Ocean Engineering 161 (2018) 168–186

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical study of liquid mixing in microalgae-farming tanks with baffles

Bang-Fuh Chen[a,*], Hung-Kai Yang[a], Chih-Hua Wu[a,b], Tzu-Chiang Lee[a], Bing Chen[c]

[a] Department of Marine Environment and Engineering, National Sun Yat-sen University, Kaohsiung, Taiwan
[b] Institute of High Performance Computing, Department of Fluid Dynamics, A*STAR, Singapore
[c] School of Ocean Science and Technology, Dalian University of Technology, China

ABSTRACT

One possible renewable energy source is microalgal biomass, representing sunlight-driven cell factories. Experimental studies have shown that a uniform mixture of microalgae is required for their access to sunlight. Generally, running paddlewheels for raceway ponds and mechanical pumps for photobioreactors are used, and their electricity consumption increases the costs of algae-mediated biodiesel production. To reduce the consumption of electricity, we developed the idea of using a floating automatic mix system based on the mixing nature of liquid sloshing in a baffled tank floating in the ocean. When oscillatory water waves induce liquid sloshing in the tank, vortices form and shedding is generated by horizontal baffles installed in the tank. Wave motions may be employed to enhance the mixing of microalgae in the tank and dramatically reduce the electricity consumption that is required in traditional algae-mediated biodiesel production. The vortex generation of sloshing liquids in a tank with horizontal baffles was numerically and experimentally investigated in this study. The effects of baffle location and length were systematically studied.

The study and application of liquid sloshing in a tank with baffles are usually used to tune liquid dampers for vibration control of a structure. The particles movement and mixing in the tank are seldom discussed. The simulation results found in this study introduce a new application for liquid sloshing in a tank with baffles. Appropriately allocated horizontal baffles in the tank can significantly enhance the mixing of liquid particles and can be applied in microalgae cultivation.

1. Introduction

Because of the gradual reduction in the use of fossil fuels, national energy policies in various countries are focused on expanding the use of energy efficiency and accelerating the development of renewable energy industries. One possible renewable energy source is microalgal biomass, representing sunlight-driven cell factories. Some of the advantages of microalgal biomass are outlined as follows: (1) The derived oil yield per unit area may exceed the oilseed crops; (2) less water is required, compared with ground crops; (3) herbicides or pesticides are not required; (4) the remainder is used as feed or fertilizer, or is fermented; and (5) direct biological fixation of carbon dioxide is executed. To reduce expenditure, biodiesel production must rely on freely provided sunlight, despite daily and seasonal changes in light levels. Experimental studies have shown that a uniform mixture of microalgae is required for their access to sunlight. Generally, running paddlewheels are used for raceway ponds, and mechanical pumps are used for photobioreactors, and their electricity consumption increases the costs of algae-mediated biodiesel production. In addition, the optimal temperature for algal production is from 20 °C to 30 °C. To maintain a cold temperature, the uses of bioreactors in raceway ponds and thermal regulation are always necessary. These facilities and the corresponding operations result in further electricity consumption. Therefore, the idea of a floating automatic mixing system was developed because of the mixing nature of liquid sloshing in a baffled tank when floating in the ocean. Liquid sloshing is produced when the tank is under oscillatory water waves. Vortices form and shedding are generated by horizontal baffles installed in the tank. Wave motions may be employed to enhance the mixing of microalgae in the tank and dramatically reduce the electricity consumption that is required in traditional algae-mediated biodiesel production.

Sloshing waves in tanks have been studied experimentally, analytically, and numerically in recent decades. A comprehensive review of early analytical and experimental studies of liquid sloshing for application in the aerospace industry was reported by Abramson (1966). If the interior of a tank is smooth and no wave breaking occurs, an inviscid/irrotational potential flow solution in combination with viscous boundary layer flow is suitable for describing sloshing (Faltinsen and Timokha, 2009). If the tank excitation is not too small and the excitation frequency is in the vicinity of the lowest natural frequencies, free-surface nonlinearities play a dominant role. Other potential flow analyses have been reported by Nakayama and Washizu

* Corresponding author.
E-mail address: chenbf@mail.nsysu.edu.tw (B.-F. Chen).

https://doi.org/10.1016/j.oceaneng.2018.04.088
Received 3 March 2017; Received in revised form 25 April 2018; Accepted 29 April 2018

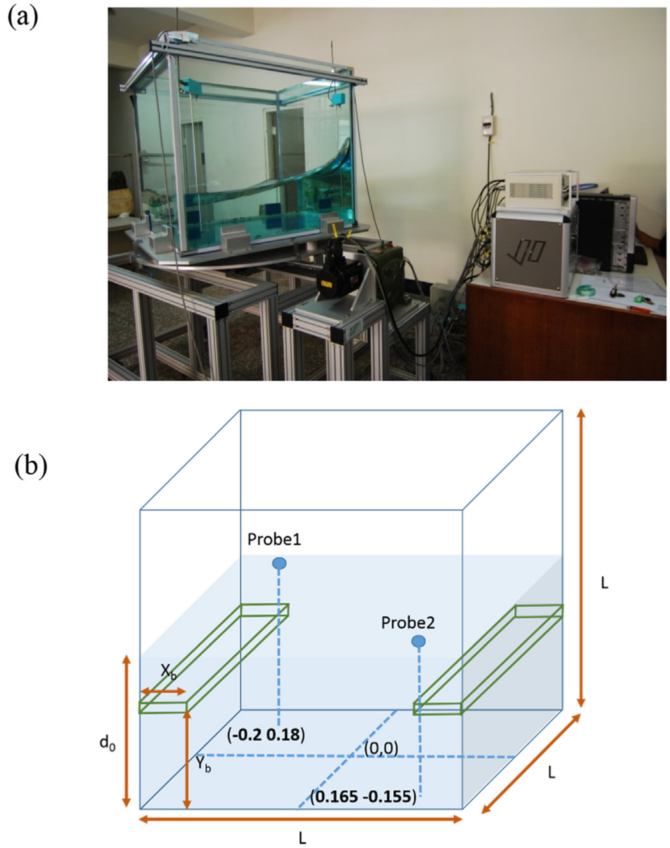

Fig. 1. (A) Photograph of the experimental setup; (b) The schematic diagram of the symmetric horizontal baffle configurations in the tank.

(1980) and Flipse et al. (1980). For a comprehensive review of methods and results of asymptotic sloshing analysis, readers are referred to Ibrahim (2005). Faltinsen and Timokha (2002) developed an adaptive multimodal approach to study nonlinear sloshing in a rectangular tank. Faltinsen et al. (2006) extended their asymptotic modal system to model nonlinear sloshing in a three-dimensional rectangular tank. In addition to potential flow approaches, multiple numerical studies of the problem using primitive variables have been made, particularly when the fully nonlinear effects of the waves on the free surface are included. Papers describing the modeling of two-dimensional sloshing include Chen and Chiang (1999), Aliabadi et al. (2003), Chen and Nokes (2005), Liu and Lin (2008), and Wu and Chen (2009, 2012).

In this study, a numerical baffled tank was established. The tank size is small and it can be designed as a series of self-buoyant floaters chained together and may ride over the wave. The sloshing liquid in a tank under a regular exciting motion takes 400 s to reach the steady state. However, it is not easy to reach the steady state sloshing when the tank is under irregular exciting motions, and the sloshing response in the transient stage is a little bit larger than that in the steady state (Wen et al., 2016). The results reported in the paper are, therefore, primarily those in the transient stage. However, we may understand the fundamental phenomena of the vortices generation and mix of the liquid sloshing in the tank with baffles according to the sloshing history in the transient stage. The software Fluent was used to investigate the effects of horizontal baffles on vortex generation in the tank. Single and dual baffles were included in the study. The effects of baffle positions and lengths were extensively studied. Experimental studies were also conducted to verify the numerical results. Section 2 briefly describes the numerical method used in this study, and the experimental setup is presented in Section 3. Section 4 describes the results and discussion, and the final section provides the conclusions of this study.

2. Numerical method

The research was conducted using the commercial code of ANSYS Fluent 6.3, which is a computational fluid dynamics (CFD) code. The flow was assumed to be incompressible with constant density and molecular viscosity. The Navier–Stokes equation and the continuity equation were solved to acquire the velocity and pressure fields. The continuity equation is

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

and the Navier–Stokes equations of a 3D incompressible flow can be written as

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\upsilon\nabla^2 u \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=-g-\frac{1}{\rho}\frac{\partial p}{\partial y}+\upsilon\nabla^2 v \tag{3}$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\upsilon\nabla^2 w \tag{4}$$

where t is the time and u, v, and w are the velocity components in the x, y, and z directions, respectively; p is the pressure, ρ is the fluid density, and υ is the kinematic viscous coefficient. The free surface was considered to be the air-liquid interface, and a pressure inlet with standard temperature and pressure was used to model the air-sea interface. Fluent uses the volume of fluid method (Hirt and Nichols, 1981) to trace and locate the instant sea–air interface and the interface between two liquid layers. The volume fraction of each fluid phase was tracked through every cell. In brief, the following equation was used:

$$\frac{\partial C_q}{\partial t}+u\frac{\partial C_q}{\partial x}+v\frac{\partial C_q}{\partial y}+w\frac{\partial C_q}{\partial z}=0 \tag{5}$$

where C_q is a fraction function used to define and calculate the volume ratio of q-phase fluid in the computational mesh. In the present study, $q=0$ for air, 1 for freshwater, and 0.5 for water-air interfaces. A staggered grid was used to define the pressure p at the mesh center, whereas the velocity components u, v, and w were $0.5\Delta X 0.5\Delta Y$, and $0.5\Delta Z$ behind, above, or backward of the cell center, respectively. The discretization techniques used to iteratively obtain the flow velocity and pressure field were the quadratic upwind interpolation for convective kinematics method and the pressure implicit with the splitting of operator algorithm. To improve the accuracy of the obtained results, the second-order upwind scheme was used. AutoCAD and Solidwork codes were used to construct the geometry of the computational domain and to store the results in *.sat or *.iges files, and then the Gambit code was used to establish computational meshes. The postprocessing of the output was conducted using Matlab to provide the plots and figures for this paper.

3. Experimental setup

Fig. 1 illustrates a schematic of the tank attached to a shaking table, which could be moved back and forth through various