Numerical Calculations of Resistance and Running Attitude of a Planing Hull

Emre Kahramanoglu*, Burak Yildiz* and Huseyin Yilmaz*

*** Yildiz Technical University, Istanbul, Turkey,** emrek@yildiz.edu.tr, buraky@yildiz.edu.tr, hyilmaz@yildiz.edu.tr

Abstract

High-speed planing vessels require good hydrodynamic performance for safe operations in rough seas. Therefore accurate prediction of resistance and running attitude is vital during the design stage. The predictions have traditionally relied on model tests which are expensive and time-consuming. During the last decades, Computational Fluid Dynamics (CFD) simulations have become an important tool for the estimation of the hydrodynamic performance of planing hulls. In this study, the hydrodynamic performance of a high-speed planing hull has been investigated with CFD (Computational Fluid Dynamics) approach based on finite volume method (FVM). The numerical analyses have been carried out by solving the RANS (Reynolds Averaged Navier Stokes) equations. Verification study has been implemented by Grid Convergence Index (GCI) method to determine the optimum grid number and time-step size. For validation purpose, results obtained from CFD computations have been compared with the results those acquired from Savitsky method and experimental data in terms of resistance, sinkage and dynamic trim at different Froude numbers.

Keywords: Planing Hulls, CFD, Savitsky Method, RANS

1. Introduction

As interest in high-speed vessels has grown in the navies and maritime industry, the number of researches to predict their hydrodynamic performance has also become popular in recent years. Traditionally, the hull resistance and running attitude are predicted by carrying out model tests in towing tanks. However, laboratory tank tests are usually expensive and time-consuming. Some researchers have created a database by using the experimental results and paved the way with the developed empirical formulas by use of regression analysis of experimental data for the performance predictions. The most well-known empirical formula for the prediction of resistance of planning hulls was developed by Savitsky in 1964. Ikeda et al. (1993) also performed a set of captive model tests to investigate the effects of trim and sinkage in high-speed craft performance and a computer program was developed by the help of the experimental database to estimate the sinkage, trim, and resistance of a planing hulls. The main disadvantage of these methods was that they were restricted with prismatic hull shapes.

During the last decades, with the developments of computer technology, many researchers have focused on predictions of the behavior of the high-speed planing hulls using computational fluid dynamics (CFD). With the increasing of computer power, the accuracy of numerical results have been improved continuously. Azcueta et al. (2003) used a commercial software to simulate the dynamic sinkage and trim of a high-speed planning vessel with the computed resistance for the whole Froude number range. Brizzolara and Serra (2007) performed the series of numerical simulations of planning hulls by using a CFD code and compared the results with Savitsky method and experimental results. They also extended the study for stepped hulls with the partially ventilated bottom (2010). The CFD results showed better agreement than Savitsky method with the experiments and showed the capability of CFD simulations. Cao (2008) and Wang et al. (2009) used numerical simulations to predict the resistance of a planning vessel and the results showed good accuracy with the experimental results and empirical formula. Caponnetto et al. (2013) and Fu et al. (2014) also performed numerical simulations for the predictions of hydrodynamics of high-speed planing crafts in calm water and waves. They found that CFD simulations of planning hulls can yield robust results that differ from experimental results by less than 10%. Frisk and Tegehall (2015) evaluated the performance of CFD simulations for planning hulls by using two different commercial softwares: Fluent and Star-CCM+. They predicted the steady resistance, sinkage and trim angle of a semi-planing and a planning hull in calm and unrestricted water. The analyses at different Froude numbers between 0.447 and 1.79 were carried out and the results followed the same trends as what is seen in the experimental data where the differences of the predicted values are below 10%. Sukas et al. (2017) investigated the hydrodynamics of a planing hull with overset grid technique. They showed that the results obtained from this method satisfactorily agree with the experimental data. They also pointed out that Savitsky's method is not a reliable method for the calculation of resistance components while it predicts the total resistance with a good accordance. De Luca et al. (2016) conducted an extended verification study for planing hulls. They implemented different verification assessments for various planing hulls.

The main focus of this study is to determine the hydrodynamic characteristics of a planing hull from Fn=0.355 to Fn=0.904 with computational fluid dynamics analyses which are based on finite volume method. The comparative results are presented with available experimental data (Kimoto et al., 2014) and Savitsky Method.

2. The Planing Hull Model

The main particulars of the hull are listed in Table 1 and the 3-D geometry of the model is demonstrated in Figure 1. The hull has no appendages.

Table 1. Main Particulars of the model hull

Length	1.0	m
Breadth	0.333	m
Depth	0.101	m
Draft	0.041	m
Initial Trim Angle	0.0	deg
Projected area	0.041	m^2
Deadrise Angle	12.0	deg
Fn	0.355-0.904	-

Fig. 1. 3D view of the hull

3. Savitsky Method

The most important and famous study on planing hulls was conducted by Savitsky (1964). In this study, a semi-empirical method was proposed to determine the hydrodynamic characteristics of a planing hull. This method can be shortly summarized as follows:

$$c_{L,\beta} = \frac{\Delta g}{0.5\rho B^2 V^2} \tag{1}$$

$$c_{L,0} = c_{L,\beta} + 0.0065\beta c_{L,0}{}^{0.6} \tag{2}$$

$$\frac{l_p}{B\lambda} = 0.75 - \frac{1}{\frac{5.21 c_V{}^2}{\lambda^2} + 2.39} \tag{3}$$

$$c_{L,0} = \tau^{1.1}\left(0.012\lambda^{0.5} + 0.0055\left(\frac{\lambda}{c_V}\right)^2\right) \tag{4}$$

Here, $c_{L,0}$ and $c_{L,\beta}$ are the lift coefficients when the deadrise angle equals to zero and β, respectively. β is deadrise angle, c_V is the dimensionless velocity coefficient based on the breadth, B is the breadth of the hull, l_p is the center of the pressure, τ is the trim angle, c_F is the friction coefficient calculated via Schoenherr's formula and λ is the aspect ratio. V_1 and V are the velocities at mean bottom and hull, respectively.

Pressure resistance (R_P), viscous resistance (R_V) and the total resistance (R_T) can be calculated as follows:

$$R_V = \frac{0.5\rho V_1^2 c_F \lambda B^2}{\cos\tau \cos\beta} \tag{5}$$

$$R_P = \Delta g \tan\tau \tag{6}$$

$$R_T = R_V + R_P \tag{7}$$

Detailed information about this method can be found in the paper of Savitsky (1964).

4. Numerical Method

4.1. Physical Model

In this study, the hydrodynamic analysis of planning hull was done using unsteady RANS (Reynolds Averaged Navier Stokes) approach based on finite volume method. The flow was considered as incompressible, fully turbulent and unsteady. k-ε turbulence model was selected. During the analyses, the effects of the free surface were taken into consideration using volume of fluid (VOF) approach. SIMPLE algorithm which is based on pressure-velocity coupling was applied to solve the pressure field. In the calculations, the hull was assumed to free to sinkage and trim. Therefore, Dynamic Fluid-Body Interaction (DFBI) method was implemented to represent the 2-DOF (degree of freedom) motion. In all 2-DOF analyses, there were two main coordinate systems. One of them was the global coordinate system and the other was fixed to centre of the planing hull. The second one was allowed to move with motion of the planing hull.

4.2. Mesh Structure and Boundary Conditions

The main dimensions of the computational domain are given in Figure 2. The computational domain extends 2.75 L in front of the overset region, 7.75 L behind the overset region, 2.7 L to the side, 1.9 L the under and 0.9 L the top of the boundaries of the overset region.

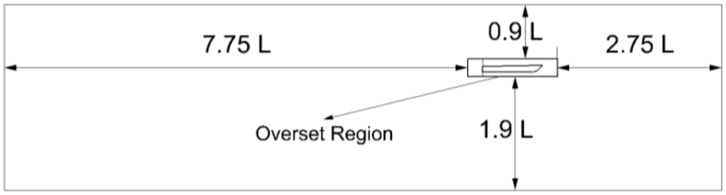

Fig. 2. Size of the computational domain

As seen Figure 3, the left and right side of the computational domain was defined as velocity inlet and pressure outlet, respectively. The rest of the surfaces except for symmetry plane were defined as velocity inlet. The hull surface was selected as a wall to impose the kinematic boundary condition.

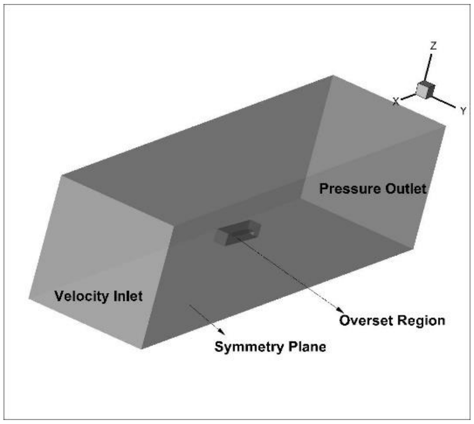

Fig. 3. Boundary conditions of the computational domain

The unstructured trimmer mesh was applied to discretize the computational domain. Since planing regime of the hull has high translations and orientations, overset grid technique was used. The unstructured mesh on the planing hull is given in Figure 4a and Figure 4b. The total mesh number in the fine grid equals to 1.2 x 10^6. In addition, to reduce the mesh numbers Kelvin adopted mesh structure was created and the only half-domain was taken into consideration (Figure 4).

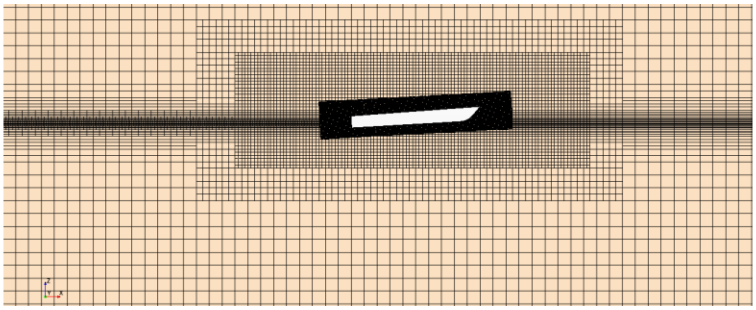

Fig. 4a. The profile view of the grid

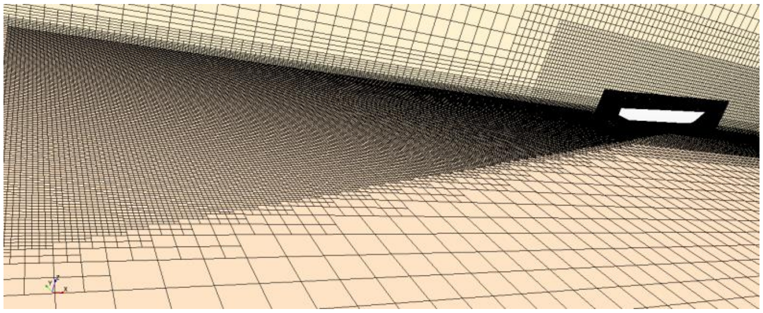

Fig. 4b. The mesh structure

4.3. Choice of the Time Step

In present study, time step size was determined according to ITTC (2011) recommendations. The fine time step size was selected as 0.005 s.

4.4. Verification Assessment

In the present paper, GCI (Grid Convergence Index) method, which is commonly used in several studies (Sezen et al. 2018, Cakici et al. 2017), was used for verification purpose. This method which is based on Richardson Extrapolation (Richardson 1910) was put forward by Roache (1998) and developed with numerous studies (Stern et al. 2001, 2006).

The methodology proposed by Celik et al. (2008) was implemented to determine the grid size and time-step uncertainties. For each verification case, three solutions were obtained by refining grid or time step size, systematically. The refinement factor was selected $2^{1/2}$ as often used in verification of numerical studies (Sezen et al. 2018, De Luca et al. 2016). The methodology proposed by Celik et al. (2008) can be summarized as follows:

i) In order to calculate the difference between two solution equation (8) can be used:

$$\varepsilon_{21} = \varphi_2 - \varphi_1 \quad \varepsilon_{32} = \varphi_3 - \varphi_2 \tag{8}$$

In equation (8), φ_1, φ_2 and φ_3 refer the value of any scalar of fine, medium and coarse time step or grid size, respectively.

ii) The convergence condition can be calculated as follows:

$$R = \frac{\varepsilon_{21}}{\varepsilon_{32}} \tag{9}$$

The convergence condition must be in the range between 0 and 1. This range is also called monotonic convergence range (Stern et al. 2006).

iii) The apparent order (p) and approximated relative error (e_a^{21}) can be calculated as follows:

$$p = \frac{\left| \ln \left| \frac{1}{R} \right| \right|}{\ln(r_{21})} \tag{10}$$

$$e_a^{21} = \left| \frac{\varphi_1 - \varphi_2}{\varphi_1} \right| \tag{11}$$

iv) Finally, the GCI index is:

$$GCI_{FINE} = \frac{1.25 e_a^{21}}{r_{21}^{p} - 1} \tag{12}$$

If the convergence condition (R) is between 0 and -1, often more than three solutions are needed and the uncertainty (U_K) of the implementation can be calculated as follows (Stern et al. 2001, De Luca et al. 2006):

$$U_K = \frac{\left| S_U - S_L \right|}{2} \tag{13}$$

The verification assessment was applied for Fn = 0.645. In Table 2, the numerical uncertainties of the time step and grid sizes are shown.

Table 2. Numerical Uncertainties of the Total Resistance

	Grid Spacing	Time Step
φ_1	6.59	6.59
φ_2	6.36	6.57
φ_3	6.89	6.40
R	-0.43	0.081
GCI_{FINE}	3.99 %	0.02 %

In Table 2, convergence condition of the grid spacing uncertainty is between 0 and -1. Therefore finer grid resolution was created and it was observed that the convergence condition was still in the same regime. Because of that the uncertainty of the grid spacing (U_K) was calculated using equation (13) and the percentage of uncertainty was obtained by dividing uncertainty (U_K) to fine grid solution (φ_1).

6. Results

In this paper, the numerical results of the planing hull are presented for the Fn range 0.355 and 0.904. The obtained result from CFD and Savitsky's method are given in Table 3. In order to get the non-dimensional resistance value, the projected area was used as in Table 1. The results revealed that the CFD results has more remarkable results than Savitsky's method for the total resistance coefficient (C_T) and the dynamic trim angle.

Table 3. Comparison of Numerical Method with Experiment and Empirical Method

	EFD			CFD			Savitsky	
Fn	CFx	Rise (mm)	Trim(deg)	CFx	Rise (mm)	Trim(deg)	CFx	Trim(deg)
0.355	0.089	-4.106	0.237	0.087	-3.20	0.450	-	-
0.484	0.122	-5.069	2.527	0.114	-5.58	2.689	-	-
0.645	0.083	2.704	3.662	0.079	2.28	3.683	0.035	4.302
0.904	0.051	14.616	4.559	0.045	13.38	4.651	0.022	4.560

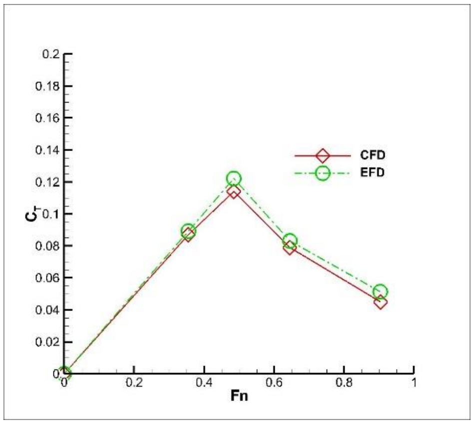

Fig. 5. Comparison of the total resistance coefficient

In Figure 5, the total resistance coefficients obtained from CFD are compared to the experimental results. It is obvious that, although the computational fluid dynamics results under estimate the total resistance coefficients, the results are satisfactorily agree with the experimental data.

The dynamic trim angle and the negative sinkage values are shown in Figure 6 and Figure 7. For dynamic trim, the results obtained from CFD are higher than the available experimental data. However, results obtained from CFD underestimates the sinkage values compare to experiments. After Fn=0.35, the CFD results are getting closer to the experiments for both hydrodynamic characteristics. However, it can be seen from the figures, the results are again in good agreement with experiments.

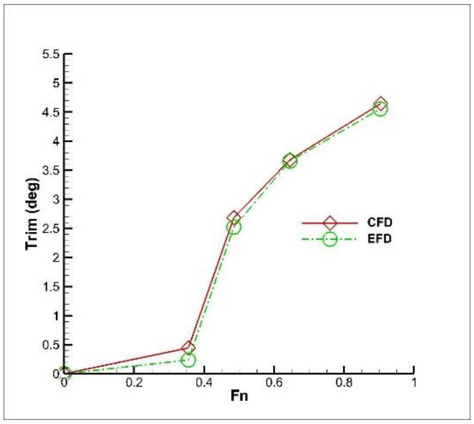

Fig. 6. Comparison of the dynamic trim angle

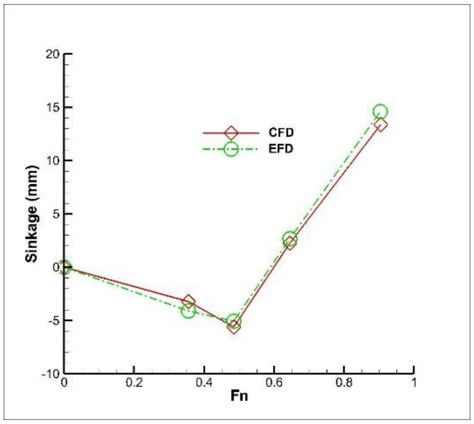

Fig. 7. Comparison of the sinkage value

In all numerical analyses, the Kelvin waves was captured. In Figure 8, the Kelvin wave patterns are shown for Fn=0.645.

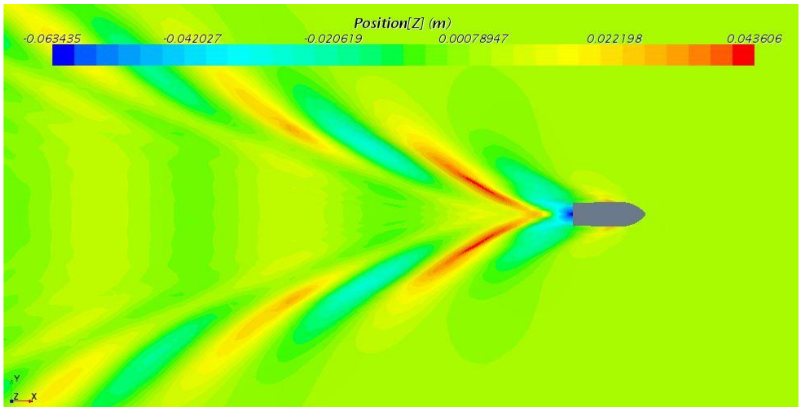

Fig. 8. Kelvin wave pattern for Fn=0.645

6. Conclusion

In the present study, the hydrodynamic performance of a planing hull is determined by using Unsteady Reynolds Averaged Navier-Stokes approach. The results are compared with experimental results and Savitsky method. The uncertainty assessment of the numerical application is done with Grid Convergence Index (GCI) method in terms of grid and time step size. The results obtained from CFD are satisfactorily agreed with experimental data.

The results show that Savitsky method and the applied CFD are useful tools to estimate the hydrodynamic characteristics of a planing hull. On the other hand CFD results have more remarkable results than Savitsky method. Although CFD results underestimate the total resistance and the sinkage values, the dynamic trim angles are overestimated. However, the results obtained from numerical method are in satisfactorily good agreement with experimental data.

References

Azcueta, R. (2003). Steady and unsteady RANSE simulations for planing CRAFTS, 7th Conference on Fast Sea Transportation, FAST'03, Abano Terme, Italy.

Brizzolara, S. and Serra, F. (2007). Accuracy of CFD Codes in the Prediction of Planing Surfaces Hydrodynamic Characteristics, 2nd International Conference on Marine Research and Transportation, Ischia.

Brizzolara S. and Federici A. (2010). CFD Modeling of Planning Hulls with Partially Ventilated Bottom, Proceedings of The William Froude Conference: Advances in Theoretical and Applied Hydrodynamics Past and Future, Royal Institution of Naval Architects, vol. 1, Portsmouth, VA, USA.

Cakici, F., Kahramanoglu, E. and Alkan A.D. (2017). Numerical Prediction of Vertical Ship Motions and Added Resistance, Transactions of the Royal Institution of Naval Architects Part A: International Journal of Maritime Engineering. Vol: 159.

Cao, H. (2008). The Computation and research on resistance of planing craft based on the software FLUENT. PhD thesis, Harbin Engineering University, Harbin, 44-53.Dong Wencai.

Caponnetto, M., Bučan, B., Pedišić-Buča, M., Perić, M. and Pettinelli, C. (2013). Simulation of Flow and Motion of High-Speed Vessels, Proceedings of the 12th International Conference on Fast Sea Transportation. Amsterdam.

Celik, I., Ghia, U., Roache, P., Fretias, C.J., Coleman, H. and Raad P.E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications, J. Fluids Eng.-Trans. ASME, vol. 130, no. 7.

De Luca, F., Mancini, S., Miranda, S. and Pensa, C. (2016). An Extended Verification and Validation Study of CFD Simulations for Planing Hulls, Journal of Ship Research, Vol. 60, No 2, pp: 101-118.

Frisk, D. and Tegehall, L. (2015). Prediction of High-Speed Planing Hull Resistance and Running Attitude, Master Thesis, Chalmers University of Technology.

Fu, T. C., Brucker, K. A., Mousaviraad, S. M., Ikeda, C. M., Lee, E. J., O'Shea, T. T., et al. (2014). An Assessment of Computational Fluid Dynamics Predictions of the Hydrodynamics of High-Speed Planing Craft in Calm Water and Waves. 30th Symposium on Naval Hydrodynamics. Hobart.

Ikeda, Y., Yokomiza, K., Hamasaki, J., Umeda, N. and Katayama, T. (1993). Simulation of Running Attitude and Resistance of High-Speed Craft Using a Database of Hydrodynamic Forces Obtained by Fully Captive Model Experiments, FAST 93. Second International Conference on Fast Sea Transportation.

ITTC (2011). Practical Guidelines for Ship CFD Applications. Rio de Janeiro, Brazil

Kimoto, R., Katayama, T. and Ikeda, Y. (2004), Effects of Running Attitude on Hydrodynamic Forces for Oblique Towed Planing Craft, Proc. of the 2nd Asia-Pacific Workshop on Hydrodynamics (APHydro2004), pp.115-122, Busan, KOREA, 2004.

Richardson L.F. (1910). The approximate arithmetical solution by finite differences of physical problems involving differential equations, with an application to the stresses in a Masonry dam. Philos. Trans. R. Soc. Lond. Ser. A, 210 , pp. 307-357.

Roache P.J. (1998). Verification of Codes and Calculations. AIAA J., vol. 36, 5: 696-702.

Savitsky, D. (1964). Hydrodynamic design of planing hulls, Mar. Tch., Vol. 1, p.71-95.

Sezen, S., Dogrul, A., Delen, C. and Bal, S. (2018). Investigation of self-propulsion of DARPA Suboff by RANS method, Ocean Engineering, 150: 258-271.

Stern, F., Wilson, R., Coleman, H. W. and Paterson, E.G. (2001). Comprehensive Approach to Verification and Validation of CFD Simulations—Part 1: Methodology and Procedures, Journal of Fluids Engineering, Vol. 123(4), pp: 793-802.

Stern, F., Wilson, R. W. and Shao J. (2006). Quantitative V&V of CFD simulations and certification of CFD codes, International Journal for Numerical Methods in Fluids, 50:1335-1355.

Sukas, O.F., Kinaci, O. K., Cakici, F. and Gokce M. K. (2017). Hydrodynamic assessment of planing hulls using overset grids, Applied Ocean Research, 65: 35-46.

Wang, Z., Niu, J., Qin, Z. and Pang, Y. (2009). The computation resistance of planing craft based on the CFD techniques, 14th Conference on China Ocean Engineering, Hohhot, 309-315.