Ocean Engineering 136 (2017) 304–313

Contents lists available at ScienceDirect

Ocean Engineering

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

A numerical assessment of the interaction of a supercavitating flow with a gas jet

Michael P. Kinzel[a,*], Michael H. Krane[a], Ivan N. Kirschner[b], Michael J. Moeny[a]

[a] Applied Research Laboratory, The Pennsylvania State University, 214 GTWT Blg., University Park, PA 16802, USA
[b] Applied Physical Sciences Corporation, Groton, CT, USA

ARTICLE INFO

Keywords:
Supercavitation
Jet

ABSTRACT

In this work, the interaction between a ventilated supercavity and a jet are examined using computational fluid dynamics (CFD). CFD results compare favorably to experimental data describing bulk cavity behavior. These validated models are used to develop a number of novel insights into the physical characteristics of the interaction. These interactions are described by: (1) the jet ventilation gas appears to dominate the gas attached to the cavity shear layer, (2) the jet appears to cause additional gas leakage by transitioning the cavity from a recirculating flow to an axial flow, (3) the jet creates more slender cavities, and (4) with sufficient momentum, the jet invokes wake instabilities that drive cavity pulsation.

1. Introduction

This article describes the interaction between a ventilated cavity and a gas jet, both originating from the same body. Isolated supercavities forming downstream of a blunt body (or cavitator) are reasonably well understood. In general, these flows are comprised of three regions: (1) the outer liquid flow that behaves like a nearly inviscid fluid flow, (2) the cavity itself, comprised of a low temperature, weakly compressible, turbulent gas flow that surrounds the body, and, (3) the cavity closure region, which can assume one of several well-known topologies (typically twin-vortex or toroidal vortex closure patterns) (Campbell and Hilborne, 1958; Epshtein, 1973). The effect of this jet on the behavior of an established cavity has received far less attention.

In the absence of a jet, ventilated cavity flows are described by several non-dimensional numbers: the ventilation flow rate, $C_Q=Q/D_N^2 V_\infty$, Froude number, $Fr_N = V_\infty/\sqrt{gD_N}$, and cavitation number, $\sigma_c = (p_\infty - p_c)/(1/2\rho_\infty V_\infty^2)$ (Epshtein, 1973; May, 1975). In these relations, Q is the gas ventilation rate, D_N is the cavitator diameter, V_∞, is the free-stream velocity, g is gravity, p_∞ is the free-stream pressure, p_c is the mean pressure in the supercavity, and ρ_∞ is the free-stream liquid density. The basic cavity dimensions, i.e., cavity length (L_c) and cavity diameter (D_c), inversely relate to σ_c (Campbell and Hilborne, 1958; Epshtein, 1973; May, 1975; Semenenko, 2001). One such correlation, of many options, yields a cavity aspect ratio given by

$$\frac{L_C}{D_c} = \frac{A}{\sqrt{k}}\frac{1}{\sqrt{\sigma_c}}, \tag{1}$$

where A and k are empirical constants (Semenenko, 2001). This correlation indicates that the cavity size increases as σ_c decreases. In addition, for steady conditions, C_Q is equivalent to the rate air escapes the supercavity (referred to as air entrainment rate in supercavitation).

The physics of isolated supercavities has been examined experimentally, theoretically, and numerically. Experimental efforts (Lee et al., 2008; May, 1975; Stinebring et al., 2001; Wosnik et al., 2003, and others) are ideal to characterize the visually accessible aspects of the flow. In supercavitation, visual access can sometimes be a challenge. Theoretical models are well established, largely based on the work of Epshtein (1973), and form the basis for semi-empirical theory. An excellent summary of such work is presented in Semenenko (2001). A more advanced modeling framework includes potential flow models (Kirschner et al., 2001), which require submodels that patch in unresolved physics such as air entrainment. Computational fluid dynamics (CFD) based on the numerical solution to the Reynolds averaged Naiver Stokes (RANS) has also been applied to supercavitation. The method has had success in both accurately simulating supercavitation conditions (Kinzel et al., 2007; Kunz et al., 2000; Lindau et al., 2003) and yielding insight into the flow physics, especially regarding air entrainment processes (Kinzel et al., 2009). This CFD methodology is ideal to develop a physical understanding, and to bridge potential flow methods and semi-empirical methods.

Abbreviations: CFD, computational fluid dynamics; DDES, delayed-detached Eddy simulation; LES, large eddy simulation; RANS, Reynolds-averaged Navier stokes

* Corresponding author.
E-mail address: mpk176@psu.edu (M.P. Kinzel).

http://dx.doi.org/10.1016/j.oceaneng.2017.03.042
Received 12 August 2016; Received in revised form 18 March 2017; Accepted 20 March 2017
Available online 23 March 2017

Current understanding regarding how a gas jet affects a ventilated cavity is summarized in the semi-empirical theory of Paryshev (2006), who summarized the jet-supercavity interaction using two additional non-dimensional parameters. The first is the non-dimensional jet momentum flux at the jet nozzle exit: $\overline{J} = \dot{m}_{Jet} V_{Jet}/D$. Note that D is the drag on the cavitator, i.e., the nose on the front of the vehicle from which the cavity initiates. The second, $P = \rho_\infty V_\infty^2/(\rho_{jet} V_{Jet}^2)$, is the ratio of stagnation pressure in the liquid flow exterior to the cavity, to that in the gas jet, at the jet nozzle exit. P is a measure of the tendency of the jet to penetrate the cavity wall in the closure region. Paryshev (2006) identified three cavity behavior regimes in terms of these parameters. The first regime (Efros, 1946) occurs when $P < 1$. In this case, the jet stagnation pressure exceeds the exterior liquid flow pressure and the jet penetrates the cavity such that air entrainment rates are significantly increased no matter the jet momentum $\overline{J}$. The other two regimes occur for $P > 1$, where the effect of the jet on cavity size is further mediated by $\overline{J}$. For low jet momentum (which Paryshev termed a "soft" jet), jet mass flux supplements that of the ventilation gas used to establish the cavity, increasing cavity size proportionally to jet momentum. For high jet momentum (a "hard" jet), the jet interaction with the closure region is more complex, with the result that the rate at which air leaves the cavity in increased, though far less than when $P < 1$. In this regime, the cavity size *decreases* in proportion to jet momentum. Paryshev's momentum-theory-based analysis suggested that the break point between these regimes is $\overline{J} = 1/2$. Comparison to limited data appeared to confirm these trends. Paryshev's model thus identified the principle regimes of jet-cavity interaction. However, it does not explicitly account for the dynamics of the gas jet, in particular diffusive processes that govern jet entrainment of cavity gas, or that would decrease the local stagnation point of the gas jet flow relative to that at the nozzle exit. Simulations that incorporate these effects will thus more precisely determine the bounds of the regimes Paryshev has identified.

In this paper, the jet-supercavity interaction is modeled using a high-fidelity CFD method to help improved the understanding of the physical characteristics in this interaction. One main objective is to evaluate the modeling developed by Paryshev (2006) and evaluate the correctness of the physics, and direct future efforts to refine the model. The paper begins by comparing the results of a CFD model with experiment to verify and validate the model. Using the validated model, the results are interrogated against the hypothesized flow characteristics of Paryshev. The physical flow characteristics are presented and discussed, which refines the insight into the interaction.

2. Methods

2.1. Numerical Method

In the present work, simulations are modeled using the commercial CFD code, Star-CCM+ (CD-Adapco, 2014). In this work, a homogeneous, Eulerian-based multiphase-modeling approach is used. The method conserves the mixture mass and momentum, whereas an isothermal assumption is made such that we can neglect the energy equation. The phase mass for the gases are conserved using a volume-of-fluid-like formulation. In this model, we use two gases, one for the ventilation gas, and a second for the jet gas. Sharp interfaces are maintained between the gas and liquid phases using the HRIC scheme (Muzaferija et al., 1998). HRIC is not used for the gas-gas discretization, and the diffusion between the gas phases is not explicitly modeled. The CFD numerical formulation is a pressure-based, segregated-flow model based on the SIMPLE-C scheme. The method is not based on a conservative form in its multiphase flow formulation, thus for this compressible multiphase flow, some model errors are expected and assumed to be relatively small. There are three-fluid-phases included in this model. The first is an incompressible liquid phase for the water (ρ_l=1000kg/m^3, μ_l=0.001$Pa\,s$, $c = 4181 J/(KgK)$), where this incompressible assumption should be reasonable based on gas jets expelled into liquid (Fronzeo and Kinzel, 2016). The ventilation and jet gases assume an isothermal compressible gas model. Both the jet gas and ventilation gas use air properties ($\gamma = 1.4$, μ_l=1.85 $\times 10^{-5}$$Pa\,s$, c_P=1003$J/(kg - K)$). Note that two gases are modeled to help understand how the jet gas interacts with the supercavity. The numerical scheme is formally 2nd-order accurate in space and time. In terms of the turbulence simulation, a hybrid Reynolds Averaged Navier-Stokes (RANS)/Large Eddy Simulation (LES) turbulence model is used. The model is based on the Spalart-Allmaras Delayed-Detached Eddy Simulation (DDES) formulation (Shur et al., 2008). The usage of such a DDES approach is based on previous work that suggests such formulations provide reasonable accuracy in the interaction of a gas-liquid interface with slip (Kinzel et al., 2009, 2007).

2.2. CFD model geometry

A validation effort is performed using a recent test campaign of cavity-jet interactions. Tests were performed in the 1.22 m Garfield Thomas Water Tunnel at The Pennsylvania State University - Applied Research Laboratory (Moeny et al., 2015). The model consists of a sting-mounted, 30°, 31.75 mm diameter, conical-shape cavitator shown in the blow-up in Fig. 1. Just aft of the cavitator is a ventilation port that is used to ventilate the supercavity. Although not depicted in Fig. 1, a gas-deflector was used in the experiment to minimize gas-jet impingement on the cavity interface. Aft of the deflector is a pseudo body that is used to support the gas-jet assembly. The jet forms from a converging nozzle that is fed gas through a dedicated line. Mass flow and thrust are altered by varying the total pressure in the supply line and varying the nozzle diameter. The experiments are performed by initially establishing a supercavity (of roughly the same σ_c) and then activating the gas jet to observe the interaction. A video of showing the procedure and interaction from the experiment is available online for reference (Krane et al., 2015). The interaction is measured both visually (using high-speed video) and via a pressure measurement on the body to provide a direct measurement of σ_c. Additional details of the experiment are available in (Moeny et al., 2015).

2.3. Computational mesh description

The numerical domain is intended to approximate the water tunnel facility. The computational domain can be visualized in Fig. 1. The strut-mounted body in the tunnel is modeled to represent the experiment. In Fig. 1, the 11 mm diameter nozzle configuration is shown. For

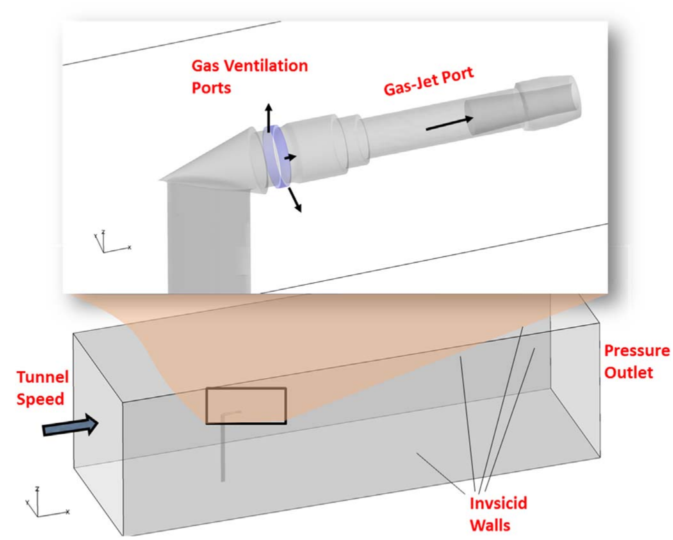

Fig. 1. Diagram of the computational domain used to represent the experimental setup.