Desalination 276 (2011) 191–198

Contents lists available at ScienceDirect

Desalination

journal homepage: www.elsevier.com/locate/desal

# CFD simulation of turbulence promoters in a tubular membrane channel

Saber Ahmed [a,*], M. Taif Seraji [b], Jonaid Jahedi [c], M.A. Hashib [d]

[a] Faculty of Sciences, Engineering and Health, CQ University, Queensland 4702, Australia
[b] Dept. of Mathematics, National University, Bangladesh
[c] Dept. of Civil Engineering, KUET, Bangladesh
[d] Dept. of Ecological Engineering, Toyohashi University of Technology, Japan

ARTICLE INFO

*Article history:*
Received 5 January 2011
Received in revised form 15 March 2011
Accepted 16 March 2011
Available online 22 April 2011

*Keywords:*
Computational fluid dynamics
Modeling
Fouling
Membrane flux
Baffle

ABSTRACT

This study aims to investigate a computational fluid dynamics (CFD) simulation of a baffled tubular membrane channel containing a concentric rod for predicting turbulent flow. An array of baffles was inserted either on the rod or on the membrane wall for simulation. The qualitative and quantitative properties of fluid dynamics in a baffled membrane has been successfully predicted which is useful to understand better the flow pattern, behavior and feature. The distribution of local parameters such as stream function, velocity, static pressure, wall shear stress, turbulent kinetic energy, and turbulent dissipation energy upon the membrane surface was obtained using CFD code FLUENT. The simulation results indicate that the presence of baffle can improve the local shear stress on the membrane surface and produce eddy activities which enhance the filtration performance. The observed flux enhancement can be attributed to the intense fluctuations of wall velocity and shear stress which can disrupt the growth of boundary layer on the membrane surface. The experimental evaluation was performed through cross flow microfiltration of titanium dioxide suspension which is consistent with the CFD simulations. The simulation results under various baffle arrangements suggest the significance of baffle specifications for optimization of a baffled tubular membrane.



## 1. Introduction

In recent years, the application of pressure driven membrane processes in wastewater remediation has gained wide interest due to their effectiveness over the conventional water treatment processes. The decline in permeate flux during membrane filtration is coupled with the problems of concentration polarization and fouling. Concentration polarization leads to the formation of a concentrated boundary layer near the membrane surface and may result in the formation of a gel layer over the membrane surface. This phenomenon largely depends on system hydrodynamics, nature and size of solute molecules, membrane pore size (Molecular weight cut off) and characteristics (hydrophobic, hydrophilic, surface charge etc.). The lack of adequate shear stress on the membrane surface expedites the formation of cake-layer and membrane fouling. This phenomenon limits the back diffusive transport of the solute to the main stream. Promoting turbulence by introducing internals such baffles is indicated to enhance the solute mass transfer in the bulk stream. Considerable efforts have been devoted to the development of strategies for controlling the build-up of these additional resistances during operation. Of the approaches used, the changing of hydrodynamics of the system, use of external body force and membrane surface modification is of noteworthy. The use of turbulence promoters [1,2], pressure pulsation [3,4] etc. alters the system hydrodynamics, suitable electric field for charged species [5,6] are commonly employed as body force and plasma treatment [7,8], and surfactant treatment [9,10] are techniques for surface modifications. The efficiency of these methods is determined by the increase in permeate flux, quality of separation and ease of membrane fouling. Turbulence promoters are most widely used to improve the hydrodynamic conditions on the membrane surface and enhance the performance of membrane module. Recent research has demonstrated the applicability of CFD to the modeling of turbulence promoters. Various geometric configurations have been tested as turbulence promoters in numerous membranes applications. Using CFD, Cao et al., [11] tested the effects of various arrangements of cylindrical turbulence promoters on fluid flow hydrodynamics. The detailed flow pattern, velocity distributions and turbulence kinetic energy distributions in a spacer filled channel are shown in their study. The location of the high shear stress region and eddies, is found to be strongly related to the spacers and their position in the channel. Significant flux enhancement was achieved using turbulence promoters in a position perpendicular to the flow direction for ultra filtration of synthetic fruit juice [12]. The coupled fluid flow and mass transfer problems were modeled in their study. Through CFD and experimental approach, Santos et al. [13] investigated flow patterns and mass transfer in membrane channels filled with 12 different flow aligned spacers under various hydrodynamic conditions. CFD results are analyzed in terms of shear stress and mass transfer at the walls. Rahimi et al. [14] employed CFD for predicting the permeate flux through a MF

* Corresponding author. Tel.: +61 749309634.
*E-mail address:* s.ahmed@cqu.edu.au (S. Ahmed).

0011-9164/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.desal.2011.03.045

membrane. The predicted pressure distribution was used instead of average input–output pressures in Darcy equation to determine the local permeate flux. In a separate study, they predicted the deposition pattern on the membrane surface using discrete phase modeling technique [15]. Ahmad et al. [16] predicted concentration polarization, mass transfer coefficient and wall shear stress at different conditions in an empty narrow membrane channel. Their simulation results indicate that the concentration polarization phenomenon can be reduced by increasing the feed *Re* and decreasing the wall shear stress. Based on the unsteady hydrodynamic profile of a reciprocating membrane channel, Ahmad et al. [17] reported that the membrane surface shear stress and TMP changes sinusoidally with the reciprocating motion. The use of inserts, such as metal grills [18], static rods [19], spiral wire [20], disk and doughnut shape inserts [21] and helical baffles [22–27], in a tubular membrane has been studied experimentally in different membrane processes. Of the tested approaches, the use of baffle at wall and central positions of the tubular membrane is shown to be simple and promising to improve permeates flux [22–28]. These findings enunciate the necessity for further investigation on this aspect of a baffled membrane system to generate vortices by local flow instability mechanism, which considerably increases flux. The use of baffles in a membrane can act as a continuous static mixer which offers several advantages such as reduction of overall device dimensions, reduction of energy consumption by replacing moving parts, better process control, shorter residence time and improved selectivity. The degree of mixing largely depends on the relative distribution of mean and turbulent kinetic energy. There have been an enormous amount of investigations [13,17] that have emphasized on the experimental research of mass transfer in a baffle-filled membrane system. However, a few of them are focused on the numerical simulation of baffle-filled tubular membrane system [21,28]. This might be related to the complexity of the problem and convergence difficulty of numerical simulation. While the use of baffles has been shown to promote the filtration flux considerably, the fundamental mechanism of the flux enhancement by the baffle is still not clearly understood. The qualitative and quantitative properties of fluid hydrodynamics in a baffled tubular membrane have been investigated much less extensively. Understanding the detailed flow field in a baffled membrane system is of paramount importance from the design and the operational points of view. This study focuses on the hydrodynamic performance of a baffled tubular membrane, which has recently attracted considerable attention [29,30] because of its ability to generate large-scale stream-wise vortices, enhancing mass and momentum-transfer phenomena and turbulent energy dissipation in the flow over those of the simple un-baffled one. In this study, the fluid flow in a tubular membrane containing a concentric rod with an array of baffles either on the rod or on the wall, was simulated using a commercial CFD code FLUENT, in order to gain an insight on the effect of baffles on the flow pattern and behavior. In addition, the distributions of velocity, wall shear stress and turbulent characteristic, the eddy formation, pressure loss along the channel on the membrane surface were also evaluated respectively.

## 2. Model development

In the present study, it is assumed that the fluid (water) is Newtonian, incompressible, isothermal, with constant physical properties and under turbulent steady state flow. Under these assumptions and following the Reynolds averaged Navier–Stokes (RANS) turbulence modeling approach [31], the CFD model involves solving the continuity Eq. (1), Reynolds averaged Navier–Stokes Eq. (2) which are expressed as

$$\frac{\partial \rho}{\partial t} + \Delta\cdot(\rho U) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \nabla) + \nabla\cdot\left(\rho \overline{uu}\right) = -\nabla\,\overline{P} - \nabla\cdot\tau \tag{2}$$

where the over bar indicates a time-averaged value, $\rho$ is the density (kg m$^{-3}$), $U$ (m/s) is the velocity of fluid, $P$ (Pa) is the pressure, and $\tau$ is the viscous stress tensor. A turbulence model is a computational method to close the system of mean flow equations and solve them, so that a more or less wide variety of flow problems can be computed. There are six known classical turbulence models including mixing length, standard $k$–$\varepsilon$, RNG $k$–$\varepsilon$, realizable $k$–$\varepsilon$, Reynolds stress and algebraic stress models [32,33]. For a channel with obstructions across the flow, the RNG $k$–$\varepsilon$ model is more appropriate compared to other $k$–$\varepsilon$ models due to its ability for precise capturing of streamline curvatures at relatively low Reynolds number [11,14]. In contrast to the Reynolds stress and algebraic stress models, this model requires lower computational time. In the RNG $k$–$\varepsilon$ model, the effect of small-scale turbulence is represented through a random forcing function in the Navier–Stokes equations. The RNG model systematically removes all small scales of motion from the governing equations by considering their effects in terms of larger scale motion and a modified viscosity. This model was derived using a rigorous statistical technique and it is similar in form to the standard $k$–$\varepsilon$ model presented by Launder and Spalding [32]. The model provides the transport equations for turbulent kinetic energy ($k$) and its dissipation rate ($\varepsilon$) as follows

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \rho\epsilon \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}P_k - C^*_{2\epsilon}\rho\frac{\epsilon^2}{k}$$
$$C^*_{2\epsilon} = C_{2\epsilon} + \frac{C_\mu \eta^3(1-\eta/\eta_0)}{1+\beta\eta^3}. \tag{4}$$

In this model, Boussinesque approximation has been used to relate the Reynolds stresses with turbulent viscosity ($\mu_t$) and mean velocity gradient. In Eqs. (3) and (4), $P_k = -\rho\overline{u_i' u_j'}\frac{\partial ui}{\partial xi}$ or $P_k = \mu_t S^2$ is the production term of turbulent kinetic energy due to mean velocity gradient, where S is the modulus of mean rate-of-strain tensor, defined as $S = (2SijSij)^{1/2}$

The turbulent viscosity ($\mu_t$) is the effective viscosity of the fluid due to turbulence and is calculated using the following relationship,

$$\mu_t = \rho C_\mu \frac{\kappa^2}{\varepsilon}, C_\mu = 0.0845$$

The values of model constants are: $C_{1\varepsilon} = 1.42$, $C_{2\varepsilon} = 1.68$, $\eta = Sk/\epsilon$, $\eta_o = 4.38$ and, $\beta = 0.012$.

In this investigation, when the RNG $k$–$\varepsilon$ model was used to perform CFD simulations, the enhanced wall treatment available in the commercial CFD software was enabled. This treatment is a near-wall modeling method that combines a two-layer model applicable in regions with fine near-wall meshes, with enhanced wall functions used in regions with coarse meshes. When using the enhanced wall treatment, it is necessary thus to construct a proper fine mesh where the viscosity-affected near-wall region is desired to be fully resolved [34]. For the annular region, where the mass transfer takes place, a boundary-layer mesh accomplishing a $y^+ < 0.5$ and having at least 10 cells within the viscosity-affected near-wall region was setup. The utilized grids for all baffled membrane arrangements had approximately 3, 13, 537 cells and they were verified to give mesh-independent results. The results showed that a $y^+$ smaller than 0.5 at the wall-adjacent cell and at least 10 cells within the viscosity-affected near-wall region ($Re_y < 200$) were needed to obtain consistent and accurate results. These near-wall mesh requirements were then utilized in all the simulations performed in this investigation.