CFD-DEM simulations of hydrodynamics of combined ion exchange-membrane filtration

Naukkarinen Tomi, Nikku Markku, Turunen-Saaresti Teemu

This is a Final draft version of a publication
published by Elsevier
in Chemical Engineering Science

DOI: 10.1016/j.ces.2019.08.009

Copyright of the original publication: © 2019 Elsevier

Please cite the publication as follows:

This is a parallel published version of an original publication. This version can differ from the original published article.
CFD-DEM simulations of hydrodynamics of combined ion exchange-membrane filtration

Tomi Naukkarinen\textsuperscript{a,b},*, Markku Nikku\textsuperscript{a}, Teemu Turunen-Saaresti\textsuperscript{a}

\textsuperscript{a}School of Energy Systems, Lappeenranta University of Technology, Lappeenranta, Finland

Abstract
The combined ion exchange-membrane filtration system, whereby an active filtration layer is formed using a slurry flow of resin beads and water, has the potential to reduce energy usage and increase the quality of purified wastewater. The hydrodynamic design of these systems is in its infancy and there is a need for numerical methods to analyze such systems. In this work, computational fluid dynamics coupled with the discrete element method (CFD-DEM) is utilized to analyse the hydrodynamics of the combined ion exchange-membrane filtration system, and the formation of the filtrating bed and the parameters affecting it are studied by varying the flow conditions. As a result, the details of the bed formation with insights on the parameters affecting these phenomena and suggestions for an improved hydrodynamic design of the combined ion exchange-membrane filtration system are presented.

Keywords: CFD-DEM, combined-filtration, wastewater, resin beads

1. Introduction

Pharmaceuticals and personal care products (PPCPs) (e.g. hormonal contraception) as well as endocrine disturbing compounds (EDCs) (e.g. bisphenol A [BPA]) are passed in drinking water and food due to the reusing of wastewater as a source of drinking water ([1], [2]). According to the literature ([3], [4], [5] and [6]), these contaminants are common in municipal wastewater as a result...
of the extensive use of BPA and hormonal contraception. Population growth and the climate change are reducing available water resources per inhabitant, which increases the attractiveness of reusing the wastewater. Diminishing water resources have received attention from some public organizations, for example, the EU has a policy to promote wastewater reclamation and reuse to improve the state of the environment and to reduce greenhouse gas emissions.

According to Snyder et al. [7] nanofiltration or reverse osmosis is required to significantly reduce PPCP/EDCs levels in the wastewater. Nanofiltration and reverse osmosis require high pressure levels [8], leading to high energy consumption. On the other hand, Aldulgader et al. [9] reviewed hybrid processes for water resources, desalination and treatment of organic pollutants. They found that the hybrid processes have the potential to decrease the amount of discharge, as well as reduce the energy usage and costs. In a typical hybrid process, ion exchange resins are placed in vessels as steady packed beds, but Kabay et al. [10] introduced an exception. In their process, boron selective resins are mixed with feed water to sorb boron. After the sorption, saturated resins are separated from water with a membrane, and the permeate fulfills boron concentration requirements. Separated resins are regenerated and recycled back to the sorption phase. Aldulgader et al. [9] concluded that hybrid ion exchange-membrane processes have a place in solving the demands of wastewater treatment. In addition, a combination of ion exchange resin beads with membranes is one method to reduce PPCP/EDC levels in the wastewater ([11] and [12]).

Experimental research is a useful but labor intensive and expensive way to study the fluid-solid flow phenomenon in wastewater applications. However, the numerical methods offer a feasible tool for evaluating the flow phenomena before or in conjunction with experiments ([13] and [14]). Low costs, adaptability and possibility to obtain detailed information are the main reasons why computational fluid dynamics (CFD) are used to study wastewater applications, e.g. [15] and [16].

There are two main approaches in the modeling of the particulate phase in a dense fluid-particle system: the continuum approach, e.g. two fluid model
(TFM), or the discrete approach. Both of these methods base on so-called four-way coupling, which takes account fluid’s effect on the particles, the particles’ effect back to the fluid phase, the void fraction, and particle-particle interactions. The continuum approach assumes that the particles form an artificial continuum. The continuum approach loses the information of single particles and the closures that stem out from the kinetic theory of granular flow ([17] and [18]), are required to close the system of equations, e.g. momentum equation. A discrete approach, such as the discrete particle model (DPM), can overcome the limitations of TFM as particles are tracked in a Lagrangian frame and particle-particle interactions can be described with a simple spring-dashpot model. On the other hand, the DPM is computationally expensive compared to TFM or merely a one-way coupled discrete approach. Computational fluid dynamics-discrete element method (CFD-DEM), also referred to as a combined continuum and discrete model (CCDM) ([19] and [17]), is commonly used to obtain detailed information of the flow fields and the particle-particle interactions in gas-solid fluidized beds and pneumatic conveying ([20]). Liquid fluidized beds have been also studied with good agreement ([21], [22], [23], [24] and [25]). In recent years, CFD-DEM has also been used to study particle batch settling ([26] and [27]), sediment transport ([28], [29], [30] and [31]), solid-liquid mixing ([32]) and slurry transport in pipes ([33], [34] and [35]). However, in the wastewater applications, due to the large size of the wastewater processing equipment and the low solid loading, the effect of the disperse phase on the carrier phase is neglected and one-way coupling is applied ([15] and [14]) to reduce the demand of computational resources.

The hybrid processes offer more compact and more energy efficient alternatives to wastewater treatment and reduction of PPCP/EDC levels ([9]). However, their development is still in its infancy and suitable validated computational tools are needed for the design and analysis. In the combined ion-exchange-membrane filtration, a layer of resin beads, called a bed, rests on top of a membrane and removes the contaminants before water permeates through the membrane film. The saturated ion-exchange resin beads can be removed
from the membrane, for example with backward or forward flushing (i.e. changing
membrane flux direction or preventing membrane flux), and the be replaced
with the slurry of the ion-exchange resin beads and water. In addition, to
achieve a uniform pressure drop, the bed height and packing must be uniform.
These requirements mean that to realize a working combined cycle, the hydro-
dynamics of the fluid-particle system and the effects of different parameters on
its performance should be understood in detail.

Previously published studies show the CFD-DEM approach is mature and
accurate method to model solid-liquid flows, therefore in this paper CFD-DEM
method is applied to a new research area, wherein it has significant potential
to enable reduction of energy usage and an increase in the wastewater quality.
The novelty of the manuscript is twofold. Firstly, this study extends the utiliza-
tion of CFD-DEM in the hydrodynamics of a combined ion exchange-membrane
filtration, where it has not been used previously. Secondly, details on the bed
formation and packing with insights on the parameters affecting these phenom-
ena are presented. Finally, suggestions for improved hydrodynamic design of a
combined ion exchange-membrane filtration are presented. The fouling of the
membrane, as studied by Lohaus et al. [36] and ion exchange are not studied
in this work.

Firstly, the applied CFD-DEM method is presented and method’s capability
to model solid volume fractions in heterogeneous slurry pipe flow is shown. Sec-
ondly, the investigated domain is introduced. Thirdly, the mesh independency
study is presented and discussed. Then, the particle settling and bed formation
is studied by changing the flow inlet velocity, the permeate flux, and the position
of the particle insertion. Fourthly, the removal of the bed is investigated by back
flushing. Finally, the results are summarized and the conclusions presented.

Nomenclature

Greek alphabet

\( \alpha \) volume fraction \([-]\)
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\omega$</td>
<td>angular velocity</td>
<td>[rad/s]</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>turbulence dissipation rate</td>
<td>[$m^2/s^3$]</td>
</tr>
<tr>
<td>$\kappa$</td>
<td>turbulence kinetic energy</td>
<td>[$m^2/s^2$]</td>
</tr>
<tr>
<td>$\delta$</td>
<td>overlap at the contact point</td>
<td>[m]</td>
</tr>
<tr>
<td>$\mu$</td>
<td>dynamic viscosity</td>
<td>[Pas]</td>
</tr>
<tr>
<td>$\mu_C$</td>
<td>Coulomb friction coefficient</td>
<td>[-]</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson ration</td>
<td>[-]</td>
</tr>
<tr>
<td>$\phi$</td>
<td>packing fraction</td>
<td>[-]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>density</td>
<td>[$kg/m^3$]</td>
</tr>
<tr>
<td>$\tau$</td>
<td>stress tensor</td>
<td>[Pa]</td>
</tr>
</tbody>
</table>

**Latin alphabet**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F$</td>
<td>force</td>
<td>[N]</td>
</tr>
<tr>
<td>$g$</td>
<td>gravity</td>
<td>[m/s$^2$]</td>
</tr>
<tr>
<td>$I$</td>
<td>rotational inertia</td>
<td>[kgm$^2$]</td>
</tr>
<tr>
<td>$R$</td>
<td>momentum exchange term</td>
<td>[N/m$^3$]</td>
</tr>
<tr>
<td>$u$</td>
<td>velocity</td>
<td>[m/s]</td>
</tr>
<tr>
<td>$x$</td>
<td>coordinate</td>
<td>[m]</td>
</tr>
<tr>
<td>$C$</td>
<td>concentration</td>
<td>[-]</td>
</tr>
<tr>
<td>$c$</td>
<td>damping coefficient</td>
<td>[kg/s]</td>
</tr>
<tr>
<td>$D$</td>
<td>diameter of pipe</td>
<td>[m]</td>
</tr>
<tr>
<td>$d_{50}$</td>
<td>the midpoint of particle size distribution</td>
<td>[mm]</td>
</tr>
<tr>
<td>$e$</td>
<td>coefficient of restitution</td>
<td>[-]</td>
</tr>
</tbody>
</table>
\( F_L \) parameter in the Durand equation [-]

\( k \) spring stiffness [N/m]

\( K_f \) momentum exchange coefficient [kg/(m^3s)]

\( m \) mass [kg]

\( r \) radius [m]

\( R_\mu \) rolling friction parameter [-]

\( T \) torque [N\( \text{m} \)]

\( t \) time [s]

\( V \) volume [m^3]

\( Z \) overall solid loading [-]

**Subscripts**

\( \nu \) viscous

\( \text{ave} \) average

\( c \) contact

\( d \) drag

\( f \) fluid

\( i \) impact

\( n \) normal

\( \text{nor} \) normalized

\( o \) overall

\( p \) particle

\( pr \) pressure
2. Numerical methods

In this work, the translational and rotational motion of the solid phase are solved explicitly by the DEM package LIGGGHTS [37]. A particle trajectory is calculated based on the force and torque balance:

\[ m_p \ddot{x} = F_n + F_t + F_d + F_{pr} + F_\nu + F_g + F_b \] (1)

\[ I_p \dot{\omega}_p = r_{p,c} \times F_t + T_r \] (2)

where \( m \), \( x \) and \( r \) are mass, position and radius. \( F_n \) and \( F_t \) are the normal and tangential force due to particle-particle or particle-wall contact. \( F_d \) is the drag force that arises from the velocity difference of the fluid phase and particles. Particles are also affected by viscous force \( F_\nu \), pressure force \( F_{pr} \), buoyancy force \( F_b \) and gravity \( F_g \). \( T_r \) represents torque due to rolling friction.

Spring-dashpot model by Cundall and Strack [38] is used to define the particle contact forces. The spring represents elastic deformation while the viscous
dissipation is represented by the dashpot. The spring tension is calculated based on the overlap $\delta$ of the two colliding particles. The normal force $F_n$ and the magnitude of tangential force $F_t$ are calculated as:

$$F_n = -k_n \delta_n + c_n \Delta u_{p,n}$$

and

$$F_t = \min\left(\left| k_t \int_{t_{c,0}}^{t} \Delta u_{p,t} dt + c_t \Delta u_{p,t} \right|, \mu_C F_n \right)$$

where $\Delta u_{p,n}$ and $\Delta u_{p,t}$ are the relative normal and tangential velocities at contact, and $t_{c,0}$ is the start time of the contact. $c_n$ and $c_t$ are the normal and tangential damping coefficients. Coulomb friction coefficient $\mu_C$ is the limiting factor for maximum tangential force. The stiffness components and damping coefficients are calculated with simplified Hertz-Mindlin contact model ([39] and [40]).

Cohesive forces may have an effect on inter-particle forces in a certain application, for example with Geldart group C particles or in the membrane fouling study of Lohaust et al [36]. In this study, the size range of particles indicates that cohesive forces are negligible and are therefore neglected.

Rolling friction is a natural torque that occurs in every rotating motion, which is one way to take into account non-sphericity of the particles. Ai et al. [41] categorized rolling resistance models into three classes: directional constant torque models, viscous models and elastic-plastic spring-dashpot models. In this study, one of the most common models, the directional constant torque model is used. This model uses a rolling friction parameter $R_\mu$.

$$T_r = R_\mu k_n \delta \frac{\omega_{rel}}{|\omega_{rel}|} r_p$$

where the relative angular velocity $\omega_{rel}$ is defined with angular velocity $\omega$ and radius $r$:

$$\omega_{rel} = \frac{r_i \omega_i + r_j \omega_j}{r_i + r_j}$$

The fluid phase is modeled with a non-resolved approach. This means that the particles are not resolved in the solution of the fluid phase, but their momentum
and mass are conserved. The volume-averaged Navier-Stokes equations (VANS) for the fluid phase is derived by Anderson and Jackson \cite{42} and revised by Zhou et al. \cite{43}. By following Zhou et al.’s \cite{43} notation, the format set II of the VANS is used in this study:

\begin{equation}
\frac{\partial \alpha_f}{\partial t} + \nabla \cdot (\alpha_f \mathbf{u}_f) = 0
\end{equation}

\begin{equation}
\frac{\partial (\alpha_f \rho_f \mathbf{u}_f)}{\partial t} + \nabla \cdot (\alpha_f \rho_f \mathbf{u}_f \mathbf{u}_f) = -\alpha_f \nabla p + \mathbf{R}_f + \nabla \cdot (\alpha_f \mathbf{\tau}_f)
\end{equation}

\(\alpha_f\) and \(\mathbf{u}_f\) are the volume fraction and the velocity, \(p\) is the pressure, \(\rho_f\) is the density and \(\mathbf{\tau}_f\) is the viscous stress tensor of the fluid. \(\mathbf{R}_f\) is momentum exchange term that accounts for forces between the fluid and particulate phase. The set II assumes that the pressure and viscous forces of the fluid are shared between the phases. This means that the particles are affected by pressure gradients and viscous shear forces, which are the independent of the drag force.

These VANS are applied in CFD-DEM software by Goniva et al. \cite{44} and the software is used in this study.

Though the solid stress tensors do not require a closure in the DEM method, an empirical or semi-empirical drag force has to be applied to the particles. In this study, the drag relation based on Koch and Hill’s \cite{45} Lattice Boltzmann simulations is applied:

\begin{equation}
\mathbf{F}_d = \frac{\nu \beta_f}{\alpha_s} (\mathbf{u}_f - \mathbf{u}_p)
\end{equation}

\begin{equation}
\beta_f = \frac{18\mu(1 - \alpha_f)\alpha_f^2}{d_p^2} (F_0(\alpha_s) + 1/2F_3(\alpha_s)Re_p)
\end{equation}

\begin{equation}
Re_p = \frac{\alpha_f|\mathbf{u}_f - \mathbf{u}_p|d_p}{\nu}
\end{equation}

\begin{equation}
F_0(\alpha_s) = \frac{1 + 3\sqrt{\alpha_s/2} + (135/64)\alpha_s \ln(\alpha_s) + 16.14\alpha_s}{1 + 0.681\alpha_s - 5.48\alpha_s^2 + 8.16\alpha_s^3}, \alpha_s < 0.4
\end{equation}

\begin{equation}
F_0(\alpha_s) = \frac{10\alpha_s}{(1 - \alpha_s)^{17}}, \alpha_s > 0.4
\end{equation}
\[ F_3 = 0.0673 + 0.212\alpha_s + \frac{0.0232}{(1 - \alpha_s)^5} \]  \quad (13)

Pressure forces \( F_{pr} \) and viscous forces \( F_\nu \) can be expressed as (Crowe et al. [46]):

\[ F_{pr} = -V_p \nabla p \]  \quad (14)

and

\[ F_\nu = -\nabla \cdot \tau_f V_p \]  \quad (15)

where \( V_p \) is particle volume, \( p \) is pressure and \( \tau \) is viscous stress tensor.

Gravity and buoyancy forces are formulated

\[ F_g + F_b = (\rho_p - \rho_f)gV_p \]  \quad (16)

The relative velocity between the particles and fluid induces the drag force, which has a major impact on many solid-fluid flows, but other forces, e.g. virtual and Basset force, may have an impact on specific flows.

Added mass force adds inertia when relative acceleration occurs between particle and fluid. If the density of particles is decades larger than the density of fluid such as in gas-solid fluidization, the added mass force can be neglected (47). However, it is shown by Zhang et al. [48] that the added mass force has a strong effect to particle trajectories when the continuous phase is liquid. The added mass force was adopted in the liquid fluidization study by Potic et al. [22]. Similarly, the added mass force is also used by Sun and Xiao [30], and Uzi and Levy [35] in sediment and slurry transport studies. On the other hand, Di Renzo and Di Maio [23], Arolla and Desjardins [33], Capecelatro and Desjardis [34], and Blais et al. [49] have neglected the added mass force. The most common reason for neglecting the added mass force is the lack of expression for non-dilute particle flow regime, as Arolla and Desjardins [33] noted. The ratio between the drag force and the added mass force is estimated with a bead collision on a bead bed, which is a typical event in this study. In this collision,
the bead loses all of its kinetic energy and stays on the bed. The nominal fluid velocity of this study is used for the collision velocity and the timescale is estimated by contact duration. The contact duration is calculated by following the note of Legendre et al. [50]. The force ratio $F_{\text{ad}}/F_d$ is $<< 1$, which indicates that the added mass force can be neglected. In addition, a simulation with the added mass force is performed and the averages of the drag force and added mass force are compared. The comparison indicates that the added mass force are two decades smaller than the drag force. Consequently, the added mass force is not taken into account in this work.

Formation of the viscous boundary layer on a particle surface generates the Basset or history force. As the Basset force being a result of time integration, it is computationally demanding ([51]). This force may be significant in the fluid field with strong oscillation, and its formations are derived for a single particle ([46]). The force analysis is done to estimate the significance of the Basset force. A bead is accelerated from stationary to the nominal velocity of this study by using the particle relaxation time $\tau_{re,p}$. The drag force is several decades larger than the estimated Basset force, which indicates that the Basset force is not a major force. For this reason, the Basset force is neglected in this study.

The Saffman lift and Magnus effect are induced by velocity difference around a particle, the Saffman lift by the shear of fluid field, and the Magnus effect by the relative rotational velocity between a particle and the fluid. There are several expression for these forces in literature ([52], [53], and [54]). These models are derived for a single particle without the effect of volume fraction, and by neglecting the lift force, the deposition velocity is in better agreement with measurements ([55]). For these reasons, the lift forces are not included in this work. However, some of the lift effects are accounted by the pressure and viscous forces ([54]). By neglecting the Magnus effect, the rotational velocity of the particles is changed only during collision with other particles or a wall.

The momentum exchange term lays on the third Newtonian axiom and the same force acts on the solid and fluid phase but in the opposite directions. The
momentum exchange term $R_f$ has two parts: explicit and implicit (14).

$$R_f = K_f u_f - K_f \langle u_p \rangle$$

which uses the ensemble averaged particle velocity $\langle u_p \rangle$ and the momentum exchange coefficient $K_f$

$$K_f = -\frac{|\sum F_d|}{V_{cell} |u_f - \langle u_p \rangle|}$$

where $V_{cell}$ is the volume of a fluid cell and the term $\sum F_d$ sums all the particles’ drag forces in the fluid cell. The viscous and pressure forces are not included into summation because these forces are taken into account directly in the VANS (13).

The simulation requires the CFD-DEM coupling routine and the algorithm to solve the pressure and velocity fields. The PISO method, which includes explicit velocity corrections, is used to solve the pressure and velocity fields of the fluid. Only a brief explanation about the coupling between CFD and DEM is presented here. The DEM solver calculates particle data, which is then transferred to CFD solver, where the forces and momentum of the fluid are calculated and new fluid fields are solved, after which the fluid-particle forces are sent to the DEM solver. Blais and Bertrand [49] have verified that the CFD solver produce divergence free solution. More details about the CFD-DEM coupling routine and the fluid solver can be found in ref. [37]. $\kappa - \epsilon$ turbulence model by Launder and Spalding [56] is used to model the fluid phase turbulence. This simple turbulence model is selected because the effect of turbulence in four-way coupling regime needs more clarification (57, 58).

The above-mentioned coupling routine includes steps which require the cell void fraction to have been defined before proceeding further. In other words, the Lagrangian property, a particle volume, has to be mapped to the Eulerian grid. In the simple mapping method, if the particle center is inside a cell, the whole volume of the particle is assumed to be inside the cell (59). This approach has two weaknesses. First, when a particle is close to a cell boundary, a major part of the particles volume can be appointed to the wrong cell. The
other weakness involves the violation of the non-resolved approach, i.e., when the particle size is close to the cell size, incorrect void fractions can occur, which have a negative effect to the model outputs and stability. To avoid the incorrect void fractions, several approaches have been applied. Peng et al. [21] used a particle meshing method (PMM) in which the particle is meshed to the small parts before each sub-part is mapped to cells according to the location of their centroids. In the offset method, used by Alobaid et al. [60] with a momentum and void fraction grid, the void fraction is defined with the Eulerian cell displacements and averaged across the calculated void fractions. In this study, a divided method, which has similarities with the PMM, is used to define the void fraction. In the divided method, a series of points that represent partial volumes are distributed in a particle and each point is mapped to the Eulerian grid [37].

3. Validation

The present method’s capability model a horizontal pipe flow with solid material is tested against measurements by Gillies and Shook [61]. A pipe with inner diameter 0.0532 m is modeled with sand and water. The particle size distribution is presented by a single particle diameter to reduce computational demand. The measurements were performed on a closed loop device, but to decrease computational costs the, loop is replaced by a pipe with cyclic boundary conditions. The length of the simulated pipe was taken as 10 times of the inner diameter. Material properties are adopted from the study of Uzi and levy [35] and presented in Table 1. There are few exception from the material properties compared to Uzi and levy’s [35] study. The Young modulus is decreased to increase the time step size, which is common way to decrease simulation time [59]. The friction coefficient between particles is increased by following Wensrich and Katterfeld’s [62] study and assuming an angle of repose of $\sim 30 \ deg$ for sands.
<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Geometry</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pipe diameter</td>
<td>0.0532</td>
<td>m</td>
</tr>
<tr>
<td>Pipe length</td>
<td>0.532</td>
<td>m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>DEM</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(d_p)</td>
<td>2.4</td>
<td>mm</td>
</tr>
<tr>
<td>(\rho_p)</td>
<td>2650</td>
<td>kg/m(^3)</td>
</tr>
<tr>
<td>(Y_p)</td>
<td>(4 \times 10^9)</td>
<td>Pa</td>
</tr>
<tr>
<td>(Y_w)</td>
<td>(20 \times 10^9)</td>
<td>Pa</td>
</tr>
<tr>
<td>(\nu_p)</td>
<td>0.17</td>
<td>-</td>
</tr>
<tr>
<td>(\nu_w)</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>(e_{p-p})</td>
<td>0.95</td>
<td>-</td>
</tr>
<tr>
<td>(e_{p-w})</td>
<td>0.737</td>
<td>-</td>
</tr>
<tr>
<td>(\mu_{p-p})</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>(\mu_{p-w})</td>
<td>0.2</td>
<td>-</td>
</tr>
<tr>
<td>(R_{u,p-p})</td>
<td>0.25</td>
<td>-</td>
</tr>
<tr>
<td>(R_{u,p-w})</td>
<td>0.3</td>
<td>-</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Fluid</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(\rho_f)</td>
<td>998</td>
<td>kg/m(^3)</td>
</tr>
<tr>
<td>(\mu)</td>
<td>(1.0 \times 10^{-3})</td>
<td>m(^2)/s</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Mixture</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>(u_{mix})</td>
<td>1.8 and 3.1</td>
<td>m/s</td>
</tr>
<tr>
<td>(\alpha_{s,tot})</td>
<td>0.15</td>
<td>-</td>
</tr>
</tbody>
</table>

The particles are randomly initialized to the flow field that is driven by the pressure gradient between the ends of the pipe. Wall boundary condition is non-slip for the fluid phase. A particle-wall contact in DEM calculation is modeled similar way to a particle-particle contact, but the walls have infinite mass and radius in the contact calculation \[63\]. For the turbulent quantities of the fluid,
the wall functions by Kalitzin et al. [64] are applied. The mean fluid and solid velocities and the overlap are monitored during simulation to prevent excessive overlapping (> 0.01d_p) between particles and ensure the stable mean values. Over 100 mixture residue times, defined as \( L_{pipe}/u_{mix} \), are used to form the mean values.

The simulated mean solid volume fractions and the measured results by Gillies and Shook [61] are presented in Figure 1 with mixture velocities 1.8 m/s and 3.1 m/s. In the same figure, Uzi and Levy’s [35] CFD-DEM simulation results are also presented. In the measurements, on the bottom of pipe there is a dense region of solids, which decays along the height of pipe. The decay rate changes over pipe height, and the mean solid volume fractions form tilted S-shape. With the higher mixture velocity, the mean solid volume fraction is more uniform and the higher amount of particles are suspended in the upper part of the pipe. This is typical for heterogeneous suspended pipe flow. The present simulations can produce the trends of solid volume fraction profiles with both mixture velocities similar as Uzi and Levy’s [35] CFD-DEM study. However, there is some disagreement with the measured results at the bottom and the upper parts of the pipe. The present simulations underestimate the mean solid volume fraction at the upper half of the pipe and overestimates at the bottom wall, especially with the mixture velocity 1.8 m/s. The results of Uzi and Levy’s [35] CFD-DEM study are more align with the empirical results at mixture velocity 1.8 m/s. By following the force analyses of Uzi and Levy [35], the major part of difference in the results can be explained with different drag model and lower particle-particle friction fraction. As a result, the present method is capable of simulating and producing representative profiles of the solid volume fraction in solid-liquid flows, thus the present method can be utilized in the hydrodynamic design of the combined ion exchange-membrane filtration.
Figure 1: Simulated mean solid volume fractions with mixture velocities a) 1.8 m/s b) 3.1 m/s are compared to the measurements of Gillies and Shook [61].

4. Computational domain, material properties, and studied cases

A pseudo-2D calculation domain is used. The flow is modeled in 2D and the resin beads are calculated with the 3D slice. The 3D slice is used in a similar way as in the study by Ketterhagen et al. [65] to reduce the computational costs and include three dimensional effects. The depth of the 3D slice is 5\(d_p\).

The computational domain is shown in Figure 2. The fluid flow with the resin beads is introduced in the left channel. A membrane that subtracts a part of the fluid is located between the backward- and forward-facing steps. The facing steps with the membrane form a region (red lines outline this volume in Figure 2) where some of the beads should settle on the membrane. This region is called a settling area. After the settling area, the rest of fluid and resin beads exit the domain on the right. A list of dimensions of the model domain is shown in Table 2. Solid phase boundary condition is same for the wall as in the validation simulation, periodic boundary condition is applied in the slice depth direction. The boundary conditions of the fluid solver side are presented in Table 3.

The resin bead’s young modulus is estimated from the study by Tiihonen et al. [66]. The friction coefficient \(\mu_c\) is 0.5 for bead-bead and bead-wall contact,
base on polystyrene’s coefficient of friction. The valuation of the coefficient of restitution is more complicated in solid-liquid flow. Davis et al. [67] proposed a dependency between the Stokes number, defined as $St_s = (\rho_p/\rho_f)Re_p/9$, and the coefficient of restitution. Gondret et al. [47] confirmed in experiments that the coefficient of restitution is strongly depend on Stokes numbers. Similar behavior was also found by Kidanemarium and Uhlmann [29] with direct numerical simulation. Legendre et al. [50] proposed a model between the Stokes number and the coefficient of restitution. Recently Biegert et al. [68] published a collision model with a lubrication force that can model the squeezing of the liquid film between particles or a wall. Despite of the lubrication force, the higher order time integration and time sub-stepping was implemented by Biegert et al. [68] to achieve good agreement with the experiments of Gondret et al. [47]. In addition, the lubrication force increases significantly computational demand ([30]). Therefore, the reduced coefficient of restitution is used to model the lubrication force, as done by Kidanemarium and Uhlmann [69] and Deshpande et al. [70] in their studies. Same approach is applied in this study and the coefficient of restitution, which is input parameter for DEM, is reduced and set to 0.3.
Table 2: Geometry and material properties

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Geometry</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>H</td>
<td>0.01</td>
<td>m</td>
</tr>
<tr>
<td>h</td>
<td>0.01</td>
<td>m</td>
</tr>
<tr>
<td>L1</td>
<td>0.1</td>
<td>m</td>
</tr>
<tr>
<td>L2</td>
<td>0.1</td>
<td>m</td>
</tr>
<tr>
<td>L3</td>
<td>0.05</td>
<td>m</td>
</tr>
<tr>
<td>z</td>
<td>0.001</td>
<td>m</td>
</tr>
<tr>
<td><strong>DEM</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$d_p$</td>
<td>200</td>
<td>μm</td>
</tr>
<tr>
<td>$\rho_p$</td>
<td>1200</td>
<td>kg/m³</td>
</tr>
<tr>
<td>$Y_p$</td>
<td>$6.7 \times 10^6$</td>
<td>Pa</td>
</tr>
<tr>
<td>$\nu_p$</td>
<td>0.34</td>
<td>-</td>
</tr>
<tr>
<td>$c_{p-p-p-w}$</td>
<td>0.3</td>
<td>-</td>
</tr>
<tr>
<td>$\mu_{p-p-p-w}$</td>
<td>0.5</td>
<td>-</td>
</tr>
<tr>
<td>$R_{u,p-p,p-w}$</td>
<td>0.0175</td>
<td>-</td>
</tr>
<tr>
<td><strong>Fluid</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\rho_f$</td>
<td>998</td>
<td>kg/m³</td>
</tr>
<tr>
<td>$\mu$</td>
<td>$1.0 \times 10^{-3}$</td>
<td>m²/s</td>
</tr>
</tbody>
</table>
The effects of different variables and boundary values are investigated. In the first case, the boundary values for the superficial inlet velocity of the fluid are varied without allowing any flow through the membrane boundary. In the second case, a constant superficial inlet velocity is used and the fluid flow rate out of the system through the membrane is varied. In the third case, the height and the location of insertion of the beads is varied at the inlet channel. Finally, in the fourth case, flushing the resins off the membrane is studied with different fluid velocities through the membrane to the system. By default, the solid loading $Z$ is fixed to 0.25 and the number of the inserted beads is set to 300,000, unless otherwise mentioned. The sum of beads’ volume is 125% of the settling area’s volume (red lines outline this volume in Figure 2). By assuming that the maximum packing fraction is 0.61 (71), the resin beads could fill the settling more than twice. A steady state fluid flow simulation without the beads is used to initialize the simulations. The membrane is modeled with a constant velocity boundary condition, similar to the study of de Jong et al. 72. This simplification means that the pressure loss of the resin bead bed is negligible compared to transparent membrane pressure (TMP). The constants and variables are presented in Table 4 for each simulation.
Table 4: Different cases with fixed and varying parameters

<table>
<thead>
<tr>
<th>Case 1</th>
<th>Changing variable:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$V_{inlet,su}$, Range: 0.1–1 [m/s]</td>
</tr>
<tr>
<td>Fixed Variables:</td>
<td></td>
</tr>
<tr>
<td>Z</td>
<td>0.25 [-]</td>
</tr>
<tr>
<td>Permeate flux</td>
<td>0 [m/s]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case 2</th>
<th>Changing variable:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Permeate flux, Range: 0–0.025 [m/s]</td>
</tr>
<tr>
<td>Fixed Variables:</td>
<td></td>
</tr>
<tr>
<td>$V_{inlet,su}$</td>
<td>0.5 [m/s]</td>
</tr>
<tr>
<td>Z</td>
<td>0.25 [-]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case 3</th>
<th>Changing variable:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Position of bead insertion, Range: Top, mid and bottom</td>
</tr>
<tr>
<td>Fixed Variables:</td>
<td></td>
</tr>
<tr>
<td>$V_{inlet,su}$</td>
<td>0.5 [m/s]</td>
</tr>
<tr>
<td>Permeate flux</td>
<td>0.015 [m/s]</td>
</tr>
<tr>
<td>Z</td>
<td>0.125 [-]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case 4</th>
<th>Changing variable:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Back flush; Permeate flux, Range: 0, Stokes settling velocity (-0.0044) and -0.015 [m/s]</td>
</tr>
<tr>
<td>Fixed Variables:</td>
<td></td>
</tr>
<tr>
<td>$V_{inlet,su}$</td>
<td>0.5 [m/s]</td>
</tr>
<tr>
<td>Z</td>
<td>0.25 [-]</td>
</tr>
</tbody>
</table>

Convection terms are spatially discretized with the quadratic upwind interpolation convection kinematics (QUICK) scheme. The time integration is
achieved using the backward scheme. The linear scheme is used for Laplacian terms.

Results are analyzed with normalized time $t_{nor}$ and average packing fraction $\phi_{ave}$. The normalized time is simulation time divided by the total simulation time $t_{nor} = t_{si} / t_{tot,si}$. Total simulation time $t_{tot,si}$ is the sum of the bead insertion time, which is the function of overall solid loading $Z$ and the superficial velocity of water, and fluid residence time. The fluid residence time is the fluid's travel time from the inlet to the outlet with the inlet superficial velocity. The average packing fraction is the sum of resin beads’ volume divided with the volume of the settling area $\Phi_{ave} = \sum \frac{V_{bead}}{V_{settling}}$.

5. Mesh independency

A mesh independency study is more complicated to perform with multi-phase simulations than single-phase simulations. The selected grid size affects the numerical stability through the exchange fields, i.e. void fraction and momentum transfer. In addition, drag closures are developed with an averaging process, whereby a certain amount of particles is assumed to fit into an averaged volume, e.g. Koch and Hill’s Lattice Boltzmann simulations. This combination can lead to a state where the mesh independency of the simulations results cannot be obtained. An example of this behavior can be seen from Figure 3 for case 2 with permeate flux 0.015 m/s. The mesh selected for further simulations contains 7,014 cells, using 3$d_p$ spacing, which is same as in Ozel et al. study. The selected mesh spacing is a tradeoff between discretization errors and void fraction errors, which arise from mapping between Lagrangian and Eulerian frames. The mesh is presented in Figure 4.

6. Results and discussion

6.1. Case 1: varying the superficial inlet velocity

The results of varying the superficial inlet velocity without any permeate flux are presented in Figure 5. The average packing fraction in the settling area
Figure 3: The influence of mesh to average packing fraction with permeate flux 0.015 m/s.

Figure 4: The mesh used in the study.
is presented as a function of the normalized time. With the velocity 0.1 m/s, the average packing fraction increases steeply after the beads reach the settling area. The settling rate decreases at 0.2 in the normalized time, but the settling continues to $t_{\text{nor}} = 0.8$. After the bead insertion ends, the average packing fraction decreases. With the higher velocities 0.5 and 1.0 m/s, there are the same three phases as with the velocity 0.1 m/s, but the settling rates are lower and the average packing fractions stay under 0.15. Additionally, after the bead insertion the average packing fraction decreases more than with the velocity 0.1 m/s. The beads at the normalized time 0.5 are presented in Figure 6 with two velocities 1 m/s (a) and 0.1 m/s. With the higher velocity, there are no beads on the membrane surface and only a few beads are captured by a recirculation eddy, while with the lower velocity 0.1 m/s the corners are partially filled with beads and there is a thin bed on the membrane. The results are compared with the deposition velocity $v_D = 0.15 m/s$ and with the Froude number. Durand’s correlation for the deposition velocity is defined as:

$$v_D = F_L \sqrt{2gD(\frac{\rho_p}{\rho_f} - 1)}$$  (19)

where $F_L$ is the parameter that takes into account the solid volume fraction and the diameter of the particles, and $D$ is the characteristic diameter of the channel. Parameter $F_L$ is calculated with Schiller and Herbich equation [75]:

$$F_L = 1.3C_V^{0.125}(1 - e^{-6.9d_{50}})$$  (20)

where $C_V$ is solid concentration by volume and $d_{50}$ is the midpoint of particle size distribution. The Froude number lies between 0.5 and 50, depending on the fluid velocity. The Froude and Durand’s deposition velocity indicate that at the velocity 0.1 m/s, the gravitational force has a major effect on the beads and the beads are settling. On the other hand, the small Stokes number, defined as $St = \tau_{re,p}/\tau_{re,f}$ similarly to ref. [46], of $< 1$ explains the bead trajectories after the backward-facing step at the higher velocities. In the Figure 5 the decay of average packing fractions after the peak value can be explained by
Figure 5: Overall packing fraction in the settling area with different inlet fluid velocities.

the recirculation eddy, which transports the beads to the main flow, and those beads are then transported out of the domain. Generally, the resin beads are held poorly in the settling area if the fluid velocity increases over the deposing velocity, and the other methods are required to form the resin bed.
Figure 6: Resin beads are inserted with velocities 1 m/s (a) and 0.1 m/s (b). Snapshots are at the halfway point in the normalized time.

6.2. Case 2: fixed superficial inlet velocity with changing permeate flux

To achieve higher volumetric flow rates and to decrease the effect of the gravity on the bed formation, a higher superficial inlet velocity (0.5 m/s) than Duran’s \[76\] deposing velocity correlation \((v_D = 0.15 m/s)\) was selected for the permeate flux test. Figure 7 shows the average packing fraction in the settling area with varying permeate flux. The permeate flux varies from 0-0.025 m/s. In terms of the volumetric flow rate, the permeate flow rate is from 0% to 50% of the inlet flow rate. The permeate flux 0.015 m/s captures a major part of the inserted beads and a resin bead bed is formed. In other words, all the water does not need to pass through the membrane in order to form the bed in the settling area. Furthermore, with a permeate flux 0.01 m/s or less, there is a maximum in the average packing fraction. After the maximum, the non-settled beads are transported by the fluid from the settling area and the average packing fraction is stabilized. A similar phenomenon is not observed with the higher permeate.
flux. Instead of a local maximum, steady average packing fractions are observed at the end of the bead insertion. With the highest (0.025 m/s) permeate flux, the settling area is fully packed with beads and the average packing fraction is in line with the maximum random packing fraction of spheres without the cohesive forces ([77]). The permeate flux has a major effect on the formation of the resin bead bed on the membrane. Figure 8 illustrates the shapes of the formed resin bead beds and supports the analysis of Figure 7. The height of the bed increases while the permeate flux increases, but the bed is not uniform. Filling starts next to the facing steps; see Figure 8 (a). The bed gets thicker from the backward-facing step to the forward-facing step, and the recirculation with the particle-particle friction forms a gap next to the backward-facing step; see Figure 8 (b). The recirculation brings beads next to the backward-facing step and the beads are in contact with other beads on the bed. In the contact, friction between beads stops the incoming bead and beads pile up. The slope of the pile, i.e., angle of repose, is the function of the fluid forces, gravity, and particle-particle forces. After the settling area is packed, the thickening bead bed decreases the flow area at the end of the inflow channel and the fluid velocity increases. The increasing fluid velocity prevents a total blocking of the flow channel and the bed stabilizes to its maximum thickness. In addition, the gap next to the backward-facing step is filled; see Figure 8 (c). As a result, the active resin bead bed can be achieved without dead-end packing, i.e., pushing all fluid through the membrane.
Figure 7: Average packing fraction in the settling area in time with varying permeate flux and constant inlet superficial flow velocity.
Figure 8: Constant inlet superficial velocity with permeate flux (a) 0.005 m/s (b) 0.015 m/s (c) 0.025 m/s. At the end of the simulation.

6.3. Case 3: the position of bead insertion in the inlet channel

The vertical position of the bead insertion is varied to study its effect to the bed formation. The bead insertion area is set to half of the inlet surface area, which means the overall solid loading $Z$ needed to be decreased by 50%. To point out the influence of the overall solid loading, a decreased overall loading $Z = 0.125$ with the whole inlet surface area is compared to a reference simulation, which is the case 2 with permeate flux 0.015 m/s. After comparing
the reference simulation and the decreased overall solid loading simulation, the
decreased bead insertion area is varied in the vertical direction and compared
to the decreased overall loading simulation. There are three positions for the
half insertion area: the bottom, the middle and the top of the inlet channel.

Figure 9 presents the average packing fraction with the different bead inser-
tion positions, the reference simulation and the decreased overall loading sim-
ulation with the respect to the normalized time. Comparing the settled beds
at the end of the simulations, there is a negligible difference with the reference
simulation and the decreased loading simulation. Based on this observation, the
overall solid loading has little effect in the formation of the resin bead bed in
this geometry in this case. However, the position of the bead insertion has a
major effect to the formation of the bed (see Figures 9 and 10). If the beads are
inserted from the bottom half of the inlet channel, there is a linear packing rate
until \( \Phi_{ave} \) is 0.5 and the rate decays to zero as \( \Phi_{ave} \) reaches its maximum. With
the insertion area in the middle of the inlet, a lower average packing fraction
\( \Phi_{ave}(0.2) \) is achieved when compared to other simulations or to the simulation
where the beads are inserted at the bottom of the inlet channel. After the initial
burst of beads to the settling area, the beads are settled on the membrane at
a nearly linear rate until bead insertion ends. The insertion from the top of
the inlet channel is an exception to the other cases. After the initial burst, the
average packing fraction does not increase and nearly all the beads escape from
the settling area after the end of the insertion. This can be explained by the
Stokes number, as at low Stokes numbers (\(<1\)), the tracking error of the parti-
cles on the streamlines of the flow is small. The streamlines, which mainly start
at the top of the inlet and end at the exit, carry the beads through the device
without depositing the beads. In the other words, only those streamlines that
end at the membrane transport beads on the membrane. In addition, all bead
trajectories cannot be explained with low Stokes number because the collisions
between the beads and walls affect to trajectories. To conclude, the position of
the bead insertion has a major effect on the formation of the resin bead bed
on the membrane, and it is recommended to insert beads near the bottom wall

29
of the inlet channel. Technically, inserting the resin beads into a certain area without mixing can be difficult.

Figure 9: Influence of the bead insertion position in the inlet channel.
Figure 10: Comparison of the bottom (a) and top insertion positions at reduced time = 0.5. The major part of the inserted beads are settled on the bed with the bottom insertion but with the top insertion beads are carried away by the fluid.

6.4. Case 4: flushing

Back flushing is a common treatment method for restoring the membrane permeability. In this case, the aim of the back flushing is to remove the saturated or contaminated resin beads from the settling area. This can be done with and without back flushing. The bead bed is packed similarly to case 2 with a constant superficial inlet velocity of the fluid (0.5 m/s) and permeate flux 0.015 m/s. After the bed is formed, the direction of the fluid velocity at the membrane surface is reversed and three different velocities are used, namely 0 m/s (no back flushing), the Stokes settling velocity 0.0044 m/s, and the same velocity (0.015 m/s) as the in the packing phase. Figures [11] and [12] show the results of the packing and flushing phase. The packing phase is explained in the case 2. In the beginning of flushing, the rate of bead removal is high in Figure [11] After the initial burst, the rate of bead removal decays. With higher (0.015
back flushing, the initial burst removes more beads compared to other simulations and the decay rate is steeper. Practically, all beads are removed at 0.8 in the normalized time. The corners of the settling area are not cleared of the beads without back flushing (see Figure 12(a)) and some of the beads are captured in the recirculation eddy. With the Stokes settling velocity, there are no settled beads in the backward-facing step corner, recirculation holds the part of the beads, and few beads are remaining in the forward-facing step (see Figure 12(b)). Back flushing at 0.015 $m/s$ removes the beads form the settling area, as can be seen from Figure 12.

The flushing phase can be explained with two factors: the drag of flushing velocity and the forming recirculation eddy. In the simulation without the back flushing, the formation of the recirculation eddy starts after the uppermost beads are transported away from the bed by the fluid flow. The forming recirculation eddy lifts beads from the bed, which affects the flow fields of fluid. This phenomenon is visible in Figure 12(a). With a back flushing at 0.015 $m/s$, the back flush pushes beads away from the settling area and the back flushing flux is high enough to prevent the formation of the recirculation eddy. With the Stokes settling velocity, the back flushing pushes beads from the corners, but it is not enough to prevent the formation of the recirculation eddy. The recirculation eddy captures some of the beads and loses them slowly to the main flow. To summarize, back flushing is needed to remove the beads from the corners of the settling area. The Stokes settling velocity as back flushing velocity is enough to remove the beads if there is enough time to empty the recirculation eddy. On the other hand, higher back flushing velocities than the Stokes settling velocity prevent the formation of the recirculation eddy and remove the bed faster.
Figure 11: Removing beads from the settling area with and without back flushing
Figure 12: At the end of back flushing a) with zero velocity b) with Stokes settling velocity (0.0044 m/s) c) with velocity 0.015 m/s

7. Conclusion

This study demonstrates how the computational fluid dynamic-discrete element method is applied to analyze and study the hydrodynamics of combined ion exchange-membrane filtration, which is a new application area for this method. The test scale model of this filtration method is selected for the demonstration. The details on the bed formation and the parameters affecting it are analyzed
by varying the fluid velocity, membrane flux, position of bead insertion, and back flushing. Based on the analyzed parameters, the following suggestions for the hydrodynamic design can be presented:

- Inlet velocities under the Durand’s depositing velocity have to be used to form the resin bead bed when there is no flow through the membrane.

- Higher inlet velocities can be used if a part of the fluid is flowing through the membrane.

- The position of the bead insertion has a large influence on the formation of the filtration bed. This is because particles with small Stokes number follow the streamlines of the fluid. Thus, for the bed formation, it is beneficially to insert the beads closer to the membrane.

- Back flushing has to used for the removal of the beads from the corners, and Stokes settling velocity is in the correct range for the back flushing velocity.

- Settling velocity plays a major role in the design of the combined ion exchange-membrane filtration system.

While detailed measurements of the flow fields are required to validate the simulations, the presented results are reasonable, especially when considering the Stokes settling velocity and Durand’s depositing velocity. The presented computational approach is capable of capturing phenomena in the solid-liquid flow at a very detailed level, even granular phase’s capability to form force-chains, i.e., form piles. The presented method seems highly applicable for studying the hydrodynamics of particle-water mixtures in wastewater applications that are aimed for the reduction of harmful emissions and improve water quality.
References


[13] D. L. Huggins, R. H. Piedrahita, T. Rumsey, Use of computational fluid dynamics (cfd) for aquaculture raceway design to increase settling ef-


43


[70] R. Deshpande, S. Antonyuk, O. Iliev, Study of the filter cake formed due to the sedimentation of monodispersed and bidispersed particles using discrete
element method computational fluid dynamics simulations, AIChE Journal 0 (0). doi:10.1002/aic.16529


