APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO OPTIMISATION OF MEMBRANE MODULE CONFIGURATIONS IN SUBMERGED MEMBRANE BIOREACTORS

Yuan Wang 1,2, Xuifei Liu 1, Jing Guan 3, T. David Waite 2, Greg Leslie 1
1. UNESCO Centre for Membrane Science & Technology, University of New South Wales, Sydney, NSW
2. Water Research Centre, University of New South Wales, Sydney, NSW
3. Beijing Origin Water Technology, Beijing, China

ABSTRACT

A three-dimensional Computational Fluid Dynamics model, coupled with sludge rheology models to account for the behaviour of mixed liquor and porous media models to represent the resistance of membranes to the fluid field, was developed and used in estimating shear stress in the filtration zone of membrane bioreactors. The influence of a suite of factors on shear stress including aerator position, nozzle direction, bubble size, membrane orientation, packing density and the geometry of the filtration chamber was examined using the CFD model. The highest shear stress was predicted to occur for vertically oriented membranes aligned in parallel with air diffuser nozzles generating 10 mm diameter bubbles constrained to travel in the filtration zone by solid baffles encasing the membrane unit.

INTRODUCTION

The membrane bioreactor (MBR) process is a suspended growth activated sludge system that utilises microporous membranes for solid/liquid separation in lieu of secondary clarifiers for the treatment of municipal wastewater (Judd 2006). MBR processes include a filtration zone, which includes the tankage containing the membrane modules and aeration system. The filtration zone has a smaller footprint and delivers superior effluent quality compared with clarifiers used in conventional wastewater treatment but is attended by higher power consumption associated with the operation of the blowers used to scour the membranes to control fouling and maintain hydraulic capacity. Commercial MBRs use a variety of membrane module and aerator configurations. (Wang et al. 2009), each with their particular aeration power requirements for maintenance of the hydraulic capacity of the filtration process. For example, air bubbles can be supplied through air diffusers incorporated into the base of each cassette (Figure 1a) or entrained in the mixed liquor and introduced into the module using a two-phase mixing system (Figure 1b). The hollow fibres can be arranged in parallel rows within a cassette (curtain shape) with central permeate collector pipes between them (Figure 1a) or vertically oriented in tubes (Figure 1b). The fibres can be fixed at both ends of the module or just at one end (Figure 1c). The configuration of the membrane module and the layout of the aeration systems will affect the geometry and the size of the filtration tank in which the membrane modules are positioned. As such, design of the membrane filtration zone is a complex process that must accommodate these variables and it is often difficult to compare the performance of alternative designs.

In this paper we explore the use of Computational Fluid Dynamics (CFD) to assess alternative designs for the filtration zone in MBRs. Using CFD techniques it is possible to build numerical models to study the effect of different design variables by changing the geometry of the computing domain and boundary conditions in complex turbulent multiphase flow (Wang et al., 2013). CFD models have been previously built to model the gas bubbles passing up through tubular or flat sheet membrane modules (Cabassud et al., 1997; Ghosh and Cui, 1999; Cui and Wright, 1994; Mercier et al., 1997), to describe the spatial variation of shear with time for rising bubbles and also to relate shear rate induced by gas bubbles to the permeate flux (Cui et al., 2003; Taha et al., 2006), and to evaluate the impact of bubble frequency, size and shape on liquid velocities and shear stress on the membrane surface (Ndinisa et al. 2006; Prieske et al. 2007; Drews et al. 2010; Martinelli et al. 2010; Buetehorn et al. 2011). Contradictory results with regard to the optimal aeration pattern (gas flow rate) and its ability to reduce fouling (Drews et al. 2010) are evident from these various studies. Review of the literature indicated a shortage of definitive recommendations with regard to the most effective bubbling regime for reducing membrane fouling. In this paper,
various design variables that could affect the effectiveness of bubbling, including characteristics of fibre bundles (fibre diameter and packing density), module geometry (orientation and size) and configuration of aeration systems were systematically evaluated using Computational Fluid Dynamics. The objective of the study is to identify an optimal hollow fibre module geometry design with regard to aeration system power demand and shear in the filtration zone.

METHODS

Pilot Scale MBR

CFD models were built for the membrane filtration chamber of a pilot scale MBR with a working volume of 1250 L and operated at the Sydney Water Bondi Sewage Treatment Plant treating municipal wastewater. The effective volume of the membrane filtration chamber is 250 L (L x W x H = 0.5 m x 0.5 m x 1.0 m). Four curtain shaped PVDF hollow fibre membrane modules (Beijing Origin Water, China) were positioned in vertical orientation. The membrane modules were surrounded by a baffle with a projected area of 0.49 m x 0.28 m. Coarse bubbling was provided through three porous circular pipes placed vertical to the membrane modules at an aeration intensity of 4.70 Nm$^3$/hr (Table 1).

Table 1 Configurational parameters of membrane filtration tank in pilot scale MBR.

<table>
<thead>
<tr>
<th>Reactor dimension (L x W x H)</th>
<th>0.5mx0.5mx1.0m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Membrane nominal pore size</td>
<td>0.3μm</td>
</tr>
<tr>
<td>Fibre o.d./i.d.</td>
<td>2.4mm/1.3mm</td>
</tr>
<tr>
<td>Total membrane area</td>
<td>10.4m$^2$</td>
</tr>
<tr>
<td>Packing density</td>
<td>350m$^2$/m$^3$</td>
</tr>
<tr>
<td>Aeration for membrane scouring</td>
<td>4.70Nm$^3$/hr</td>
</tr>
<tr>
<td>Recirculation flow rate</td>
<td>45mL/s</td>
</tr>
</tbody>
</table>

The pilot plant MBR was designed for chemically enhanced phosphorus removal with chemical addition required to reach the discharge requirement for total phosphorus (TP) of less than 0.05 mg/L. Ferrous iron solutions were pumped to the pilot reactor at a constant flowrate, yielding a molar ratio of Fe(II) to orthophosphate of 2.01 to 4.10. Details of operational conditions, influent and effluent water quality and fouling behaviour of the pilot plant can be found elsewhere (Wang et al. 2014).

Model Geometry and Design Variables

The CFD code ANSYS CFX 14.5 was used to simulate the three dimensional flow field (Figure 2) and different design variables evaluated (Table 2). Each simulation was performed by changing one variable while keeping other conditions the same (as the standard scenario). A hexahedron mesh was created for the membrane module as well as the near membrane zone. Five inflation layers (first layer thickness of 2 mm and a growth rate of 1.5x per layer) were used to increase the local resolution of the boundary layer for shear stress calculations. The remaining domain was meshed using tetrahedron methods, resulting in a total of 660,000 elements. A mesh independent test was performed by refining the grid size of the membrane module from 5 mm to 3 mm. A maximum error of 5% in the flow quantity (gas hold-up and membrane wall shear stress) was observed between the finer grid and coarser grid.

![Figure 2: Geometry of model domain and design variables](image)

Table 2 Design variables used in standard scenario and modified scenarios in CFD simulations

<table>
<thead>
<tr>
<th>Design variables</th>
<th>Standard scenario</th>
<th>Modified scenarios</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Bubble size (mm)</td>
<td>5</td>
<td>3, 10</td>
</tr>
<tr>
<td>2. $\alpha$ (degree)</td>
<td>180</td>
<td>0</td>
</tr>
<tr>
<td>3. $\varphi$ (degree)</td>
<td>90</td>
<td>0</td>
</tr>
<tr>
<td>4. $d_1$ (mm)</td>
<td>N/A</td>
<td>0.23</td>
</tr>
<tr>
<td>5. $d_2$ (mm)</td>
<td>50</td>
<td>20, 80, 100</td>
</tr>
<tr>
<td>6. $\theta$ (degree)</td>
<td>0</td>
<td>90</td>
</tr>
<tr>
<td>7. Baffle</td>
<td>present</td>
<td>absent</td>
</tr>
<tr>
<td>8. $w_1/w_2$</td>
<td>0.3</td>
<td>0.2, 0.4 – 0.7</td>
</tr>
</tbody>
</table>
Model Development

The CFD model developed in this work was composed of a core numerical hydrodynamics model for the liquid and gaseous phases, empirical sludge rheology models to account for the behaviour of the mixed liquor (Liu et al., 2015) and porous media models to represent the resistance of membrane modules to the fluid flow (Wang et al., 2010). The Eulerian model was implemented to model the liquid and gas components of the multiphase flow with the water and sludge mixture treated as a single liquid phase (Liu et al., 2015). A RNG k-ε turbulence model was employed to simulate the turbulence flow while a zero equation model was used for the dispersed gas phase. It was assumed that air bubbles in the dispersed gas phase were i) spherical; ii) had a constant diameter; and iii) did not collide, coalesce or fracture/divide. This is consistent with assumptions made in the development of other CFD models of MBRs that have been validated using tracer studies (Brannock et al., 2010) and particle image velocimetry (Liu et al., 2015).

Sludge rheology, in the absence and presence of iron(II) salts, determined using a Malvern Kinexus Rheometer® with electronic gap control system, exhibited shear-thinning characteristics. An Ostwald-de Waele model was used to describe the rheological behaviour of sludge with different MLSS concentrations in the CFD model (Liu et al., 2015).

For mixed liquor without iron(II) salts addition, the relationship between shear stress and shear rate can be expressed by:

\[ \tau = 0.0431\text{MLSS}^{0.89}\gamma^{0.68} \text{(MLSS conc. } = 3 - 8 \text{ g/L)} \]  \hspace{1cm} (1)

\[ \tau = 0.0412\text{MLSS}^{1.64}\gamma^{0.45} \text{(MLSS conc. } = 8 - 16 \text{ g/L)} \]  \hspace{1cm} (2)

For mixed liquor with iron(II) dosed to the membrane zone:

\[ \tau = 0.0219\text{MLSS}^{0.52}\gamma^{0.83} \text{(MLSS conc. } = 3 - 8 \text{ g/L)} \]  \hspace{1cm} (3)

\[ \tau = 0.0263\text{MLSS}^{1.43}\gamma^{0.49} \text{(MLSS conc. } = 8 - 16 \text{ g/L)} \]  \hspace{1cm} (4)

The rheological behaviour of activated sludge at known mixed liquor suspended solids (MLSS) concentrations, in the absence and presence of iron(II) salts, was incorporated into the continuous phase modelling using the above correlations.

The pressure loss through the hollow fibre membrane bundle can be described using a porous media approach (Wang et al., 2010):

\[ S_i = \sum_{j} C_{ij} \frac{1}{2} \rho v_{avg} v_j \]  \hspace{1cm} (5)

where \( C_i \) is the inertial resistance factor which can be determined by measuring pressure drop across hollow fibre bundles with different geometrical characteristics (i.e. fibre diameter and packing density) at different flow velocities and viscosities. For flow direction parallel to the hollow fibre bundles (Liu et al., 2015):

\[ \frac{\Delta p}{L} = 1.98 \text{Re}^{-0.51} D_h^{-1.51} \frac{1}{2} \rho u^2 \]  \hspace{1cm} (6)

where \( D_h \) is the hydraulic diameter of the hollow fibre membrane bundle (Wang et al., 2010).

For flow direction perpendicular to the hollow fibre bundles:

\[ \frac{\Delta p}{L} = 13.5 \left( \frac{A\sqrt{\mu}}{V\sqrt{u}} \right)^{0.905} \cdot \frac{1}{2} \rho u^2 \]  \hspace{1cm} (7)

where \( A \) is the total membrane surface area. \( V \) is the free volume of the module (volume not occupied by hollow fibres), and \( \mu \) is the viscosity of the liquid phase.

The empirical correlations shown here were calibrated for hollow fibres with diameters ranging from 1.3 to 2.4 mm and packing densities from 200 to 560 m²/m³, and for non-Newtonian fluids with viscosities from 0.8 to 2.1 mPa.s at 100 s⁻¹ at velocities from 0 to 0.35 m/s. Therefore, the macroscopic characteristics of the hollow fibre membrane module in the fluid domain can be incorporated in the CFD models by adding a source term to the momentum equations in \( x \), \( y \) and \( z \) directions.

Calculation of Shear Stress

Shear stress on the membrane surface can be calculated from the tangential flow velocity outside the boundary layer using the wall-function approach (ANSYS, Inc., 2012):

\[ u^+ = \frac{U_t}{u_r} = \frac{1}{k} \ln \left( \frac{y^+}{C} \right) + C \]  \hspace{1cm} (8)

where \( u^+ \) is the velocity inside the boundary layer, \( U_t \) is the velocity tangent to the wall at a distance of \( \Delta y \) from the wall, \( u_r \) is the friction velocity, \( k \) is the von Karman constant and \( C \) is a log-layer constant depending on wall roughness (natural logarithms are used). The dimensionless distance \( y^+ \) from the wall is defined as:

\[ y^+ = \frac{\rho A y u_r}{u} \]  \hspace{1cm} (9)

while the wall shear stress, \( \tau_{w} \), can be calculated from the friction velocity:
\[ u_r = \left( \frac{\tau_w}{\rho} \right)^{1/2} \]  

(10)

In the current simulations, automatic wall treatment was used, i.e. \( \Delta y = \Delta n \)

where \( \Delta n \) is the distance between the first and the second grid point off the wall (i.e. membrane surface). Therefore, in the current paper, \( U_t \) was calculated and used to describe the shear stress on the membrane surface.

RESULTS AND DISCUSSION

Effect of Bubble Size

The CFD simulated area weighted membrane surface shear stress for the MBR in the absence of iron addition was 0.68 (± 0.30) Pa for the standard scenario when bubble size was 5mm. This value was consistent with the values obtained from an earlier study in which a similar aeration intensity had been used (Fulton et al. 2011). Ratkovich et al. (2012) reported CFD simulated shear stress in the range of 0.3 – 0.9 Pa. Maintaining a constant number of bubbles while increasing the bubble size from 5 mm to 10 mm resulted in an increase in area weighted average liquid velocity tangent to the membrane surface (i.e. \( U_t \), abbreviated as liquid velocity in later sections) from 0.13 m/s to 0.40 m/s (Figure 3). This indicated that a single coarse bubble could produce larger shear stress on the membrane surface than a number of fine bubbles. The increase in liquid velocity (and shear stress) was found to occur at all sections along the length of the membrane (i.e. in positive \( y \) direction; Figure 4).

![Figure 3: Comparison of liquid velocity with different bubble sizes](image_url)

![Figure 4: Liquid velocity along the membrane surface in \( y \) direction with different bubble size](image_url)

Configuration of Air Diffusers

Air diffusers can be oriented either with the nozzle facing the membrane elements (\( \alpha = 0^\circ \)) or with the nozzle facing the bottom of the tank (\( \alpha = 180^\circ \)). When \( \alpha = 0^\circ \) the development of dead zones in the lower section of the membrane modules was more pronounced when liquid velocities were less than 0.05 m/s (with minimum shear of 0.03 Pa) (Figure 5a). Changing the nozzle direction to 180\(^\circ\) was found to reduce the occurrence of dead zones and increase membrane shear stress by 12% (Figure 5b). Directing the nozzles toward the bottom of the tank facilitates the distribution of air bubbles and creates vortices and therefore promotes turbulence before the bubbles enter the hollow fibre bundles.

![Figure 5: Comparison of liquid velocity near membrane surface with different nozzle direction: (a) \( \alpha = 0^\circ \); (b) \( \alpha = 180^\circ \)](image_url)
In the standard scenario (i.e. the current design of the air diffusers and membrane module of the pilot plant), high liquid velocity (i.e. shear stress) was confined to the region above the diffusers but not widely spread in the horizontal direction, leading to the presence of an unsparged area in the lower half of the membrane cassettes that may cause severe clogging (Figure 6a). This uneven distribution of near membrane liquid velocity was greatly improved when locating the air diffusers parallel to the membrane cassettes (φ = 0°) (Figure 6b and 6c). Compared to perpendicular aerators, the average flow velocity was slightly increased in the upper section of the membrane cassettes for the case of φ = 0° and d1 = 0 mm (i.e. aerators parallel and in-line with membrane modules). Membrane modules configured in “top-out” permeation mode exhibited higher local permeate flux and hence higher fouling potential at sections closer to the permeate header (i.e. the upper sections) (Chang and Fane 2001). Therefore, any increase in shear stress over this region would reduce cake layer buildup. Interestingly, a 6.7% decrease in average flow velocity in the upper half of the membranes was observed when the aerator was positioned parallel to the membranes (φ = 0°) while alternating the location in between two membrane elements (i.e. d1 = 23 mm). Therefore, by improving the distribution of near membrane flow velocity without decreasing the shear stress in the upper half of the membranes and the aeration diffusors positioned parallel and in-line with membrane modules exhibited superior performance compared with other aerator orientations and layouts.

**Distance of Aerator to Membrane Module**

Area-weighted average flow velocity was found to decrease by 25 % (0.129 m/s vs. 0.097 m/s) when changing θ from 0° to 90° under the same aeration intensity (Figure 8), indicating a lower aeration efficiency for a horizontally aligned membrane module. These findings were consistent with the conclusion drawn by Chang et al. (2002) from bench scale experiments. The presence of hollow fibre membranes caused a significant resistance to the flow field. Pressure drop caused by the membrane bundle with fibres orientated vertically to the main liquid flow direction was much larger than the y-direction) (Figure 7). Increasing d2 to 100 mm facilitated the distribution of shear on the lower half of the membrane without the shear on the upper half of the module being compromised (Figure 7). This is because the additional distance between the diffusers and modules provided more space for air bubbles to disperse horizontally before entering the membrane module which helps constrain the bubbles to travel along the membranes with resultant more uniform air volume fraction and more even shear stress profile.

**Orientation of Membrane Module**

Hollow fibres in commercial modules can either be horizontally aligned θ = 90° or vertically aligned θ = 0° relative to the side walls of the tank. The area-weighted average flow velocity was found to decrease by 25 % (0.129 m/s vs. 0.097 m/s) when changing θ from 0° to 90° under the same aeration intensity (Figure 8), indicating a lower aeration efficiency for a horizontally aligned membrane module. These findings were consistent with the conclusion drawn by Chang et al. (2002) from bench scale experiments. The presence of hollow fibre membranes caused a significant resistance to the flow field. Pressure drop caused by the membrane bundle with fibres orientated vertically to the main liquid flow direction was much larger than
that through a bundle parallel to the flow at the same packing density (Wang et al. 2010). Consequently, horizontally aligned membrane modules cause larger flow resistance which retards the liquid velocity up through the filtration zone and reduces circulation of liquid flow around the membrane modules.

**Use of Baffles in the Filtration Zone**

Baffles can be placed in the filtration zone between module racks or cassettes to concentrate the flow of bubbles in the volume occupied by the membranes. CFD simulations indicate that the area-weighted liquid velocity decreased from 0.129 m/s in the presence of a baffle to 0.113 m/s in the absence of a baffle (Figure 9). This 14% decrease suggested the important role of baffles in maintaining an efficient bubble induced shear. Moreover, a more significant increase of 32% was found for the upper section of the membrane module. The upper section of a vertical membrane module is closer to the suction pump. Consequently the upper section of the membrane fibre often experiences a higher transmembrane pressure which can make this region more susceptible to fouling (Chang and Fane 2001). These results suggest that the use of a baffle affords the opportunity to reduce the fouling potential in this region. In systems with a baffle, the liquid in the region above the aerators is displaced by the rising bubbles travelling to the top of the tank within the membrane channel. The baffle constrains and directs the rising flow and divides the flow field into two regions: an ascending flow region inside the baffle and descending flow region outside the baffle (Figure 9a and c). When the baffle was removed, the ascending flow region became more dispersed and moved towards the wall (dark blue zone in Figure 9b and d) resulting in a lower flow velocity in the ascending flow zone which reduced the effectiveness of the aeration system.

We can therefore conclude that baffles prevent the movement of air bubbles away from the membrane modules and promote high shear conditions at the membrane surface. The construction of the baffle in full scale MBRs however would incur additional cost. An alternative solution would be to optimise the distance between the wall of the filtration tank and the membrane module in order to achieve a similar effect in the absence of a baffle. CFD simulations were carried out on a series of module to wall distances. In these simulations, air diffusers were positioned parallel to membrane elements (i.e. $\phi = 0^\circ$, $d_i = 0$ mm). The design variable assessed here is the ratio of distance of membrane module to wall ($w_1/w_2$) to the width of membrane module ($w_2$). Results from simulations suggested highest liquid velocity occurred when $w_1/w_2 = 0.6$. The area-weighted liquid velocity decreased by 30% (from 0.050 m/s to 0.035 m/s) when this ratio decreased from 0.6 to 0.2 because the circulation of upward flow was significantly inhibited by the wall (Figure 10). No further improvement in shear conditions
was observed by increasing this ratio to 0.8. However, it was found that, even at the optimum value of 0.6, the average liquid velocity was still lower than that obtained in the presence of the baffle (Figure 11).

![Figure 11: Average liquid velocity as a function of $w_1/w_2$.](image)

**CONCLUSION**

A three-dimensional two-phase CFD model, which incorporated rheology models to represent the behaviour of mixed liquor, and porous media models to describe the macroscopic characteristics of hollow fibre membrane module to the fluid field, was developed to assess the effects of different design variables, including aeration systems, module configuration, and filtration tank geometry on the efficiency of bubble induced shear at the membrane surface.

CFD simulated results suggested higher shear conditions can be achieved via:

- Maintaining the same number of bubbles while increasing the size of the bubbles;
- Locating the air diffusers 100 mm below, parallel and in line with the membrane module, with nozzles facing down; and
- Having the hollow fibres orientated vertically in the filtration tank and baffles surrounding the membrane module.

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 and features including baffles.

**ACKNOWLEDGMENT**

Funding provided by the Australian Research Council, Water Research Australia, Sydney Water and Beijing Origin Water through ARC-Linkage project LP100100056 is gratefully acknowledged. The authors would like to thank Sydney Water for facilitating the access to the wastewater treatment plants and Beijing Origin Water for providing membrane modules. Special thanks to Mr Keng Han Tng and Mr Ben Zhou for their assistance in the operation and maintenance of the pilot plant at Bondi STP.

**NOMENCLATURE**

- $A$: Membrane total surface area
- $C$: Log-layer constant
- $C_{ij}$: Three-dimensional inertial resistance factor
- $D_h$: Hydraulic diameter of module (see Wang et al. 2010 for details)
- $d_1$: Distance between aerators and the adjacent membrane in $x$ direction
- $d_2$: Distance between aerators and the bottom of membrane module
- $k$: von Karman constant
- $L$: Length
- $\Delta n$: Distance between grid point
- $\Delta p$: Pressure drop
- $S_i$: Source term in porous media model
- $U_t$: Flow velocity tangent to the wall at a distance of $\Delta y$ from the wall
- $u$: Flow velocity
- $u'$: Flow velocity inside boundary layer
- $u_s$: Friction velocity
- $V$: Free volume in membrane module
- $v_{mag}$: Maximum velocity in the channel
- $v_j$: Velocity vector
- $w_1$: Distance of membrane module to wall
- $w_2$: Width of membrane element
- $\gamma$: Dimensionless distance from wall
- $\Delta y$: Depth of boundary layer

**REFERENCES**


flows through submerged membrane units with irregular fibre arrangement, Journal of Membrane Science 384: 184-197


Fulton, B.G., Redwood, J., Tourais, M. and Berube, P.R. (2011) Distribution of surface shear forces and bubble characteristics in full-scale gas sparged submerged hollow fiber membrane modules. Desalination 261, 128-141


