Applied Ocean Research 65 (2017) 35–46

Contents lists available at ScienceDirect

Applied Ocean Research

journal homepage: www.elsevier.com/locate/apor

Full length article

Hydrodynamic assessment of planing hulls using overset grids

Omer Faruk Sukas, Omer Kemal Kinaci*, Ferdi Cakici, Metin Kemal Gokce

Yildiz Technical University, Naval Architecture and Maritime Faculty, Istanbul, Turkey

ARTICLE INFO

Article history:
Received 6 January 2016
Received in revised form 27 January 2017
Accepted 25 March 2017

Keywords:
Planing hull
Overset grid
Savitsky
Ship resistance
Squat
Hydrodynamic lift
URANS

ABSTRACT

In conjunction with high performance computers, recent developments in computational science paved the path to more accurate representation of body motions inside fluids. Small motions inside the flow can be computationally approximated by using rigid body motion but it is incapable of accurately predicting the large motions of a planing vessel. The implementation of overset grid has made it possible to better approximate the complex fluid-structure interaction problem of the planing regime. The focus of this study was to evaluate the opportunity of using an overset grid system to numerically solve the flow around a planing hull and to understand the planing regime with this invaluable tool. It was shown in this study that the overset grid better captures the large motions of the planing hull at high Froude numbers. Then, the results obtained by overset grid were used to calculate the resistance components of a planing hull in a wide Froude number range. The resistance components were discussed with respect to values generated by Savitsky approach. Using the benefits that the computational science brings, the flow was visualized to explain some underlying physics relevant to the planing regime.

1. Introduction

Planing hulls have been one of the most challenging problems for the naval architecture society as the large motions of the hull complicate hydrodynamical calculations and hull optimization. Researchers have tried to approach the problem experimentally or computationally, though with many assumptions. The large motions of the hull created problems generalizing the experimentally or computationally derived results. It was Savitsky [15] who was one of the first (but definitely the most famous) who succeeded to formulate and generalize the motions of the hull (trim) and the drag (total resistance) it encounters in the flow. There are many studies that came after his. Some tried to formulate the pressure of the underwater hull to calculate slamming effects [13]; others tried to enhance Savitsky's method by introducing whisker spray drag [16] or estimated the wake profile at the aft of the hull [17] to understand the underlying hydrodynamics behind planing regime. But these studies were all empirical approaches to the planing hull problem. They are all valid in a limited range as advised by the researchers who performed the experiments and compiled the results. None of these empirical approaches cover the whole aspects of the flow around a planing hull. If a designer wanted to work with an unconventional planing hull geometry in a Froude (*Fr*) number that is not covered by these works, the remaining options were to conduct experiments or approach the problem with computational fluid dynamics (CFD). Experiments are not economically feasible to do hull optimization of a planing hull as the costs of conducting experiments are substantially higher than approaching the problem computationally. Plus, high speed computers started becoming widespread and the costs of computational methods gradually decreased in the last few decades. Together with commercially available softwares or other in-house codes that are capable of solving viscous flow involving fluid-structure interaction, it has become possible to solve the complicated flow around a planing hull even for the most non-traditional geometries with high accuracy.

There are many computational approaches to hydrodynamically solve the planing hull problem but in general, all of these methods can be classified into two. Likes of Ghassemi [9,6,8] and Matveev [11,12] implement potential theory to solve the inviscid flow around the planing hull and obtain faster results, owing to the practicality of boundary element methods. Stern [14,21] and many others [10,20] use Unsteady Reynolds-Averaged Navier Stokes Equations (URANS) to solve the viscous and transient flow which consumes considerably higher amount of time. In his book *Hydrodynamics of High Speed Marine Vehicles*, Faltinsen [24] broadly covers the literature and explains many flow aspects of planing vessels. In their review article, Yousefi et al. [23] mention the anal-

Abbreviations: URANS, unsteady Reynolds averaged navier stokes; CFD, computational fluid dynamics; DOF, degree of freedom; DFBI, dynamic fluid body interaction; LCG, longitudinal center of gravity; VOF, volume of fluid.

* Corresponding author.
E-mail address: kinaci@yildiz.edu.tr (O.K. Kinaci).

http://dx.doi.org/10.1016/j.apor.2017.03.015
0141-1187/

Nomenclature

B	Breadth of the vessel
C	Courant number
C_F	Frictional resistance coefficient
C_{L0}	Lift coefficient of flat plate
$C_{L\beta}$	Lift coefficient with a deadrise angle
C_Δ	Displacement coefficient
Fr	Froude number
Fr_B	Froude number based on breadth
g	Gravitational acceleration
L_{hd}	Hydrodynamic lift
L_{hs}	Hydrostatic lift
l_p	Center of hydrodynamic pressure
R_F	Frictional resistance
R_P	Pressure resistance
Rn	Reynolds number
S	Sinkage
V_l	Local velocity
V_f	Freestream velocity
V_B	Bottom velocity
β	Deadrise angle
θ	Dynamic trim angle
λ_W	Mean length-to-beam ratio
ν	Kinematic viscosity

ysis techniques of planing hulls referring to many numerical and experimental studies in the literature.

Planing hulls operate at high Froude numbers where the effects of cavitation are likely to occur and disturb the flow. In this study, cavitation effects were not included in the numerical approach. Boundary element methods are handy to solve the flow around cavitating bodies. Bal has selected works on cavitating bodies inside the fluid at high speeds. A numerical model is given in [1] for cavitating hydrofoils. Very high-speed surface-piercing hydrofoils were investigated in [2].

This study covers a thorough discussion on an example planing hull and its resistance components with experimentally, numerically and empirically generated results. The results obtained numerically by implementing overset grid and rigid body motion system in the fluid domain were compared with experiments and Savitsky's empirical approach. Shortcomings of the empirical approach and numerical methods were explained with respect to the experimental data. After validation of numerical methods with a benchmark Fridsma hull, the resistance characteristics of the example planing hull were investigated. Underlying physics of the planing regime were tried to be revealed with the help of computational visualization.

2. Experimental and numerical approach

2.1. Hydrostatics of the hull

A 1/9 scaled model of a prototype hull with a length of 13.374 m hull was numerically and experimentally investigated in this study. Different views of the model can be seen in Fig. 1. The hydrostatic and geometric properties of the hull are given in Table 1.

2.2. Experimental procedure

All towing tests of the model planing hull were performed in the Ata Nutku Ship Model Laboratory of Istanbul Technical University (ITU). The capabilities of the towing tank and its main dimensions are given in Table 2.

Table 1
Hydrostatic and geometric properties of the model planing hull.

	Model Scale: $\lambda = 1/9$		
Length between perpendiculars	L_{BP}	m	1.4860
Maximum breadth	B	m	0.4405
Draft	T	m	0.0844
Displacement volume	∇	m^3	0.0234
Wetted area	S	m^2	0.5718
Block coefficient	C_B	–	0.4170
Longitudinal center of gravity	LCG	m	0.5390
Kinematic viscosity	ν	m^2/s	$1.202 * 10^{-6}$
Density	ρ	kg/m^3	999.6

The towing tank facility has a manned carriage that speeds up to 6 m/s manually. It is equipped with a force dynamometer to measure resistance (X and Y force) and also a computer with some connection equipment for data acquisition.

Tests were performed in calm water and effect of wind resistance is included but not separately calculated in the experiments. For trim and sinkage measurements the model was towed with two free degrees of freedom (2DOF), namely heave and pitch. A 2DOF force dynamometer was mounted between the model and tow post. The tow post was attached to the model at its longitudinal centre of gravity. Values of sinkage and dynamic trim angle were tracked with the help of a laser range finder. All data signals were acquired using a data logger at a certain sample rate and saved on a laptop.

The planing hull was manufactured from wood and scaled by 1/9 to model. The model was towed in calm water at speeds ranging from 0.7 to 5.5 m/s. At the end of the runs, beaches in the towing tank were manually lowered to calm the water. Approximately 15 min of waiting time between two consecutive runs was obligatory to dampen the waves generated during the experiments. The waiting time might be extended up to 1 h to make sure that there were no reflecting waves from the side walls of the tank. Special attention was paid to calibration and misalignment of the dynamometer. The tests were repeated at least 3 times. In spite of all these precautions taken to guarantee high quality test results; if there was a mismatch in integral variables such as resistance, sinkage or trim between the sets, the experiments were repeated 3 more times at least.

2.3. Numerical implementation

A commercial software Star CCM+ was used in this study to model the hydrodynamics of the planing hull. An implicit unsteady solver was selected implementing URANS with k–ε turbulence model. The two-phase flow involving air and water was solved using the Volume of Fluid (VOF) approach that tracks the free surface boundary. The dynamic fluid-body interaction (DFBI) model in the code was activated to have 2DOF for the hull. The planing vessel was free to heave and pitch as in experiments. Time step size was selected according to the instructions set by (ITTC 7.5-03-02-03). The time step sizes used in numerical calculations at each Fr are given in Fig. 2.

Ship motions in the fluid domain were represented using rigid body motion and overset grid systems. The grid was discretized with respect to CFL condition for the highest Froude number and

Table 2
Properties of the Ata Nutku Ship Model Laboratory in ITU.

Length of the channel	m	160
Width of the channel	m	6
Depth of the channel	m	3.4
Max towing speed	m/s	6
Max model length	m	5