<table>
<thead>
<tr>
<th>Title</th>
<th>Simulation studies of process scale membrane aerated biofilm reactor configurations</th>
</tr>
</thead>
<tbody>
<tr>
<td>Authors(s)</td>
<td>Casey, Eoin</td>
</tr>
<tr>
<td>Publication date</td>
<td>2007-01</td>
</tr>
<tr>
<td>Item record/more information</td>
<td><a href="http://hdl.handle.net/10197/4455">http://hdl.handle.net/10197/4455</a></td>
</tr>
</tbody>
</table>
Simulation studies of process scale membrane aerated biofilm reactor configurations

E. Casey*
* School of Chemical and Bioprocess Engineering, University College Dublin, Belfield, Dublin 4, Ireland.
(E-mail: eoin.casey@ucd.ie)

Abstract
In the membrane aerated biofilm reactor oxygen diffuses through the membrane into the biofilm where oxidation of pollutants, supplied from the biofilm side of the membrane takes place. Despite numerous studies at the laboratory scale showing the potential of the technology, efforts to scale-up the technology to process scale have been hampered by problems such as excessive biofilm growth and consequent flow distribution problems. This paper presents results of simulation studies which utilise Computational Fluid Dynamics (CFD) to examine performance of several technical scale MABR design configurations. The simulations suggest that plate-and-frame membrane configuration with a suitable liquid inlet distributor will deliver superior performance compared to hollow fibre configuration with respect to liquid flow distribution.

Keywords
Biofilm, reactor, mixing, CFD.

INTRODUCTION
One of the key advantages of biofilm-based wastewater treatment processes is the potentially high volumetric reaction rate that can be attained due the high specific biomass concentration. Unfortunately, this advantage is rarely exploited in full-scale processes due to the difficulty in supplying oxygen at a sufficient rate to the pollutant degrading microorganisms. Under typical conditions, the depth of oxygen penetration into a biofilm is in the order of 100 microns, if the biofilm is greater than this thickness, then a fraction of the biomass is unavailable for pollutant oxidation. In the membrane aerated biofilm reactor (MABR) an oxygen permeable membrane supports biofilm growth. Oxygen diffuses across the membrane into the biofilm where oxidation of pollutants, supplied from the biofilm side of the membrane, takes place. The direct contact between the biofilm and the membrane affords oxygen transfer directly into the biofilm matrix without bubble formation. The advantages of the concept are twofold: by pressurizing the membrane with pure oxygen, sufficient oxygen to meet the need of the microorganisms can be provided, and up to 100% oxygen transfer efficiency can be achieved. This membrane-aerated biofilm concept was first described by Yeh and Jenkins (1978) who reported performance data demonstrating superiority over both conventional biofilm reactors and activated sludge systems under conditions of high organic loading. Several subsequent studies at laboratory scale have demonstrated that the process to outperform high-rate biotreatment processes, with respect to both COD removal rate and oxygen transfer efficiency (Debus et al 1992, Pankhania et al, 1994, Rothenmund et al, 1994, Brindle et al, 1999, Casey et al, 1999, Semmens et al, 2003, Terada et al, 2004). Although the MABR concept was developed over 30 years ago, efforts to scale-up to process scale
have been unsuccessful. The reasons for this can be largely attributed to excess biomass which can ‘choke’ the reactor, reducing the contact between the wastewater and the biofilm and deterioration in performance. Semmens et al (2003) reported a pilot-plant study where the membrane fibers became attached to each other with eventual choking of the reactor with excess biomass. The difficulty in controlling biofilm thickness resulted in a drastic reduction in the interfacial area between the wastewater and the microorganisms.

The objective of this study is to assess the performance of process scale conceptual MABR configurations using Computational Fluid Dynamics (CFD). CFD is essentially the numerical computation of the governing equations which describe fluid flow and conservation equations. The advantage of the approach is that it provides the flexibility to readily simulate changes in design and operational parameters in order to predict system response.

METHODS
Reactor Conceptual Design
The design objective is to maximise the volumetric biomass concentration whilst simultaneously ensuring that the biofilm-liquid interfacial area is maximised. Previous MABR studies (for example, Pankhania et al, 1999) based on the HF design have tended to adapt proprietary hollow-fibre membrane modules which are designed to maximise the area to unit volume. Because of the minimal intra-fibre spacing, clogging has been a significant problem.

![Figure 1 Schematics of (a) hollow fibre configuration and (b) plate and frame.](image)

In this study, the conceptual design of the MABR was based on the objective of achieving an optimal balance between the need for retention of thick biofilm, good contact between wastewater and biofilm (voidage in the region of 40% at maximum operational biomass hold-up) and the ability to supply sufficient oxygen to the entire biofilm. Although dependent on several factors, oxygen penetration into the biofilm is expected to reach 2.0 mm with pressurised silicone membranes. Laboratory data and reaction-diffusion modelling supports these figures (Casey et al, 2000). Based on these assumptions and allowing for a membrane thickness of 0.2 mm and appropriate
spacing, calculations suggest that under optimum conditions biofilm will occupy up to 40% of the entire reactor volume with a voidage in the region of 40% allowing the potential for good liquid distribution and sufficient overall reasonable high biomass hold-up. Assuming a typical biofilm dry weight density of 50 kg m\(^{-3}\), the overall biomass concentration in the reactor is expected to be in the region of 20 kg m\(^{-3}\).

Two broad categories of membrane module were considered, a hollow fibre (HF) and plate-and-frame (PF) as shown in figure 1. The HF design incorporates 388 3mm tubes in a cylindrical module of 170cm diameter and 500 cm length. Simulations assumed a 2mm thick biofilm evenly distributed on the membranes. Under these conditions the volume fraction of liquid in the module would be 0.38. Simulation HF-1 considered a single liquid inlet adjacent to the outer wall of the module (inlet 1 on Figure 1a), Simulation HF-2 considered two inlets (inlet 2 on Figure 1a) located close to the centreline of the cylindrical module. The PF design incorporates 36 flat-sheet membranes in a horizontally aligned plate-and-frame configuration. Each thin plate comprises a gas compartment with an inlet and outlet and is sandwiched between two planar membranes, which seal the gas compartment. Multiple plates are stacked in a frame apparatus (Figure 1b). The plate spacing creates a channel in which wastewater flows via a manifold in the frame. The wastewater comes into direct contact with the biofilm, which grows on the outer surfaces of the membrane.

Simulation PF-1 considered a single liquid inlet adjacent to the outer wall of the module (inlet 1 on Figure 1b), Simulation PF-2 considered triple inlet manifold (indicated on Figure 1b) equally spaced across the inlet region of the module.

For all simulations the wastewater feed rate was set to correspond to a one hour liquid retention time. Liquid density and viscosity was assumed to be that of water. The flow was assumed to be at steady-state and it was assumed that the membrane material was impermeable to liquid.

**CFD Simulations**

The commercial CFD package FLUENT (Fluent Inc. Lebanon NH, USA) was employed for all simulations and post-processing. The computational grids were generated in the FLUENT pre-processor GAMBIT. The mesh size was approximately 500,000 and 400,000 for the HF and PF configurations respectively. The node spacing was refined adjacent to regions where velocity gradients were expected to be high, such as the inlet and outlet of the modules. Grid independence checks were undertaken prior to running the final simulations.

**Figure 2**: Representative sections of the computational mesh for both HF and PF configurations.
Figure 2 shows representative sections of the computational mesh. Laminar, steady-state conditions were applied. Solution discretisation levels were set at 2nd order for pressure and momentum, in conjunction with the segregated solver. The SIMPLE algorithm was applied for pressure–velocity coupling in the segregated solver. The PRESTO algorithm was used for pressure discretisation. In order to investigate short-circuiting and mixing conditions in the membrane modules a tracer stimulus–response experiment was simulated. Once the fully converged steady-state flow simulations were completed the species transport model was employed in an unsteady-state simulation. By using a tracer with identical physical properties of the bulk fluid, a short concentration pulse at the inlet of the computational domain was introduced, computing the time dependent concentration of the tracer at the outlet.

RESULTS
Figure 3 shows representative velocity vectors in a two vertical planes in the HF module. Liquid velocity can be seen to be highest adjacent to the module outer wall. In the vertical plane closest to the liquid inlet there may be enhanced convection in the intra-membrane spaces when compared with the main body of the module.

![Velocity vectors in the HF module](image)

**Figure 3** Velocity vectors in the HF module for (a) a plane adjacent to the liquid inlet and (b) in a plane corresponding to half the axial distance.

In order to visualise regions of poor velocity distribution, iso-surfaces showing regions in the computational domain where the velocity exceeds a certain value were prepared. Figures 4 and 5 shows iso-surfaces for both PF and HF configurations. In the HF module regions of higher than average flow velocity are found at the module axial periphery, i.e. adjacent to the walls of the module and in the general areas of the inlet and outlet. For the PF configuration, the liquid flow follows a more uniform distribution. In order to further investigate flow distribution, tracer injection simulations were undertaken by solving the transient species equations. A species monitor was configured at the liquid out flow and the time dependent tracer response was recorded and converted to normalised tracer response curves. Figure 6 shows the results of these simulations, where it is apparent that there are significant differences between flow distribution in HF and PF modules and for the PF modules, flow
distribution can be improved by employing multiple inlet manifolds to distribute influent liquid across the PF inlet region.

**Figure 4** iso-surfaces of velocity magnitude set at 0.12 mm/s in the hollow fibre configuration for simulations HF-1 (a) and HF-2 (b). Regions of the computational domain where the velocity exceed the iso value are dark, light regions correspond to low velocity.

**Figure 5** iso-surfaces of velocity magnitude set at 0.21 mm/s in the plate-and-frame configuration for simulations PF-1 (a) and PF-2 (b).
Figure 6 Normalised tracer curves for a step input for (a) hollow fiber configuration and (b) plate-and-frame configuration. Dashed line indicate simulation 1 and continuous line indicates simulation 2 in each case respectively.

DISCUSSION
The effect of liquid flow distribution in MABR reactors can be categorised into both reaction rate effects and propensity for biomass choking.

With regard to reaction rate, some previous experimental work has investigated these parameters. Brindle et al (1999) in a pilot scale MABR operated under both plug-flow and completely mixed conditions. Completely mixed operation is associated with high velocities and a corresponding reduction in the boundary layer thickness at the biofilm liquid interface hence promoting reaction rate, on the other hand this operation mode will result in uniform but low COD concentrations throughout the reactor which, if approaching the COD saturation constant, may result in substrate limitation and thereby diminished performance. Plug flow operation may suffer from increased mass transfer resistance but inevitable COD axial and radial gradients in the membrane module. If MABRs are to be implemented at process scale it seems likely that operating conditions with respect to fluid mixing need to be related to the organic loading and removal rates in order to achieve satisfactory process efficiencies.

Regarding the effect of flow distribution on biofilm overgrowth, the simulation results reported here suggest that there will be regions of higher and lower than average velocity within the membrane modules. The effect of velocity on biofilm thickness is a complex relationship encompassing both shear effects as well as mass transfer effects. However, it seems certain that uneven velocity distribution will lead to regions of biofilm overgrowth and consequent localised choking of the reactor. Experimental evidence of this effect was reported by Semmens et al, 2003 in a hollow fibre module. Attempts to ameliorate the choking with air scouring were only partly successful. To overcome biomass overgrowth, several methodologies have been proposed including occasional scouring, backwashing or high shear regimes (Yeh and Jenkins, 1978, Rothemund et al, 1994, Pankhania et al, 1994, Brindle et al, 1999.) While Brindle (1999) reported effective biofilm control with regular washing and air
scouring, it is important to highlight that process performance in an MABR inevitably linked to the time taken for the biofilm to recover from any cleaning regime. In this regard, module design with a high specific surface area and relatively thin biofilm may recover relatively quickly. Brindle et al, (1999) reported removal efficiencies returning to normal within one half to two HRTs.

Conclusions
Scale-up of the MABR process is currently hampered by problems with biomass clogging. This study is an effort to simulate the performance of MABR conceptual designs to overcome this problem. It is expected that prioritisation of this area will lead to progress on the long-term route to the development of a viable and effective MABR process. The results of this preliminary study suggest that liquid distributor design may make the most significant impact on efforts to ensure good flow distribution in MABR reactors, but is also important to consider that the achievement of uniform mixing in membrane modules may impact not only the propensity for biomass choking, but also the local reaction rate by reducing the COD level to a levels close to the saturation constant.

References