Applied Ocean Research 59 (2016) 451–458

Contents lists available at ScienceDirect

Applied Ocean Research

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

Analysis of interaction between ship bottom air cavity and boundary layer

E.L. Amromin*

Mechmath LLC, Federal Way, WA 98003, USA

ARTICLE INFO

Article history:
Received 23 August 2015
Received in revised form 4 February 2016
Accepted 14 March 2016

Keywords:
Ship air bottom cavity
Boundary layer
Air demand
Multi-phase compressible flow

ABSTRACT

The successful designs of hulls for ships employing drag reduction by air bottom cavitation have been based on solutions of inverse problems of the theory of ideal incompressible fluid. However, prediction of the drag reduction ratio, the air demand by ventilated cavities and the cavity impact on the hull–propeller interaction is impossible in the framework of this theory because all mentioned characteristics depend on interaction of air cavities with the ship boundary layers. Because the known CFD tools are not fitted to ventilated cavitation at low Froude numbers, an analysis of this interaction requires a novel flow model. This model includes the incompressible air flow in the ventilated cavity, the compressible flow of a water–air mixture in the boundary layer on cavities and downstream of them and the curl-free incompressible outer water flow. The provided 2D computations employing this model allows for explanations of the earlier observed effects and for prediction of the air demand by ventilated cavities. The computed velocity profiles downstream of cavities are in the accordance with the available experimental data.

1. Introduction

Air bottom cavitation has been proven as the effective ship drag reduction technology since 1960s. The drag reduction rates up to 20% were reported by Butuzov et al. [1] for river ships and barges. Later the power saving rate close to 25% were predicted by Amromin et al. [2] and Gorbachev et al. [3] on the basis of towing tank tests for models of various sea ships with air bottom cavities. Sverchkov [4] and Amromin and Gorbachev [5] provided more detail on these successes. The interest to this technology has recurred during the last decade in various countries and the experimental results were reported also by Foeth [6], Kopriva et al. [7], Lay et al. [8], Mäkiharju et al. [9], Zverkhovskyi et al. [10].

This technology became successful because the special design of ship bottoms allowed for simultaneous elimination of the wall friction under the air cavity and suppression of the cavity tail pulsations. The proven methods of the successful ship hull design or the hull retrofits to this technology are based on solving inverse problems of ideal fluid theory similar to the linear problem considered by Butuzov [11] a half century ago. However, the effects related to interaction of the ventilated cavities with the ship boundary layers were left behind solutions of these problems, though some of these effects are significant and very important practically.

First, the drag reduction and power saving rates cannot be predicted in the framework of ideal fluid theory. This theory can satisfactory predict the shape of ventilated cavities (like done by Choi and Chahine [12]), but these rates are not directly proportional to the ratio of the hull surface area covered by the cavity to the total hull wetted surface area. In particular, these rates were lower in the experiment [10], but they were higher in the experiment [2].

Second, the friction reduction by the cavity and an air escape from the cavity affects the thicknesses of ship boundary layers and wakes, as well as the velocity profiles across them. The profile changes influence the propeller inflow field and should be taken into account during design of ship propellers.

Third, there are substantial scale effects on air demand by partial ventilated cavities. The power saving rate evidently depends on this demand. As was reported in [1] for very small Fr and later confirmed in [2] for much higher Fr, there is a saturation of drag reduction by ventilated cavitation with an increase of air supply and an excessive air supply gives even negative results. However, as seen in Fig. 1 combining experimental data of Arndt et al. [13], Amromin et al. [2] and Mäkiharju et al. [9], the common trend of the air demand as a function of Fr for various tests does not exist even for $Fr < 1$ (the shapes of models the tested in [2,9,13] are shown in Figs. 2–4). Moreover, as reported in [13], the ventilated cavities under small models can be maintained at small Reynolds numbers even without any air supply.

Further, as one may find in the paper of Kawakami and Arndt [14], i.e., it looks impossible to derive a dependency of Q on

---

* Tel.: ++1 952 402 9642.
E-mail address: Amromin@aol.com

http://dx.doi.org/10.1016/j.apor.2016.03.009
0141-1187/

Nomenclature

B	ship (model) beam
C_D	drag coefficient
$c=\sqrt{dP/d\rho}$	sound speed
$Fr=U_\infty/\sqrt{gL}$	Froude number
g	gravity acceleration
H	cavity thickness
L	ship (model) length
Lc	cavity length
M	Mach number
N	normal to S^*
P_c	pressure in cavity
P_∞	ambient pressure
Q	volumetric air demand by cavity
q	$10^4 Q/LBU_\infty$
$Re=LU_\infty/\nu$	Reynolds number
$r(\beta)=\beta\varepsilon+1-\beta$	mixture normalized (dimensionless) density
S^*	combination of all boundaries of inviscid flow
Sc	the part of S^* under the cavity
$U=\lvert grad\Phi\rvert/U_\infty$	velocity of inviscid flow
U_∞	ship (model) speed
u	velocity in the boundary layer
u_0	velocity on the cavity surface
u_a, v_a	air flow velocity components
v^*	friction velocity
X_2	abscissa of the cavity end
v^*	friction velocity
y	lateral coordinate in boundary layer
z_C	vertical coordinate of the cavity surface normalized by L
β	void fraction in the boundary layer
δ	thickness of water boundary layer
$\delta^*=\int_0^\delta (1-ru/U)dy$	displacement thickness
$\delta^{**}=\int_0^\delta ru/U(1-u/U)dy$	momentum thickness
$\delta_a^*=\int_0^H (1-u_a/u_0)dz$	the airflow displacement thicknesses
ε	the ratio of air density to water density
ρ	water density
ν	water kinematical viscosity
$\sigma=2(P_\infty-P_c)/\rho U_\infty^2$	cavitation number
Φ	velocity potential

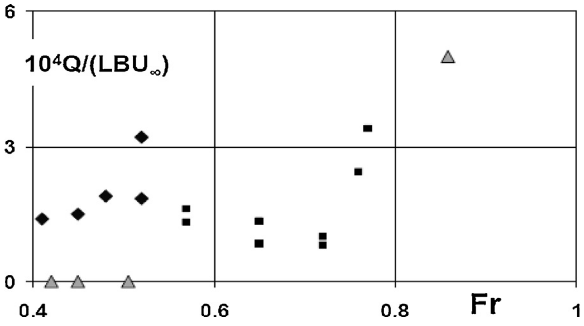

Fig. 1. Air demand to maintain cavities under a 12.9 m 2D model in a water tunnel (squares, from [9]), a 0.5 m 2D model in another water tunnel (triangles, from [13]) and a 4.6 m ship model towed in a tank at *Fr*-depended pitch (rhombs, from [2]).

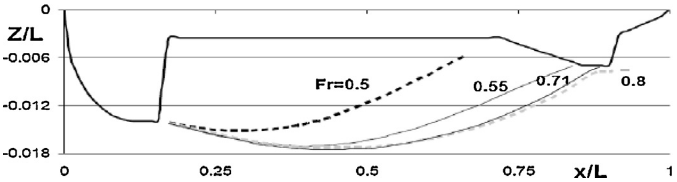

Fig. 2. Computed sections of the water tunnel model with cavities tested in [9] at various *Fr*.

cavitation number from experimental data even for the same body. So, the known experimental data on interaction of ventilated cavities with boundary layers are insufficient for their empirical generalization and there is the practical necessity of a numerical analysis of the interaction of ventilated cavities with boundary layers over the already designed hull.

Kinzel et al. [15] numerically manifested a high impact of turbulent mixing on air demand by ventilated supercavitating flows at moderate *Re* and $Fr \gg 1$ and the necessity to include a boundary layer impact in the analysis of ventilated cavitation became clear. One may think that such analysis is a simple task because the numerous computational studies on natural cavitation in viscous fluids have been already described in the literature. Indeed the flow models employed in these studies are not suitable for the air ventilated cavitation. First of all, they did not give a satisfactory description of air escape through the cavity surface. One group of these computational studies has been carried out using various versions of viscous–inviscid interaction methods; this group neglected by the gas flows within the cavities and its escape from them (as in Brewer and Kinnas [16] or in Amromin [17]). Another group of computational studies has been carried out using for the whole flow the model of a bubble cloud. Such model was suggested by Kubota et al. [18] for the cavity closure zone and there is no cavity surface in this model, though one can clearly see such surface in photos of ventilated cavities (as by Kawakami and Arndt [14]) and no bubbly clouds inside ventilated cavities are seen there. Further, all computational studies are based on semi-empirical approaches

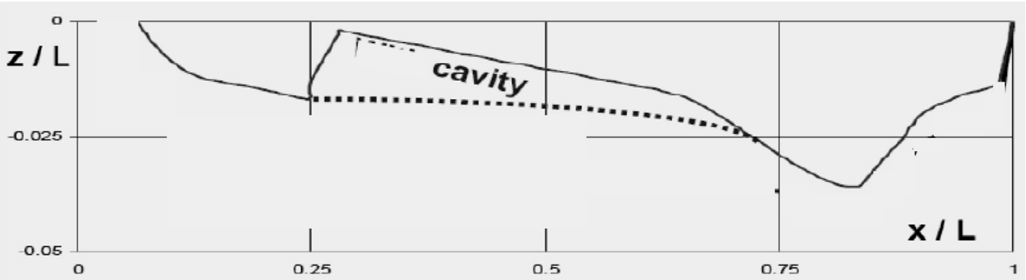

Fig. 3. Sections of the ship model towed [2] with computed cavity at *Fr* = 0.48 (at the model measured pitch).