Fluid dynamics in a full-scale flat sheet MBR, an experimental and numerical study

Lasse Sørensen and Thomas Ruby Bentzen

ABSTRACT

Fluid dynamics is used for fouling mitigation in membrane bioreactors (MBRs), whereby a proper understanding of the fluid dynamics is of great interest. The influence of fluid dynamics has led to the use of computational fluid dynamics for optimizing MBR systems. In this work, a model has been validated for flat sheet membranes, with use of the Eulerian multiphase method. The model is validated against a comparable setup where the liquid velocities are measured with a laser Doppler anemometer (LDA). Furthermore, the Eulerian multiphase approach is validated against the more numerical direct volume of fluid (VOF) approach with sludge properties for the liquid, resulting in an error between the models of less than 2% for the wall shear stresses. The VOF model further showed that the horizontal components contribute significantly to the total wall shear stresses. The model has been applied to a full-scale setup for studying the effect of deflecting membranes as deflections have been seen in production. Minimizing the deflection of the membrane sheets was crucial to achieve a good operating condition as a deflection of 2 mm in a setup with a gap of 7 mm decreased the wall shear stresses with as much as 40% on average on the specific membrane surface.

Key words | flat sheet membranes, fluid dynamics, fouling mitigation, MBR, rising bubbles, shear stress

INTRODUCTION

Membrane bioreactors (MBR) have become more popular for treating wastewater during the last decade. In China, the first +10,000 m$^3$/d plant was installed in 2006, while the number of plants at the end of 2014 had increased to around 130 (Xiao et al. 2014). With MBR being deployed on a larger scale for municipal wastewater treatment, the research in optimization has followed, and different types of configurations are applied. One of the common types of configurations is the hollow sheet or flat sheet (FS) configuration, where hollow sheets with membranes are submerged in the activated sludge. Such systems are typically operated with a low transmembrane pressure (TMP) from the hydraulic head which can be assisted by applying suction to the channels in the hollow sheets. The low-pressure setup used for flat sheet membranes gives a low flux through the membranes leading to a low transport of particles to the membranes as the drag force increases with flow velocity (Koustrup Jørgensen 2014).

A general drawback for MBR is the fouling of the membranes, which leads to increased energy consumption and expenses for maintenance. To reduce the fouling of the membranes, one typically used approach is air scouring, where bubbles are induced to scour the membrane surfaces. Aeration is the major energy consumer in operating MBR, typically using 35% to 50% of the total energy and in some cases even higher fractions (Krzeminski et al. 2012).

The antifouling effect from the air scouring is correlated with wall shear stresses, which is described from the shear rate \( \dot{\gamma} \) and the effective viscosity \( \mu_{\text{eff}} \) with Equation (1). For non-aerated systems, an increase in flux with higher wall shear stresses has been found (Jørgensen et al. 2014). For setups where rising bubbles induce the shear stress, the determination of wall shear stresses is a more complicated task. It has, though, been shown for clay suspensions in flat sheet configurations that the bubble-induced wall shear stresses to enhance the flux of permeate through the membrane (Ducom et al. 2002). In general, there is a consensus in the literature that higher wall shear stresses have a positive effect regarding fouling mitigation of membranes, though at some point a threshold is reached where the positive effect will fade or even have adverse effects (Böhm et al. 2012).

\[
\tau = \dot{\gamma} \mu_{\text{app}}
\] (1)
The shear stress force is acting parallel to the membranes and depends on the shear rate as seen in Equation (1). Whereby increasing wall shear stresses also entails increasing shear rate, which is relevant as the lift of particles depends on the shear rates (Saffman 1965). The relation of the effect on fouling mitigation from shear stress and shear lift force is not clear from the literature, and only the shear stress is studied in this work, which is also an indication for the shear lift.

Several studies have been published regarding optimization of hollow sheet systems, regarding geometry and aeration strategies. It has been found that larger bubbles increased the mass transfer until a threshold with bubbles larger than 60 mL where no further increase was found for a setup with a 20 mm gap (Zhang et al. 2009). In the same study, it was found that a frequency of bubbles up to 0.4 Hz increased the mass transfer, where afterwards only a small effect was found. It has been found that the smaller the gap size, the higher wall shear stress on the surface on the membranes for gaps in the range of 3–7 mm for different bubble sizes (Prieske et al. 2012). This was the result of a study with fixed liquid velocity, which will be difficult to achieve in full-scale setups as the gap size will also influence the liquid. A positive effect on transmembrane pressure from larger bubbles and higher aeration intensity has been shown for a setup with a yeast suspension surrogate for sludge (Ndinisa et al. 2006).

As it is difficult to study the local flow patterns in full-scale systems experimentally, computational fluid dynamics (CFD) is a useful tool. The use of CFD gives the possibility to study parameters which are difficult to measure experimentally as the wall shear stresses.

CFD is a well-known tool for studying fluid dynamics in MBR systems. The multiphase modelling can be split into different types of modelling being the volume of fluid (VOF) approach, the Euler–Lagrange approach and the Eulerian multiphase method (EMP). In this work, the EMP and the VOF approach are used. The VOF method resolves the interface between the two phases and the entire velocity profile around every single resolved bubble. This makes it a very computationally heavy model which is typically used for single bubble modelling, where it can describe the shape of the bubble and the wall shear stresses from the bubble (Wei et al. 2013). Since the VOF model resolves the bubbles, it requires small cells and low timesteps, making it too computationally heavy for full-scale setup. The VOF method does not include coalescence hindrance, meaning that care should be taken when used for multi-bubble setups, as it might overpredict the coalescence.

The rheology of the sludge is also an important parameter when modelling MBRs, as it influences the flow of the sludge and the oxygen transfer (Amaral et al. 2018). Due to the high aeration demand for antifouling, no problems have been found with oxidation in active plants with the current setup and the focus in this work has not been on the oxidation but rather on the flows. Sludge has been found to be thixotropic (Baudez 2008) and visco-elastic (Chhabra et al. 2008), though the most important behaviour is the shear thinning property (Rosenberger et al. 2002), which is the only rheological property included in this work.

For modelling the flows in full-scale flat sheet setups, the EMP has been used (Ndinisa et al. 2006; Khalili-Garakani et al. 2011; Amini et al. 2015; Yang et al. 2016). The EMP does not resolve the bubbles but relies on modelling of interphase momentum transfer, which in some degree is based on empirical correlations. All the listed works are using the EMP, but the exact method of implementation of interphase momentum transfer and turbulence differs from study to study. As there is no consensus of how to model FS MBR systems, the validation of the model is a crucial part of this study. Especially the fact that the interphase transfer models are not developed for geometries confined in one direction makes the accuracy uncertain. For validating the numerical models, an experiment has been set up which is comparable to the full-scale modules. In the experimental setup, the velocities are measured with a laser Doppler anemometer (LDA).

During the production of FS membrane systems it has been observed that the sheets tend to start deflecting if the hollow sheets are exposed to heat during the production. This deflection occurs if, e.g. if the membranes are welded on the sheets. An example of the deflecting membranes is illustrated in Figure 1. In this study, the validated model has been applied to a full-scale setup to evaluate the influence of such imperfections, e.g. areas with a lower crossflow velocity due to changed flow will also have a lower back transport of particles in the retentate resulting in less efficient fouling mitigation.

The study evaluates the usability of different approaches for CFD in FS MBR. Where the high resolution but computation heavy VOF model is used to give data on small scales, including a study of the different components of the wall shear stresses around the single bubbles in a bubble swarm setup. This is an area which is only understood for single bubble setups due to the very small spatial scale and high temporal scale making it difficult to study experimentally. The new knowledge about the flow on the small scale can help to increase the understanding of fouling mitigation from aeration.
The current problems with the use of the Eulerian multiphase model are that it is mainly validated for unconfined geometries, making the accuracy uncertain when used for modelling of the narrow gaps like in FS MBRs. The validation of this model gives the ability to model larger setups and even full tanks where it can be used in the areas of membranes, while other models will still need to be validated for the areas outside the membranes. In this work, the Eulerian multiphase model is used to evaluate overall flow patterns in a full-scale setup and the influence of the deflecting sheets that are seen in the production of membrane sheets.

**METHOD**

**Experimental setup**

The experimental setup for validation was made in a box of $0.76 \times 0.189 \times 1.36$ m with the rest of the dimensions illustrated in Figure 2. The sheet with the height of 1 m was used to create a gap of 7 mm, corresponding to a typical gap in an FS membrane setup. The liquid used in the setup was tap water at around 23°C. In the top and the bottom of the setup, there was a free passage for a recirculating flow. Air was blown into the system through a pipe with five holes to release the bubbles into the system where they were forced into the gap with a sloping sheet. The experiments were made with hole sizes of 2, 4 and 8 mm. The experiments were conducted with three different flow rates, which were 9.8 L/min, 11.8 L/min and 13.8 L/min. The flow rates correspond to a specific aeration demand per membrane area ($\text{SAD}_m$) of 0.39, 0.47 and 0.54 m$^3$/m$^2$/h. This is in the range of typical values for existing systems where most of the setups are in the range 0.30–0.80 (Prieske et al. 2012). It is important to note that the aeration demand depends on the height of membranes, as higher units have a lower aeration demand as the air travels a longer path on the membrane surface. For the setup from Alfa Laval, the $\text{SAD}_m$ is in the range of 0.18–0.54 m$^3$/m$^2$/h, with the flow normalized to the height to be in the range 0.48–0.72 m$^3$/m/h. The different aeration demands are listed in Table 1. The normalized flow is the specific aeration demand per membrane length $\text{SAD}_l$ which for the experiments was 0.39, 0.47 and 0.54 m$^3$/m/h corresponding well with the setup from Alfa Laval.

### Table 1: Aeration flows for a different number of stacked modules for Alfa Laval’s MBR modules

<table>
<thead>
<tr>
<th>Setup</th>
<th>$\text{SAD}_m$ [m$^3$/m$^2$/h]</th>
<th>$\text{SAD}_l$ [m$^3$/m/h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 module</td>
<td>0.48–0.54</td>
<td>0.48–0.54</td>
</tr>
<tr>
<td>2 modules</td>
<td>0.24–0.30</td>
<td>0.48–0.60</td>
</tr>
<tr>
<td>3 modules</td>
<td>0.18–0.24</td>
<td>0.54–0.72</td>
</tr>
</tbody>
</table>

Figure 1 | Illustration of deflecting sheets on test setup.

Figure 2 | Illustration of experimental setup.

Downloaded from https://iwaponline.com/wst/article-pdf/78/10/2077/516353/wst078102077.pdf by guest
The data for validation were obtained with a LDA mounted on a traverse system to measure in a grid of predefined points. The measurements were made in the centre of the gap with 3.5 mm to each wall. The location of the measurement was 250, 500 and 750 mm above the bottom of the membrane with 15 points equally distributed above the centre outlet. The measurements were made over a period of 60 s per measurement point. This resulted in between 1,000 and 15,000 velocity measurements for each point, from which the mean vertical velocity was found.

**Setup for CFD**

For validation of the model, the measurements from the experimental setup have been used. It is sought to validate the EMP as it is computationally light and applicable for full-scale modelling. The EMP is in some degree based on empirical correlations; the model has also been validated against the results of a VOF model which is a much more direct numerical method, giving the possibility to validate the EMP model with sludge properties. The approach for the modelling of the fluids was with the Reynolds-averaged Navier–Stokes equation. The turbulence has been modelled with the realizable k-epsilon model for all setups. The wall treatment was made with the two-layer with the Wolfhstein variant (Wolfshtein 1969).

For the EMP, a coarse mesh is used to make it applicable to full-scale modelling, entailing that the wall-layer is not resolved. Therefore, wall shear stresses have been validated against the result of the VOF model where the wall-layer is resolved by the mesh.

Four different setups are used, which is the EMP with both water and sludge properties as well as the VOF method with both water and sludge properties. The setups with water have been modelled to validate the modelling setups against measurements, while the sludge properties have been used as that is close to what would be found in treatment plants. The rheology of the sludge is described with the power law function in Equation (2).

\[
\mu_{eff} = k \gamma^n
\]

where \( \mu_{eff} \) is the effective viscosity, \( k \) is the consistency factor, \( \gamma \) is the shear rate and \( n \) is the power law exponent.

The rheology is determined corresponding to a sludge concentration of 10 gTS/L (Rosenberger et al. 2002). The properties for the different fluids are listed in Table 2.

The settings specific to the different multiphase methods are described in the following.

### Table 2 | Properties of fluids

<table>
<thead>
<tr>
<th>Fluid</th>
<th>Density [kg/m³]</th>
<th>Dynamic viscosity [Pa s]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>1.18</td>
<td>1.86 ( \cdot 10^{-5} )</td>
</tr>
<tr>
<td>Water</td>
<td>998</td>
<td>0.89 ( \cdot 10^{-3} )</td>
</tr>
<tr>
<td>Sludge</td>
<td>1020</td>
<td>0.17 ( \cdot \gamma^{0.46-1} )</td>
</tr>
</tbody>
</table>

**Setup Eulerian multiphase method**

The setup for the EMP uses a continuous phase for the liquid and a dispersed phase for the gas. The two phases have their own set of momentum equations, while the pressure is shared between the phases. The model was run with a timestep of 0.05 s. For interphase momentum transfers, a drag force is applied. The used drag formulation is the Tomiyama drag, with a bubble size of 3 cm, which was a typical bubble size based on the experiments. The Tomiyama formulation operates with a contamination level of the liquid, where the contaminated state was used, which is comparable to tap water (Tomiyama et al. 1998). For the turbulence of the models, the realizable k epsilon model is used for both phases.

The confined geometry between the membranes leads to different physics compared to the rest of the setup where the flow is much less confined. Since several of the models build on empirical correlations, no model is valid for all the setups. Due to this, the physics are divided into two groups, where the one is used for the area between the membranes, while the other is valid for the flow in the rest of the volume.

A turbulent dispersion force is used in the entire setup, with a calibration constant of 100 between the membranes, while it is 1 in the rest of the setup. With the turbulent dispersion force, the added mass is also used. Furthermore, the wall lubrication force is used to force the air away from the walls on the side. It has been shown that a force works in the direction normal to the wall at a distance of a few diameters of the bubble (Antal et al. 1993). Their formulation has been modified and used in the membrane area, where the component normal to the membranes is set to 0 as the bubbles fill the entire gap, entailing that it has no physical meaning in this direction. The wall lubrication has been implemented with Equation (3).

\[
F_{wall} = -C_{WL}(y_w) \alpha_d \rho_c |v_r - (v_r \cdot n)n|^2 \left/ n \right.
\]

where \( y_w \) has been modified from the original expression to be the distance the nearest wall which is not a membrane,
rather than just the distance to the nearest wall as the force is only working in this direction. The vector \( n \) is the outward facing unit normal of the wall which is not a membrane. \( \alpha_d \) is the volume fraction of the dispersed phase, \( \rho_c \) is the density of the continuous phase, \( d \) is the bubble diameter and \( v_r \) is the relative velocity. The wall lubrication coefficient is described with Equation (4).

\[
C_{WL} = \max\left(C_{w1} + \frac{C_{w2}}{y_w}, 0\right)
\]

(4)

The coefficients have been calibrated to \( C_{w1} = -2 \) and \( C_{w2} = 10 \), resulting in a force of zero from five bubble diameters from the wall.

The mesh for the Eulerian multiphase model was made with a hexahedral mesh, where only two cells of 3.5 mm in the direction normal to the membranes were used to describe the gap between the membranes for the model to be able to run on a full-scale setup. In the other directions parallel to the membranes, the cells were 7 mm, while the cells outside the membrane area had a maximum size of 28 mm.

**Setup volume for VOF model**

The VOF method is used with the high-resolution interface capturing scheme (HRIC) with a sharpening factor of 0.5 to keep a sharp interface between the two phases. For a momentum convection scheme the MUSCL 3rd order/CD convection scheme, which gave numerical errors on the pressure boundary when bubbles were present. The walls are modelled as smooth with the no-slip condition. For the VOF model, the contact angle is 0.

**Boundary conditions**

For the Eulerian multiphase model, the air inlets are made at the locations of the actual inlets, they are though made quadratic rather than circular for increased mesh quality. The inlets are made with a volume fraction of only air, while the velocity is made to fit the given setup. For the VOF model, the inlet is located at the end of the pipe, and thereby also calculating the flow inside the distribution pipe.

For the top boundary, a pressure outlet is used for the Eulerian multiphase approach with a volume fraction of 100% liquid. For the VOF model, the mesh has been extended 20 cm to model the free water surface. The boundary is a pressure boundary with a volume fraction of only air. This method had to be used due to the MUSCL 3rd order/CD convection scheme, which gave numerical errors on the pressure boundary when bubbles were present. The walls are modelled as smooth with the no-slip condition. For the VOF model, the contact angle is 0.

**RESULTS**

**Experiments**

The measurements were conducted for different inlet hole sizes and different inlet flow rates. The experiments showed that there was only a minor influence of the hole sizes in the range of 2 to 8 mm. All the setups have bubbles in the sizes of around 3 cm in diameter in the narrow gap area, while only a low amount of the air was present as bubbles smaller than the size of centimetres. The velocities measured with the LDA for the different holes sizes and different flows rates are plotted in Figure 3, where no clear trend separates the different hole sizes. Due to this, the models have only been validated for the setups with inlet holes of 4 mm. For the influence of the flow rate, it was clear that the higher flow rates did also result in higher vertical velocities.

**Validation of CFD**

The calibrated Eulerian multiphase model showed to yield slightly lower mean velocities than the measured. The modelled flow was 77–92% of the measured flow, as listed in Table 3. From the measured velocities, it is also clear that the velocities were increasing with the height in the measurement points. As the measurement points were in the centre of the setup, the higher velocities there must result in lower velocities elsewhere due to the mass balance. This flow pattern substantiates the assumption of migration of bubbles towards the centre which was also visually found during the experiments. This agrees with what has been reported in Tomiyama et al. (2002), where it is found that large bubbles migrate towards the centre of the column. It has not been implemented in this setup as the point
sorces of air gives high values for the curl around the areas where the bubbles are rising, resulting in a migration of a local scale rather than the global scale that was observed in the experiments and with the VOF method.

It is clear from Figure 4 that the shapes of the modelled velocity profiles are comparable to the measured profiles. In Figure 4, the results of the model without the wall lubrication force and the increased turbulence dispersion force between the membranes is shown. Figure 4 shows that the increased turbulence is crucial to achieving the correct shape of the velocity profiles. This is likely because the geometry is very confined in one direction and thereby minimizing the turbulence in the model. In the experiments, a chaotic raising behaviour of the bubbles was observed, which will also lead to transport of scalars in this plane.

The VOF model does also describe the shape of the velocity profile well, but as the case for the Eulerian multiphase approach does tend to underestimate the mean velocities. It is difficult to conclude if this is due to the wrong physics in the model or irregularities in the model setup. The VOF model does not use any tuning of constants or empirical correlation but resolves the flow on the scales of interest which

---

**Figure 3** | Measured vertical velocities for different hole sizes and flow rates.

**Table 3** | Measured and modelled velocities and percentage of modelled velocities compared to the measured

<table>
<thead>
<tr>
<th>Position</th>
<th>Air flow – 9.8 L/min</th>
<th>Air flow – 11.8 L/min</th>
<th>Air flow – 13.8 L/min</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Velocity [m/s]</td>
<td>Velocity [m/s]</td>
<td>Velocity [m/s]</td>
</tr>
<tr>
<td></td>
<td>Exp.</td>
<td>CFD</td>
<td>Error [%]</td>
</tr>
<tr>
<td>z = 750</td>
<td>0.81</td>
<td>0.64</td>
<td>21</td>
</tr>
<tr>
<td>z = 500</td>
<td>0.72</td>
<td>0.62</td>
<td>14</td>
</tr>
<tr>
<td>z = 250</td>
<td>0.75</td>
<td>0.61</td>
<td>19</td>
</tr>
</tbody>
</table>

**Figure 4** | Vertical velocities in the narrow gap.
in general makes it a reliable model. The VOF method does though have uncertainties concerning the coalescence of the bubbles as it is not possible to model any coalescence hindrance. It has not been thoroughly studied in this work, but it was found that a significant percentage of the air was present as bubbles in sizes in the range of 2–5 cm, while a little part of the volume fraction was present as smaller bubbles, corresponding to what was observed in the experiments.

For the setup with sludge, no experiments were conducted, as the LDA is not suitable to use for liquid that is not clear. Since the wall shear stresses are of interest, the results from the Eulerian multiphase model has been compared the more numerical direct VOF model. The ability of the VOF method to model the wall shear stresses have been validated by comparison of the velocity profile in the direction normal to the membranes. Figure 5 shows that the VOF method models the correct velocity profiles in this direction, which implies that the wall shear stresses must also have a high precision as they depend on the shear rate. The same method could not be used for the Eulerian multiphase model where the mesh does not resolve the velocity profile between the membranes but is described by the law of the wall.

Due to this, the mean values of the surface wall shear stresses from the VOF model have been compared with the EMP, where the results are listed in Table 4. With an error between the modelled wall shear stresses of less than 2% for the two different models, the Eulerian multiphase model is assumed able to model the wall shear stresses even though it does not resolve the velocity profiles.

With the shape of the velocity profiles which are comparable to the measured profiles and a model that has shown to be able to model the same wall shear stresses as the VOF model where the wall layer is resolved, the model has been used to model full-scale MBR setups, which is not suitable to do with the VOF method due to the high computational cost.

### Evaluation of fluid dynamics in FS MBR

The use of the VOF model does provide information on a much smaller temporal and spatial scale than the EMP. This high-resolution data give the possibility to study correlations, which are difficult to study experimentally.

The resolved bubbles allow the study of the different components of the wall shear stresses illustrated in Figure 6. The results showed that the vertical components of the wall shear stresses were the largest as would also be expected, but it was also found that the other components were providing a significant part of the wall shear stresses as well. With the mean value of wall shear stresses on the surface of the membrane of 2.73 Pa, the horizontal components alone provided 1.05 Pa of this. In the positive vertical direction, the maximum value of the wall shear stresses was 24 Pa, while the maximum value in the negative vertical direction and the horizontal direction was between 9 and 10 Pa. The highest wall shear stresses were found in the wake of the bubbles which has also been shown to be the case for single bubble setups (Wei et al. 2013). The wide variety of wall shear stresses is expected to be positive for fouling mitigation as the lift of deposited particulates depends on the shape of the particle, and some particles will be more sensitive to shear from other directions. This also highlights one of the main issues with the Eulerian multiphase model, which by definition does not include other components of the wall shear stresses than the ones in the general flow direction. Even though the model is the most used model for multiphase systems in FS MBR, this is an issue that has not yet been addressed.

The ratio between the viscosity of the fluid and the turbulent viscosity is also shown in Figure 6, for the setup with sludge properties. The turbulence is present in the

---

**Table 4** | Surface average wall shear stresses from CFD with a flow rate of 11.8 L/min

<table>
<thead>
<tr>
<th>Model</th>
<th>$\tau_w$ [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>EMP (water)</td>
<td>1.62</td>
</tr>
<tr>
<td>VOF (water)</td>
<td>1.60</td>
</tr>
<tr>
<td>EMP (sludge)</td>
<td>2.77</td>
</tr>
<tr>
<td>VOF (sludge)</td>
<td>2.73</td>
</tr>
</tbody>
</table>

---

**Figure 5** | Velocity profiles between the membranes in $x=230$ and $z=500$.
The present turbulence still showed that despite the narrow geometry and high viscosity the turbulence is present in the system. Thereby does the model substantiate the assumption of using the turbulence in the Eulerian multiphase model. From this, it also seems likely that the extra turbulent dispersion used with the Eulerian multiphase model is rather compensating for the chaotic behaviour of the bubbles on a large scale than actual turbulent dispersion in the system. The results show that the VOF model has some strong advantages on the smaller scales compared to the Eulerian multiphase model. With the development of computational power, this makes it an obvious choice for the part of a validation if experiments are not possible.

**Full-scale setup**

For studying the effect of the different changes, a full-scale setup is modelled. The full-scale setup consists of a setup with 85 sheets where membranes would be attached on both sides in an active system. The geometry of the module is illustrated in Figure 7. In this study, there is no
flux through the membranes as they are assumed to be walls. The sheets are 1 × 1 m with a thickness of 5 mm with a gap of 7 mm between each membrane as standard. The air inlet is modelled from 7 × 7 inlets on the downside of the manifold pipe illustrated in Figure 7. The size of the inlet is quadratic 4 × 4 mm, where the flow is equally distributed on all the inlet surfaces. The total inlet is 100 m³/h equalling a SADₘ of 0.59 m³/m²/h; this is in the interval for running conditions for the Alfa Laval flat sheet MBR system, which for a setup with a height of 1 m also equals a SADₙ of 0.59 m³/m²/h.

For reducing the computational cost, the module is modelled with symmetry boundaries on all the sides, equalling a setup with an infinite number of modules with a fixed distance between the modules. The distance between the modules is 10 cm in both directions.

The deflection of the sheets was made, so all the edges of the sheets still were fixed with a distance of 7 mm, while they deflected toward the centre of the sheets. For studying the effect of the deflected sheets, sheet numbers 42 and 43 of the 85 sheets have been modified with a deflection. The deflections have been made with total deflection of 1 and 2 mm of the sheets, corresponding to deflections in the range that have been seen in the production. This deflection leaves a gap of 5 and 3 mm, respectively, in the centre where the distance is smallest.

Effect of membrane deflection

The model with the deflected membranes showed that the deflections have a significant effect on the flow between the membranes as well as the wall shear stress on the surface of the membranes. The results for the different setups are listed in Table 5, where they have also been compared with the ideal setup.

The wall shear stress was reduced as much as 40% with a deflection of 2 mm on both membranes, while it was 23% for a deflection of 1 mm of both membranes. Furthermore, the mass flow reduced 78% and 49% for the two setups. There is not sufficient knowledge about the relationship between sludge composition, shear stresses and its influence on fouling of the membrane to make a precise quantification of the influence on the operation of a full-scale system.

<table>
<thead>
<tr>
<th>Membrane setup</th>
<th>τₚ-wall [Pa]</th>
<th>τₚ-wall Percentage of parallel [%]</th>
<th>Liquid mass flow [kg·s⁻¹]</th>
<th>Liquid mass flow Percentage of parallel [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Parallel sheets</td>
<td>2.92</td>
<td>100</td>
<td>2.10</td>
<td>100</td>
</tr>
<tr>
<td>1 mm deflection narrow gap</td>
<td>2.26</td>
<td>77</td>
<td>1.08</td>
<td>51</td>
</tr>
<tr>
<td>2 mm deflection narrow gap</td>
<td>1.76</td>
<td>60</td>
<td>0.47</td>
<td>22</td>
</tr>
<tr>
<td>1 mm deflection wide gap</td>
<td>3.17</td>
<td>109</td>
<td>3.46</td>
<td>165</td>
</tr>
<tr>
<td>2 mm deflection wide gap</td>
<td>3.51</td>
<td>120</td>
<td>3.76</td>
<td>179</td>
</tr>
</tbody>
</table>

Figure 8 | Wall shear stresses for (a) parallel membranes, (b) 1 mm deflection of membranes and (c) 2 mm deflection of the membranes.
Overall, the results show that the parallelization is an essential factor when producing the modules.

For illustrating the distribution of the wall, shear stresses on the surface of the membranes a snapshot is taken from the model and shown in Figure 8.

Figure 8 shows that the entire surface of the sheet is affected by the deflection. It is not only at the centre where the sheets are closest but also in the top and the bottom where the distance is kept close to the 7 mm for the ideal setup. The snapshots of the wall shear stress also show that there are lower wall shear stresses on the sides of the sheets close the walls.

CONCLUSION

A setup with computational fluid dynamics has successfully been validated with the EMP. With the relatively low computational cost due to the coarse mesh with only two cells across the gap and the large timesteps of 0.05 s, the model is applicable to full-scale studies. It has been validated against measured velocities to give the correct shape of the velocity profile, though it tends to underestimate the vertical velocities slightly. To achieve realistic velocity profiles between the membranes it was found that the inclusion of an extra turbulent dispersion force was needed.

For evaluating the wall shear stresses, the EMP has been compared with the VOF method that resolves the layer between the bubble and the wall. The difference in the average wall shear stress between the two approaches was less than 2%. The Eulerian multiphase model was not able to describe the horizontal components of the wall shear stresses, which were significantly lower than for the VOF model.

The VOF model showed that the horizontal components of the wall shear stresses with values as high as 10 Pa contribute significantly to the overall wall shear stresses, meaning that this should be included when studying wall shear stresses.

With these results, it is also clear that even though the VOF model and Eulerian multiphase model gave similar wall shear stresses it cannot be concluded that the Eulerian multiphase model is able to model correct wall shear stresses and more work should be done to verify the capability of this model. From the application on the full-scale setup, it was found that the deflection of membranes had a large impact on both recirculating flow and wall shear stresses. With a deflection of 2 mm, there was a reduction of as much as 78% of the flow between the membranes and reduction of 40% for the wall shear stresses. This influence shows the importance of choosing production methods that minimize the deflected the sheets.

REFERENCES


First received 4 April 2018; accepted in revised form 9 November 2018. Available online 20 November 2018.