Engineering Analysis with Boundary Elements 63 (2016) 15–29

Contents lists available at ScienceDirect

Engineering Analysis with Boundary Elements

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

A three-dimensional vortex method for the hydrodynamic solution of planing cambered dihedral surfaces

Stefano Brizzolara, Giuliano Vernengo*

MIT i-Ship Lab. (Innovative Ship Design Laboratory) Massachusetts Institute of Technology, Cambridge, MA, USA

ARTICLE INFO

Article history:
Received 9 June 2015
Received in revised form
7 September 2015
Accepted 8 October 2015
Available online 16 November 2015

Keywords:
Vortex Lattice Method (VLM)
Boundary Element Method (BEM)
Hydrodynamic Performance Analysis
High Speed Planing Hulls
Cambered Dihedral Planing Hulls
Dynamic Lift and Induced Resistance

ABSTRACT

A new numerical approach based on the Vortex Lattice Method (VLM) for the solution of the hydrodynamic performances of cambered hulls in steady planing is formulated and validated. Due to its fully 3D formulation, the method can be applied to both cambered and un-cambered dihedral planing surfaces of any shape without any further approximation. The exact three-dimensional wetted surface of the hull is where the body boundary condition is fulfilled. The sprays region detaching both in front of the stagnation root line and from the wet portion of the chine are modeled in the numerical scheme by means of additional vortex lattice regions. The dynamic boundary condition at the stern of the hull is non-linear with respect to the perturbation potential. Results show the dynamic pressure consistently accounts for the 3D features of the flow especially in the case of cambered planing surfaces. The numerical method is verified by a systematic analysis against semi-empirical methods and it is finally validated with experimental results on prismatic as well as cambered dihedral planing surfaces. Excellent correlations are found for both types of planing surfaces that range in the same confidence interval of higher fidelity numerical models, such as RANSE solvers.

1. Introduction

While the basic problem of the hydrodynamic design of prismatic planing surfaces has been solved since more than half a century with the elegant semi-empirical method of Savitsky [26,27], very few theoretical methods have been focused on the effect of longitudinal and transversal camber, which are usually considered as an empirical correction to the hydrodynamic characteristics calculated for prismatic surfaces [1]. New opportunities to study the effects of camber may derive from parametric investigation studies by means of CFD methods. In this respect, two categories of numerical methods, corresponding to two different fundamental theoretical approaches, which imply different levels of approximation and related computational times, can be considered to solve the steady hydrodynamic flow on planing surfaces: potential flow or free-surface Reynolds Averaged Navier–Stokes solvers. We like to refer to the first class of methods as *medium fidelity* and to the second as *higher fidelity*, reserving the connotation of *lower fidelity* models to quasi-3D or 2D+t methods.

A new research project focused on the hydrodynamic design of very high-speed, deep-V, stepped planing hulls with cambered bottom created the opportunity to initiate the development of a new medium fidelity CFD code, PLAMIT. The code, first presented in [35] is based on a Boundary Element Method, whose theoretical formulation is given in full detail in this paper and it is complemented by more extensive validation cases. The proposed numerical method solves the hydrodynamic steady free surface potential flow around 3D planing hulls to eventually get estimates of hydrodynamic forces and moments and use them for parametric design by optimization of planing hulls with cambered bottom.

If from one side, numerical RANSE solvers are a valid option for the performance analysis of planing hulls [6,30,5], on the other, they are not the right numerical tool to be integrated into parametric design optimizations when longitudinal and transverse camber distributions are considered.

The way we envisage to study these effects is based on global optimization methods using 3D full parametric description of the planing hull surface [23,24] and a medium fidelity CFD tool. This technique initially applied with success to SWATH hull forms [3,4], has been more recently extended to high-speed displacement ships [22] and to fast unconventional multihull hydrodynamic design [36] at the MIT-iShip.

With particular regards to planing hulls, different numerical models have been developed with various level of approximation of the numerical scheme used to compute the solution and to impose the physical boundary conditions, especially at the side and at the leading edge of the planing surface.

---

* Corresponding author.
E-mail address: gverneng@mit.edu (G. Vernengo).

http://dx.doi.org/10.1016/j.enganabound.2015.10.008
0955-7997/

Nomenclature

U_∞	modulus of free stream velocity
τ	trim angle of the hull
d	transom depth
L_K	wetted keel length
L_C	wettedchine length
L_M	wettedchine length, $\frac{(L_K+L_C)}{2}$
B	hull beam
λ	mean wetted length-beam ratio, $\frac{L_M}{B}$
β	deadrise angle
τ	trim angle
C_V	speed coefficient, $\frac{U_\infty}{\sqrt{gB}}$
Fn_∇	Froude number relative to the volume, $\frac{U_\infty}{\sqrt{g\nabla^{\frac{1}{3}}}}$
Fn_L	Froude number relative to the length, $\frac{U_\infty}{\sqrt{gL}}$
Φ	total velocity potential
ϕ	perturbation velocity potential
u, v, w	perturbation velocity components
ζ	function of the hull surface
δ	spray root line angle in the {x,y} plane
$\overrightarrow{V}_{tot}$	total velocity vector projected in the {x,y} plane
p_T	relative pressure at transom
V_T	(total) velocity at transom
p_w	relative pressure in the far field wake
V_w	(total) velocity in the far field wake
ΔX_L	longitudinal distance between two consecutive vortex ring centers
ΔV_y	local lateral velocity at the *wet chine* of the hull
$\overrightarrow{V}_i$	velocity induced by a vortex ring
$\overrightarrow{v}_i$	velocity induced by a vortex filament
$\overrightarrow{v}_t$	self-induced tangential velocity of a vortex ring
A_i	area of the ith vortex ring
Γ	circulation strength of a vortex filament
$\overrightarrow{n}$	normal vector to a vortex ring in its centroid
$\overrightarrow{F}$	force vector acting on a vortex ring centroid
G	circulation strength of a vortex ring
C_L	lift coefficient, $\frac{L}{\frac{1}{2}\rho U_\infty^2 B^2}$
	C_{Di} induced drag coefficient, $\frac{D_i}{\frac{1}{2}\rho U_\infty^2 B^2}$
C_p	pressure coefficient, $\frac{P}{\frac{1}{2}\rho U_\infty^2}$
	C_{Df} frictional drag coefficient
A_{Tot}	area of the wet bottom hull surface
$\overrightarrow{V}_{tot,i}$	total velocity on the ith vortex ring centroid
$\overrightarrow{V}_{mean}$	mean velocity on the wet bottom hull surface

The developments of 3D methods can be traced back to first works of Doctors [10,11] based on higher order pressure patch finite elements. This method has been recently re-appraised and extended in [38]. A quasi-3D method, based on a distribution of point sources on longitudinal strips along the undisturbed free surface has been proposed in [19] and applied to different types of planing surfaces including warped hard chine hulls and negative dihedral [20]. Neumann–Kelvin free surface boundary condition is applied aside the wet chine as well as in front of the stagnation root line. Although this is evidently not a fully 3D method, it has the advantage of determining as part of the solution the position of the stagnation line. Another interesting quasi-3D method is thorough extension of the original 2D impact vortex method of Vorus [37] to 3D proposed in [25] that has been effectively validated on different types of low aspect ratio planing surfaces. A three dimensional BEM coupled with a two dimensional boundary layer technique for friction drag and some empirical corrections has been used to assess the performance of planing flat plates and various prismatic planing hulls with constant and variable deadrise in [15].

The idea to treat the hydrodynamic planing in analogy with an aerodynamic lifting surface was systematically introduced by [8]. From a numerical point of view, Lai and Troesch were the first to proposed a fully 3D linear Vortex Lattice Method in [16] and [17]; in their formulation, the problem has been desingularized moving the vorticity on the projected plan form surface and boundary conditions have been linearized at the stagnation root line, the wet chine region and the dry transom. They show validation of the method only on prismatic planing hulls, not considering the effect of camber.

This paper intends to contribute to this last class of methods, proposing a new and effective way to collocate non-linear boundary conditions on the exact 3D hull surface where vortex singularities are also distributed, eventually obtaining improved accuracy in the prediction of the effect of camber on the steady hydrodynamic characteristics of planing surfaces. The focus of this study in on the analysis of the three dimensional effects produced by a variable camber on the pressure distribution on the hull, so the position of the leading edge line of the pressure area under the hull is assumed, leaving its numerical prediction to further studies. This in the intent of preserving the coherence of the proposed three dimensional VLM when applied to different cases, while avoiding further approximations that may arise from the introduction of low-fidelity techniques to predict the position of the stagnation line as in [25,17]. This way, when the numeric results are compared with those from empirical approaches the stagnation root line is consistently taken from the latter formulation [26], while for the comparison with towing tank tests, the pressure area observed in the experiments is considered in the computation.

The paper is structured as follows: a brief description of the physics of the fluid flow around planing hulls is done in Section 2. We then detail the theoretical model in Section 3, focusing on the formulation of the boundary conditions for the specific 3D problem. In Section 4 the numerical solution by VLM is described and the discretization technique used for the boundary conditions is detailed. The verification and validation of the numerical method by comparison with different experimental results of *V*-shaped planing surfaces, both with and without camber, is presented in Section 5.

2. Physics of the planing hull hydrodynamic

A hard chine hull running at constant forward speed in calm water reaches the purely planing regime when its weight is supported by a large portion by the dynamic lift generated by the pressure distribution over its bottom wetted surface (i.e. the *pressure area*). For opportunely design planing hulls [1] this happens at sufficiently high Froude numbers, typically $Fn_\nabla > 3.0$ and $Fn_\mathrm{L} > 1.0$. Considering only half of the hull, the *pressure area* is bounded within four edges, indicated in Italic in Fig. 1: the symmetry plane, the transom at the aftermost port of the hull (i.e. the trailing edge), the chine wet (i.e. the side edge) and the so-called *spray root line* (*SRL*) at the foremost location of the *pressure area* (i.e. the leading edge). The steady fluid domain around such a lifting body can be divided into four major zones too, numbered in Fig. 1: