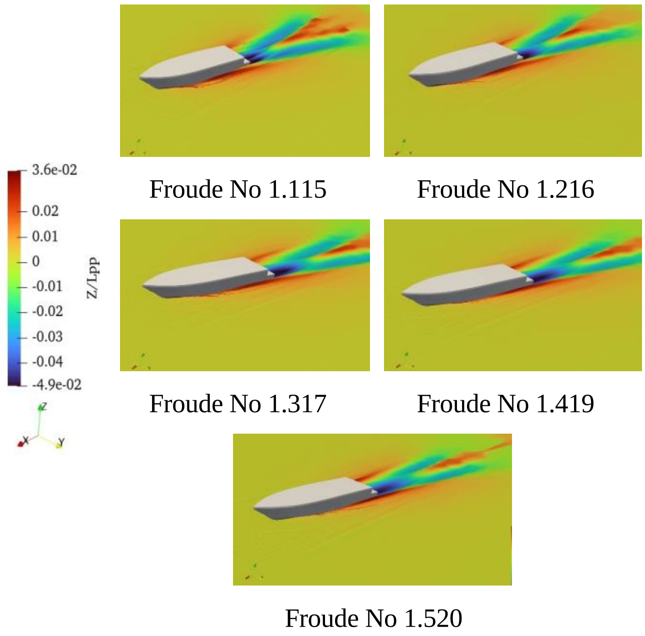


Fig.11 Wave elevation at different Froude numbers of Patrol boat.

A high speed boat generates spray at the bow, which impacts the pressure distribution, which can be seen as a sharp spray angle near the bow. With increasing lift force at high speed, the dynamic immersed volume of the hull decreases. So, hydrodynamic pressure is distributed over a small area of the bottom of the hull at high speed.

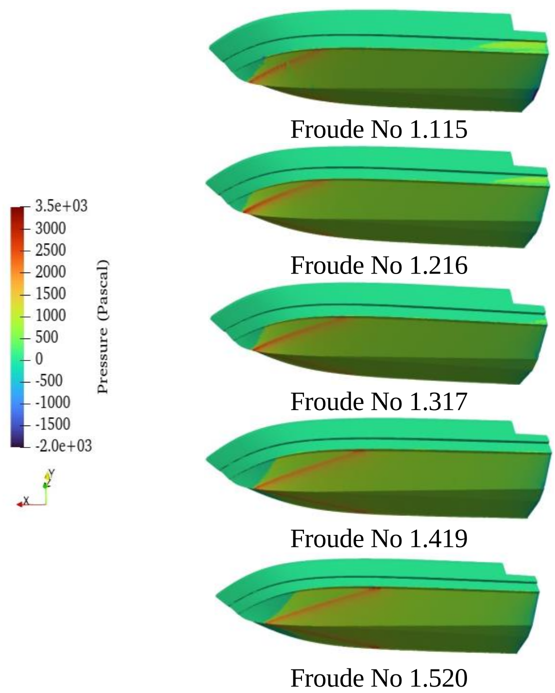


Fig.12 Distribution of hydrodynamic pressure on the hull at different Froude numbers of Patrol boats.

5. Conclusion

In the present study, hydrodynamic characteristics of planing hull are evaluated at various speeds. The main focus of the study is to predict the required break power, trim and sinkage of the planing hull using the well-known Savitsky's method and CFD analysis. Savitsky's method is used for fast calculation with some simple data input that can be done on a spreadsheet or even by hand calculation. Savitsky's method has a limitation in that it yields reliable results only for a triangular-shaped cross-section of the hull with a constant deadrise angle, whereas in practical scenarios, the cross-section of the hull varies along the length. Besides, the complex hydrodynamic nature of the hull, such as wave elevation and distribution of hydrodynamic pressure on the hull, cannot be predicted by Savitsky's method. On the contrary, the CFD method can provide reliable resistance prediction. However, the time required for meshing, calculation, and post-processing is a significant drawback in CFD. The comparison between Savitsky's method and CFD demonstrates that resistance outcomes are quite accurate at a higher Froude number, with a deviation of 5.09%. The deviation for trim calculation decreases from 21.57 % to 8.80 % as the Froude number increases. In the case of power prediction, even though Savitsky's method shows good agreement at high speed, a CFD analysis is necessary to gain a comprehensive understanding of the hydrodynamics of the planing hull.

References

[1] Savitsky, D., "Hydrodynamic design of planing hulls," *Journal of Marine Technology*, vol. 1, no. 1, pp. 71-95, 1964.

[2] Brizzolara, S., Serra, F., "Accuracy of CFD Codes in the Prediction of Planing Surfaces Hydrodynamic Characteristics," in *2nd International Conference on Marine Research and Transportation*, Napels, Italy,, 2007.

[3] Clement, E.,P., Blount, D.,L., "Resistance tests of Systematic Series of Planing Hull Forms," *SNAME Transactions*, vol. 71, pp. 491-579, 1963.

[4] Q. W. M. a. S.-L. Y. Swaine L. Chen, "Numerical Investigation On Hydrodynamic Characteristics of a Planing Hull," in *The Twentieth International Offshore and Polar Engineering Conference*, Beijing, China, 2010.

[5] Brizzolara, S., Villa, D., "CFD SIMULATION OF PLANING HULLS," in *Seventh International Conference On High-Performance Marine Vehicles*, Melbourne, Florida, USA, 2010.

[6] Lotfi, P., Ashrafizaadeh, M., Esfahan, R., K., "Numerical investigation of a stepped planing hull in calm water," *Ocean Engineering*, vol. 94, pp. 103-110, 2015.

[7] Mancini, S., Luca, F., D., Ramolini, A., "Towards CFD guidelines for planing hull simulations based on the Naples Systematic Series," in *VII International Conference on Computational Methods in Marine Engineering*, Nantes, France, 2017.

[8] Avci, A., G., Barlas, B., "An Experimental and Numerical Study of a High Speed Planing Craft with Full Scale Validation," *Journal of Marine Science and Technology*, vol. 26, no. 5, pp. 617-628, 2018.

[9] Cucinotta, F., Sfravara, F., "Experimental and Numerical Numerical Hydrodynamic Analysis of a Planing Hull Pleasure Boat," *International Journal of Applied Engineering Research*, vol. 13, no. 1, pp. 30-39, 2018 .

[10] Molland, A.,F.,Turnock,S., R., Ship Resistance and Propulsion, Cambridge University Press, 2017.

[11] Celik, I., Ghia, U., Roache, P.J., Freitas, C.J., Coloman, H., Raad, P.E., "Procedure for Estimation and Reporting of Uncertainity Due to Descrertization in CFD Applications," *Journal of Fluids Engineering*, vol. 130, no. 7, p. 078001(4 pages), 2008.

[12] Stern, F., Wilson, R., Shao, J., "Quantitive V&V of CFD simulations and certification of CFD codes," *INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS* , vol. 50, no. 11, pp. 1335-1355, 2006.

NOMENCLATURE

ρ : Density ($kg\ m^{-3}$)
μ : Dynamic viscosity ($kg\ m^{-1}s^{-1}$)
u : Velocity ($m\ s^{-1}$)
g : Gravitational acceleration ($m\ s^{-2}$)
Fr : Froude number