Algal Research 27 (2017) 318–324

Contents lists available at ScienceDirect

Algal Research

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

Hydrodynamic performance of two air nozzles diameters on the massive microalgae culture: Computational and experimental approaches

Bruno Galler Kubelka[a,b,*], Waldir Terra Pinto[b], Paulo Cesar Abreu[a]

[a] *Aquaculture Marine Station, Institute of Oceanography, Federal University of Rio Grande, Brazil*
[b] *Laboratory of Fluid-Structure Interaction, Engineering School, Federal University of Rio Grande, Brazil*

ARTICLE INFO

Keywords:
Nozzle size
Microalgae cultivation
Dead zones
Scaling up
Water column mixing

ABSTRACT

The scaling of microalgae production in culture systems depends on preventing cells to settle down and assure a well-mixed water column. Doing so, each cell would have better exposure to light and nutrients. Computational Fluid Dynamics (CFD) can be an effective tool for understanding and predicting the multiphase flow inside tanks. In this study, we have used CFD to simulate the velocity field and the turbulent kinetic energy inside a computational tank to find best hydrodynamic performance for two nozzles configurations: case 1) one single 3 mm air nozzle; case 2) a set of nine 1 mm air nozzles. Computational results showed that smaller air nozzles are more efficient in reducing dead zones inside culturing tanks. Further, mean turbulent kinetic energy field suggested a more homogeneous dissipation, and higher intensities of turbulence were observed in the smaller nozzle simulation. The application of present numerical observations on culture systems was investigated by several experiments in 300 L cylindrical photobioreactors and 1600 L open circular tanks, both with different air nozzles sizes (1 and 3 mm) where the marine microalga *Nannochloropsis oceanica* was grown. Microalga growth was determined by measurements of water absorbance at 750 nm. Higher cell densities were determined in the photobioreactors and tanks with 1 mm nozzles. Thus, smaller nozzle was efficient to improve the microalga production even with increasing volume and different culture system designs, supporting the numerical analyses predictions by CFD.

1. Introduction

Economic viability for commercial-scale microalgae culture depends on the production of these photosynthetic microorganisms in large-scale culture systems [1]. Scaling-up microalgae production from laboratory controlled conditions to thousands of liters' tanks subject to environmental conditions is an obstacle to attain high productivity [2]. Optimization of productivity has been a topic of intense study in recent years, mainly by improvements in strain performance through genetic engineering [1] and development of new technologies of cultivation that could reduce energy cost [2].

High production of microalgae mass culture depends on preventing microorganisms settling. Moreover, a well-mixed water column is a guarantee that cells have uniform exposure to light, CO_2 and other nutrients [1]. This can be done by transferring momentum to the water and creating a circulation pattern. Two typical mechanisms can perform this hydrodynamic circulation. First, paddlewheels that creates horizontal velocities on water. Weissman et al. [3] previously studied this system and it has been recently optimized with the aid of numerical models [4]. The second mechanism makes use of air bubbles to create water circulation and to prevent organisms from settling. Air bubble is injected in the tank's bottom and rise to the surface, resulting in vertical momentum transfer to the water.

Studies on bubbling applied to microalgae culture are rare. Nevertheless, the latest tools for hydrodynamic analyses allow the development of innovative approaches and optimization of massive culturing systems. Utilization of Computational Fluid Dynamics (CFD) and Particle Image Velocimeter (PIV) by Bitog et al. [5] pointed out the best design for 30 L tanks, where tank geometry, air flow rate, nozzle diameter and configuration were optimized. Yang et al. [6] also using PIV measurements investigated air flow rate and air nozzles distribution for a 15.45 L tank and found optimum culture conditions through the determination of velocity field and turbulent mixing. Similarly, CFD simulations of inner structures of photobioreactors were analyzed by Yu et al. [7] with the evaluation of hydrodynamics parameters as turbulent kinetic energy and percentage of dead zones.

However, the relationship between air bubble hydrodynamics behavior and mixing performance is still poorly known, especially for

* Corresponding author at: Aquaculture Marine Station, Institute of Oceanography, Federal University of Rio Grande, Brazil.
E-mail address: brunokubelka@furg.br (B.G. Kubelka).

http://dx.doi.org/10.1016/j.algal.2017.09.024
Received 14 June 2017; Received in revised form 30 August 2017; Accepted 26 September 2017

culture tanks above 100 L [8]. Although, it is well accepted that bubble injection systems are potentially energy saving and highly productive [9], the study of bubble hydrodynamics is necessary for production improvement and energy optimization in large-scale systems.

Air bubble injection in water tanks is a multiphase flow, where air and water interact through superficial tension forces that arise from the presence of a pressure gradient [10]. When air bubbles enter the water tank from the bottom, they present four stages: i) Forming; ii) Shedding; iii) Rise; and iv) Breakup at surface. All stages are dependent on some flow properties that are extremely important when trying to improve culture tank mixing conditions [11]. These properties are air injection velocity (or flow rate), air nozzle diameter and distribution. The modification of air nozzle diameter with the maintenance of the same open area, and installation of rotameters to make sure that the same air flow rate were adjusted for both cases, appears to enhance productivity with no extra energy costs. As previously suggested by Zimmerman et al. [10], smaller air bubbles injection can result on a higher interfacial surface area leading to higher mass and momentum transfer to the water.

The Laboratory of Microalgae Production (LMP) at the Federal University of Rio Grande (FURG), Southern Brazil, has been culturing the marine microalgae *Nannochloropsis oceanica* in 300 L cylindrical photobioreactors (PBR) and 1600 L open circular tanks (OCT), both with bubbling made by 3-mm air nozzles distributed on the tank's bottom, and raceway tanks with water mixed by paddlewheels. Roselet et al. [12] demonstrated the efficiency of air injection in circular tanks in comparison to paddlewheels in raceways at LMP-FURG. However, no further optimization studies were done to improve the mixing efficiency in PBR and OCT, to achieve better microalgae cell exposure to light and to reduce dead zones in both systems.

In this study, vertical water velocity and turbulent kinetic energy were evaluated on two nozzles diameters (3 and 1 mm) through numerical analyses using Computational Fluid Dynamics. Main objective was to determine which nozzle size most enhances momentum transfer to water column, minimizing dead zones and improving water column mixture. Afterwards, experiments to evaluate the growth of *N. oceanica* were conducted in 300 L cylindrical photobioreactors and 1600 L open circular tanks with 1 and 3 mm air nozzles.

2. Materials and methods

2.1. Computational analysis

2.1.1. Tank geometry

The program *STAR-CCM+* includes a *3D-CAD* tool that allows computational geometry creation inside program interface. Culture tanks configurations were reproduced numerically, the acrylic cylindrical PBR had a 55 cm' diameter, and 120 cm is filled with salt water. Through this, tanks and nozzles designs were developed for two simulation cases (Figs. 1 and 2). The first case aimed to reproduce nozzle configuration found at the photobioreactors used in the Microalgae Production Laboratory (Federal University of Rio Grande), where a single 3-mm (1 × 3 mm) air nozzle is the standard mixing system (Fig. 1) [12]. In the second case, we tested an optimized nozzle system with a set of nine 1 mm (9 × 1 mm) nozzles (Fig. 2). Cases 1 and 2 have different nozzle geometry, although nozzles total open area, or air inlet area, was the same two rotameters were installed to be certain that air flow rate was the same in both cases.

2.1.2. Mesh

Mesh generation was also done using the *STAR-CCM+* interface. Results quality for cases 1 and 2 was guaranteed by comparing the residuals for three different mesh element sizes, or three different total numbers of cells (Table 1). Steady simulations were run for each mesh case, and residuals from solving governing equations dropped $1.0E^{-3}$ for all cases, attesting for a good physical setup [13].

Mesh selection was done using mesh diagnostic parameters [14], shown in table 1. According to the *STAR-CCM+* users guide, direction of mesh selection must have the following characteristics: i) MSA needs to be lower than 90°; ii) Higher MVC corresponds to better meshes and it needs to be higher than $1.0\ e^{-4}$, for a satisfactory mesh. These mesh diagnostics indicated the best mesh choice for case 1 has been the intermediate and case 2 also the intermediate. These mesh choices are shown in Figs. 1 and 2 and were used for the transient simulations.

2.1.3. Initial and boundary conditions

The tank did not present any air passing through the water column at the beginning of the simulation, and the water was at rest. As the transient simulations runs, air enters through nozzles to the water column. Air bubbles are formed, then detaches from bottom, rise and exit through the top of the tank. Top wall allow only air exit and tank walls are not permeable for neither air nor water. Air velocity, for both cases was set to as the rise velocity of the bubble [14], which was $0.76\ m\ s^{-1}$ for case 1, and $0.72\ m\ s^{-1}$ for case 2 [15]. Also, bubble diameter was entered on interface following experimental results from Neto et al. [15], for the first case bubble diameter was 9 mm, and for the second was 6.5 mm.

2.1.4. Numerical models

The Computational Fluid Dynamics (CFD) was employed for solving fluid dynamics equations. Air-water interactions were considered as multiphase problems, where air momentum is transferred to water, and turbulence is also involved. To predict water circulation resulting from nozzle air injection in the water tank *STAR-CCM+*, a CFD commercial program, was used. Through the Multiphase Segregated Flow model a set of conservation equations for each Eulerian phase was solved [13].

2.1.4.1. Fluid dynamics equations. The following integral equation (Eq. (1)) represent the Navier-Stokes equations for continuity and momentum for each phase [13]:

$$\frac{\partial}{\partial t}\left(\int_V \rho \boldsymbol{v}\right)dV + \int_A \rho \boldsymbol{v} \otimes (\boldsymbol{v} - \boldsymbol{v}_g)\cdot d\boldsymbol{a} = -\int_A p\boldsymbol{I}\cdot d\boldsymbol{a} + \int_A \boldsymbol{T}\cdot d\boldsymbol{a} + \int_V (\boldsymbol{F})dV \tag{1}$$

where t stands for time, V is the cell volume, ρ is phase density, $\boldsymbol{v}$ is the velocity vector, $\boldsymbol{v_g}$ is the grid velocity vector, $\boldsymbol{a}$ is the face area vector, p is the pressure, $\boldsymbol{I}$ is the identity matrix, $\boldsymbol{T}$ is the viscous stress tensor and $\boldsymbol{F}$ represents all body forces that should be incorporated to the equation to be solved (including buoyancy).

From Eq. (1) volume discretization method was applied to solve the equation numerically. Current simulations used first order temporal discretization with a $5.0E^{-4}$ time-step and velocity convection was accounted on second order. Solver SIMPLE algorithm was used to control the overall solution [14]. When each phase solving is done, phase interaction models (Eulerian Multiphase) are used to define the influence that one phase exerts upon another across the interfacial area between them.

2.1.4.2. Turbulence models. Navier-Stokes equations for the instantaneous velocity and pressure fields were decomposed into a mean value and a fluctuating component, resulting on equations for the mean quantities that are identical to the original equations, except that an additional term now appears in the momentum transport equations. This additional term is a tensor quantity, known as the Reynolds stress tensor (**T**), which has the definition of Eq. (2) [13]:

$$\boldsymbol{T} = -\rho\begin{pmatrix} \overline{u'u'} & \overline{u'v'} & \overline{u'w'} \\ \overline{u'v'} & \overline{v'v'} & \overline{v'w'} \\ \overline{u'w'} & \overline{v'w'} & \overline{w'w'} \end{pmatrix} \tag{2}$$

where u', v' and w' stands for local velocity fluctuations.

A Standard K-Epsilon turbulence model was selected to provide