Bioresource Technology 175 (2015) 633–637

Contents lists available at ScienceDirect

Bioresource Technology

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

Short Communication

Hydraulic optimization of membrane bioreactor via baffle modification using computational fluid dynamics

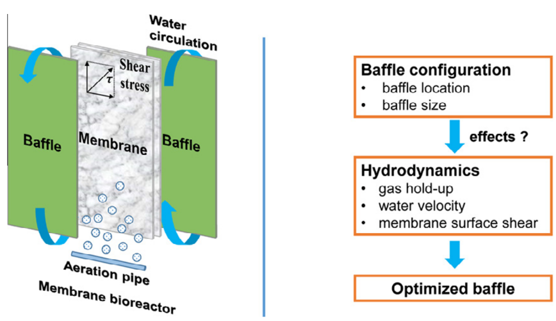
Xiaoxu Yan [a], Kang Xiao [a,b], Shuai Liang [a], Ting Lei [a], Peng Liang [a], Tao Xue [c], Kaichang Yu [c], Jing Guan [c], Xia Huang [a,*]

[a] State Key Joint Laboratory of Environment Simulation and Pollution Control, School of Environment, Tsinghua University, Beijing 100084, China
[b] College of Resources and Environment, University of Chinese Academy of Sciences, Beijing 100049, China
[c] Beijing Origin Water Technology Co., Ltd, Beijing 102206, China

HIGHLIGHTS

- 3D distribution of hydrodynamics in a bench-scale MBR was simulated by CFD.
- Baffle location (front, side, or both) and size had significant effects.
- Side baffles were more effective in elevating membrane surface shear.
- Maximum shear was obtained with both front and side baffles of optimized size.
- Role of baffles was more prominent at lower aeration intensities.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 11 August 2014
Received in revised form 23 October 2014
Accepted 27 October 2014
Available online 4 November 2014

Keywords:
Airlift membrane bioreactor
Baffle
Hydrodynamics
Membrane surface shear
Computational fluid dynamics

ABSTRACT

Baffles are a key component of an airlift membrane bioreactor (MBR), which could enhance membrane surface shear for fouling control. In order to obtain an optimal hydraulic condition of the reactor, the effects of baffle location and size were systematically explored in this study. Computational fluid dynamics (CFD) was used to investigate the hydrodynamics in a bench-scale airlift flat sheet MBR with various baffle locations and sizes. Validated simulation results showed that side baffles were more effective in elevating membrane surface shear than front baffles. The maximum average shear stress was achieved by adjusting baffle size when both front and side baffles were installed. With the optimized baffle configuration, the shear stress was 10–30% higher than that without baffles at a same aeration intensity (specific air demand per membrane area in the range of 0–0.45 m^3 m^{-2} h^{-1}). The effectiveness of baffles was particularly prominent at lower aeration intensities.

1. Introduction

Membrane bioreactor (MBR) has been accepted as a promising technology for wastewater treatment due to its advantages, such as relatively high effluent quality, small footprint, and low excess sludge production, over conventional activated sludge processes (Huang et al., 2000; Huang et al., 2010; Judd, 2011). However, membrane fouling remains as a challenge for the practical application of MBRs. An important issue associated with membrane fouling is the high energy consumption involved in aeration in membrane tank for fouling control (Liu et al., 2000; Prieske et al., 2008). The power used for aeration in membrane tank could take up about 30–50% of the overall energy consumption for MBR operation. In order to control fouling with a minimal energy cost, it should be of great significance to optimize the hydraulic condition in the aerated membrane tanks.

* Corresponding author. Tel.: +86 10 62772324.
E-mail address: xhuang@tsinghua.edu.cn (X. Huang).

http://dx.doi.org/10.1016/j.biortech.2014.10.133

Baffles play a vital role in the commonly applied airlift MBRs (Liu et al., 2003; Prieske et al., 2008). They divide the membrane tank into a downcomer and an aerated riser section where membranes are usually located. The difference in gas hold-up, and hence in fluid density, between these sections drives the bulk liquid to circulate, creating cross flow over the membrane surface to depress deposition of sludge particles. It is conceivable that geometric configuration (such as location and size) of the baffles should have profound impacts on the hydraulic conditions in the membrane tank and thus on the membrane fouling. Therefore, optimization of baffle configuration deserves careful consideration.

Despite the importance of baffles, there has been little information for an optimal design of baffles in airlift MBRs, and there is also a lack of in-depth understanding about the detailed effects of baffle configuration on the hydraulic condition in the membrane tank. Experimental studies on this topic have been scarcely reported, which could probably arise from either the difficulty in measuring the hydrodynamics (concerning intrusive impacts on the test flow (Yamanoi and Kageyama, 2010) and interference from sludge flocs (Bérubé et al., 2006; Chan et al., 2011)), or the onerous effort required to create a series of representative scenarios for the experiments (involving reactor building and baffle resizing). In this context, mathematical modeling studies have been conducted more often, with several simplified empirical models proposed to predict some collective properties of hydrodynamics (e.g. average cross flow velocity in the riser section) (Liu et al., 2003; Liu et al., 2000). However, most of these models cannot provide adequate quantitative information (e.g. spatial distributions of gas hold-up, water velocity and membrane surface shear) for elaborate baffle design.

With the development of computer science, computational fluid dynamics (CFD) provides possibilities for quantification of three-dimensional flow field distribution that could hardly be detected experimentally. It uses numerical methods to solve and analyze flow problems, and has been gradually applied to the field of membrane filtration processes. A few examples of successful utilization of this technique for MBR systems have been reported (Naessens et al., 2012; Ratkovich et al., 2012).

The present study aims to investigate the hydrodynamics in a bench-scale flat sheet MBR using CFD and find an optimized baffle configuration to improve the aeration efficiency for fouling control. Three scenarios of baffle locations were applied, including front location (i.e. parallel with the membrane sheet), side location (i.e. perpendicular to the membrane sheet), and coupled front and side locations. Baffle size under each scenario was varied to find its effect on the hydrodynamics in the reactor. Gas hold-up distribution, water motion and membrane surface shear characteristics were calculated. Baffle location and size were optimized to obtain maximum membrane surface shear with even distribution. The optimized condition was then compared with the no-baffle condition to assess the effectiveness of baffles in reducing aeration demand. Additionally, the reliability of the CFD simulation results were confirmed by velocity profile measurement based on particle image velocimetry (PIV).

2. Methods

2.1. CFD description and implementation

Two plates made of acrylic glass were fixed in the center of the reactor standing for flat sheet membranes (Fig. S1(I)). The reason for using acrylic glass plates instead of real membranes is that the measurement of water velocity by PIV requires the reactor to be transparent so that the camera can capture the whole reactor. Another reason is that the simulation did not contain filtration process, thus it is also not necessitated to use real membrane. The distance between the plates was 7 mm, which was applied in most commercial flat sheet MBR modules (Judd, 2011). An aeration pipe with 8 nozzles (with diameter of 2 mm) at an interval of 25 mm was installed in the center of the reactor, 50 mm below the membrane bottom. The distance between the front baffle and the outer surface of a membrane was also 7 mm (Fig. S1(II)). Side baffles were placed adjacent to membranes (Fig. S1(III)).

For each of scenarios II–IV, the effect of baffle size was investigated. The baffle size was varied by changing the distance between the upper edges of the baffles and water surface (D_u), and the distance between the lower edges of the baffles and tank bottom (D_d) (Fig. S1(II) and (III)). For each scenario, D_u and D_d were scaled from 20 to 100 mm, both with a step increase of 20 mm. D_u/H defines the opening degree of the upper wetted cross-sectional channel (OD_u) where H is water level. Similarly, D_d/H defines the lower opening degree (OD_d).

The flow filed of each scenario was calculated by ANSYS CFX 14, the detailed description about the simulation models was presented in Supplementary materials.

2.2. Experimental validation

Water velocity profiles obtained from PIV was compared with simulated data to validate the CFD methodology. The general principle of PIV is to illuminate the tracer particles in a flow field with a laser sheet, capture images of the particles at a certain time interval, and derive the water velocity profiles from the trajectories of the particles by cross-correlating the consecutive images. More detailed information about PIV system setup was presented in Supplementary materials.

3. Results and discussion

3.1. Overview of fluid dynamics in the MBR tank under different scenarios

CFD simulation of the fluid dynamics in the aerated MBR tank yielded spatial distributions of gas hold-up (Fig. S3), water velocity (Fig. S4(A)), and shear stress on the membrane surface (Fig. S4(B)), under different scenarios of baffle configuration.

For all the scenarios, air phase was mostly observed between the membranes and in the space vertically above them (Fig. S3, lateral view). This zone formed the riser section, while the other part of the tank forms the downcomer section. The water circulation velocity between the riser and downcomer sections has been reported to depend closely on the motion of bubble swarms (Heijnen et al., 1997), and could in turn affect the gas hold-up in these sections. Almost no air was observed in the downcomer section for all the scenarios (Fig. S3, lateral view), which should be ascribable to the inertia–buoyancy balance. The inertial force provided by the fluid circulation seemed not strong enough to push the bubbles against buoyancy into the downcomer.

It is also notable that air bubbles tended to be pinched toward the center of the tank while moving upward in the riser section (Fig. S3, front view). As a consequence, less air was distributed near the outer part of the membrane surface. Similar phenomenon was observed by other researchers (Gresch et al., 2011; Spicka et al., 2001). It could be explained by the pressure variance in bulk liquid caused by aeration. Water above the nozzles could obtain a higher speed than around, resulting in a lower pressure, which would force water around to flow in, thereby pushing bubbles toward the center.

The evenness of gas hold-up distribution varied with height in the tank. Fig. 1 depicts the gas hold-up distributions along the x