Introducing a ‘slow’ cross-flow at the capillary outlet in comparison to conventional dead-end mode—a trajectory analysis of the effects

A. Lerch

ABSTRACT

A model has been developed, based on the finite element method (FEM) of computational fluid dynamics (CFD), for the description of the complete flow field and concentration distribution inside a membrane capillary, driven in inside-out and dead-end or ‘slow’ cross-flow mode, sometimes referred to as ‘bleed flow’. Particle or floc transport and deposition have been described by trajectory analysis, i.e. superimposing the calculation of forces and torques acting on the particles or flocs, based on the previously modelled fluid flow field. The model is used to give an overview of deposition behaviour and fouling layer formation of particles and flocs of a certain size in dead-end and cross-flow filtration. Example results are shown for different sized flocs. It is shown that the choice of dead-end or cross-flow operation is more significant if small floc aggregates have to be filtered by the membrane. Small flocs will be deposited more or less homogeneously along the membrane wall after some significant distance to the capillary inlet, leaving the first part of the membrane area unused for deposition. A ‘slow’ cross-flow could be used to transport small flocs out of the capillary which entered the capillary cross section area in the neighbourhood of the axis. The faster the chosen cross-flow velocity, the larger the area. Larger flocs will be ‘accumulated’ in one resulting equilibrium trajectory and are transported to the rear end of the capillary, independent of their starting radial position at the inlet and operation conditions. It was calculated, that larger flocs will not be significantly transported out of the capillary lumen by introducing ‘slow’ cross-flow velocities at the capillaries outlet only.

Key words | CFD, FEM, floc transport, membrane, modelling, MF, trajectory analysis, UF

INTRODUCTION

Generally, membrane filtration performance is mainly limited by an increase of the required transmembrane pressure (TMP) to maintain constant flux operation due to fouling phenomena. One of these phenomena is the formation of a fouling layer on the membrane surface by the retained particulate and colloidal matter. Considering micro- and ultrafiltration (MF and UF) of coagulated raw waters, fouling and layer formation are generally formed by deposition of flocs, deposition of unflocculated particulate and colloidal matter as well as adsorption of dissolved matter such as humic substances. For the derivation of appropriate operational performance in respect of economic and procedural engineering aspects, it is necessary to understand how limiting factors occur and how they may be avoided by design and choice of operating parameters of the complete process. For instance, knowing the transport and deposition behaviour of particles and flocs inside the capillaries of a membrane module enables the proper choice of operation conditions, process design and cleaning procedures. This becomes of further importance when treating surface waters.
by the hybrid process coagulation and membrane filtration, where clogging of the capillaries can become a major problem (Lerch et al. 2003, 2005; Heijman et al. 2007). This is, because UF and MF membrane processes for potable water treatment are often operated in dead-end mode in order to reduce energy consumption in comparison to cross-flow mode operation. As suggested by Panglisch (2001) for filtration of uncoagulated raw waters, clogging might be avoided by introducing a slow cross-flow velocity at the end of the capillary, sometimes referred to as ‘bleed flow’. Cross-flow operation in this respect would just be used to transport and push larger flocs out of the capillary which otherwise would be deposited at the dead-end.

The flow profile within the capillary lumen will still be laminar. Please note that this is different to commonly understood cross-flow operation, where flow along the membrane is often kept turbulent to minimise deposition and where the extracted water volume is mostly governed by the critical flow concept (Bacchin et al. 2006).

In addition to an earlier publication of the author in this journal (Lerch et al. 2007), the focus of this work will be set on theoretically investigated effects of such slow cross-flows on the deposition behaviour of flocs in a capillary lumen. Hence, the following shows chosen results from models with different average outlet velocities from 0%, i.e. dead-end and 1%, 5%, 10% and 20% of the inlet average velocity to prove the effects of an introduced slow cross-flow. The theoretical investigations were based on a developed numerical finite element method (FEM) model, used for the description of the complete flow field and concentration distribution inside a membrane capillary of arbitrary diameter and length, driven in inside-out mode (Lerch 2008). The capillaries considered here were 1 m in length and 0.8 mm in diameter. Particle or floc transport and deposition was described by superimposing the calculation of force and torque equilibria on a previously modelled fluid flow and floc volume concentration field, delivering floc trajectories inside the capillary lumen. This attempt describes and predicts at least qualitatively the loss in permeability as a function of the coagulated raw water and membrane properties and the chosen operating conditions. Hence, the model was used to give an overview of the deposition behaviour of flocs of different size in dead-end and cross-flow filtration mode.

**MATERIALS AND METHODS**

**Forces and torques**

Deposition during filtration of coagulated raw waters occurs if a floc intersects the membrane surface, hence building up a fouling layer. The rate and coordinates of floc deposition can be determined by trajectory calculations, describing the floc’s pathway inside the capillary lumen, but essentially require some knowledge of the governing fluid flow. These hydrodynamic conditions can be expressed by the calculation of the complete velocity flow field, i.e. the velocity components in axial and radial direction dependent on capillary length and radius at any point inside the capillary lumen. Fundamental solutions for the hydrodynamics of flow through a membrane capillary with its porous walls start from the Navier–Stokes (NS) equations and leads to a complete description of the isothermal fluid motion in the capillary. Due to the given symmetry of the capillary it was chosen to use cylindrical coordinates as indicated in Figure 1. Here, \( z \) denotes the longitudinal or axial distance coordinate and \( r \) denotes the radial distance coordinate of the capillary. Figure 1 shows the coordinate system and 2D section plane for illustration of trajectories in the capillaries of the considered cylindrical capillary membrane whose porous walls are separated by a distance \( d_c \), the diameter of the capillary.

The flow regime was modelled in dimensional and dimensionless form by using finite element method (FEM) software (COMSOL Multiphysics, Version 3.3a). The floc trajectories were derived from balances of the acting forces and torques due to the modelled and calculated hydrodynamic conditions inside the capillary lumen and physical properties of the flocs, usually dominant in the far-field region, and due to electrochemical properties of the flocs and the membrane, usually dominant in the near-field region.

![Figure 1](https://iwaponline.com/ies/article-pdf/8/4/389/418946/389.pdf)
The acting forces and torques under consideration arise from drag incl. wall interactions, virtual mass, buoyancy and sedimentation, Brownian and shear induced diffusion, lateral migration and DLVO interactions. The considered forces are not given in detail here but are described in detail in Lerch (2008) and are illustrated in Figure 2.

**Porous floc aggregate**

Flocs are always thought of as being aggregates consisting of iron or aluminium hydroxide and embedded particles and colloids. They are considered to be rigid and ideally spherical and of constant shape, but reveal basic floc properties such as porosity, low density and low zeta potential. This approach is reasonable as a first approximation because of the more macroscopic model investigated in this work. Hydrodynamic acting forces, such as drag forces are influenced by the inner porosity of flocs, because the fluid flow is around and through the porous aggregates. The floc porosity is assumed to be homogenous and was considered by the derivation of drag coefficients, based on the work of Masliyah et al. (1987) and Veerapaneni & Wiesner (1996). The drag coefficient was defined as the force exerted by the fluid on a permeable aggregate of radius \(a_F\), normalised by the force exerted by the fluid on an impermeable sphere of radius \(a_F\). The drag coefficient was then used to derive a hydrodynamic floc of radius \(a_{hF}\), being smaller than the physical radius \(a_F\) of the floc but accounting for the inner permeability. The hydrodynamic floc radius was used for the calculation of the forces and torques as far as hydrodynamic aspects were concerned. Furthermore, while filtering floc suspensions in an inside-out driven capillary membrane, concentration and therefore effective suspension viscosity will change depending on the axial and radial coordinates in the capillary. An increased concentration leads to a change of the fluid velocities in longitudinal and radial directions (Liu & Masliyah 1996). This additional aspect was considered in the transport model by correction functions based on the work of Vainshtein & Shapiro (2006).

**Model geometry and boundary conditions**

The membrane capillary was modelled in 2D, but the independent variables were chosen to be \(r, \phi\) and \(z\) for the radial, angular and longitudinal cylindrical coordinates in a 3D flow field respectively. An axial symmetry is naturally given at the capillary centre axis, i.e. at \(r = 0\), which was used to save computational memory consumption during modelling. However, the 2D results obtained can be transformed into 3D using a general transformation technique of the software for coupling variables. The geometry was adapted to the geometry of the capillaries used in experiments (Single UFC M5 ultrafiltration capillaries of the company X-Flow B. V.) and could be determined in dimensional or dimensionless form. Hence, in order to analyse the model in dimensional form, the capillary radius was set to \(r_C = 0.4 \times 10^{-4}\) m and the length of the modelled capillary was set to \(l_C = 1\) m if not stated otherwise. In order to analyse the model in dimensionless form, the geometry was normalised by some scaling factors \(SR\) and \(SZ\). The scaling factor \(SR\) was chosen to be \(r_C\), denoting for the parameters in the radial direction and the scaling factor \(SZ\) was chosen to be \(l_C\), denoting for the parameters in the longitudinal direction, respectively. For both cases the boundaries of the 2D model were chosen to be: boundary 1 as the capillary centre axis, representing the axial symmetry; boundary 2 and 5 as the capillary inlet; boundary 3 and 6 as the capillary dead end or outlet in cross-flow mode respectively and boundary 7 as the membrane wall. An interior boundary 4 was introduced to account for the fouling layer formation, see red dashed line in Figure 3, dividing the modelled domain into two subdomains. The distance to the membrane wall was always set to the physical radius of the flocs to be modelled, hence representing the distance when contact occurs, while compaction of the flocs was neglected.
The determination of the calculated flow field was modelled for single membrane capillaries operated in both dead-end and cross-flow mode. The flux was chosen to be 80 L/m²/h in dead-end and the fluid flow, showing a laminar flow profile at the lumen entrance, entered the capillary lumen with an inlet velocity of 0.111 m/s. The membrane resistance \( R_M \) of the capillaries was set to \( 5.24 \times 10^{11} \) 1/m, as an average value evaluated from experimental data. If the operation condition was chosen to be cross-flow, the outlet velocity at boundary 3 and 6 was modelled with 1%, 5%, 10% and 20% of the inlet average velocity, still possessing a laminar flow profile at the lumen outlet, as compared in Figure 4. These velocities represent just a very slow cross-flow at the capillary outlet, keeping the flow inside the lumen laminar, but were assumed to ensure floc transport out of the capillary and not to achieve turbulent flow in the capillary.

Table 1 summarises some of the initially used model parameters. Further details will be found in Lerch (2008).

RESULTS AND DISCUSSION

Model flow fields and fluid streamlines

The delivered results of the numerical calculation for the determination of the initial steady state fluid flow field and trajectories are shown in the following figures as 2D and 3D dimensionless plots. Please note that the results shown here are not influenced by floc concentration at the membrane wall nor inside the lumen, hence representing initial conditions at the start of operation. In the following, the type of illustration used for presenting the floc trajectories was chosen to be in 2D at a certain section plane within the capillaries geometry. The section plane considered here is spanned by the complete capillary length and radius at \( \phi = 270^\circ \), i.e. below the symmetry axis at \( r = 0 \) as indicated in Figure 1. Generally, horizontally aligned capillaries are considered. For purposes of clarity, the direction of the action of gravity was introduced in the following 2D plots too. However, the considered section plane in these cases is similar to the one shown above. In the figures, the dimensionless capillary radius is usually considered as the abscissa. Hence, the floc trajectories usually start at the bottom line and will end at the right hand side, i.e. the membrane wall.

Figure 5 presents a calculated 2D axial symmetric flow field and fluid streamlines plot in a single membrane capillary driven in dead-end mode. The bar at the right hand side in Figure 5 represents the magnitude of the fluid flow field velocity, increasing from black to white. The fluid streamlines represent the transport pathways of fluid elements. In the case of dead-end operation, all fluid elements are transported towards the membrane wall and finally pass it.

The calculated flow field velocity can be further used to validate the accuracy of the numerical model by integration of the velocity on boundary 2, i.e. the capillary inlet, and boundary 4, i.e. the capillary membrane wall. The integration
of the surface integrals deliver the volume flow through the boundaries and can be used to control if no volume of the fluid is 'lost'. Integration of the surface integral on boundary 2 delivers $5.585054 \times 10^{-8}$ m$^3$/s and $5.585529 \times 10^{-8}$ m$^3$/s on boundary 4. Because the difference of the result is smaller than 0.009%, mass conservation can be assumed.

Figure 6 presents the calculated fluid streamlines in a single membrane capillary driven in different cross-flow modes, i.e. different cross-flow outlet velocities at boundary 3 and 6, clockwise from the upper left: 1%, 5%, 10% and 20%. It can be seen that fluid elements entering the capillary in the neighbouring area of its axis will leave the capillary at the end. The faster the chosen cross-flow velocity, the larger this neighbouring area at the axis.

**Floc trajectories**

Generally, the software generates the trajectories by solving the fundamental equation of motion:

$$m_F \frac{\partial^2 \vec{x}}{\partial t^2} = F \left[ t, \vec{x}, \frac{\partial \vec{x}}{\partial t} \right]$$

(1)

for $\vec{x}(t)$ being the trajectory. Here, $m_F$ is the flocs mass, $F$ equals the force component acting on the floc as introduced above and $t$ is time. This delivers a system of ordinary differential equations (ODE) for $\vec{x}$, which the software solves using a pair of Runge-Kutta methods of orders four and five (compare Finlayson 2006), which are iterative numerical methods for the approximation of solutions of ODE systems. The solver advances the algorithm with the solution of order five and uses the difference between the order-five and order-four solutions to obtain the local error estimate. For the used 2D axis-symmetric model, all three force components were used. Hence, the algorithm solved for a trajectory in 3D in cylindrical coordinates, but only the projection on the axis-symmetry plane is plotted in 2D.

The forces acting on the particles or flocs and velocities derived can be used to determine the floc trajectories.

---

**Table 1 | Constants and calculated initial parameters used in the model**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gravitational acceleration in positive radial direction</td>
<td>$g_r$</td>
<td>$9.81$ m/s²</td>
</tr>
<tr>
<td>Floc density</td>
<td>$\rho_F$</td>
<td>$1,050$ kg/m$^3$</td>
</tr>
<tr>
<td>Initial floc volume concentration in the bulk flow</td>
<td>$\phi_{F,0}$</td>
<td>$9.524 \times 10^{-4}$ m$^3$/m$^3$</td>
</tr>
<tr>
<td>Fluid temperature</td>
<td>$T$</td>
<td>$20$ °C</td>
</tr>
<tr>
<td>Initial average fluid inlet velocity</td>
<td>$V_0$</td>
<td>$0.111$ m/s</td>
</tr>
<tr>
<td>Initial average fluid outlet velocity (1%)</td>
<td>$V_{CF,1%}$</td>
<td>$0.00111$ m/s</td>
</tr>
<tr>
<td>Initial average fluid outlet velocity (5%)</td>
<td>$V_{CF,5%}$</td>
<td>$0.00555$ m/s</td>
</tr>
<tr>
<td>Initial average fluid outlet velocity (10%)</td>
<td>$V_{CF,10%}$</td>
<td>$0.0111$ m/s</td>
</tr>
<tr>
<td>Initial average fluid outlet velocity (20%)</td>
<td>$V_{CF,20%}$</td>
<td>$0.0222$ m/s</td>
</tr>
<tr>
<td>Initial fluid inlet pressure</td>
<td>$p_0$</td>
<td>$1.348 \times 10^4$ Pa</td>
</tr>
<tr>
<td>Membrane filtration area</td>
<td>$A_M$</td>
<td>$2.513 \times 10^{-3}$ m$^2$</td>
</tr>
<tr>
<td>Initial dynamic fluid viscosity at 20°C</td>
<td>$\eta_0$</td>
<td>$1.003 \times 10^{-3}$ Pa·s</td>
</tr>
<tr>
<td>Initial fluid density at 20°C</td>
<td>$\rho_0$</td>
<td>$998.204$ kg/m$^3$</td>
</tr>
</tbody>
</table>

---

**Figure 5 | Axial symmetric velocity field and fluid flow streamlines in a capillary driven in dead-end mode.**
The trajectories can be calculated starting from any position within the capillary, even backward integration is possible. It has to be pointed out that the shown trajectories represent the moving centre of mass of the flocs. Floc agglomeration along the trajectories is not considered. Further, there is initially no effect on the modelled flow field by motion of the flocs. Therefore, the results shown here are neither influenced by floc concentration at the membrane wall nor inside the lumen, hence representing initial conditions at the start of operation. The influence of increasing floc concentration within the capillary and floc deposition, resulting in change of the acting resistance against fluid flow, and the effect on the trajectories of following flocs were considered and discussed in Lerch (2008). Generally, the findings obtained by the developed model are in strong agreement with findings shown by Chellam & Wiesner (1992), Song & Elimelech (1995) and Panglisch (2001) for instance.

The convective transport of dissolved and colloidal matter up to particulate micro sized flocs can be generally described according to the transport of fluid elements and will therefore follow the fluid streamlines as shown in Figure 5 for dead-end and Figure 6 for cross-flow operation mode. Due to this flow behaviour, the complete inner

Figure 6 | Calculated fluid flow streamlines (axial symmetric, dimensionless model, 2D) for models with different cross-flow outlet velocities at boundary 3 and 6. Clockwise from the upper left: 1%, 5%, 10% and 20%.
surface of the capillary will be fouled by micro flocs. In case of dead-end operation, all micro sized flocs are transported towards the membrane.

If the floc size is increased to radii larger than a certain ‘limiting radius’, the transport behaviour will change even under the same flow conditions and physical properties and an equilibrium trajectory will be developed into which individual trajectories merge. At the equilibrium trajectory, the acting forces directed towards the membrane are equalised by forces directed away from the membrane wall towards the capillary axis. The corresponding limiting floc radius can be determined by balancing the radial velocity component of the floc. For being deposited all over the membrane wall of the capillaries, the radial velocity has to be always larger than zero along the complete capillary length. The first radius which does not fulfil this inequality is called the limiting radius and was calculated to be $a_F = 3.27 \mu m$ for the given flow conditions and physical properties in dead-end mode. This effect is due to the acting repulsive forces, which usually increase with increasing floc sizes, closer distances to the membrane wall, and increasing longitudinal flow velocities. The developed theoretical model shows that this is mainly caused by lateral migration.

Figure 7 illustrates the trajectories of different sized flocs inside the capillary, driven in dead-end mode. It can be seen, that for smaller floc sizes this equilibrium trajectory can be usually found at closer distances to the membrane wall, approaching the wall only slightly until deposition occurs. This is due to the occurrence of floc wall interactions such as repulsive double layer forces and lateral migration effects. In case of dead-end mode, this equilibrium trajectory is usually much closer to the wall than in cross-flow mode operation, as shown by Chellam & Wiesner (1992), which is likely to be due to the higher local maximum longitudinal fluid flow velocities. In contrast, fluid elements are not exposed to those interactions and additional forces, and hence their streamlines are approaching the membrane wall directly as indicated in Figures 5 and 6. Due to the equilibrium trajectory, flocs of radius $5 \mu m$ will be deposited at the membrane only after passing approximately the first 61% of its length. Hence, according to the modelled results, only 29% of the membrane area is used for deposition of flocs of this size.

For further increasing floc sizes, the front contact point, i.e. the longitudinal coordinate where contact between the equilibrium trajectory and the membrane occurs, is shifted more inwards. Further, due to the increased floc size the equilibrium trajectory is moved away from the membrane wall towards a position closer to the capillary axis, eventually towards a position at approximately two thirds of the capillary radius for flocs of $30 \mu m$ in radius. For further increased floc sizes, this position will be located eventually at $r/\text{SR} = 0.622$, which is in very good agreement to the findings of Segré & Silberberg (1962). As illustrated in Figure 7 for flocs with a radius of about $12 \mu m$ and larger, all flocs entering the capillary, independent of their starting radial position at the inlet, will merge to a final equilibrium trajectory and travel along it until deposition occurs at the rear lumen end. Hence, most of the membrane area is initially not affected by floc deposition of larger flocs. The larger the floc size, the faster the flocs will merge and accumulate on this final equilibrium trajectory.

Because the floc trajectories merge together it can be assumed that further floc growth will take place while the flocs are travelling along the equilibrium trajectory. This is, because the flocs will be brought into unavoidable contact while merging and are not exposed to high shear rates except at the capillary inlet. Hence floc growth will be more dominant than floc destruction. This will lead to a transport of larger flocs downwards to the dead-end of the capillary.

This effect is even more pronounced if sedimentation effects are considered as shown in the 3D plots in Figure 8. For an increasing floc size, more and more flocs will not be able to stay in the upper part, being transported into the

![Figure 7](https://iwaponline.com/ies/article-pdf/8/4/389/418946/389.pdf)
lower part ($r/\text{SR}$ from 0 to 1), i.e. below the symmetry axis of the horizontal aligned capillary, and will be deposited there. It was found that almost no flocs will be retained by the upper part of the membrane capillary for flocs with a radius larger than approximately $14.75\ \text{mm}$ under the given flow conditions, as shown in Figure 8.

Here, the flow is from the rear to the front and gravity acts from top to bottom. The diameter of the tubes was scaled to the modelled floc radius $a_F/\text{SR}$ and the colour bar at the right hand side of each plot indicates the floc velocity. Due to repulsive DLVO forces and sedimentation, the flocs will slide close to the membrane wall along its surface to the bottom. If the floc radius is increased further even areas of the lower capillary part will not be available for floc deposition and the flocs will finally merge for deposition at the rear end into one trajectory at the bottom of the capillary.

In contrast to solid particle suspensions, compare Panglisch (2001), it could be shown that the more widely homogeneous distribution and preferential deposition of smaller and mid sized particles was reduced. Thus, it still comes to the formation of zones with different composition, porosity and thickness, but the majority of particles embedded in generally larger flocs will be transported towards the dead-end which may cause clogging of the capillary.

As suggested by Panglisch (2001) for filtration of uncoagulated raw waters, clogging might be avoided by introducing a slow cross-flow velocity at the end of the capillary. The concentrate flow might be further treated in a second membrane stage, adapted in geometry and operation parameters on the better defined performance, i.e. reduced or more tightened particle size distribution. Cross-flow operation in this respect would just be used to transport and push larger flocs out of the capillary as mentioned above. In the following, the chosen results of models with different average outlet velocities at boundary 3 and 6 are shown to prove the effects of an introduced slow cross-flow. The average velocities at the capillary outlet were chosen to be 1%, 5%, 10% and 20% of the inlet average velocity, whereas...
all other previously used parameters were kept constant. Figure 9 shows floc trajectories of different sized flocs for models with 5% (left) and 20% (right) average cross-flow velocity at the capillary outlet, whereas just the inner- and outermost floc trajectories and resulting equilibrium trajectories within the capillary for flocs of radius $a_F = 29.47 \, \mu m$ and $a_F = 60 \, \mu m$ are illustrated.

It can be seen, that in case of slow cross-flow operation smaller flocs entering the capillary in the neighbouring area of its axis at the inlet will leave the capillary at the end. The faster the chosen cross-flow velocity the larger this neighbouring area and the more flocs are able to be transported out of the capillary lumen without being deposited.

For flocs of radius $a_F = 5 \, \mu m$ and 1% average cross-flow outlet velocity it was found that all flocs entering between the capillary axis and a dimensionless capillary radius $r/\text{SR} = 0.03$ will be transported out of the system. Applying an outlet velocity of 5% will deliver $r/\text{SR} = 0.11$ and at 20% will deliver $r/\text{SR} = 0.273$. Hence, assuming an evenly distributed floc suspension entering the capillary, about 0.09%, 1.23% or 7.67%, respectively, of all entering flocs will leave the capillary. This amount will be higher for smaller flocs and lower for larger flocs. For example, the values for flocs of radius $a_F = 12.25 \, \mu m$ at 20% outlet velocity were calculated to be 0.02% and for 5% and slower velocities no floc of any size will leave the capillary.

Even at high flux operation, for instance 200 L/m$^2$/h, larger flocs will not be driven out of the system to any great extent, as indicated in 3D in Figure 10. Just flocs from the very upper part might be able to escape deposition and leave the capillary. Here, the flow is from the rear to the front and gravity acts from top to bottom. The diameter of the trajectories was scaled to the modelled floc radius $a_F/\text{SR}$ and the colour bar at the right hand side of each plot indicates the magnitude of floc velocity. Larger flocs will not be able to be significantly transported out of the capillary lumen only by introducing slow cross-flow velocities at the capillaries outlet. This shows that clogging caused by larger floc aggregates cannot be avoided unless the flux is dramatically increased.

However, these findings are just based on the developed theoretical model, which does not claim to be complete yet. For instance, the model presumes a stagnant fouling layer, neither compressible nor expandable. Once a floc is deposited, it will not be moved to another position. Further, no tangential flow through the fouling layer is included in the model so far. However, in real operation the composition of the fouling layer will be much more multifaceted. For instance, it was shown by Marselina et al. (2007) using direct observation...
techniques, that the upper part of a fouling layer is fluidised by the passing fluid during operation. This viscous flow of the upper part of the fouling layer, tangential along the membrane, could be probably used to avoid clogging at slow cross-flow velocities at the capillary outlet.

CONCLUSION

The developed model can be used to model arbitrary flow fields in capillary membranes, depending on certain process and membrane parameters such as filtration flux and membrane resistance for instance. Further, the model can be used to investigate particle or floc deposition by trajectory analysis inside the capillary of any kind of solid and spherical particles of a certain size to derive deposition behaviour and fouling layer formation.

The results discussed here show that the choice of dead-end or slow cross-flow operation conditions is more significant if small particles or floc aggregates have to be filtered by the membrane. Dissolved and colloidal matter will follow the fluid streamlines as far as possible and will therefore eventually be deposited homogeneously along the membrane wall, if retained by the membrane, whereas the deposited mass depends on the chosen cross-flow velocity. The higher this velocity, the more dissolved and colloidal matter will leave the system at the outlet.

Small sized flocs or particles will be deposited more or less homogeneously along the membrane wall after some significant distance to the capillary inlet, leaving the first part of the membrane area unused for floc deposition. This area was found to be about 31% for flocs with radius 5 \( \mu \text{m} \) under the given flow conditions. It could be shown that this distance increased with increasing floc size; the larger the floc, the larger this distance will be. A slow cross-flow could be used to transport small flocs out of the capillary which entered the capillary cross section area in the direct neighbourhood of the axis. The higher cross-flow velocity will give a larger area.

Larger flocs will be ‘accumulated’ in equilibrium trajectories and are transported to the rear end of the capillary, independent of their starting radial position at the inlet and...
the chosen operation conditions. It was calculated that larger flocs or particles will not be significantly transported out of the capillary lumen only by introducing slow cross-flow velocities at the capillaries outlet. This shows that clogging caused by larger floc aggregates cannot be avoided unless the flux is dramatically increased. Nevertheless, it has to be considered that in reality, high flux conditions are generally more fouling since the particle or floc concentration in the lumen will increase, since higher flux brings more particles or flocs. As shown by Pearce & Field (2007), for a given feed, fouling will increase, and it has been observed that fouling increases exponentially with flux at commercial fluxes of interest above a certain threshold flux. Hence, the flux to be chosen in real operation should be limited to the so-called sustainable flux, which is that flux at which there is an acceptable degree of fouling, but that the fouling is easily removed in a cleaning procedure of acceptable frequency.

However, the presented results here are the findings on the basis of the developed theoretical model, which does not claim to be complete yet and therefore will be enhanced in future work. For instance, future modelling should include the implementation of the mass fractal dimension $d_F$ into the equations utilized to consider the impact on transport and deposition behaviour of more realistic floc structures. Considering the formed fouling layer, additional modelling work should be done on floc and layer compressibility as well as the detachment behaviour of deposited flocs. This will lead to a more realistic approach to model the viscous flow of the upper part of the fouling layer, tangential along the membrane. This flow is assumed to be responsible for the transport of larger flocs out of the capillary at the end as found in real membrane systems using small cross-flows.

REFERENCES