Resistance prediction of a high speed craft by using CFD

Sadık ÖZÜM[a],* Bekir ŞENER[a], Kaan ÜNLÜGENÇOĞLU[b],

[a]*Department of Naval Architecture and Marine Engineering, Yildiz Technical University, Istanbul, 34349, Turkey*
[b]*Department of Marine Operation Engineering, Yildiz Technical University, Istanbul, 34349, Turkey*

Abstract One of the most important parameter for resistance prediction of a high speed craft is hydrodynamic forces during planing. Because of these forces, the draught and displacement of the hull changes, and it affects to reduce resistance. In this study, resistance analyses of a high speed craft is carried out for 35 knots speed by using new CFD techniques. Vertical plane motions and lift force is taking into account by means of 6-DOF Motions approach. As a consequence, the resistance of the craft can be calculated more realistically with considering hydrodynamic effects.

Keywords: High speed craft, CFD, vertical plane motions.

1. Introduction

Nowadays, numerical simulations are becoming a common way for assessment of ship performance in early design stages. Although experimental approach is still very useful it has its own restrictions and model tests are long dated and expensive. Taking into account the advances in computer hardware, use of Computational Fluid Dynamics (CFD) is becoming the best choice in many cases. In this way CFD simulation plays an important role in hull form design, performance analyses and form optimization etc.

The prediction of resistance for planing craft is a complex problem. Planing crafts are high-speed marine vehicles with applications ranging from small pleasure boats to large military crafts. Generally in a properly configured planing hull form the deadrise angle diminishes from bow towards stern. High-speed planing crafts have hard chine, and may have both longitudinal and transverse steps at intermediate positions over the wetted region. The planing craft run typically with a small bow-up trim or attack angle. The flow around the planing hull is characterized by a large spray, a dry transom at design speed, sharp separation of the flow at the chine with eventual reattachment along the sidewalls. The main problem encountered in resistance prediction of a high speed craft is considering the hydrodynamic lift forces during the analyze process. With developments in CFD, the 6-DOF motions can be considered in flow analyses.

The prediction of resistance of planing craft has been the subject of many investigations [1,2]. Caponnetto (2001) studied a CFD code for the analysis and design of planing crafts and compared the results with Savitsky method. Subramanian et al (2007) investigated pressure and resistance characteristics of a single chine high speed planning hull. They used two hull forms, one with a tunnel (called also "propeller pockets") and one without, and finally compared the results with results obtained from towing test. Ozdemir et al (2007) analyzed a high speed craft with different turbulent models. They investigated the effects of turbulent models on solution and compared the experimental result with the result obtained from CFD analyses. Moreover, Brizzolara and Serra (2007) investigated the accuracy of CFD codes in the prediction of planing surfaces. They used a wedge shaped planing hull to CFD analyses by varying the running trim angle systematically, and compared the results with model tests and semi-emprical theories.

In this study, resistance analyses of a high speed planing craft is carried out for 35 knot by using new CFD techniques. Vertical plane motions and lift force is obtained by means of 6-DOF Motions approach. Thus, hydrodynamic effects are considered and it is aimed to predict the resistance of the high speed craft more realistic. Finally the result obtained from CFD analyses is compared with the result of statistical Savitsky resistance prediction to evaluate the solution.

2. Physical Model

A CFD work involves three different activities; preprocessing, analyzing the problem by using a solver and post-processing to demonstrate the results.

In this study, the hull form, used for analyses, is designed by using MAXSURF commercial software. The hull form is exported to RHINO commercial software to generate the exterior domain and to make some modifications. Finally the hull and domain exported to STARCCM+ commercial CFD software which is used for mesh generation and to prescribe boundary conditions. The main dimensions and form coefficients of the hull are given in Table 1. Fig. 1 shows the body plan of the hull.

Table 1: Main dimensions and form coefficients of the hull.

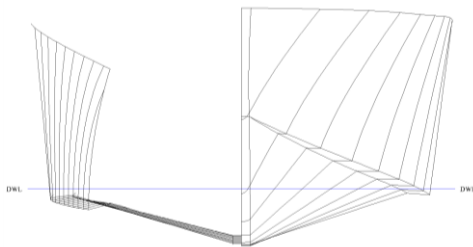| Displacement	3250	kg
Draft (T) (from baseline)	0.38	m
Waterline length (L_{WL})	9.08	m
Prismatic coefficient (C_P)	0.697	
Block coefficient (C_B)	0.38	
Mid-ship section coefficient (C_M)	0.577	
Water plane area coefficient (C_W)	0.718	

Fig. 1: Body plan of the planing hull

3. Mathematical Model

STARCCM+ uses Volume of Fluid (VOF) model to take into account free surface in CFD calculations. VOF model assume two different fluids, water and air, that do not get into each other. The transport equation is solved for the volume fraction of water. Besides, heave and pitch motions from 6-DOF Motions are considered during analyze process.

3.1 Computational domain and grids

The grid system for CFD calculation is generated using the STARCCM+ software. Discretization of the flow equations requires the subdivision of the computational domain into a grid of sufficiently small cells as shown in Fig. 2a. This figure shows the unstructured grid for the hull and whole domain. An unstructured grid provides more flexibility in geometry and grid generation. Multi-block technique is used for generation of the grid and mesh structure is clustered near the hull surfaces. Grid structure around the hull can be seen in Fig. 2b. It must be noticed that generation of a grid around the hull is time consuming and must be generated to describe the hull geometry realistic. For the hull body and whole domain, approximately 875.000 trimmer cells are used. Analysis are then performed on a cluster consists of four parallel computer.
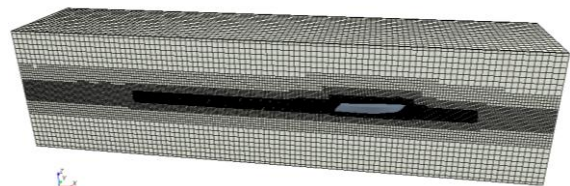
(a)

(b)

Fig. 2: (a) the mesh structure of whole domain (b) clustered mesh structure near hull surface

3.2 Boundary conditions

Inlet: forward, top and bottom of the domain, uniform flow is given, 18 m/s
Outlet: backward of the domain, pressure outlet
Walls: hull body, no-slip condition
Symmetry: centerline boundary and side of the domain

3.2 Turbulence model

In this paper, three-dimensional, incompressible, two phase flow of air-water is considered. The governing equations for the fluid are the equation of continuity (Eq. 1) and momentum balance (Eq. 2) in Cartesian tensor notation.

$$\frac{\delta \rho U_i}{\delta x_i} = 0 \tag{1}$$

$$-\frac{\delta P}{\delta x_i} + \frac{\delta}{\delta x_j}\left[\mu(\frac{\delta U_i}{\delta x_j} + \frac{\delta U_j}{\delta x_i}) - \rho\overline{u_i u_j}\right] = 0 \tag{2}$$

The standard $k - \omega$ model is an empirical model based on transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (ω), which can also be thought of as the ratio of ε to k.

As the $k - \omega$ model has been modified over the years, production terms have been added to both the k and ε equations, which improved the accuracy of the model for predicting free shear flows. The turbulence kinetic energy, k, and the specific dissipation rate, ω, are obtained from the following equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial}(\rho k u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k + S_k \tag{3}$$

and

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial\omega}{\partial x_j}) + G_\omega - Y_\omega + S_\omega \tag{4}$$

G_ω represents the generation of ω. Γ_k and Γ_ω represents the effective diffusivity of k and ω due to turbulence. All of the above terms are calculated as described below. S_k and S_ω are user-defined source terms.

The effective diffusivities for the $k - \omega$ model are given by

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}, \quad \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \tag{5}$$

where σ_k and σ_ω are the turbulent Prandtl numbers for k and ω, respectively. The turbulent viscosity, μ_t, is computed by combining k and ω as follows:

$$\mu_t = \alpha * \frac{\rho_k}{\omega} \tag{6}$$

4. Results and Discussions

In this paper it is presented the results of the study obtained after two weeks running. It should be noted that, to take into account the 6-DOF motions in analyses, center of gravity (C_g) of the hull have to be entered to the program. In solution, the coordinate system is overlapped on the C_g.

As a result of CFD analyses, pressure contours on the hull surface are given in Fig. 3. As shown in the figure, it is obtained high pressures around the amidships and low pressure at the stern of the hull. The bow of the hull rises in time and stays above the water plane. This motion of the hull is common for the planing hulls. Moreover, it can be seen the streamlines around the hull in Fig. 4.
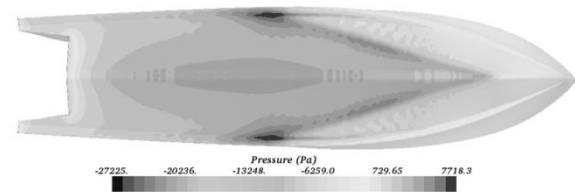

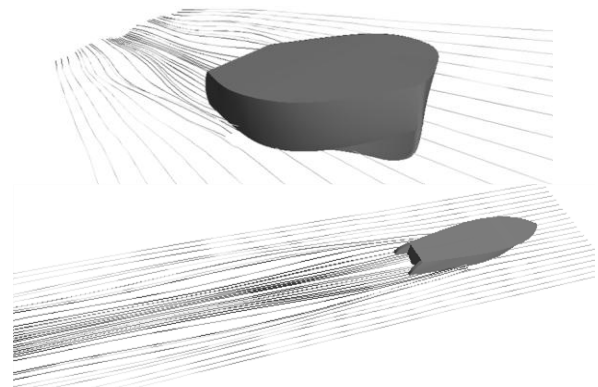Fig. 3: Pressure contours on the hull surface

Fig. 4: Streamlines around the hull

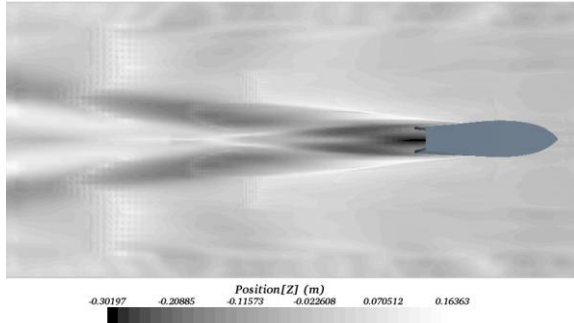Figure 5 shows the waves generated from the hull.

Fig. 5: Waves generated from the hull (35 knots)

Resistance and running trim angle of the craft is calculated by Savitsky method to compare the results obtained from CFD. Compared values are given in Table 2.

Table 2: Resistance and trim comparison

Type of analyses	Total resistance (kN)	Running trim (deg.)
Savitsky method	7.32	2
CFD method	6.94	2.85

As shown Table 2, result of CFD analyses is in good agreement with Savitsky method.

5. Conclusions

The main objective of this study is to show the capability of the general-purpose CFD solver of STARCCM+ to use 6–DOF motions on resistance prediction of high speed crafts. CFD analyses is performed with considering the hydrodynamic lifting forces for a planing craft and the study is concluded with a comparison between the CFD result and the result obtained from Savitsky method.

On the contrary of displacing ships, where the resistance curves are very steep, planing hulls have a flatter resistance curve; this means that larger differences of speed can be gained or lost with minor changes of the hull efficiency. As a consequence, with more real-like resistance prediction for a planing craft, it is possible to give to the designer useful suggestions for the improvement of its hull at the earliest stage of design process.

6. References

[1] R.H. Akers, *Dynamic analysis of planing hulls in the vertical plane*, Ship Motion Associates, Portland, Maine (1999).

[2] R. Akers, S. Hoeckley, R. Peterson, A. Troesch, *Predicted vs. measured vertical-plane dynamics of a planing boat*, FAST'99, pp. 91-105, Seattle, USA, (1999).

[3] M. Caponnetto, *Practical CFD simulations for planing hulls*, HIPER, 2001.

[4] V.A. Subramanian, P.V.V. Subramanyam, N.S. Ali, *Pressure and drag influences due to tunnels in high-speed planing craft*, International Shipbuilding Progress 54 (2007) 25–44.

[5] Y.H. Özdemir, S. Bayraktar, T. Yılmaz, *Computational investigation of a hull,* 2nd International Conference on Marine Research and Transportation, Naples, Italy, 2007.

[6] S. Brizzolara, F. Serra, *Accuracy of CFD codes in the prediction of planing surfaces hydrodynamic characteristics,* 2nd International Conference on Marine Research and Transportation, Naples, Italy, 2007.