Desalination 351 (2014) 109–119

Contents lists available at ScienceDirect

Desalination

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

Analysis of wall shear stress on the outside-in type hollow fiber membrane modules by CFD simulation

Recep Kaya [a], Gokhan Deveci [a], Turker Turken [a,b], Reyhan Sengur [a,c], Serkan Guclu [a], Derya Y. Koseoglu-Imer [a,b], Ismail Koyuncu [a,b,]*

[a] National Research Center on Membrane Technologies, Istanbul Technical University, Maslak, 34469 Istanbul, Turkey
[b] Department of Environmental Engineering, Istanbul Technical University, Maslak, 34469 Istanbul, Turkey
[c] Department of Nanoscience and Nanoengineering, Istanbul Technical University, Maslak, 34469 Istanbul, Turkey

HIGHLIGHTS

- CFD simulation of hollow fiber membrane module has been studied.
- Tangential module configuration had higher shear stress distribution.
- Experimental results are in accordance with CFD simulations.

ARTICLE INFO

Article history:
Received 7 February 2014
Received in revised form 22 July 2014
Accepted 24 July 2014
Available online 10 August 2014

Keywords:
Hollow fiber membrane module
Shear stress
CFD
Simulation
Crossflow filtration

ABSTRACT

In this study the effects of shear stress distribution and pressure loss on two different hollow fiber module types through have been investigated. The CFD simulations are based on the numerical solutions of the Reynolds averaged Navier–Stokes equations on three dimensional module geometries. The fluid flow inside modules is modeled using a realizable k-ε turbulence model. Module geometries consist two different types of inlet and outlet. One of the modules has normal and the other has tangential inlet and outlet. These two module types are investigated by CFD simulations and results are verified with experimental studies. Based on the simulation results, it has been observed that tangential inlet and outlet create rotational flow inside the module and this causes higher shear stress when compared to normal module geometry. The velocity profiles inside the modules and average pressure drop between inlet and outlet ports are presented. For tangential module configuration, the distribution of velocity inside the module is more homogeneous than the normal module configuration. Average pressure drop between inlet and outlet ports for both module configurations is nearly the same in steady state simulations. The results of the experimental studies are in accordance with the simulation results.

1. Introduction

Membrane filtration has a wide range of applications for water and wastewater treatment. However the performance of most pressure driven membrane systems suffers from concentration polarization, fouling and scaling, which decreases module productivity by reduced flux, increase of energy consumption, etc. [1,2]. Membrane fouling occurs by particle accumulation on the membrane surface, which forms a cake layer that plugs the pores entirely. These effects are highly dependent on suspension composition, membrane properties and hydrodynamic conditions [3]. To alleviate these adverse effects, shear stress should be investigated as a hydrodynamic condition in the membrane module. Computational Fluid Dynamics (CFD) is a powerful tool for understanding the flow dynamics inside a module body.

CFD is the science of predicting fluid flow, heat transfer, mass transfer, and related phenomena by solving the mathematical equations which govern these processes using a numerical algorithm. It significantly reduces the cost, time and risk associated with running repeated experiments [4]. CFD can characterize flow conditions in various situations and has been widely used for studying flow dynamics. For estimating important parameters such as turbulence conditions and shear stress on membrane surface, CFD can be a useful simulation tool. Shear force is an important parameter for scraping the cake layer particles from the membrane surface [5,6]. It is well known that high shear rates on the membrane influence particle back transport from the membrane surface which in turn reduces the concentration polarization and cake formation in cross-flow filtration [1,7]. Many studies

* Corresponding author at: National Research Center on Membrane Technologies, Istanbul Technical University, Maslak, 34469 Istanbul, Turkey. Tel.: +90 212 2853473; fax: +90 212 285 6667.
E-mail address: koyuncu@itu.edu.tr (I. Koyuncu).

http://dx.doi.org/10.1016/j.desal.2014.07.033
0011-9164/

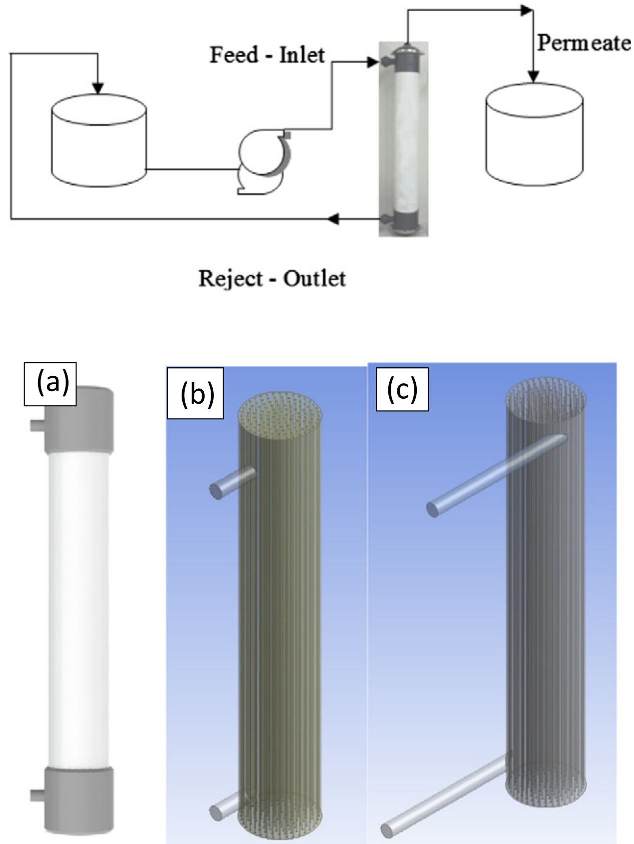

Fig. 1. (a) CAD view of the module and CFD calculation domain extracted from this CAD data, (b) classical inlet outlet configuration, (c) second inlet outlet configuration.

[4,8–13] have been conducted to find the effect of shear stress on membrane fouling using CFD. In these studies, effects of air sparging [8,9], different phase flows [10,11], spacers [14–16], etc. were investigated.

In one of the studies, Ochoa et al. [12] found that non-uniform distribution of shear stress on biofilm has been more effective than uniform shear force distribution on the membrane surfaces. They concluded that scattered and discrete shear force has an important role in removing accumulated particles from the membrane surface. Fulton et al. [9] focused on the shear forces that occurred on the membrane surface from gas sparging. According to their findings, module geometry such as module spacing, played an important role in the distribution of sparge bubbles, bulk liquid flow and surface shear forces. Chan et al. [1] found that the magnitude, duration and frequency of the shear conditions have an impact on the fouling rate. In another study, Wray et al. [8] studied the effect of surface shear stress on submerged hollow fiber membranes using different surface waters. The effect of continuous and intermittent coarse bubble air sparging, large pulse bubble air sparging, and no air sparging has been tested. When shear stress conditions were compared between bubble types, large pulse bubbles reduced fouling up to 80%. Wei et al. [17] investigated the hydrodynamic effect of slug bubble flow in an industrial size flat sheet membrane bioreactor using CFD modeling techniques. Their results showed that shear stress increased with increase in bubble size. It was found that the gap size affected wall shear stress significantly, and the optimal gap size may be 8 mm for an industrial flat sheet MBR unit.

Shear stress distribution depends on different phase flows. S. Laborie and Cabassud [10] investigated the effect of wall shear stress generated from gas–liquid flow inside a capillary tube by using mathematical modeling and electrochemical technique. They found that film thickness was independent from gas velocity; however, it is controlled by the solid/liquid surface tension effect. Buetehorn et al. [18] used irregular fiber arrangement and studied CFD simulations of single and multiphase flows and according to the results cross flow velocity and turbulent viscosity were affected from single phase flow. Berube et al. [13] found that shear force can be lower for tightly configured multi-fiber modules.

Researchers also concentrated on the effects of spacer on shear stress. Fimbres-Weihs et al. [16] used the CFD model to simulate the unsteady flow conditions with mass transfer in narrow zigzag spacers. For the zigzag geometry, unsteady flow has been observed at a hydraulic Reynolds number between 526 and 841. Mass transfer has been enhanced due to the vortices near membrane walls which led to an increased wall shear. In another study, Shakaib et al. [19] investigated the effect of spacer geometry (diamond and parallel) on fluid dynamics of spiral wound membranes. CFD simulations showed that spacer geometry parameters such as filament spacing, thickness and flow attack angle have effects on wall shear rates. Parallel spacers have been found to be more suitable for spiral wound applications. Li et al. [20] investigated the fluid flow through a spacer-filled disk type membrane module and according to their findings, properties of fluid flow depend on membrane module geometry, collection tube size and the spacer thickness. Also, flow rate changes and distribution of permeation rates due to geometry have been investigated for six different collection

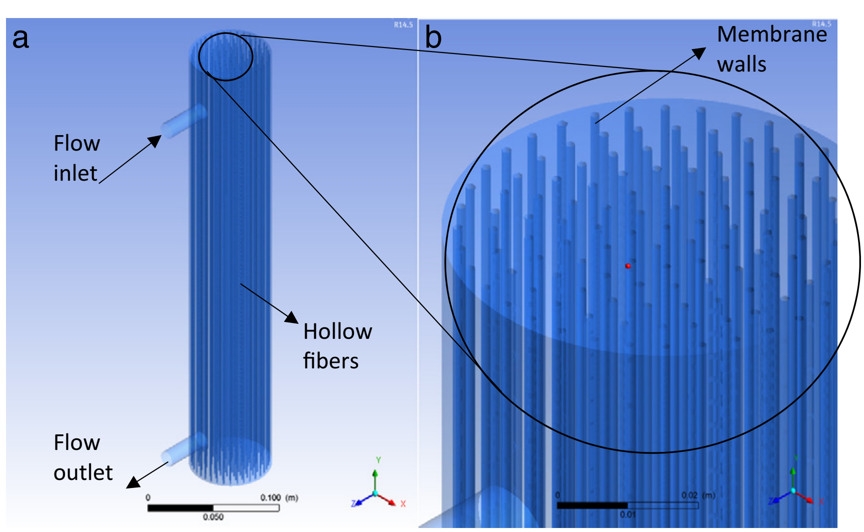

Fig. 2. The geometry and the boundary conditions used in the simulation, a) whole geometry, (b) detailed cross section of module.