Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 53(Special Issue 1) (2021) 97-100
DOI: 10.22060/mej.2020.15917.6238

Investigate of Hydrodynamic and Mass Transfer in the Spacer-Filled Channel of Reverse Osmosis Module

N. Mansouri, M. Moghimi*

Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran

ABSTRACT: The feed channel spacers cause the membrane plates to be separated. These mesh spacers increase the pressure drop in the channel and, in contrast, improve the mass transfer process. In this study, investigate hydrodynamics and mass transfer in the spacer-filled channel in the reverse osmosis module by using the simulation of computational fluid dynamics coupled with the response surface method. Input parameters include the average inlet velocity, the attack angle, the mesh angle, and the output parameters include the pressure drop over the computational domain and the water flux across the membrane walls. The Latin hypercube sampling design method was used to sample the input parameters and the Kriging model has been used for the response surface model. Also, genetic algorithms and screening were used to determine the optimal output parameters. The sensitivity analysis of the input parameters on the output parameters indicates that the average inlet velocity and the attack angle are the most and the least influential parameters, respectively. The optimum configuration geometry taking the values of both output parameters (pressure drop and water flux) into account was stood up at the attack angle of 72.74 degrees, the mesh angle of 85.19 degrees, and the inlet velocity of 0.13 m/s.

Review History:

Received: 2019-03-09
Revised: 2019-10-01
Accepted: 2019-11-05
Available Online: 2020-01-08

Keywords:

Reverse osmosis
Feed spacers
Optimization
Hydrodynamics
Mass transfer

1. INTRODUCTION

In the spiral-wound membrane, the feed channel spacer causes the membrane plates to be separated. The spacers increase the pressure drop in the channel and improve the mass transfer process in the channel. There are many numerical studies that exist to simulate in the spacer-filled channels. Gu et al [1] developed twenty different geometric models by varying the attack angles and the angle between filaments (mesh angle) in four different configurations and evaluated the hydrodynamic flow and water flux from the membrane wall. Li et al [2] studied hydrodynamics and mass transfer in a feed channel with and without the spacer. In this simulation, they considered a permeable membrane wall with five cells to better investigate the relationships between flow velocity and pressure drop, as well as between Reynolds number and Shard number. Their geometric model derived from the Bucs et al.[3], which focused on the precise modeling of spacer geometry. The geometrical model of the separators is also used in this study.

The surrogate-based modeling process can reduce the cost of numerical simulation. The surrogate-based modeling can also be used to understand the effects of parameters and to sensitivity analysis and optimization of the system [4]. Box and Wilson presented a statistical tool called the Response Surface Methodology (RSM), which is very suitable for evaluating several design parameters in the design space (experimental region).

*Corresponding author's email: moghimi@iust.ac.ir

In this study, using Computational Fluid Dynamics (CFD) and the RSM, the effect of the parameters of the angle between filaments (mesh angle), attack angle and average inlet velocity on hydrodynamics and mass transfer is investigated in a design space. The membranes are permeable making the numerical scheme closer to the reality and give the presented model to implement into the problem the effect of input parameters on permeation.

2. METHODOLOGY

2-1 Numerical procedure and boundary conditions

In this simulation, the fluid is Newtonian and the flow is laminar and steady-state. The fluid flow physics is expressed using the continuity and Navier-Stokes governing equations, as well as mass transfer (under diffusion-convection relations) using the following equations:

in which $\nabla = (\partial/\partial x, \partial/\partial y, \partial/\partial z)$ and $u = (u, v, w)$ represent the velocity vector along x, y and z. Also, in Eq. (3), D is the salt diffusivity and C is the salt concentration.

$$\nabla . u = 0 \tag{1}$$

$$\rho(u.\nabla)u = \nabla.\left[-PI + \mu\left(\nabla u + (\nabla u)^T\right)\right] \tag{2}$$

$$\nabla.(D\nabla C) = u.\nabla C \tag{3}$$

For the inlet of the computational domain, the inlet fluid flow is considered as a fully developed flow. The static pressure of zero was used to output the computational domain. The side walls are periodic in terms of both hydrodynamic and mass transfer. The top and bottom membrane walls with non-slip boundary conditions are considered as permeability membrane walls and the amount of water flux is calculated from Eq. (4):

$$J_w = L_P\left(P - P_P - f_{os}c\right) \tag{4}$$

In the Eq. (4), L_P, the hydraulic permeability, P is the hydraulic pressure on the membrane surface in feed channel, P_P is the hydraulic pressure in the permeate channel, f_{os} is the Van't Hoff coefficient used for the relationship between concentration and osmotic pressure.

2-2 Response surface methodology

Due to the high computational time and cost for numerical simulation in studying the effects of different parameters especially in the optimization process, in this study the response surface method is used to reduce the computational cost-effectively.

Latin Hypercube Sampling (LHS) design was used to select the samples. This method is an advanced mode of Monte Carlo sampling and is the most efficient method to select samples. After selecting the samples, a function must be obtained so that the independent variables are the design parameters and the dependent variables are the outputs of the problem. The method used in this study is a Kriging method. The Kriging model consists of two parts, interpolation and regression. The method of Kriging is a combination of a polynomial model with fluctuations around the general trend such as the following:

$$\mathrm{L}(\lambda) = \mathrm{f}(\lambda) + g(\lambda) \tag{5}$$

In the above equation, $\mathrm{L}(\lambda)$ is the unknown function (final response) of the purpose of the design variable λ, $\mathrm{f}(\lambda)$ is a polynomial function of λ, and $g(\lambda)$ is a Gaussian distribution of normalized with zero mean, σ^2 and non-zero covariance.

Evolutionary screening and optimization methods (in particular genetic algorithms) were also used to obtain the optimal response.

3. RESULTS AND DISCUSSION

In the velocity contours, high velocities in the up and down filaments (where the flow passes through a filament and at an appropriate distance from the membrane plate) can be seen, as well as the velocity of flow decreases near the filaments. The results show (Fig. 1) that in areas where the lower flow velocity (the flow goes towards stagnation), salt concentration increases and favorable areas are created to cause fouling; thus it is important to create a physical structure that can prevent the increase in concentration polarization in some regions of the tip and around the membrane plate.

Following the response surface process and the prediction of the output values in the entire design space, with a very low computational cost, the optimum points in the design space are identified using the genetic algorithm.

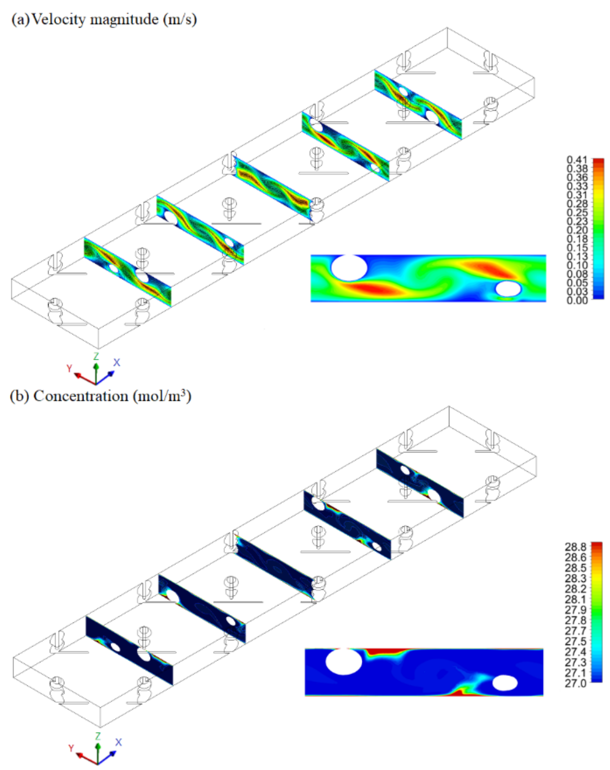

Fig. 1. Contours of (a) velocity and (b) salt concentration in a fluid in five *yz*-sections

Attack angle (deg)	Mesh angle (deg)	Inlet velocity (m/s)	Pressure drop (Pa)	$J_{w,ave}$ (m/s)$\times 10^6$
73.32	53.44	0.041	39.29	-
86.3	114	0.163	-	8.68
72.74	85.19	0.13	236.65	8.645

4. CONCLUSION

The main findings from the results are:

1. The inlet velocity has the greatest impact on the output parameters in the design space.

2. Water flux can be affected more with respect to the input parameter variations. Although the low water flux changes in the design space, water flow can be greatly affected due to the large surface area of the membrane plates.

3. The optimal design point with respect to both output parameters $\theta_\alpha = 72.74$, $\theta_\beta = 85.19$ and $u_0 = 0.13$ m/s.

4. In the regression study and the interaction effect of both output parameters, the coefficient of influence of the inlet velocity is greater.

REFERENCES

[1] B. Gu, C.S. Adjiman, X.Y. Xu, The effect of feed spacer geometry on membrane performance and concentration polarisation based on 3D CFD simulations, J Journal of Membrane Science,

527 (2017) 78-91.
[2] M. Li, T. Bui, S. Chao, Three-dimensional CFD analysis of hydrodynamics and concentration polarization in an industrial RO feed channel, J Desalination, 397 (2016) 194-204.
[3] S.S. Bucs, A.I. Radu, V. Lavric, J.S. Vrouwenvelder, C. Picioreanu, Effect of different commercial feed spacers on biofouling of reverse osmosis membrane systems: a numerical study, J Desalination, 343 (2014) 26-37.
[4] N.V. Queipo, R.T. Haftka, W. Shyy, T. Goel, R. Vaidyanathan, P.K. Tucker, Surrogate-based analysis and optimization, J Progress in aerospace sciences, 41(1) (2005) 1-28.

HOW TO CITE THIS ARTICLE

M. Moghimi, N. Mansouri, Investigate of hydrodynamic and mass transfer in the spacer-filled channel of reverse osmosis module, Amirkabir J. Mech Eng., 53(Special Issue 1) (2021) 99-100.

DOI: 10.22060/mej.2020.15917.6238