Journal of the Taiwan Institute of Chemical Engineers 000 (2017) 1–12

Contents lists available at ScienceDirect

Journal of the Taiwan Institute of Chemical Engineers

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

Numerical heat transfer analysis in a two-phase microorganism malodor removing system with the effect of internal structures

Haider Ali, Cheol Woo Park*

School of Mechanical Engineering, Kyungpook National University, 80 Daehakro, Bukgu, Daegu 41566, South Korea

ARTICLE INFO

Article history:
Received 15 March 2016
Revised 2 March 2017
Accepted 9 March 2017
Available online xxx

Keywords:
Malodor removing system
Heat transfer
Two-phase flow
Heat transfer coefficient
Temperature

ABSTRACT

Malodorous emissions from industrial and household sewage systems cause harmful effects on nearby human populations and the environment. Eco-friendly malodor removing systems utilize bacteria to control unpleasant odors from sewage systems. Heat transfer is a vital factor in industrial system designs because microorganisms require an optimum water temperature to function efficiently. A numerical study was conducted to estimate the heat transfer rate in the water basin of a malodor removing system. Flow-deflecting baffles were utilized to reduce heat loss in the basin. Heat transfer was reduced by installing a rotating drum to increase gas dissolution by an improved liquid circulation. The average and local heat transfer coefficients, Nusselt number, and temperature distribution were evaluated to estimate heat loss in the water basin. This study proposes an empirical relation to generalize the numerical results for calculating heat transfer rates. Results showed that using baffles and rotating drum reduced heat transfer in the basin. Furthermore, the heat transfer coefficient was increased by raising the superficial gas velocity and bubble size.

1. Introduction

An eco-friendly malodor removing system is a newly developed two-phase flow industrial application that utilizes a group of effective microorganisms (EM) to absorb odor-producing gases. EM cells in the water basin of an odor removal system interact with malodorous gases and decompose them into organic materials for cell reproduction [1,2]. The thermal control of industrial odor removal systems is necessary because an optimum water temperature (303–350 K) of EM cells is required for their growth. Water must be maintained at the preferred temperature to improve the performance of malodor removal systems [2,3]. Heat transfer between malodorous gases and liquids at various temperatures is important because it affects the work rate of EM cells. Therefore, heat transfer must be estimated and quantified to minimize thermal losses in the water basin. Heat transfer in two-phase flow applications (chemical reactors and bubble columns) has been experimentally studied to improve their designs and operations. The gas bubble size, reactor geometry, superficial gas velocity, sparger type, hydrodynamic properties, and operating temperature affect heat transfer [4]. The heat transfer rate increases with the increase of superficial gas velocity and is higher in the center than in the reactor walls. The heat transfer coefficient decreases when the gas exhibits large bubbles and increased holdup [3,5–8]. Adding microorganism cells in the bubble column increases the heat transfer coefficient [9,10]. The presence of internal structures affects the heat transfer coefficient, and the baffles can alter the heat transfer coefficient in the bubble column [11,12]. Large-scale experimental multi-phase flow systems require a considerable amount of construction time and cost. Therefore, most experimental studies were conducted using small-scale bubble columns and few selected locations to calculate heat transfer coefficients [8–10,13,14].

Computational fluid dynamic (CFD) codes are new and reliable choices to model complex and large-scale multi-phase flow systems. Generally, commercial CFD codes are incorporated into four different numerical models, namely, Euler–Lagrange, Euler–Euler, volume of fluid (VOF), and mixture models, to solve two-phase flows. The Euler–Lagrange model tracks the individual trajectory of each particle of dispersed phase, resulting in an accurate but expensive process. This model is suitable for flows with low volume fraction of the dispersed phase, particularly in spray dryers, coal, and liquid fuel combustion, and some particle-laden flows [15–17]. The Euler–Euler model is commonly used when the volume fraction of the dispersed phase is high. The Euler–Lagrange model ignores the displacement of the continuous phase because of the low volume fraction of the dispersed phase. Tracking the overall or average phase concentration of particles, rather than tracking individual particle, renders the Euler–Euler model less expensive. The Euler–Euler model is widely used in modeling two-phase flows

* Corresponding author.
E-mail address: chwoopark@knu.ac.kr (C.W. Park).

http://dx.doi.org/10.1016/j.jtice.2017.03.012
1876-1070/

Nomenclature

a	interfacial area (1/m)
c	gas concentration (mol/m^3)
C_d	drag coefficient (1)
d_b	gas bubble diameter (m)
D	diffusion coefficient (m^2/s)
Fr	Froude number (1)
g	gravity vector (m/s^2)
h	heat transfer coefficient (W/m^2 K)
H	Henry's constant (Pa•m^3/mol)
k	liquid thermal conductivity (W/m•K)
k_m	mass transfer coefficient (m/s)
m_{gl}	mass transfer rate from gas to liquid (kg/m^3•s)
M	gas molecular weight (kg/mol)
n	bubbles per volume (1/m^3)
Nu	Nusselt number (1)
p	pressure (Pa)
Pr	Prandtl number (1)
P_{tot}	total power heat source (W)
R	ideal gas constant (J/mol•K)
Re	Reynolds number (1)
St	Stanton number (1)
T	temperature (K)
U	velocity vector (m/s)
∇	gradient operator
Greek symbols	
α	thermal diffusivity (m^2/s)
ρ	density (kg/m^3)
φ	volume fraction as (1)
μ	dynamic viscosity (Pacs)
μ_T	turbulent or eddy viscosity (Pa•s)
ω	angular velocity (rad/s)
σ	surface tension coefficient (N/m)
ν	kinematic viscosity (m^2/s)
Subscripts	
g	gas phase
l	liquid phase

in fluidized beds, bubble columns, risers, and particle suspensions [18–28]. The VOF model is applicable for two or more immiscible and non-interpenetrating fluids with the dispersed phase that is well separated from the continuous phase with a distinct interface. The VOF model is accurate and provides a clear picture of the flow field; however, it is not always practical because of its high computational cost. Generally, the VOF model is appropriate for microfluidic problems in which only few droplets or bubbles are tracked [29–31]. The mixture model is a simplified form of the Euler–Euler model and is preferable when a wide distribution of the dispersed phases exists (*i.e.*, if the particles vary in size). The low cost of the mixture model enables its application in particle-laden flows with low loading, bubbly flow, sedimentation, and cyclone separators [32,33]. The EM basin contains a large amount of malodorous gas bubbles; thus, tracking the average phase concentration is more effective than tracking each particle. Moreover, gas and liquid phases are interpenetrating, and the size of gas bubbles is consistent because of the unchanged diameter of the sparger holes. The Euler–Euler approach is the appropriate choice for modeling a two-phase flow with heat transfer in the EM removing system [27,34]. The two-phase flow in the EM odor removing system was previously modeled by applying the two-fluid Euler–Euler model [34]. However, our research only focused on gas–liquid modeling with mass transport and excluded heat transfer analysis. Therefore, a detailed numerical study must be conducted to completely characterize heat transfer in the entire EM model by considering the effects of superficial gas velocity, gas bubble size, reactor inner structure, and operating temperature [4].

This study aims to investigate heat transfer with two-phase flow in the EM malodor removal system. The Euler–Euler method was used to model the two-phase flow of the malodorous gas and water. Flow-deflecting baffles were used to reduce the heat transfer rate in the water basin to maintain the optimum water temperature for EM cells. The number of baffles was increased to examine their effect on heat transfer properties. A rotating drum was also used to reduce heat transfer by increasing gas–liquid flow mixing in the water basin. Various internal structures of basin, superficial gas velocities, gas bubble sizes, and rotating drum speeds were employed to determine their effects on average and local heat transfer coefficients, Nusselt number, and temperature distribution. This study proposes a general empirical relation to calculate heat transfer in the EM malodor removal system by considering the effects of different internal geometric structures, superficial gas velocities, and gas bubble sizes.

2. Mathematical modeling

The present study used a 3D geometric model of the water basin with a length (L) of 0.4 m, a width (W) of 0.25 m, and a height (H) of 0.8 m (Fig. 1). Four horizontal flow-deflecting baffles were installed in the basin to reduce heat transfer rates (Fig. 1(a)). The baffle exhibits a length (l) of 0.28 m, a width (w) of 0.25 m, and a height (h) of 0.001 m. The vertical distance (d) between two baffles is 0.04 m (Fig. 1(e)). The number of flow-deflecting baffles was increased to 8 and 12 to further decrease the heat transfer in the water basin (Fig. 1(b) and (c), respectively). A rotating drum (diameter = 0.12 m) with four blades was also installed in the eight-baffle model to maintain the optimum water temperature for EM cells by improving the gas–liquid flow mixing in the water basin (Fig. 1(d)). The blade of the rotating drum has a length (X axis) of 0.4 m, a width (Y axis) of 0.01 m, and a height (Z axis) of 0.1 m. Four round-shaped gas spargers (diameter (ϕ) = 0.01 m) were employed to inject air into the water basin (Fig. 1(d)). The EM cells were not physically considered in this study. The effects of the two-phase flow on heat transfer were described using heat transfer coefficient, Nusselt number, and temperature distribution.

2.1. Hydrodynamic and heat transfer equations

A two-phase turbulent bubbly flow problem is represented by an EM water basin filled with water and air bubbles that are injected from the bottom. As a result of buoyancy, the gas bubbles rise, thereby inducing a circulating motion of the liquid. Gas is dissolved from the bubbles into the liquid as the bubbles rise through the water [35]. The Euler–Euler method was used to model this turbulent bubbly flow problem because it can solve average phase concentrations compared with tracking individual gas bubbles [20,27,34]. The governing equations for the continuity and momentum transport for the liquid and gas phases are as follows [31,36]:

$$\varphi_l \rho_l U_l \times \Delta U_l = -\Delta p + \Delta \times [\varphi_l \mu_l (\Delta U_l + (\Delta U_l)^T)] + \varphi_l \rho_l g \quad (1)$$

$$\rho_l (\nabla \times U_l) = 0 \quad (2)$$

$$\nabla \times (\rho_g \varphi_g U_g) = -m_{gl} \quad (3)$$

where φ_l is the volume fraction of the liquid phase (1), ρ_l is the liquid density (kg/m^3), U_l is the liquid phase velocity (m/s), p is