Journal of Membrane Science 520 (2016) 201–213

Contents lists available at ScienceDirect

Journal of Membrane Science

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

Numerical simulations of impact of membrane module design variables on aeration patterns in membrane bioreactors

Xuefei Liu [a], Yuan Wang [a,b], T. David Waite [b], Greg Leslie [a,*]

[a] UNESCO Centre for Membrane Science & Technology, School of Chemical Engineering, University of New South Wales, Sydney 2052, Australia
[b] Water Research Centre, School of Civil & Environmental Engineering, University of New South Wales, Sydney 2052, Australia

ARTICLE INFO

Article history:
Received 23 March 2016
Received in revised form
6 July 2016
Accepted 7 July 2016
Available online 14 July 2016

Keywords:
Membrane bioreactor
Computational Fluid Dynamics
Aeration
Membrane module design
Particle Image Velocimetry

ABSTRACT

Computational Fluid Dynamics (CFD) models incorporating sludge rheology model and porous media sub-models were used to simulate the impact of eight design variables, including fibre orientation, filtration tank geometry and aeration design, on the hydrodynamics and aeration patterns in a pilot scale membrane bioreactor (MBR) fitted with commercially available membrane modules. Comparison of simulated with experimental grid averaged flow velocities measured using Particle Image Velocimetry (PIV) in an aerated bench scale MBR differed by approximately 6%. The assessment of different turbulence models revealed that it was difficult to accurately simulate local variations in turbulence induced by air bubbles due to limitations of Reynolds averaging Navier Stokes (RANS) models. Simulation results revealed that MBR's fitted with hollow fibres in a vertical orientation in the filtration zone experienced 25% more membrane surface shear than horizontally oriented fibres at the same aeration intensity. The inclusion of baffles around the membrane modules promoted turbulence and increased shear in the upper section of the membrane module by approximately 30%. This is important for control of fouling along sections of the hollow fibre membrane that experience the highest transmembrane pressure and are more susceptible to fouling. Rotating the nozzle aperture to face the bottom of the tank increased the homogeneity of shear stress on the lower half of the module and increased shear on the upper half of the membrane surface. This effect was amplified by locating the air diffusers 100 mm below the bottom of the module and aligning the aeration pipe in parallel with the direction of the fibre bundle. This research demonstrates the capability of using Computational Fluid Dynamics to optimise the design of the filtration zone comprising the membrane module, aeration system and MBR tank dimensions.

1. Introduction

Membrane bioreactors (MBR) use microporous membranes in lieu of secondary clarifiers and media filters for solid-liquid separation in wastewater treatment applications [1]. The membranes are located in a *Filtration Zone*, which includes tankage containing membrane modules and coarse bubble aeration system [2]. Hydraulic capacity of the MBR process is controlled via aeration induced turbulence which prevents accumulation of mixed liquor suspended solids on the membrane surface [3–5]. Commercial MBRs are available in different configurations based on membrane orientation, aerator aperture size, aerator position and free volume between membrane modules and tank walls. The overarching design goal is to create a spatially uniform velocity gradient in the filtration zone to limit localised fouling and promote even distribution of filtrate flux over all the available membrane area [4,6]. However, in the absence of performance data collected under controlled conditions, it is difficult to assess which design achieves the highest surface shear while simultaneously optimising the footprint of the filtration zone and power input for the aeration system. This can be attributed to the complex conditions in the filtration zone and the interdependence of the effect of each design variable on bubble induced shear.

The effect of bubble characteristics, membrane module orientation and the geometry of filtration tank has been investigated by empirical measurement of flux decline [7,8], liquid flow velocities using Particle Image Velocimetry (PIV) [9–11], and characterisation of membrane surface shear using electrochemical/electrodiffusion methods [12–16]. For example, a 15% reduction in flux decline was obtained by rotating fibre orientation relative to the tank wall through 90° from horizontal to vertical [17,18]. Direct measurement of shear forces on Teflon tubes representing bundles of hollow fibres using electrodiffusion methods found that the average shear increased from 0.3 Pa at a gas flowrate of 2 mL/min to 0.8 Pa at a gas flowrate of 35 mL/min [12]. Similarly, increasing the fibre packing density produced "dead zones" where fibres

* Corresponding author.
E-mail address: g.leslie@unsw.edu.au (G. Leslie).

http://dx.doi.org/10.1016/j.memsci.2016.07.011
0376-7388/

received minimal shear forces [19], however, the occurrence of these dead zones could be reduced by changing the location of the aerator port from the centre of the module to the corner of the module [9]. Similarly, the installation of baffles at the periphery of the filtration zone was found to increase the crossflow velocity in the vicinity of the fibres [20]. The shear profile in a MBR is transient and non-uniformly distributed in the filtration zone. Different statistical parameters such as time-averaged shear, standard deviation and amplitude have been used to quantify shear for different MBR configurations [21]. While these empirical studies can be used to compare different design features (such as the effect of baffles or membrane orientation) the data is not representative of the hydrodynamic conditions in a commercial MBR module and would have limited utility in the design of a full scale filtration zone [5]. A more systematic approach is required to identify the optimum design that can achieve superior shear profile whilst minimising power input for aeration [20].

Numerical modelling enables *a priori* evaluation of different design variables by changing the geometry of the computing domain and boundary conditions in complex turbulent multi-phase flow [6]. While numerical simulations of surface loading rates, settling velocities and velocity gradients have been used to design full scale conventional solid-liquid separation processes in wastewater treatment [22], these techniques have not been applied widely on MBRs. Khalili-Garakani et al. [23] modelled the use of baffles in the filtration zone to constrain air flow around flat sheet modules and found the change in baffle angle from 90° to 85° could increase the shear stress on the membrane surface. The majority of the numerical studies have focussed on the movement of gas bubbles (or slugs) along tubular or flat sheet membrane modules and have been used to model temporal changes in spatial variation of shear as a function of bubble size, channel dimension and geometry for Newtonian fluids [24–28] in order to relate shear rate induced by gas bubbles to the permeate flux [29], and to evaluate the impact of bubble frequency, size and shape on liquid velocities and shear stress on the membrane surface [30–35]. Notwithstanding this body of work, there remain inconsistencies on the ability to reduce fouling by optimising the aeration pattern (gas flowrate) [33]. Consequently, it is difficult to provide definitive recommendations with regard to the most effective bubbling regime that could be used to reduce membrane fouling. In this paper, various design variables including characteristics of fibre bundle (fibre diameter and packing density), module geometry (orientation and shape of module) and configuration of aeration systems are systematically evaluated using Computational Fluid Dynamics.

The objective of the studies described here was to evaluate which combination of features of the hollow fibre membrane module, filtration tank, and aeration system that can achieve higher and more homogeneous shear in the filtration zone at the same aeration energy input. A Computational Fluid Dynamics (CFD) approach using rheological and porous media sub-models described elsewhere [36] was used to simulate pressure drop across the hollow fibre membrane bundle for a range of conditions that are relevant for application of MBR's in municipal wastewater treatment. Particular emphasis is given to identifying the capabilities of current state-of-the-art turbulence models in capturing the high degree of turbulence induced by bubbly flow by comparing the simulated data using three different types of turbulence models with experimentally measured data using Particle Image Velocimetry (PIV).

2. Theory

2.1. Shear stress and liquid flow velocity

Shear stress on the membrane surface can be calculated from the tangential flow velocity outside the boundary layer using the wall-function approach [37,38], where the flow velocity inside the boundary layer (u^+) is given by:

$$u^+ = \frac{U_t}{u_\tau} = \frac{1}{k}\ln\left(y^+\right) + C \tag{1}$$

where u_τ is the friction velocity, U_t is the flow velocity outside the boundary layer, k is the von Karman constant and y^+ is the dimensionless distance from the wall, which is defined as:

$$y^+ = \frac{\rho \Delta y u_\tau}{\mu} \tag{2}$$

where Δy is the thickness of boundary layer. The standard definition of y^+ generally used in CFD can be written as:

$$y^+ = \frac{\rho u_\tau \cdot \Delta n}{\mu} \tag{3}$$

where Δn is the distance between the first and second grid points off the wall [38].

In ANSYS CFX®, a "Scalable Wall Function" is used for all ε-based turbulence models while an "Automatic Near-Wall Treatment" is used for all ω-based models, which result in slightly different treatments in Δy, to achieve an optimum model accuracy and robustness [38]:

For the scalable wall function, $\Delta y = \Delta n/4$; and for automatic wall treatment, $\Delta y = \Delta n$.

The friction velocity is defined as:

$$u_\tau = \left(\frac{\tau}{\rho}\right)^{1/2} \tag{4}$$

By substituting Eq. (2) and Eq. (4) into Eq. (1), we obtain

$$U_t = \frac{1}{k} u_\tau \ln\left(\frac{\rho \Delta y u_\tau}{\mu}\right) + u_\tau C \tag{5}$$

The wall shear stress, τ, is a function of U_t, μ, and ρ and is directly linked to the flow velocity outside the boundary layer, U_t. In this work U_t is used to represent the shear stress on the membrane surface for sludge with the same viscosity and density.

2.2. Porous media model

The resistance caused by the hollow fibre membrane bundle to the fluid flow can be modelled using a porous media approach [39]. A momentum source term, S_i, is added to the governing momentum equations with the pressure drop considered to be proportional to the fluid velocity; i.e.

$$S_i = -\left(\sum_{j=1}^{3} D_{ij}\mu v_j + \sum_{j=1}^{3} C_{ij}\frac{1}{2}\rho v_{mag} v_j\right) \tag{6}$$

$$D = \frac{1}{K_{perm}} \tag{7}$$

$$C = K_{loss} \tag{8}$$

where K_{perm} is permeability, K_{loss} is friction loss coefficient and D and C are prescribed matrices. The first term on the RHS of this equation is the viscous loss term and the second is the inertial loss term.