International Journal of Heat and Mass Transfer 110 (2017) 262–275

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Computational study of multiphase flows over ventilated translating blades

Cosan Daskiran [a,b], I-Han Liu [a], Alparslan Oztekin [a,*]

[a] P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA 18015, USA
[b] Turkish Military Academy, Department of Mechanical Engineering, Ankara, Turkey

ARTICLE INFO

Article history:
Received 9 October 2016
Received in revised form 11 March 2017
Accepted 12 March 2017

Keywords:
Computational fluid dynamics
Large eddy simulations
Multiphase flows
Aeration
Oxygen dissolution

ABSTRACT

Computational fluid dynamics simulations are conducted to investigate dissolved oxygen characteristics in water. The aeration process is achieved by injecting air over the surfaces of submerged, translating blades. Mathematical model and numerical methods employed are validated by comparing predicted results against prior experimental measurements. Parametric study is conducted in a two-dimensional geometry by employing the Eulerian multiphase model with k-ω SST turbulence model to assess the significance of interfacial forces and the parameters affecting features of dissolved oxygen. Multiphase mixture model with LES turbulence model is employed to study oxygen dissolution in three-dimensional geometries. A single blade and an array of blades with free ends are considered. Aeration does not influence the drag coefficient of the single blade while it can have profound influence on drag forces acting on blades in an array configuration. While achieving aeration to improve water quality this study can aid in designing and optimizing river current energy harvesting devices consisting of translating blades.

1. Introduction

Water in rivers, lakes, reservoirs and underground aquifers are vitally important to everyday life of people and to ecological systems of the world. Water quality is a major issue for use of water for different purposes. The dissolved oxygen concentration in water is a critical indicator to determine the water quality. The low level of the oxygen concentration in water can be detrimental to aquatic life including bacteria, fish and plants. According to the criterion assigned by the United States Environmental Protection Agency the dissolved oxygen level in water should not be less than 3 mg/l for fish survival [1]. The present study considers aeration of water via energy harvesting devices for marine current applications.

Aerating water by injecting air from rectangular shaped translating blades is investigated here by conducting computational fluid dynamics simulations. Flow past a single blade and arrays of translating blades near free and rigid surfaces are studied by present researchers to design and optimize potential energy harvesting devices for river current applications [2,3]. Objective of the present study is two folds: (1) to determine how effective it is to aerate streaming water via energy harvesting devices (2) to determine the influence of aeration on the potential power generation from river current systems.

Previous numerical and experimental studies on multiphase flows including oxygen mass transfer from air to water have primarily focused on the prediction of bubble size distribution and dissolved oxygen concentration in the flow field. Several studies have considered bubble column reactors, stirred tanks and ventilated hydrofoils to observe the bubble size and oxygen absorption characteristics. Bubble column reactors are mostly used in chemical and biochemical industries for various applications such as a dissolution, a fermentation and a waste water treatment. Jia et al. [4] and Zhou et al. [5] performed computational fluid dynamics (CFD) simulations to predict the bubble size and the mass transfer in a bubble column. They validated their mathematical model and the numerical method by comparing predicted results against experimental measurements. The bubble diameter at a gas inlet boundary of a bubble column is studied experimentally by Akita and Yoshida [6]. They reported empirical correlation of the bubble diameter as a function of the inlet gas velocity and the orifice diameter.

Similarly, stirred tanks are used to improve the dissolution performance through the momentum mixing of one or multiple impellers. Kerdouss et al. [7] conducted CFD simulations for a double impeller stirred tank to obtain the gas volume fraction and the bubble diameter in the tank. Their predictions agree well with

* Corresponding author.
E-mail address: alo2@lehigh.edu (A. Oztekin).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.03.038
0017-9310/

Nomenclature

A	area, m^2
C	molar concentration, $kmol/m^3$
C_D	drag coefficient
$C_{D,b}$	bubble drag coefficient
d_S	Sauter mean bubble diameter, m
$d_{S,inlet}$	Sauter mean bubble diameter at inlet, m
d_h	hole diameter at sparger, m
D	diffusion coefficient, m^2/s
DO	dissolved oxygen, mg/l
F_D	drag force, N
F_{int}	gas-liquid interfacial forces, N
g	gravity, m/s^2
H	plate height, m
He	Henry constant, Pa
K	adjustable parameter
k_l	liquid side mass transfer coefficient, m/s
L	plate length, m
M_{O_2}	molar mass of oxygen, kg/kmol
M_{H_2O}	molar mass of water, kg/kmol
$\dot{m}_{gl}$	mass transfer from gas phase to liquid phase, kg/m^3 s
$\dot{m}_{lg}$	mass transfer from liquid phase to gas phase, kg/m^3 s
N	number of cells
P	partial pressure, Pa
p	pressure, Pa
$\dot{Q}$	flow rate, m^3/s
Re	Reynolds number
S	rate of the strain tensor, 1/s
SGS	sub-grid scale
Sc	Schmidt number
Si	sink term, 1/ms
So	source term, 1/ms
t	time, s
$\vec{U}$	phase velocity, m/s
$\vec{U}_{gl}$	interfacial velocity, m/s
$\vec{U}_{slip}$	relative velocity between phases, m/s
u_t	bubble fluctuation velocity, m/s
u_r	bubble terminal velocity, m/s
w	plate width, m
We	Weber number
$\times_i$	position vector
x	mass fraction
y^+	dimensionless wall distance

Greek symbols

α	volume fraction
α_{gmax}	dense packing limit of volume fraction
α_i	interfacial area concentration, 1/m
Δ	change in variable
λ	non-dimensional time
ε	permutation symbol
μ	dynamic viscosity, kg/ms
ν	kinematic viscosity, m^2/s
ρ	density, kg/m^3
σ	surface tension coefficient, N/m
ω	vorticity, 1/s
Ω	rotation tensor, 1/s
τ_{st}	stress-strain tensor, kg/ms^2

Superscript

$\sim$	normalized

Subscripts

cr	critical
dr	drift
g	gas phase
g, l	gas or liquid phase
i, j, k	tensor indices
l	liquid phase
min	minimum
max	maximum
O_2, eq	equilibrium property of oxygen in liquid phase
O_2, g	property of oxygen in gas phase
O_2, l	property of oxygen in liquid phase
T	turbulent
TI	turbulent impact
WE	wake entrainment
RC	random collision

experimental observations and numerical predictions. Smaller bubble size is observed around the impeller discharge as a result of a bubble breakup by small eddies induced by the impeller. CFD simulations of three-impeller agitators with a constant and a variable bubble diameter were carried out by Min et al. [8]. They concluded that a single bubble size assumption which ignores the bubble breakup and the coalescence is not capable to capture a local gas holdup in the stirred tank accurately. Their findings with the variable bubble diameter model are consistent with documented experimental measurements.

Recently, experimental analyses of a ventilated hydrofoil were performed in a closed loop water channel by Karn et al. [9–12] to investigate bubble size characteristics and the dissolved oxygen level. Karn et al. [9] proposed a dispersion theory for the bubble size prediction and obtained reasonably good agreement with experimental observations within low air injection rate in the breakup dominated region. They observed that the bubble size is influenced by breakup effects in the wake region within 2.1 times length of the chord and by coalescence effects in the far wake region. Karn et al. [11] studied the influence of the free stream velocity, the injected air flow rate, and the angle of attack of the hydrofoil on both the bubble size distribution and the oxygen mass transfer rate. They reported that Sauter mean bubble diameter depends strongly on the free stream water velocity. The influence of the air injection rate and the angle of attack ranging from 0° to 8° on the bubble size is not as strong. These investigators also reported their mass transfer analyses are applicable to similar bubbly flows.

Flow past a single blade and arrays of translating blades with various arrangements is studied by Liu et al. [2,3]. These researchers conducted simulations to investigate multiphase flows past a single plate in the close proximity to a free surface [2]. Their study revealed that the drag coefficient is reduced significantly when the plate is near the surface. Liu et al. [3] conducted large eddy simulations (LES) in three-dimensional geometries to examine flows past arrays of translating plates. They demonstrated that power generation by translating blades is profoundly influenced by the spacing and the arrangement of plates. Their study aims to improve the design of energy harvesting devices consisting of translating blades for river current applications. The blade design used in Ref. [3] is utilized for the current study. Flow physics and performance characteristics predicted here are compared against their results.

In the present study, the Eulerian and the mixture model are used to predict a local bubble size and oxygen mass transfer characteristics. The Eulerian multiphase model and the numerical