Applied Ocean Research 50 (2015) 163–172

Contents lists available at ScienceDirect

Applied Ocean Research

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

Ships with ventilated cavitation in seaways and active flow control

E.L. Amromin

Mechmath LLC, Federal Way, WA 98003, USA

ARTICLE INFO

Article history:
Received 13 August 2014
Received in revised form
19 December 2014
Accepted 24 January 2015
Available online 19 February 2015

Keywords:
Hydrodynamics
Waves
Ship drag reduction
Active flow control

ABSTRACT

Bottom ventilated cavitation has been proven as a very effective drag reduction technology for river ships and planning boats. The ability of this technology to withstand the sea wave impact usual for seagoing ships depends on the ship bottom shape and could be enhanced by some active flow control devices. Therefore, there is the need in numerical tools to estimate the effects of bottom changes and to design such devices. The fundamentals of active flow control for the ship bottom ventilated cavitation are considered here on the basis of a special model of cavitating flows. This model takes into account the air compressibility in the cavity, as well as the multi-frequency nature of the incoming flow in wavy seas and of the cavity response on perturbations by incoming flow. The numerical method corresponding to this model was developed and widely manifested with an example of a ship model tested in a towing tank at Froude numbers between 0.4 and 0.7.

The impact of waves in head seas and following seas on cavities has been studied in the range of wavelengths from 0.45 to 1.2 of the model (or ship) length. An oscillating cavitator-spoiler was considered as the flow controlling devices in this study. The oscillation magnitude and the phase shift between cavitator oscillation and the incoming waves have been varied to determine the best flow control parameters. The main results of the provided computational analysis include oscillations of cavity surface, of the pressure in cavity and of the moment of hydrodynamic load on the cavitator. The major part of computations has been carried out for the flap oscillating at the frequency coinciding with the wave frequency, but the effect of a frequency shift is also analyzed.

1. Introduction

Drag reduction by bottom ventilated cavitation is a technology well-proven for several river ships operating at small *Fr* (as was first noted by Butuzov et al. [1], further by Latorre [2], and recently described in more detail by Gorbachev and Amromin [3]), for numerous fast planning boats (listed by Sverchkov [4]) and for various ship models, including designed for $Fr \approx 0.5$ (like described by Choi et al. [5] and by Amromin et al. [6]). The achieved rates of calm water drag reduction (shown in Fig. 1) are impressive, though for the Fr_D close to 1.0 the full-scale rates should be somewhat lower (due to higher fraction of wave resistance in the total drag scale effect of friction will affect this rate there). However, an expansion of this technology to seagoing ships depends on the ability to keep drag reduction by in wavy seas. The recent towing tank results [6] (presented in Fig. 2) manifested such ability. The drag reduction rates for Figs. 1 and 2 were calculated by comparison of experimental data for hulls with cavities in the niche and for the baseline hulls without niches). It was found that the cavities maintained in a deep bottom recess can mitigate the wave impact at quite high sea states.

Nevertheless, the unsuccessful attempts to use this drag reduction technology were also reported. Therefore many engineers (like Foeth [7], Mäkiharju et al. [8], Zverkovski et al. [9]) recognized that design of ship with bottom ventilated cavitation should not be based on intuitive ideas, but requires the profound comprehension of fundamentals. Such comprehension can be achieved on the basis of consideration of elementary flows allowing, however, for estimation of the effects of bottom shape changes and the capacities of eventual flow control devices at the stage of the ship design.

Two substantial flow singularities must be comprehended by engineers analyzing ship ventilated cavitation. First, cavitation number is not the governing parameter of ventilated cavitation. Generally, shapes of large cavities (which length is much larger than the local boundary layer thickness) depend mainly on interaction of three forces: Water inertia (lengthening the cavity); pressure difference between incoming flow and the cavity (compressing or extending the cavity) and gas buoyancy (deflecting up the cavity tail). The flow governing parameters are two ratios of these forces named as cavitation number and Froude number. In particular, at *Fr* values inherent to the majority of ships, just *Fr* becomes the most influential parameter of cavitating flows, whereas σ plays a secondary role (unlike to cavitating flows around propellers or missiles). Second, though the classical ideal fluid model has allowed for the successful design of ships with bottom cavities for calm

E-mail address: amromin@aol.com

0141-1187/$ – see front matter © 2015 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.apor.2015.01.017

Nomenclature

A	wave amplitude
B	ship beam
c	sound speed
D	displacement
dz	sinkage
Fr	Froude number
Fr_D	$Fr \cdot D^{-1/6}L^{1/2}$
H	cavity thickness
h	perturbation of cavity thickness
h^*	$\lvert h(lc) \rvert$
$J = \rho_0 c^2/\rho_{water} U_\infty^2$	parameter describing the cavity compressibility under the wave impact
L	ship length
l	cavitator length
$ho(t)$	cavitator deflection
lc	cavity length
P_C	pressure in cavity
P_∞	ambient pressure
Q	air supply to cavity
Q^*	variation of the air supply to the cavity
Sc	surface of unperturbed cavity
U_∞	ship speed
V	volume of unperturbed cavity
$x = Xo$	location of cavitator edge
α	pitch
λ	the wavelength
$\sigma = 2(P_\infty - P_C)/\rho_{water} U_\infty^2$	cavitation number
ρ'	the dimensionless perturbation of air density normalized by its initial value ρ_0
$\tau = tU_\infty/L$	dimensionless time
$\Phi^*(S,t)$	potential of wave-induced perturbations
Ψ	phase shift between wave and cavitator oscillations
Ω	oscillation frequency of cavitator
ω	wave frequency
$\bar{\omega}$	$\equiv \omega L/U_\infty$

sea conditions, this model does not consider the gas flow within the cavity and the cavity compressibility under wave impact. As a result, such model is insufficient for analyzing the cavity behavior and its possible collapse in waves. So, this model must be corrected.

The necessary corrections were already briefly described in [6]. Here this corrected theory will be applied to the analysis of the wave impact on a ship with the bottom cavity in head and following seas. The aim of this analysis is to determine the ship ability to keep drag reduction by cavitation in waves, as well as the possibility to enhance this ability by an active flow control.

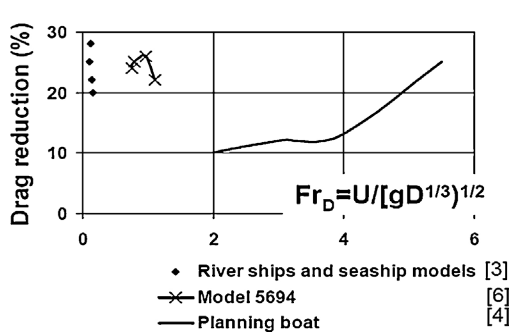

Fig. 1. The measured ratios of calm water drag reduction by bottom ventilated cavitation for ships and ship models.

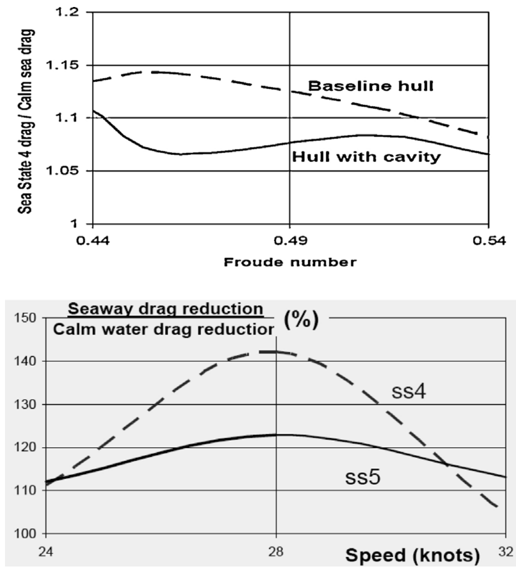

Fig. 2. Measured [6] wave impact on drag of ship model 5694 with cavity and on her baseline hull in sea state 4 (top) and percentage of drag reduction by cavitation kept in head sea states 4 and 5 by such ship of 90 m length (bottom).

The provided analysis directly relates to ships of medium size operating at medium *Fr* because of the following reasons: First, as shown by model tests in Krylov Ship Research Institute (partially reported in [3]), even sea state 5 does not cut the drag reduction of very large ships operating at small *Fr*. Second, for medium size ships operating at medium *Fr* (like ferries), even sea state 4 may be an issue. Third, there are already model test data to validate the developed computational model for cavitation under bottoms of such ships.

2. Influence of air compressibility on unsteady ventilated cavitation

The described prediction of sea state impact on ship drag reduction by cavitation is based on the numerical study of 2D cavity-wave interaction within framework of a quasi-linear perturbation theory. The ventilated cavitating flow is considered as a combination of two interacting flows. The first one is incompressible water flow out of the cavity. The second one is compressible air (or air–water mixture) flow within the cavity. The sketch of the whole flow is given in Fig. 3.

There is no the explicit analysis of hull boundary layer, though the compressible flow within the cavity is assumed depending on the air entrainment by the cavity boundary layer. This flow also depends on the air escape through the cavity tail pulsating under the wave impact, on the air supply to the cavity from a compressor and on the concentration of water drops pushed to the cavity by the reentrant jet from the cavity tail. Because $\omega \ll c/lc$, it is possible to assume that the same time-dependent air pressure is instantly

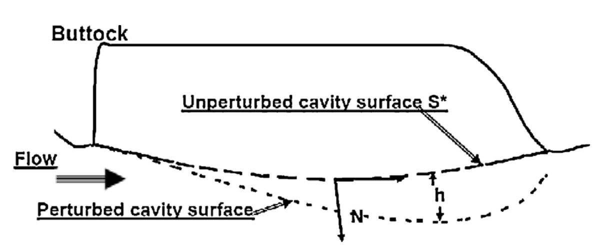

Fig. 3. Sketch of a ship buttock with a cavity in a niche; h is counted along the normal N to unperturbed cavity surface Sc.