Novel Spacer Design Using Topology Optimization in a Reverse Osmosis Channel

Seungjae Oh

Semyung Wang[1]
e-mail: smwang@gist.ac.kr

School of Mechatronics,
Gwangju Institute of Science and
Technology (GIST),
261 Cheomdan-gwagiro, Buk-gu,
Gwangju 500-712, China

Minkyu Park

Joon Ha Kim[1]
e-mail: joonkim@gist.ac.kr

School of Environmental
Science and Engineering,
GIST, 261 Cheomdan-gwagiro, Buk-gu,
Gwangju 500-712, China

The objective of this study is to design spacers using topology optimization in a two-dimensional (2D) crossflow reverse osmosis (RO) membrane channel in order to improve the performance of RO processes. This study is the first attempt to apply topology optimization to designing spacers in a RO membrane channel. The performance was evaluated based on the quantity of permeate flux penetrating both the upper and lower membrane surfaces. Here, Navier–Stokes and convection-diffusion equations were employed to calculate the permeate flux. The nine reference models, consisting of combinations of circle, rectangle, and triangle shapes and zig-zag, cavity, and submerged spacer configurations were then simulated using finite element method so that the performance of the model designed by topology optimization could be compared to the reference models. As a result of topology optimization with the allowable pressure drop changes in the channel, characteristics required of the spacer design were determined. The spacer design based on topology optimization was then simplified to consider manufacturability and performance. When the simplified design was compared to the reference models, the new design displayed a better performance in terms of permeate flux and wall concentration at the membrane surface. [DOI: 10.1115/1.4025680]

Keywords: concentration polarization (CP), spacer, topology optimization, reverse osmosis (RO), pressure drop

1 Introduction

Reverse osmosis (RO) is one of the most attractive and well-established technology to produce fresh and potable water from saline water sources, such as seawater [1]. High rejection rate of the contaminants by RO membranes ensures the reliability of produced water. Meanwhile, however, this high rejection inherently increases the extent of concentration polarization (CP). CP is a phenomenon in which the rejected salts accumulate on the surface of the membrane, thereby augmenting the osmotic pressure across the membrane and decreasing the driving force of water flux [2]. In addition, CP has long been identified as a major intractable problem that locally promotes fouling and scaling at the membrane surface and deteriorates the performance of RO systems [3]. In order to alleviate this problem, spacers are frequently employed in commercially available spiral wound membrane modules; spacers are primarily used to maintain space between membranes and improve the mass transfer, and consequently reduce the development of the concentration polarization layer. Since the specific configurations and shapes of spacers influence the hydrodynamic condition and mass transfer pattern, the accurate prediction of CP and its optimization are crucial factors in its optimization.

Studies pertaining to predictions of the CP layer are divided into two cases: one is to use an analytical model; the other is to use a numerical model. When an analytical model is used, it is inevitable that assumptions are used in the approximated systems, especially pertaining to the empty simple channel [4–6]. However, predicting the CP layer remains challenging for cases in which the channel shape is complex or if spacers are inserted into the channel. Thus, a number of studies have been conducted in attempts to approximate the CP layer using a numerical model [7–13].

In a complex case, fully coupled Navier–Stokes and convection-diffusion equations are required to numerically describe the phenomena. In other words, it is known both that fluid flow influences the concentration polarization on the membrane surface as well as that the concentration accumulated on the membrane surface affects the amount of permeate flux through the membrane. That is, interaction simulations between the Navier–Stokes and convection-diffusion equations are needed. However, it is difficult to precisely predict the CP layer based on the membrane surface due to the steep concentration gradient near the membrane, even though numerical models have been implemented [7]. Hence, Ma et al. [7,10] obtained a reliable solution (no oscillation) for determining the CP on a membrane surface by applying the streamline upwind Petrov/Galerkin (SUPG) method, which they then experimentally validated. If the SUPG method is not applied to a numerical simulation, a very fine boundary mesh is needed at the membrane surface, which eventually causes a significant increase in the computation cost, and therefore, the SUPG method is necessary.

The effect of spacers on the CP layer has also been extensively studied. For example, the impacts of only rectangle spacers with respect to their configurations were also investigated by Ma et al. [7,10]. Ahmad et al. [8] studied variations of pressure drop and CP layer along the membrane channel depending on different types of spacers (i.e., circle, rectangle, and triangle) using a turbulence model. Song and Ma [9] then showed that spacer size and shape influence the hydrodynamic conditions and mass transfer patterns in a membrane channel. Specifically, the CP layer patterns were analyzed based on their hydraulic characteristics (e.g., reattachment point, recirculation) in the membrane channel. Subsequently, Subramani et al. [11] studied the variation of pressure drop, wall shear rate, and CP layer at the membrane surface with respect to different configurations of circle spacers (i.e., cavity, zig-zag, and submerged). Santos et al. [12] simulated the concentration pattern and pressure drop along the channel using 12 different flow-aligned spacers in which an excellent agreement between the computational fluid dynamics (CFD) and experimental results were achieved. Furthermore, it was found that transverse spacers mainly determine the flow structure, and the presence of longitudinal spacers in the channel had a more limited

[1]Corresponding authors.

Contributed by the Fluids Engineering Division of ASME for publication in the Journal of Fluids Engineering. Manuscript received February 19, 2013; final manuscript received September 1, 2013; published online November 22, 2013. Assoc. Editor: Ali Beskok.