Experimental Thermal and Fluid Science 70 (2016) 397–407

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

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

Bubble coalescence and breakup in turbulent bubbly wake of a ventilated hydrofoil

Ashish Karn [a,b], Siyao Shao [a,b], Roger E.A. Arndt [a], Jiarong Hong [a,b,*]

[a] Saint Anthony Falls Laboratory, 2 Third AVE SE, University of Minnesota, Minneapolis, MN 55414, USA
[b] Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55455, USA

ARTICLE INFO

Article history:
Received 19 April 2015
Received in revised form 15 July 2015
Accepted 2 October 2015
Available online 13 October 2015

Keywords:
Bubble coalescence
Bubble breakup
Bubbly wake
Auto-venting turbines
Hydroturbine aeration
Shadow image velocimetry

ABSTRACT

Auto-venting turbines have been proposed as a promising solution to the problem of low oxygen content in the discharged downstream water of an electric power plant. The current design of these turbines relies primarily on computational simulation. The experimental studies that focus on the physical processes occurring in turbulent bubbly wake are urgently needed to improve the performance of these simulations in predicting the bubble size distribution behind auto-venting turbines. Therefore, in the current study, we conducted detailed experimental investigations into the bubble size distributions in the wake of a ventilated hydrofoil. The mean bubble statistics is measured at different liquid velocities and air entrainment rates, and then the variation in mean bubble statistics is studied at different downstream locations in the wake. The bubble size distributions at different downstream locations have revealed the presence of distinct coalescence-dominant and breakup-dominant regimes. Analytical expressions are derived for the prediction of maximum stable diameter and Sauter mean diameter of bubbles, in the breakup and coalescence regimes, respectively. The observations from high speed imaging provide support for the measurements of bubble statistics, and physical insights into different mechanisms of bubble breakup and coalescence in turbulent wake. It is hoped that these insights will aid in developing generic model of bubble size distribution, and will help researchers improve bubbly flow simulations for auto-venting turbines.

1. Introduction

Hydroelectric power, although one of the major contributors to the renewable non-polluting energy source, has a potential disadvantage [1]. In the hydropower facilities, usually the penstocks feeding the turbines are installed deep in the reservoirs. At such large depth within the impoundments of water, the oxygen concentration is significantly low. This water, which is low in dissolved oxygen, when released, adversely affects the downstream ecosystem, including the aquatic flora and fauna [2]. As a result, increasingly stringent water quality standards have been posed on the hydropower plants to improve the dissolved oxygen (DO) levels in the downstream water. Such strict requirements have impelled the development of aeration techniques to replenish the oxygen concentration in the water.

Auto-venting turbines (AVT) have been proposed as a promising solution to the problem of low oxygen content in the discharged downstream water of an electric power plant [3,4]. In an AVT, air is drawn into the water at naturally occurring low pressure points in turbines leading to increased dissolved oxygen levels in the tailraces below hydroelectric dams. However, although the aerated hydroturbines improve the water quality, yet the increasing air ventilation in the turbine has a detrimental effect on the turbine efficiency due to subsequent hydraulic losses. Accordingly, the design and operation of such AVTs have to fulfill the twin objectives of increased DO levels, and minimal drop in turbine efficiency. Currently, much of the design of auto-venting hydroturbines is performed with computational fluid dynamics (CFD) simulations. The information available to test and verify the performance of these simulations as regards bubble size distribution is scarce. Also, these simulations are challenging since capturing the evolution of bubble size distribution in the wake accurately requires an understanding of the process of air entrainment and other physical processes occurring in the turbulent bubbly wake.

The air entrainment in such a hydroturbine can be carried out by opening a port connected to the low-pressure regions in the turbine. Evidently, when air is entrained into the wake, it breaks into individual bubbles because of the presence of turbulent velocity

* Corresponding author at: Department of Mechanical Engineering, University of Minnesota, Minneapolis, MN 55455, USA.
E-mail address: jhong@umn.edu (J. Hong).

http://dx.doi.org/10.1016/j.expthermflusci.2015.10.003
0894-1777/

fluctuations in the wake. Further, the collision of bubbles might also produce bubbles with increased size due to coalescence. In a turbulent and dispersed liquid flow, breakup and coalescence processes occur continuously and interplay between these two processes determines the bubble size distribution. Finally, bubbles attain a stable size due to the establishment of a local dynamic equilibrium between these two processes. However, the underlying mechanism in which the entrained air breaks into bubbles and the change of bubble size in the turbulent wake of a ventilated hydrofoil is not very clearly understood. Also, these processes will depend upon the liquid flow, air ventilation rate and the downstream distance from the turbine. The detailed understanding of the bubble coalescence and breakup events in the wake as a function of these parameters will aid to the development of physical models for bubble size distribution.

There have been numerous reports on the bubble breakup phenomena [5–8], and similarly some studies have exclusively focused on coalescence of bubbles [9,10]. Many previous studies have reported the occurrence of coalescence and breakup of bubbles or its effect on resulting bubble size distributions in different types of flow configurations such as air-sparged bubble columns [11–16], a two-dimensional packed bed [17], pipes [18–20], water jet [21] etc. However, to the best of our knowledge, there are hardly any studies conducted in the wake of a turbine hydrofoil blade. The flow physics in the bubbly wake of a hydroturbine is markedly different as compared to these other flow configurations often reported. Specifically, there are very few reports on the prediction of bubble size in a turbulent bubbly wake flow. The recent measurements in the turbulent bubbly wake and the oxygen transfer [1,2] have employed a ventilated hydrofoil in a water tunnel for simulating the flow around a hydroturbine. However, a detailed insight into the physical processes in the wake and its effect on the resulting bubble size has not been provided. Therefore, in the current study, we conducted a detailed investigation into the breakup and coalescence processes in the wake of a ventilated hydrofoil and also proposed a theory to predict the maximum stable bubble size for dilute dispersions in a turbulent bubbly wake.

This paper is structured as follows: Section 2 provides the details for the experimental facility, the setup, the optical approach used to capture digital images and the image analysis technique employed in our study. Subsequently in Section 3, we present results and discussion on bubble size distributions in the wake and offer the insights into the coalescence and breakup mechanism involved, which is followed by a final conclusion in Section 4.

2. Experimental methodology

2.1. Experimental setup

The experiments are conducted in the SAFL high-speed water tunnel at the 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. A NACA 0015 hydrofoil similar to the study of [1] was employed in our experiments. During the experiments, the hydrofoil is installed at zero angle of attack in the test-section as shown in Fig. 1. It has a span of 190 mm and a chord length (c) of 81 mm and a narrow spanwise slot that allows air to be injected into the flow near the leading edge of the hydrofoil, resulting in a narrow spanwise bubbly wake. The injection slot is 0.5 mm in thickness and the air exits the slot at an angle of 45° to the hydrofoil chord. In order to make bubble size measurements, ventilation is restricted to a narrow 9.6 mm wide slot (5% of span) at the center of the span, thus ensuring that bubbles remain mostly within a narrow distance away from the centerline. As discussed in Karn et al. [1], the bubbles obtained in the current experiment setup provides a reasonable representation of the bubble population when the full span of the insert is ventilated. The air flow rates in the test-section is controlled by a mass flow controller, with uncertainty being less than 1% of the measured value. Uncertainty in the measurement of test-section velocity was less than 2% of the measured value.

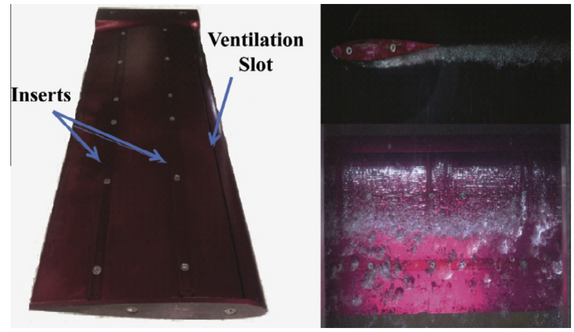

Fig. 1. Details of the NACA0015 ventilated foil. Left: photo of suction side of hydrofoil. Right: Sample image of ventilated flow. Side view and bottom view of suction side. Adapted from [2].

2.2. Experimental techniques

A Shadow Imaging Velocimetry (SIV) technique was used in our experiments, as has been reported previously by [1]. The SIV technique employs direct in-line volume illumination with a LED light source and an optical setup to produce a narrow depth-of-field for 2D plane imaging [22,23]. A pulsed LED light source from Innovative Scientific Solutions Inc. is used to illuminate the flow, which has flash rates up to 10 kHz with a 5 μs pulse width and rise and fall times around 200 ns. To ensure uniform back-lighting and to eliminate noise in the images, a light shaping diffuser is placed between the light source and the flow. A 1 K × 1 K pixel Photron APX-RS camera with a 60 mm focal length lens is used for image acquisition.

In the SIV technique, two LED light pulses separated by small time duration are synchronized with camera exposure in order to obtain two consecutive (or, double) images. In our experiments, the time duration between two pulses varied between 100 and 230 μs depending upon the free stream velocity. Using the image pairs, the instantaneous velocity field of the bubbles is obtained through a commercial software (*LaVision DaVis* 7.2), for Particle Image Velocimetry (PIV). The field of view (FOV) of the captured images is approximately 60 mm × 60 mm and such images are captured at five different downstream locations in the wake corresponding to x/c = 1.3, 2.1, 3.0, 3.7 and 4.7 as shown in Fig. 2. The imaging system is calibrated prior to the start of the experiments using a 2.5 mm × 2.5 mm calibration plate at the centerline of the test section. The use of the imaging lens in our experiments ensured that there are negligible distortions in our images, even at the edge of the FOV. Both pixel dimensions are 0.059 mm in all calibrated images. The image depth of field is determined to be approximately 15 mm at the sampling plane. The lateral spread in the bubble plume is measured at the measurement location and the calculated uncertainty in spanwise bubble location translates to a length scale (and calculated velocity) uncertainty of 1.6% near