Ocean Engineering 141 (2017) 143–153

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical analysis of ventilated cavity flow over a 2-D wall mounted fence

Luka Barbaca*, Bryce W. Pearce, Paul A. Brandner

Australian Maritime College, University of Tasmania, Launceston, Tasmania 7250, Australia

ARTICLE INFO

Keywords:
Ventilated cavity
Wall mounted fence
CFD

ABSTRACT

Ventilated cavity flow over a 2-D wall mounted fence is numerically investigated using a viscous approach. An implicit unsteady compressible solver was used with a RANS $k-\omega$ SST turbulence model and VOF approach to capture the cavity interface. The simulations were carried out for a fixed fence height based Froude number and constant outlet pressure. Cavity topology, wall pressure distributions and the resulting hydrodynamic forces were determined as a function of ventilation rate, degree of fence immersion in the oncoming wall boundary layer and degree of confinement of the flow domain. It was found that with an increase in ventilation rate, lift increases and drag decreases resulting in a greater hydrodynamic efficiency (lift to drag ratio) of the fence-wall system. With increase in immersion of the fence in the boundary layer, both lift and drag decreased, while the lift to drag ratio increased. Variation in the degree of confinement had a large influence on the flow, with the reduction in lift and hydrodynamic efficiency observed for the more confined conditions.

1. Introduction

Ventilated (also termed 'artificial') cavities can be utilized for drag reduction in marine applications. Drag can be decreased by forming an air bubble/layer between the solid surface and water to reduce the skin friction (Ceccio, 2010), or by increasing the pressure on the downstream surface of the cavitating body to reduce the form drag (Franc and Michel, 2004). The main parameter used to characterize these flows is the cavitation number, $\sigma_c = (p_\infty - p_c)/0.5\rho U_\infty^2$, where p_∞ is the reference free-stream pressure, p_c is the pressure inside the cavity, ρ is the liquid density and U_∞ is the reference free-stream velocity. Similar cavities can be formed naturally (vapour filled) when the liquid is subjected to vapour pressure (p_v), with in this case $\sigma_c = \sigma_v = (p_\infty - p_v)/0.5\rho U_\infty^2$. It has been shown that, for the same σ_c value, natural and ventilated cavities exhibit comparable behaviour except for the differences in closure physics (May, 1975; Kunz et al., 1999).

To sustain a cavity long enough to be applicable for drag reduction purposes, σ_c has to be in the order of 0.1 (Kawakami and Arndt, 2011). As p_v is small, to naturally achieve such a low σ_c in a practical flow, a high free-stream velocity (i.e. >100 knots) is required. In the ventilated cavity case p_c is controlled by the flux of injected air, and such low σ_c values can be achieved independent of free-stream conditions. Consequently, devices utilising ventilation can be efficiently used at lower operating velocities and/or higher free-stream pressures (i.e. deeper submersion), leading to a much broader range of potential application.

Depending on the location of the cavity closure, a cavity is classified as either a 'partial' cavity or a 'supercavity'. The former is defined where the closure region is located on the surface of the body and if the closure is rather downstream in the wake it is termed a supercavity (Franc and Michel, 2004). The application of both ventilated partial and supercavitation to high-speed underwater bodies has been of particular interest post the second world war for military applications (Reichardt, 1946; Waid, 1957), but also there has been some interest in a commercial context (Brentjes, 1962).

Past studies into ventilated flows have mainly focused on the reduction of skin friction in hydrodynamic applications. There has been extensive research into the use of axisymmetric ventilated cavities for drag reduction of underwater projectiles in the second half of last century, with an ongoing interest in the topic. A review of basic physical properties and calculation methods for axisymetric ventilated cavities is given by Semenenko (2002). The other application of substantial interest has been in the use of ventilated partial cavities on the underwater part of ship hulls, referred as 'air-lubrication'. The injection of air is used to create a stable cavity for ships operating in the range of speeds that are not sufficient to enable detachment of a natural cavity from a geometric discontinuity in the hull. To date several semi-displacement and planning boats using this phenomenon have been built, with a reported resistance decrease in the range of 10–30% (Latorre, 1997; Butuzov et al., 1999; Matveev et al., 2009). Additionally, some work has been done on implementing air-lubrication system on full displacement ships in a commercial context (Mizokami et al., 2010; Surveyor, 2011). A comprehensive overview

* Corresponding author.
E-mail address: Luka.Barbaca@utas.edu.au (L. Barbaca).

http://dx.doi.org/10.1016/j.oceaneng.2017.06.018
Received 12 May 2016; Received in revised form 15 February 2017; Accepted 7 June 2017

of the work done in this field is presented in a review article by Ceccio (2010). Kopriva et al. (2008) proposed the use of ventilation on low-drag partially cavitating hydrofoils with a smooth cavity reattachment (Amromin et al., 2003) for application on high-speed vessels.

Alternatively, ventilation has been also applied as a form drag reduction technique. Based on this technique a class of lifting surfaces, described as base-ventilated hydrofoils, has been developed. These hydrofoils have a wedge shaped profile, with air injected through a thick trailing edge and a cavity detaching from the trailing edge geometric discontinuities. Such foils can be utilized on hydrofoil supported boats or for propulsion by ventilated propellers (Lang and Daybell, 1961; Huang, 1965; Verron and Michel, 1984; Franc and Michel, 2004). More recently, a novel concept of an intercepted base-ventilated hydrofoil for ride control of a high-speed craft is proposed by Elms (1999). A related topic of the flow around a retractable fence fitted to the transom of a marine craft, also known as an 'interceptor' (Faltinsen, 1996), can be observed as a special case of an 'infinite' ventilated cavity. Some numerical investigations on interceptor flows are reported by Brizzolara (2003) and Molini and Brizzolara (2005).

Within the scope of the present study ventilated cavity flow over a 2-D wall mounted fence is investigated. This study is a continuation of the research on cavitating flow over a wall mounted fence that has been performed at the Australian Maritime College (AMC) Cavitation Research Laboratory (CRL). To date, a several reports on numerical (Pearce et al., 2010; Pearce and Brandner, 2014; Barbaca et al., 2014) and experimental (Barbaca et al., 2016) work have been published. The interest in this flow is based on the potential use of a retractable fence ('interceptor') attached to the trailing edge of a base-ventilated hydrofoil for a rapid generation of bi-directional lift with a minimum drag penalty. This topic has been of an ongoing interest at the AMC with a numerical (Pearce and Brandner, 2015a) and preliminary experimental (Pearce and Brandner, 2012a, 2012b) studies reported.

The main objective of the present work is to investigate the effect of fence immersion within the oncoming wall boundary layer on the flow. As the fence is attached to a flat wall (i.e. isolated from the foil), these results are also relevant for flow over an interceptor attached to a ship hull (either used for lift generation, or ventilated cavity drag reduction). Furthermore, the case studied is a canonical cavitating flow (with nominal zero streamwise pressure gradient) of basic interest and suitable for comparison of experiment and computation, which has not previously been addressed in the published literature.

Cavity topology and resulting hydrodynamic forces are examined with respect to the flux of injected air, immersion of the fence in the upstream wall boundary layer and the degree of confinement of the flow. The results are compared with potential flow predictions obtained using a Boundary Element Method (Pearce and Brandner, 2014) and analytical predictions from free-streamline theory (Brennen, 1995). Some comparison with the experimental results from the cases with similar geometries is provided. The present numerical investigations will be complemented with a future experimental study in a cavitation tunnel.

2. Numerical modelling

A schematic representation of the ventilated cavity flow over a wall mounted fence of interest here is shown in Fig. 1. A fence of a height h is immersed in the oncoming wall boundary layer of thickness δ. δ is defined as the distance from the wall where U is 99% of the free-stream velocity U_∞. Air is supplied to the wake region of the flow through the downstream face of the fence. A ventilated cavity with re-entrant jet closure is shown detaching from the fence tip at the angle β to horizontal axis. The resulting pressure signature on the wall upstream of the fence is indicated.

In the case of ventilated cavities, for constant free-stream conditions, the cavitation number, σ_c, is controlled by the air flow rate. Ventilated air flow data is presented in terms of a volumetric flow rate

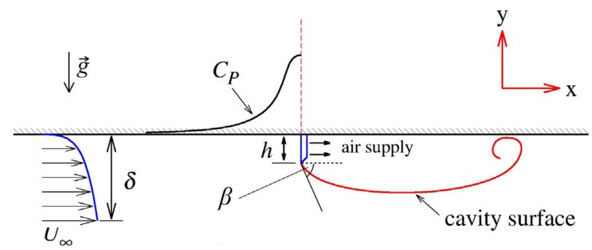

Fig. 1. Sketch of a wall mounted fence immersed in the oncoming wall boundary layer with a ventilated cavity detaching from the sharp fence tip. Air is supplied from the downstream face of the fence. The pressure distribution on the upstream wall is shown. The origin of the coordinate system is at the fence/wall junction.

coefficient, $C_{Qv} = Q_m/\rho_{air} U_\infty S$, where Q_m is the air mass flow rate, ρ_{air} is the air density and S is the reference area (fence height multiplied by unit width). For the consideration of the effect of the fence immersion in the wall boundary layer an additional dimensionless parameter is defined, the boundary layer thickness to fence height ratio, δ/h. Influence of the confinement is evaluated through the use of the dimensionless ratio between domain and fence height (H/h). A Froude number based on fence height, $Fr = U_\infty/\sqrt{gh}$, where g is gravitational acceleration, is also applicable.

Commercial Computational Fluid Dynamics (CFD) software, CD Adapco STAR-CCM+, was used in the present study. For later comparison with experimental results a rectangular computational domain of finite height was chosen for the numerical analysis (Fig. 2). The domain height, H, was varied between 40 h and 200 h, and the fence, modelled as 10 mm high and 0.1 mm thick, was attached to the upper domain boundary. The domain inlet and outlet were positioned sufficiently away from the fence to minimise their influence on the flow, located 500 h and 1000 h respectively. For spatial discretization, a structured hexahedral mesh with prism layer cells in the boundary layer region was used. To resolve the flow a first order implicit unsteady finite volume method was employed. Water was defined with constant density and air as an ideal compressible gas. The interface between the phases was captured using a VOF method based on the volume fraction equation (Hirt and Nichols, 1981). Surface tension and gravity effects were included in the model. For the consideration of viscous effects a RANS approach with the SST (Menter) $k-\omega$ turbulence model was used.

Water enters the domain through the constant velocity inlet, with the velocity set to 10 m/s (Fr=31.93) for all cases. The flow rate of the air injected through the downstream face of the fence was varied between 0.01 kg/s and 0.09 kg/s (giving $0.084 < C_{Qv} < 0.751$), except for the cases with the thinnest boundary layer where additional air was needed for the cavity to achieve the 'infinite' length. The domain outlet was defined as a constant pressure outlet, with the pressure set to 100 kPa for all cases (giving a free-stream cavitation number, based on vapour pressure, $\sigma_v \cong 1.95$). A range of unperturbed boundary layer thicknesses at the fence position (x=0) was achieved by changing the length of the no-slip wall condition (l_{ns}) upstream of the fence (Fig. 2). The part of the wall upstream of l_{ns} was prescribed as a free-slip wall with no boundary layer present. With this technique, δ/h values between 0 and 10 were achieved for a constant fence height and constant free-stream conditions. For investigating the effect of blockage four domains of different height H were created, giving confinement ratios, H/h, ranging between 40 and 200. The reference pressure (p_∞) was defined as the minimum pressure value on the tunnel wall upstream of the fence. Existence of minimum pressure value is due to the boundary layer growth imposed pressure signature. The reference free-stream velocity (U_∞) was chosen as the value on the domain centreline, 100 h upstream of the fence.

A convergence analysis was undertaken for both temporal and spatial discretization. Time steps ranging between 0.5 and 2 ms were analysed and a time step of 1 ms gave results within 1% of the