Experimental Thermal and Fluid Science 64 (2015) 186–195

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

Investigations into the turbulent bubbly wake of a ventilated hydrofoil: Moving toward improved turbine aeration techniques

Ashish Karn [a,b], Christopher Ellis [a], Jiarong Hong [a,b], Roger E.A. Arndt [a,*]

[a] Saint Anthony Falls Laboratory, 2 3rd AVE SE, University of Minnesota, Minneapolis, MN 55414, USA
[b] Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55414, USA

ARTICLE INFO

Article history:
Received 20 October 2014
Received in revised form 17 December 2014
Accepted 20 December 2014
Available online 13 January 2015

Keywords:
Hydroturbine aeration
Turbulent bubbly wake
Ventilated hydrofoil wake
Auto venting turbines
Particle shadow velocimetry

ABSTRACT

The use of aerating hydroturbines to mitigate the problem of low dissolved oxygen in the discharge of hydroelectric power plants has recently attracted a lot of attention. The design of a ventilated hydroturbine requires a precise understanding of the dependence of the operating conditions (viz. liquid velocity, air ventilation rate, hydrofoil configuration, etc.) on the bubble size distribution generated in the bubbly wake and the consequent rise in dissolved oxygen in the downstream water. In the current research, experiments are conducted in the wake of a ventilated NACA0015 hydrofoil by systematic variation of hydrodynamic conditions allowing for quantitative analysis of aeration statistics and capabilities for turbine blade hydrofoil designs. The data concerning bubble velocity distributions, bubble locations and size distribution, void fraction, etc. are reported for a chosen reference case. In addition, trends in the variation of bubbly wake are explored particularly in the light of wake physics. It is found out that an increase in Reynolds number (Re) led to greater breakup, while an increase in normalized air ventilation rate (C_Q) favored greater coalescence events in the wake. Further, the PDF($\tilde{d}$) of the normalized bubble size $\tilde{d} = d/d_{32}$, where d_{32} represents Sauter mean size distribution, is found to have a universally similar shape independent of either Re, C_Q or hydrofoil angle of attack. Finally, a numerical formulation is proposed for the bubble sizes in the hydrofoil wake. This rich dataset will also contribute to the development of a numerical turbulence model, to investigate turbulence effects on bubble size distribution and predict the rate of air entrainment and the oxygen transfer occurring in the wake at different hydrodynamic conditions.

1. Introduction

Over the past few decades, environmental concerns have broadly affected the electric power industry, including the depletion of fossil fuel supplies, the potential of global climatic changes and challenges involved in long-term nuclear waste management [1]. In response to such concerns, the electric power industry has increasingly focused attention on renewable non-polluting energy technologies such as hydroelectric power. Hydroelectric power generation is clean, inexpensive and reliable and does not significantly pollute the land, air or water. However, environmental problems resulting from the discharge of water with low dissolved oxygen (DO) levels are a concern at many hydroelectric facilities. The problem of low DO is well documented in the literature [2] and a variety of methods have been suggested to provide dissolved oxygen [3]. The low DO water when released to the river downstream can adversely affect the aquatic habitat and contribute to other water quality problems viz. dissolution of trace metals, formation of hydrogen sulfide and depression of pH [4].

One of the most attractive techniques currently being investigated to mitigate low DO while maintaining operation efficiency, is the use of an auto-venting turbine (AVT). AVT is a self-aspirating hydroturbine designed to aerate the turbine discharge through ports located at low pressure regions which are open to the atmosphere. When air is allowed to flow through these ports, it breaks down into small bubbles by the high velocity and turbulence of the water flow through the turbine. DO transfer is augmented by the high interfacial area of these bubbles and the level of turbulence in the water [5].

To maximize the DO transfer in the flow through AVT, the knowledge of the bubble sizes of the entrained air at different locations in the bubbly wake is extremely important. The size of bubbles in the wake is a result of different bubble breakup and coalescence processes which consequently affects the gas transfer at different downstream positions. Ultimately, it is desired to have

* Corresponding author.
E-mail address: arndt001@umn.edu (R.E.A. Arndt).

http://dx.doi.org/10.1016/j.expthermflusci.2014.12.011
0894-1777/

a numerical turbulence model to investigate turbulence effects on bubble size distribution and predict the rate of air entrainment and the oxygen transfer occurring in the wake at different hydrodynamic conditions. The accurate prediction of bubble sizes in the wake at a given flow field is imperative for the prediction of oxygen transfer occurring in the wake. However, both for the operation and validation of such a computational model, there is a desperate need for a high quality experimental dataset that can provide information on bubble sizes and void fraction in the wake, pressure at the air entrainment location and the rate of oxygen transfer. To the best of our knowledge such relevant experimental reports on the bubbly wakes are scarce.

Thus, in the current study, a series of ventilation experiments are conducted at several hydrodynamic conditions at the Saint Anthony Falls Laboratory (SAFL) water tunnel, allowing for quantitative analysis of aeration statistics and capabilities for turbine blade hydrofoil designs. SAFL has pioneered the study of bubble wakes [6,7] and has several hydrofoil designs specifically modified for ventilation studies. In the present study, a systematic investigation into a bubbly wake is being carried out at multiple hydraulic operating conditions such as different air ventilation rates, liquid velocities and hydrofoil configuration to serve as a test-bed for computational turbine aeration programs. The data concerning bubble velocity distributions, bubble locations and size distribution, void fraction, etc. are obtained. In addition, some simple numerical formulations are proposed for the bubble sizes in the hydrofoil wake.

This paper is structured as follows: Section 2 provides the details of the experimental facility, and the experimental methodology employed. The obtained results are presented in Section 3 along with a brief discussion which is followed by a final conclusion in Section 4.

2. Description of experimental setup

2.1. Experimental apparatus

The experiments are conducted in the SAFL high-speed water tunnel at University of Minnesota. The tunnel has a horizontal test section of 1 m (length) × 0.19 m (width) × 0.19 m (height) with three sides having Plexiglas walls for optical access. The tunnel is designed for cavitation and air ventilation studies and is capable of operating at a maximum velocity of 20 m/s. A gas-collector dome in the tunnel provides for fast removal of large quantities of air bubbles generated during cavitation and ventilation experiments, enabling bubbly flow experiments for extended periods with little effect on test section conditions.

2.2. Experimental conditions

The NACA0015 hydrofoil used in our experiments has a span of 190 mm and a chord length of 81 mm. As shown in Fig. 1, a narrow spanwise slot allows air to be injected into the flow near the leading edge of the hydrofoil and the full width of the injection slot is used for measurements of oxygen uptake. This setup results in a dense spanwise bubbly wake and thus, in order to make bubble measurements, ventilation is limited to a narrow 9.6 mm wide slot (5% of span) at the center of the span. This configuration ensures that bubbles remain mostly within a narrow distance away from the centerline. The findings of Silberman [8] indicated that the bubble diameter is independent of the orifice characteristics for a gas jet coming out of an orifice into a cross flow. Also, the bottom views of the bubbly wake show a minimal lateral drift of the bubbles. Based on our observation of the bottom views of bubbly wake, the study of Silberman and the fact that slot width remained unchanged in our experiments, we posit that the bubbles obtained are a reasonable representative sample of the bubble population that exists when the full span is ventilated.

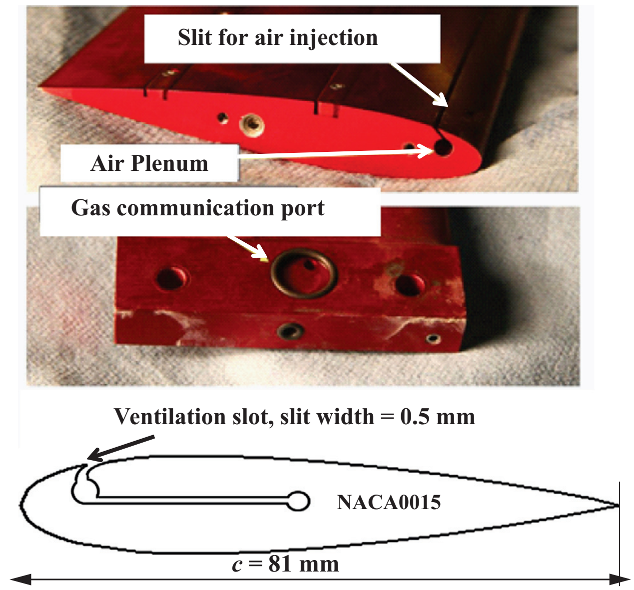

Fig. 1. Details of the NACA0015 ventilated foil (adapted in part from [10]).

To systematically investigate the physical processes occurring in the bubbly wake, an air-entrainment coefficient is defined similar to other ventilated cavitation studies viz. Laali and Michel [9], as $C_Q = Q/UcS$, where Q, U, S, and c denote ventilation air flow-rate, liquid velocity, hydrofoil span and chord, respectively. Reynolds number is defined using hydrofoil chord as length scale: $Re = Uc/\nu$. To study the effect of the experimental parameters on the resulting flow physics in the bubbly wake, three different sets of experiments are performed. In the first set of experiments, C_Q and Re are fixed ($C_Q = 1.6 \times 10^{-4}$, $Re = 4.1 \times 10^5$) and experiments are repeated at three different angles of attack ($AoA = 0$, 4 and 8 deg.). The subsequent experiments are all done at a fixed zero degree angle of attack. In the second set of experiments, a fixed value of $C_Q = 1.6 \times 10^{-4}$ was chosen and Re spanned four different values from 2.4×10^5 to 8.1×10^5. Finally, in the third set of experiments, Re was kept fixed at 4.1×10^5 and experiments were conducted at four other values of C_Q varying between 1.1×10^{-4} and 3.3×10^{-4}. Thus, these three sets of experiments comprised of eleven different experimental conditions, which were imaged at three different downstream locations in the wake ($x/c = 1.34$, 3 and 4.65), resulting in 33 experimental datasets and a total number of 66,000 bubble images captured using a shadow imaging technique.

2.3. Particle shadow velocimetry

Particle Shadow Velocimetry technique (PSV) employs direct in-line volume illumination using low power sources such as LED and an optical setup to produce a narrow depth-of-field for 2D plane imaging [11,12]. Fig. 2 shows a schematic of the experimental setup as has been reported previously by Karn et al. [10]. A 1 K × 1 K pixel Photron APX-RS camera (capable of 3000 frames/s at full sensor size) with a 60 mm telephoto lens is used to acquire images. A custom-made pulsed LED light source from Innovative Scientific Solutions Inc. is used to illuminate the flow, with a pulsed LED array having flash rates up to 10 kHz with a 5 μs pulse width and rise and fall times around 200 ns. To ensure uniform backlighting and to eliminate noise in the images, a light shaping diffuser is placed between the light source and the flow.