Numerical and experimental investigation for cleaning process of submerged outside-in hollow fiber membrane

Xingfei Guo, Yaowu Wang, Hongwei Zhang, Pengfei Li and Cong Ma

ABSTRACT

Membrane fouling has limited extensive applications for hollow fiber membranes in water treatment. Backwashing and air scouring can effectively solve this problem in the submerged outside-in hollow fiber membrane system. In this study, variation of the fouling layer on the membrane surface during backwashing and the impact of shear stress caused by air scouring on fouling removal were investigated through computational fluid dynamics (CFD) simulation. The backwashing and air scouring process were simulated using CFD and the results were verified by experimental studies. The results of experimental studies are in accordance with the simulation results. During the backwashing process, the velocity profile inside the reactor was presented, and visualization of the particle movement to illustrate the dynamic peeling process of the fouling layer on the membrane surface was also shown. The formation of uneven cleaning reveals that the upper region of the fibers has an excellent cleaning effect during backwashing. After that, the supporting role of air scouring was investigated in the study. It is concluded that the lower part and the middle region of the fibers suffer greater shear stress by analyzing the velocity contours and vectors, and the analysis results indicated that air scouring can further remove membrane fouling.

Key words | air scouring, backwashing, CFD simulation, membrane fouling, shear stress

ABBREVIATIONS

CFD computational fluid dynamics
MBR membrane bioreactor
3D three-dimensions
TMP transmembrane pressure

NOMENCLATURE

\( \ddot{a} \) particle's acceleration (m/s\(^2\))
\( D, K \) prescribed matrices
\( d \) particle's diameter (m)
\( F \) body force (N)
\( F_\text{e} \) external body force (N)
\( F_{\text{lift},q} \) lift force (N)
\( F_{\text{wl},q} \) wall lubrication force (N)
\( F_{\text{vm},q} \) virtual mass force (N)
\( F_{\text{td},q} \) turbulent dispersion force (N)
\( \ddot{g} \) gravitational acceleration (m/s\(^2\))
\( J \) membrane flux of every moment (m\(^3\)/m\(^2\)/s)
\( J_0 \) initial membrane flux (m\(^3\)/m\(^2\)/s)
\( j \) phase index
\( k \) turbulent kinetic energy (J/kg)
\( m_{pq} \) mass transfer from the \( p \)th to \( q \)th phase (kg/s)
\( \dot{m}_{qp} \) mass transfer from phase \( q \) to phase \( p \) (kg/s)
\( n \) number of phases
\( p \) pressure (Pa)
\( \tilde{R}_{pq} \) interaction force between phases (N)
\( S_i \) sink term for the \( i \)th (x, y or z) momentum equation
\( \tilde{v}_m \) mass-averaged velocity (m/s)
\( \tilde{v}_q \) velocity of phase \( q \) (m/s)
\( \tilde{v}_{pq} \) interphase velocity (m/s)
\( \tilde{v}_{d,k} \) drift velocity for secondary phase \( k \) (m/s)

Greek symbols

\( \alpha_k \) volume fraction of phase \( k \)
\( \alpha \) permeability
\( \alpha_k, \alpha_\epsilon \) inverse effective Prandtl numbers for \( k, \epsilon \)
\(e\) turbulent dissipation rate (m\(^2\)/s\(^3\))
\(\mu_m\) mixture viscosity (Pa·s)
\(\rho_m\) mixture density (kg/m\(^3\))
\(\mathbf{T}\) stress tensor (Pa)
\(\tau_p\) particle relaxation time

**INTRODUCTION**

Coagulation-membrane filtration has a wide range of applications in the treatment of drinking water and wastewater (Wan et al. 2012). Coagulation pretreatment is always an effective way to markedly decrease the organic contents in water and improve the permeability of the membrane (Sun et al. 2016). In addition, the coagulation cake layer can greatly reduce the direct deposition of organic pollutants on the surface of membranes. However, particles can still accumulate on the membrane surface, which then forms a cake layer and clogs the pores in the membranes. Membrane fouling still seriously restricts the efficient application of coagulation-membrane technology.

Therefore, it is essential to clean the fouled membrane after filtration to restore and guarantee a high membrane flux. Hydraulic backwashing and air scouring are two important methods in physical cleaning of outside-in hollow fiber membranes (Diez et al. 2012) and have been extensively used for the removal of fouling without introducing new contaminants. Computational fluid dynamics (CFD), a powerful tool to predict the flow dynamics inside a membrane device (Samstag et al. 2016), can be used to investigate physical cleaning methods and understand the hydrodynamic behaviors (e.g., fluid flow, particles movement and shear stress) during the cleaning process. As a result, CFD technology is widely applied in membrane fouling control.

In the backwashing process, the pressure applied on the permeate side pushes the permeate back through the membranes. Previous studies have suggested that backwashing can effectively remove fouling on the surface of hollow fiber membrane (Katsoufidou et al. 2007; Akhondi et al. 2014). Katsoufidou et al. (Katsoufidou et al. 2007) found a significant recovery of membrane flux (more than 24%) after each backwashing step in the single fiber module. Akhondi et al. (Akhondi et al. 2014) also suggested that the periodic backwashing pattern could effectively remove the fouling layer in the fouling region in a dead-end mode. Backwashing is also known to be particularly effective against reversible fouling and can effectively detach the fouling cake from the membrane surface (Wu & Lin 2012; Mansour et al. 2013). By means of CFD, Mansour et al. (Mansour et al. 2013) simulated the movement of particles in transverse section of capillary membranes during backwashing under two different dead-end filtration modes, and indicated that backwashing could accelerate the movement of particles and reduce the blockage in membrane pores. The simulation results from Wu and Lin’s work (Wu & Lin 2012) also suggested that backwashing operation with a side stream flow channel could effectively clean the particles deposited on the membranes. However, to the best of our knowledge, previous studies have only simulated the movement of reversible particles on the membrane surface, and the investigation that attempts to model the backwashing process using CFD is still needed. A more specific mechanism involved in the backwashing of membrane fouling needs further investigation.

After backwashing, there were still some deposits attached to the membrane surface. In order to further enhance the cleaning efficiency, air scouring, which uses air to remove the detached or the loosely bound fouling off the membrane surface, is introduced (Kayaalp & Ozturkmen 2016). Previous studies have proved that air scouring can also effectively remove fouling on the membrane surface (Serra et al. 1999). Serra et al. (Serra et al. 1999) pumped air into the hollow fiber modules and found that the removal efficiency for fouling was increased by 40%. During the air scouring, shear stress plays a predominant role in the removal of particles accumulating on the membrane surface (Yang et al. 2011), and CFD models can be used to gain more insights into the shear stress induced by air scouring (Ratkovich & Bentzen 2013). Yan et al. (Yan et al. 2016) optimized the membrane bioreactor configuration and found that the average shear stress of the membrane surface was related to the aeration intensity through CFD simulation, thus directly affecting the formation rate of membrane fouling. Although air scouring is effective in reducing membrane fouling, the hydrodynamic behaviors in the submerged hollow fiber membrane system are still unknown. CFD simulation provides a possibility to understand the flowing profiles of the gas-liquid two-phase and distribution of shear stress on the membrane surface. In this way, the removal effect of fouling during air scouring can be analyzed effectively.

Backwashing and air scouring can be coupled in the removal of membrane fouling (Qaisrani & Samhaber 2011; Ye et al. 2011). Qaisrani and Samhaber (Qaisrani & Samhaber 2011) suggested that the coupling of backwashing and bubbling was more effective than a separate one. In addition, air scouring during backwashing can prevent the formation of membrane fouling in the filtration region, thus limiting re-deposition (Serra et al. 1999; Ye et al. 2011). Ye et al. (Ye et al. 2011) directly took photographs of the fouling deposition

Downloaded from https://waponline.com/wst/article-pdf/78/6/1283/4489527/wst076061283.pdf by guest
and removal during filtration, and observed that the thickness of the fouling cake was expanded from 23 to 57 μm after backwashing during the 1st filtration cycle. At the end of the 18th filtration cycle, the fouling cake with 25 μm thickness reduced to 20 μm after backwashing coupled with air scouring. With the aid of air scouring, these results prove the improvement of backwashing effectively. CFD can also be used to simulate the fluid motion simultaneously on backwashing and air scouring, and provides a quantitative description of the hydrodynamics in three-dimensions (3D), which is difficult to achieve in conventional experiments. Moreover, the relative contributions of backwashing and air scouring in fouling removal can be identified by CFD models.

Although the coupling of backwashing and air scouring has been successfully applied in the membrane cleaning process, how to use CFD to simulate the hydrodynamic behaviors of fouling removal on the membrane surface is still lacking. In this work, the backwashing and air scouring process of the membrane have been investigated using 3D CFD simulation, where the volume fraction and average shear stress have been taken into consideration. Visualization of the peeling process of the fouling layer on the membrane surface during backwashing during the 1st filtration cycle, the fouling cake with 25 μm thickness reduced to 20 μm after backwashing coupled with air scouring. With the aid of air scouring, these results prove the improvement of backwashing effectively. CFD can also be used to simulate the fluid motion simultaneously on backwashing and air scouring, and provides a quantitative description of the hydrodynamics in three-dimensions (3D), which is difficult to achieve in conventional experiments. Moreover, the relative contributions of backwashing and air scouring in fouling removal can be identified by CFD models.

The movement of fouling particles in the backwashing process and variation of the bubble column during air scouring need to be analyzed in this work, therefore the two processes can be considered as a multiphase flow simulation of liquid-solid and gas-liquid flows. The Euler-Euler approach was chosen for the numerical calculation of multiphase flows in this study (FLUENT 2005). The basic equations of the multiphase modeling assume an incompressible fluid with the corresponding conservation of mass and momentum equations, and the energy equation was not considered.

In different Euler-Euler multiphase models (Table 1), the mixture model is designed for two or more phases (fluid or particulate), and the applications include particle-laden flows with low loading, sedimentation, and cyclone separators (FLUENT 2005; Cornelissen et al. 2007). Therefore, the mixture model was chosen to describe the motion of particles in the study. In this approach, conservation of mass and momentum provides the governing equations for the interpenetrating liquid and solid phases. The equations are given below.

(i) The continuity Equation (1):

\[ \frac{\partial}{\partial t} (\rho_m) + \nabla \cdot (\rho_m \bar{v}_m) = 0 \]  

where \( \bar{v}_m \) is the mass-averaged velocity, and \( \rho_m \) is the mixture density:

\[ \rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \]  

\( \alpha_k \) is the volume fraction of phase \( k \).

### Table 1 | The comparison between different multiphase flow models

<table>
<thead>
<tr>
<th>Multiphase models</th>
<th>Characteristics</th>
<th>Applications</th>
</tr>
</thead>
<tbody>
<tr>
<td>VOF model</td>
<td>Appropriate for stratified or free-surface flows.</td>
<td>Filling, sloshing, large bubbles, dam break, jet breakup, liquid-gas interface.</td>
</tr>
<tr>
<td>Mixture model</td>
<td>Appropriate for flows in which the phases mix or separate.</td>
<td>Particle-laden flows with low loading, bubbly flows, sedimentation, cyclone separators.</td>
</tr>
<tr>
<td>Eulerian model</td>
<td>The Eulerian model is chosen when accuracy is more important than computational effort.</td>
<td>Bubble columns, risers, particle suspension, fluidized beds.</td>
</tr>
</tbody>
</table>
(ii) The momentum Equation (3):

\[
\frac{\partial}{\partial t} (\rho_m \vec{v}_m) + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^T)] + \rho_m \vec{F} + \nabla \cdot \left( \sum_{k=1}^{n} \alpha_{k} \rho_k \vec{v}_{d,k} \vec{v}_{d,k} \right)
\]

where \( n \) is the number of phases, \( \vec{F} \) is body force, \( \mu_m \) is the mixture viscosity:

\[
\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k
\]

and \( \vec{v}_{d,k} \) is the drift velocity for secondary phase \( k \):

\[
\vec{v}_{d,k} = \vec{v}_k - \vec{v}_m
\]

The drift velocity and the relative velocity (\( \vec{v}_{pq} \)) are connected by Equation (6):

\[
\vec{v}_{d,p} = \vec{v}_{pq} - \sum_{k=1}^{n} c_k \vec{v}_{qk}
\]

where \( \vec{v}_{pq} = \vec{a} \tau_p, \vec{a} \) is the particle’s acceleration, \( \tau_p \) is the particle relaxation time:

\[
\tau_p = \frac{\rho_p d^2}{18 \mu_q}
\]

\( d \) is the diameter of the particles.

The Eulerian model is the most complicated multiphase model, and allows for the modeling of multiple separate, yet interacting phases (Table 1). The phases can be liquids, solids, or gases in almost any combination (FLUENT 2005). The applications of the Eulerian multiphase model include bubble column, floating and fluidized bed. In this study, the Eulerian model was applied to simulate the bubble flow. The governing equations are expressed as:

(i) The continuity Equation (8) for phase \( q \):

\[
\frac{\partial}{\partial t} (\rho_q \vec{v}_q) + \nabla \cdot (\rho_q \vec{v}_q \vec{v}_q) = \sum_{p=1}^{n} (m_{pq} - m_{qp}) + S_q
\]

where \( \vec{v}_q \) is the velocity of phase \( q, m_{pq} \) characterizes the mass transfer from the \( p^{th} \) to \( q^{th} \) phase, and \( m_{qp} \) characterizes the mass transfer from phase \( q \) to phase \( p \).

(ii) The momentum balance for phase \( q \) yields the following Equation (9):

\[
\frac{\partial}{\partial t} (\rho_q \vec{v}_q) + \nabla \cdot (\rho_q \vec{v}_q \vec{v}_q) = -\alpha_q \nabla p + \nabla \cdot \vec{F}_q + \alpha_q \rho_q \vec{a}
\]

\[
+ \sum_{p=1}^{n} \left( \vec{F}_{pq} + m_{pq} \vec{v}_{pq} - m_{qp} \vec{v}_{qp} \right)
\]

\[
+ (\vec{F}_q + \vec{F}_{lift,q} + \vec{F}_{vel,q} + \vec{F}_{vm,q} + \vec{F}_{td,q})
\]

where \( p \) is the pressure shared by all phases, \( \vec{F}_{pq} \) is the interaction force between phases, \( \vec{v}_{pq} \) is the interphase velocity, \( \vec{F}_q \) is the external body force, \( \vec{F}_{lift,q} \) is the lift force, \( \vec{F}_{vel,q} \) is the wall lubrication force, \( \vec{F}_{vm,q} \) is the virtual mass force, and \( \vec{F}_{td,q} \) is the turbulent dispersion force.

By comparing different turbulence models (Table 2), the RNG k-\( \varepsilon \) turbulent model was used in the simulation (Versteeg & Malalasekera 2007). An equation about the turbulent dissipation rate \( \varepsilon \) was introduced based on the equation of turbulent kinetic energy \( k \), then the \( k-\varepsilon \) model was formed, which is called the standard \( k-\varepsilon \) model (Versteeg & Malalasekera 2007). Furthermore, the RNG \( k-\varepsilon \) model is an improvement on the standard \( k-\varepsilon \) model, which enables the use of lower Reynolds numbers, and the additional term in the \( \varepsilon \) equation improves the accuracy when modeling rapidly strained flows (FLUENT 2005; Versteeg & Malalasekera 2007). In addition, the effect of swirl turbulence is included in the RNG model. These features make the RNG \( k-\varepsilon \) model more reliable and accurate than the standard \( k-\varepsilon \) model.

**Table 2 | The comparison between different turbulence models**

<table>
<thead>
<tr>
<th>Turbulence models</th>
<th>Characteristics</th>
<th>Applications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard ( k-\varepsilon ) (SKE) model</td>
<td>The SKE model is valid only for fully turbulent flows.</td>
<td>Fully turbulent flows.</td>
</tr>
<tr>
<td>RNG ( k-\varepsilon ) model</td>
<td>Better performance for more complex rapidly strained flows than SKE.</td>
<td>Rapidly strained flows, swirling flows, separated flow.</td>
</tr>
<tr>
<td>Realizable ( k-\varepsilon ) (RKE) model</td>
<td>Produces non-physical turbulent viscosities when the computational domain contains both rotating and stationary fluid zones.</td>
<td>Strong streamlined curvature, vortices, rotation.</td>
</tr>
</tbody>
</table>
The equations are given below:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i}\left(\alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{10}
\]

and

\[
\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_i}\left(\alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_j}\right) + C_\varepsilon \frac{\varepsilon}{k} (G_k + C_{\varepsilon G_b}) - C_{\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{11}
\]

where

\[
\dot{\varepsilon} = \frac{\mu_{\text{eff}}}{\mu} \tag{12}
\]

\[
\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{13}
\]

\[
C_\mu = 0.0845, \quad \alpha_k = \alpha_\varepsilon \approx 1.393
\]

\[
R_\varepsilon = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0) \varepsilon^2}{1 + \beta \eta^3} \tag{14}
\]

\[
\eta_0 = 4.38, \quad \beta = 0.012
\]

\[
C_{\varepsilon} = C_{\varepsilon t} + \frac{C_\mu \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \tag{15}
\]

\[
C_{\varepsilon t} = 1.42, \quad C_{\varepsilon} = 1.68
\]

where \( k \) is the turbulent kinetic energy, \( \varepsilon \) is the turbulent dissipation rate, and \( \alpha_k, \alpha_\varepsilon \) are the inverse effective Prandtl numbers for \( k, \varepsilon \), respectively.

The porous media model is widely used in flows through filter papers, packed beds and tube banks (Fluent 2005). Previous studies have applied the porous media model in the membrane system and have coupled it with the CFD model to investigate hydrodynamic properties (Brannock et al. 2010). Therefore, the hollow fiber membranes were set as porous media during the simulation of this study. In the porous zone, a porous media model (Fluent 2005) is utilized to apply an additional volume-based momentum sink to simulate the flow resistance through a porous region:

\[
S_i = - \left( \sum_{j=1}^{3} D_{ij} \mu_{ij} + \sum_{j=1}^{3} K_{ij} \nu_{ij} \right) \tag{16}
\]

where \( S_i \) is the sink term for the \( i \)th (\( x, y \) or \( z \)) momentum equation, \( j \) is the phase index, and \( D \) and \( K \) are prescribed matrices.

Regardless of the convective acceleration and diffusion, the porous media model is simplified to Darcy’s law:

\[
\nabla p = - \frac{\mu}{\alpha} \dot{\varepsilon} \tag{17}
\]

where \( \alpha \) is the permeability.

### Model creation and boundary conditions

The 3D modeling software of Pro/E Wildfire 4.0 was used to set up the geometry. The main computational region is a rectangular geometry that has a 0.15 m × 0.15 m × 0.65 m size and contains 13 hollow fiber membranes with a 1.1 mm diameter. The length of the membrane is 0.4 m. The main body of the model and each fiber were modeled separately. There is a simplified aerator at the bottom of the system tank for the simulation of air flow.

A detailed view of the geometry can be seen in Figure 1. The membrane module was placed at the bottom of the reactor above 0.05 m. Hollow fiber membranes were arranged in an approximately circular cross section to conform to the actual shape of the membrane module. The membrane size in the model reflects the actual diameter of the hollow fiber membranes. In addition, in view of the fact that the fibers are fixed on the support column in the real module, hollow fibers were modeled as rigid elements (Wang et al. 2014b), and the sway of fibers was not considered.

The global mesh geometry is represented through multiple views (Figure 2). The structured grids were used for meshing to obtain high quality mesh. Due to the importance of the flow around the fibers, the mesh in these regions needs to be densified to get accuracy. In the same way, all fibers require the same operation. The mesh numbers in the center of reactor are more than that in other regions to ensure the accuracy of the simulation results. The reason can be interpreted that the investigation of flow and particle movement mainly focuses on the central membrane bundle.

The fluid domain was discretized into \( 8.19714 \times 10^5 \) number of elements for geometry and \( 3.1512 \times 10^8 \) number of elements for each membrane model. The local mesh of
one hollow fiber membrane is shown in Figure 3. Especially, $1.48338 \times 10^5$ is the number of elements in the middle fiber. After meshing for the rectangular geometry and fibers, ICEM CFD was used to merge the mesh geometry. Therefore, the number of mesh elements in the entire model is $1.346196 \times 10^6$, meeting the requirements of the calculation.

Water enters the membranes from the upper end of the fibers, and the value of the velocity-inlet was set to 0.014 m/s. Air enters the system tank through the simplified aerator, and the value of the velocity-inlet was chosen to be 0.387 m/s in the air scouring process.

**Model solution and post-processing**

Remarkably, each fiber was modeled as a porous region with axial and radial resistances to flow. The corresponding assumptions will be established due to the limitation of the simulation, which is beneficial to reduce the difference between the simulation and the actual test. A thin fouling layer is initially formed by particles on the membrane surface after a filtration process, and the formation of an electrical double layer and particle compaction over time can be neglected (Li et al. 2012). After that, the membrane was assumed to be porous media, and the pore structure as well as the irreversible fouling caused by particles entering the membrane pores were also neglected.

In both cases, the flow was assumed to be unsteady. In the simulation of liquid-solid two-phase flow, the PISO pressure-velocity coupling algorithm and standard pressure method for pressure calculation were applied to the solution. For visualization purposes, every 50th time step was saved, and the time step size was set to 0.02 s. In the simulation of the gas-liquid two-phase flow, the drag force and gravity were taken into account in the model. The QUICK scheme was used for discretizing and the phase coupled SIMPLE pressure-velocity coupling algorithm was adapted for calculating the equations. The time step size was set to 0.02 s and every 100th time step was saved. During numerical calculations, a convergence criterion of $10^{-3}$ was set to guarantee a converged solution.

Due to the great computation consumption and the high-precision requirements, the computer possesses 8.0 GB ram to handle the computational region. In addition, Tecplot 360 was used for post-processing of the completed calculations and the corresponding two/three-dimensional graphics can be obtained.

**Experimental setup**

The designed experiments were conducted to validate the accuracy of the simulation results. The experimental setup used in this study is shown in Figure 4. Two different cleaning methods for the recovery of hollow fiber membrane performance had been studied. The micro-polluted lake water of the campus was used as feed water. A cylindrical-type PVDF hollow fiber membrane module was adopted in the experiment. The pore diameter of the single membrane is $0.02 \mu m$ and the operating pressure is controlled in the range of 0.01–0.05 MPa. The system is composed of the membrane module, suction and aeration pump, valve,
pipe fittings. The operation of the whole apparatus was completed in conjunction with the valve and pump.

To compare the effects of different cleaning methods, two groups of experiments were carried out in constant-flux mode (15 LMH) and constant pressure mode (20 kPa). Each experiment was conducted for 180 min. After the system is running, every 30 minutes, hydraulic backwashing (45 LMH for 2 min) and air scouring (0.15 m³/h for 3 min) were conducted. Under the constant pressure condition, the flow rate was recorded by the flowmeter. The change of pressure drop was recorded through the pressure gauge under a constant flow.

**RESULTS AND DISCUSSION**

**The flow field during backwashing**

The flow field inside the reactor changes during the backwashing process. The velocity contours inside the vessel represent the velocity field when backwashing was conducted on the 1 and the 19 s (Figure 5). There is an additional and enlarged view for better observation. At the beginning of backwashing (Figure 5(a)), water gradually spreads to the vessel walls, resulting in a greater velocity near the reactor walls, and variations of velocity are also
more apparent than those in other regions (Figure 5(a)). The overall flow rate within the reactor increases gradually as backwashing continues (Figure 5(b)). At 19 s, the flow rate near the walls compared with the beginning of backwashing has increased significantly (Figure 5(b)). There is an obviously irregular velocity field around the fibers and form a high-velocity in these regions (Figure 5(b)). Meanwhile, the distributions of the velocity field along the vertical direction are different according to different layers in the reactor (Figure 5(b)), and the greater flow rate occurs in the corner of the upper and lower areas in the reactor. Most likely, it is due to the formation of vortex in the corner induced by water flow. The distribution and variations of flow velocity in the reactor may affect the removal effect of the fouling at different locations on the membrane surface.

The peeling process of the fouling layer

The dynamic peeling process of the fouling layer on the surface of the hollow fiber membrane is presented in Figure 6. At 1 s of backwashing, the particle layer is scattered obviously as water flows down the fibers (Figure 6, 1 s). When the entire fouling layer is lifted up, the volume fraction of the particles reduces from $1.5 \times 10^{-3}$ to $5.0 \times 10^{-5}$-$6.5 \times 10^{-4}$. Within 2-5 s, the volume fraction of the particles declines gradually along the fiber direction downward, indicating that the accumulated particles are being continuously removed from the membrane surface (Figure 6, 2-5 s). Then, the particles at the lower end of the fiber are removed due to the action of the backwashing water, leading to a decrease in the volume fraction in this region (Figure 6, 6 s). After that, the continuous decrease in the volume fraction on the surface indicates that particles are being continuously removed (Figure 6, 9 s). After 10 s of backwashing, the significant segment phenomenon can be observed (Figure 6, 10 s), suggesting different effects of backwashing on the membrane surface. The upper part of the fiber suffers a strong force, leading to a rapid decrease in the volume fraction in this area (Figure 6, 10 s). The accumulation of particles in the middle portion of the fiber is relatively uneven (Figure 6, 10 s), and the highest volume fraction occurs in the lower end of the fiber (Figure 6, 10 s). This may be due to the low shear stress in these portions and the change of flow field in the reactor. With the continuous removal of the accumulated particles, the formation of uneven cleaning occurs. After 19 s of backwashing, the volume fraction on the membrane surface is already at a low level ranging from $2.0 \times 10^{-7}$ to $2.4 \times 10^{-6}$, and the lowest volume fraction can be found in the upper region of the fiber (Figure 6, 19 s). Besides the effects of backwashing on the membrane surface, the cleaning effects may also be influenced by the flow movement in the reactor. At the moment, the volume fraction of particles is low, indicating that the membrane surface fouling is effectively cleaned, and the fouling can be further cleaned with continuous backwashing. Relatively speaking, the upper region of the membrane with the excellent cleaning effect is prior to the middle and lower region. The conclusion was verified in Wang et al.’s experimental study (Wang et al. 2014a) about variation of flux along the hollow fiber.
direction during backwashing, and their results suggested that the backwashing effect gradually reduced from the outlet to the end under a certain fiber length.

The average volume fractions of particles on the membrane surface along the fiber direction in the backwashing process are illustrated in Figure 7. The particles are distributed in an irregular pattern at 1 s (Figure 7, 1 s). With the removal of particles from the membrane surface, the volume fraction of the particles is continuously reduced (Figure 7, 11 s). The layering trend of the particles on the membrane surface is becoming more apparent as backwashing proceeds. After 19 s, the volume fraction of the particles is relatively low at both ends and the middle part of the fiber (Figure 7, 19 s). However, some sections with a high volume fraction indicate that there are still fouling particles in these areas (Figure 7, 19 s). The situation is consistent with the volume fraction field on the membrane surface when backwashing reaches 19 s (Figure 6, 19 s). Combining the data graph, it can obviously reveal the changing process of the fouling layer on the surface of hollow fiber membranes. In addition, the region with a relatively poor effect of backwashing was identified, and the effect of air scouring was investigated based on the above research results.

Air scouring after backwashing

Simulations are also conducted to investigate the gas-liquid two-phase flow in different systems (Delnoij et al. 1997b), and variations in the bubbles in the system are analyzed in their studies. In this work, the vertical velocity profile in the center section of the reactor is shown through two-dimensional graphics (Figure 8). The variation of the bubble flow with time shows that the bubbles rose in a straight line at the beginning and a high velocity appeared in the center of the section (Figure 8). After a while, the movement of bubbles began to deviate from the axis region (Figure 8), and it is considered that the bubble flow led to the increase in the intensity of the liquid turbulence, which then aggravated the flow in the liquid level and promoted the distribution of air in the liquid phase. The movement of bubbles observed in this study is consistent with that in Delnoij et al.’s work (Delnoij et al. 1997a). After that, the bubble group began to swing, and the swing presented a periodicity with time (Figure 8). That is because of the momentum transfer between air and liquid in the flow field. In addition, the development of the vortex is the main reason for the movement of the bubble flow (Delnoij et al. 1997a). The air distributed in a larger range is due to the merger of bubbles during the bubble-rising process. In sum, in this study, simulation results of the bubble movement and the velocity field in the gas-liquid two-phase flow agree well with the particle image velocimetry experimental results from Besbes et al. (Besbes et al. 2015).

Velocity contours during air scouring

Velocity contours for the air scouring are shown in Figure 9. Different cross-sections were chosen to understand the flow
phenomena around the module. Along the direction of aeration, the fluid velocity increased initially, followed by a decrease (Figure 9). The small bubbles generated from the aeration holes rose upward in a relatively concentrated manner near the area of the holes. Around the aeration holes, the region with higher velocity has a limited distribution (Figure 9). Along the upward direction of the fibers, the high-velocity region gradually expands with the
continuous aeration process, concentrating around the fibers and nearly covering the entire cross-section area of the module (Figure 9). The flow velocity near the liquid level is obviously smaller than in other regions. At this point, the kinetic energy is converted into potential energy, leading to a gradual decrease in the velocity.
After being post-processed, more cross-sections were selected to reflect the water velocity in the reactor. The average flow velocity in different cross-sections of the reactor in continuous time points is presented in Figure 10. The average flow velocity in the middle of the reactor is always maintained at a larger range compared with other regions (Figure 10). As shown in Figure 10, the high flow velocity caused by air occurs in the middle section of the reactor, and a stronger shear stress is formed in the middle region of the fibers, consistent with the visualization results of the flow velocity (Figure 9).

**Velocity vectors and shear stress**

The velocity vector distribution around the module during air scouring at 20 s is given in Figure 11. The velocity vector describes the magnitude and directions of the velocity within the system, and the arrow indicates the direction of the air velocity. The flow of gas-liquid two-phase has a central effect at the bottom of the reactor (Figure 11), and the maximum liquid velocity can be found in the middle of the reactor (Figure 11), resulting in a good scouring effect on the membrane surface. It can be clearly seen that fluid forms an up-flow region in the membrane area and forms a drop-flow region between the membranes and the vessel walls, resulting in the formation of a circulation within the reactor (Figure 11). In general, a greater air rising rate between the fibers always produces greater shear stress on the membrane surface, slowing down the formation of membrane fouling.

The distribution of the average shear stress on the membrane surface along the fiber direction at
different points in time is illustrated in Figure 12. A hollow fiber membrane located in the middle of the module was chosen to simulate the average shear stress. The fiber in the central location is directly over the aerator, and can be used as a representative fiber to evaluate the shear stress on the membrane surface induced by air scouring. During the air scouring process, a strong shear stress is distributed in the lower end and middle sections of the fiber, and it changes alternately (Figure 12). The shear stress in the downward part of the fiber is significantly higher than that at the upper end (Figure 12). The high shear stress results in good cleaning effects in these sections. The same conclusion was also observed in Ding et al.’s work (Ding et al. 2016) about the impact of shear stress on the fouling layer caused by aeration. The rapid decrease of shear stress at the upper end of the fiber due to energy loss indicates an unsatisfactory cleaning effect in this section. This
trend is consistent with the velocity vector of the air distribution (Figure 11).

Given the uneven phenomenon during the backwashing process, the high shear stress generated during aeration can further promote the removal of fouling in the middle section and the lower end of the hollow fibers and effectively prevents the formation of cake on the membrane surface. In particular, the average shear stress during air scouring maintains a high level in the region where the particle volume fraction is higher, such as in the position of 0.25, 0.275 and 0.425 m on the membrane surface, effectively removing the fouling that backwashing cannot achieve. It also suggests that air scouring can be used after the backwashing process to
further clean the hollow fiber and alleviate the uneven distribution of the fouling.

**Comparison of the simulated and experimental results**

CFD simulation was used to investigate the air scouring as an auxiliary method after hydraulic backwashing for the removal of fouling on the membrane surface. According to the simulation results, air scouring can further clean up the fouling at specific locations, where the hydraulic backwashing cannot reach to have an effect. To verify the simulation results, experiments were carried out with the coupling of these two methods. Membrane-specific flux ($J/J_0$) and TMP measurements are presented in Figures 13 and 14, respectively.

Relatively speaking, the membrane specific flux of backwashing coupled with air scouring decreases less than separate backwashing, and the flux recovery of backwashing plus air scouring is superior to that of backwashing (Figure 13). Moreover, the growth of TMP after backwashing coupled with air scouring is also relatively slow (Figure 14). After 90 min filtration, the increase in TMP and the decrease of membrane specific flux are accelerated due to further membrane fouling. The recovery rate of flux after different cleaning methods becomes worse. However, backwashing coupled with air scouring keep good control of TMP growth, and the decline of membrane specific flux is relatively slow. Experimental results showed that the cleaning method of backwashing combined with air scouring removes more fouling and restores the membrane performance to a greater extent. Shear stress induced by air scouring accelerates the removal of the cake layer, which averts the blocking of membrane pores and lowers the flux decline rates during the experiment. As a result, the conclusion of air scouring playing a supporting role in backwashing derived from the simulation has good agreement with the experimental results.

**CONCLUSIONS**

The cleaning process for the submerged hollow fiber membrane was investigated by CFD simulation in the study, and the results were verified by experimental studies.

Based on the simulation results, the flow field changes within the reactor during backwashing were shown in 3D mode, and the peeling process of the fouling particle layer on the membrane surface was visualized. The formation of uneven cleaning proved that the upper region of the membranes has a better cleaning effect compared with the middle and lower regions.

After that, air scouring as an auxiliary method of backwashing can further remove the fouling on the membrane surface. Velocity contours and vectors demonstrated that greater shear stress was formed in the middle and lower section of the fibers to further clean the particles in these regions. Additionally, the average shear stress was also maintained at a higher level for the points with a higher volume fraction of
particles, such as in the position of 0.25, 0.275 and 0.425 m, which can effectively remove the fouling in these regions that backwashing cannot reach. Meanwhile, experimental studies showed that air scouring played a positive role in membrane cleaning, compared by the differences of membrane specific flux and TMP under two different cleaning methods, which proved the validity of the simulation results. Therefore, air scouring can be used after the backwashing process to restrain the uneven phenomenon of membrane cleaning effectively and improve the cleaning efficiency of hollow fiber membranes.
ACKNOWLEDGEMENTS

This study was financially supported by the National Natural Science Foundation of China (no. 51,508,390, no. 51,578,374, no. 51,508,383).

REFERENCES


FLUENT, A. 6.3. Theory Manual 2005 Fluent Inc. Central Source Park, 10 Cavendish Court, Lebanon, NH 03766, USA.


First received 2 December 2016; accepted in revised form 5 April 2017. Available online 28 April 2017.