Available online at www.sciencedirect.com
SCIENCE DIRECT®
Journal of Membrane Science 271 (2006) 1–15
journal of MEMBRANE SCIENCE
www.elsevier.com/locate/memsci

Fluid dynamics of spacer filled rectangular and curvilinear channels☆

Vivek V. Ranade [a], Ashwani Kumar [b,*]

[a] *Industrial Flow Modeling Group, National Chemical Laboratory, Pune 411008, India*
[b] *Institute for Chemical Process and Environmental Technology, National Research Council of Canada, M-12 Montreal Road, Ottawa, Ont., Canada K1A 0R6*

Received 1 December 2004; received in revised form 5 July 2005; accepted 8 July 2005
Available online 22 August 2005

Abstract

Spacers are designed to generate significant secondary flow structures and create directional changes in the flow through membrane modules. Shape of the spacers used in membrane modules strongly influences the resulting flow and therefore performance of the module. In this work fluid dynamics of rectangular channels similar to membrane modules and containing different spacers was simulated using a three-dimensional computational fluid dynamics (CFD) model. A 'unit cell' approach was evaluated and used for this purpose. In addition to predicting the pressure drop, the simulated results provided significant insight into fluid dynamics of spacer filled channels. The validated CFD model was used to evaluate performance of different spacer shapes and understand the role of spacer shape and resulting fluid dynamics. The models were extended for the first time to simulate flow in spacer filled curvilinear channels, which could be useful in understanding the fluid behavior in spiral modules. The results were compared with those obtained with the flat channel. The approach and results presented in this work will have significant implications for identifying improved spacers with higher propensities to reduce fouling in membrane modules.

Keywords: Membrane modules; Spacers; CFD; Form drag; Viscous drag; Thin channels

1. Introduction

Pressure driven membrane processes are extensively used for pre-treatment, desalination and recycling in water and wastewater treatment industry. These processes usually suffer from concentration polarization and fouling caused by gradual build-up and deposit of dissolved and/or suspended species near and onto the surface of membranes. Some degree of fouling control can be achieved by the appropriate choice of operating parameters. Accumulation of rejected species can be suppressed by creating back mixing from the membrane to the bulk of the liquid. Sablani et al. [1] in a review on concentration polarization have commented on various techniques used to study and ease the problem. There is a need to design new products including spacers that are capable of alleviating the concentration build-up at a relatively lower pressure drop across the module. Spacers are introduced in membrane modules to separate membrane leaves and reduce fouling by modifying the fluid flow behavior. These spacers or turbulence promoters for spiral-wound and flat sheet modules are often net-like materials, which enhance mass transfer as well as provide passage for feed solutions. It is obvious that back mixing is more effective in spacer-filled channels than in empty channels. However, introduction of spacers also increases the pressure drop over the feed channel, i.e. the mechanical energy dissipation in the feed channel is higher in spacer-filled channels. Therefore, it is necessary to understand the role of spacer shape in resulting fluid dynamics of membrane modules. Such an understanding will allow optimization of spacer shapes to obtain an improved performance in terms of mass transfer and mechanical energy dissipation. An analysis by Da Costa et al. [2] of the processing costs in spacer filled channels, including equipment and energy consumption, demonstrated that there are distinct possibilities to optimize net spacers.

☆ NRCC no. 57853.
* Corresponding author. Tel.: +1 613 998 0498; fax: +1 613 991 2384.
E-mail addresses: vvranade@ifmg.ncl.res.in (V.V. Ranade), ashwani.kumar@nrc-cnrc.gc.ca (A. Kumar).

0376-7388/$ – see front matter © 2005 Elsevier B.V. All rights reserved.
doi:10.1016/j.memsci.2005.07.013

In recent years CFD techniques have been used by increasing number of researchers for understanding the fluid-flow behavior in membrane modules. Cao et al. [3] and Geraldes et al. [4] first reported two-dimensional CFD simulations of net and ladder type of spacers in narrow channels, respectively. These authors concluded that location and inter-filament distance of the spacers play an important role for shear stress distribution, mass transfer coefficients and pressure inside the channel. Karode and Kumar [5] reported studies on flow visualization and pressure drop estimation in spacers-filled channels. This work was probably the first attempt at using 3D CFD for modeling a thin channel with commercial spacers. These authors reported that bulk of the fluid does not change direction at each mesh as reported by Da Costa et al. [2,6]. It was also reported that pressure drops with symmetric spacers were higher than the asymmetric spacers, across an identical thin channel. Year 2002 saw a flurry of activity in the use of CFD techniques for studying spacers for membrane modules. Geraldes et al. [7] described a two-dimensional flow in rectangular channels filled with ladder-type spacers. They also reported 2D flow visualization and pressure drop. Schwinge et al. [8,9] reported two-dimensional CFD studies on hydrodynamics, mass transfer enhancement and estimation of foulant deposits in spacer filled channels. The flow characteristics were examined for arrangements of individual filaments in three different configurations namely zigzag, cavity and submerged with variation in mesh length and filament diameters. Li et al. [10,11] reported studies on optimizing the commercial net spacers for membrane modules using three-dimensional CFD simulations including the verification of their models with experimental data. These authors have shown the existence of transversal and longitudinal vortices, vortex shedding and instable flow behavior. Wiley and Fletcher [12,13] have made an attempt to apply CFD simulation for studying fluid dynamics of feed and permeate flows simultaneously. However, most of the reported work focused on 2D simulations. Many a time the grid independence of simulations was not clearly demonstrated. Furthermore, studies of fine flow structures in small independent cells and their impact on neighboring cells has not been reported in details. Recently, Koutsou et al. [14] reported a 2D flow simulation in a rectangular channel in the presence of cylindrical turbulence promoters similar to a case reported by Schwinge et al. [8,9]. They concluded that more realistic 3D simulations are needed for optimization of spacers. In a recent review, Schwinge et al. [15] described various experimental and computational techniques for understanding and improving the performance of spiral wound modules for various applications. They emphasized that CFD techniques should be used cautiously particularly considering complex processes, such as concentration polarization and solute rejection. Review of the previous work has shown that most workers have used CFD for very simple cases where selected filament configurations have been simulated in 2D. There is also a general lack of description of the computational procedures for CFD simulations including the selection of simulation parameters. Furthermore, published literature has mainly focused on rectangular channels with the assumption that this will be a good representation of the fluid flow behavior in spirals. The validity of this assumption needs to be verified. Therefore, there is a need for 3D simulations of commercial as well as new spacers in both rectangular and curvilinear channels.

In this work, we have evaluated and demonstrated the applicability of a ‘unit cell’ approach in understanding fluid dynamics of spacer filled channels. Detailed three-dimensional CFD models were developed to understand influence of spacer shapes on fluid dynamics of channels similar to membrane modules. For the first time, CFD simulations are reported for spacer filled curved channels, which would have direct relevance for spiral membrane modules. This study also reports the influence of radius of curvature, contributions of various drags and effects of spacer shapes on fluid dynamics. The approach and results presented in this work will have significant implications for identifying improved spacers with higher propensities to reduce fouling in membrane modules.

2. Computational model

2.1. Unit cell approach

Spacer-filled channels have stream wise periodic cross-sections. This means that the channel consists of a large number of identical cells. In a recent paper of Yuan et al. [16], it was shown that the flow and heat transfer in channels with stream wise-periodic cross-sections became ‘periodic fully-developed’ after a few cycles or cells. This finding is expected to apply also to the flow in channels with non-woven net spacers, which contains several hundreds of identical cells (see Fig. 1). Obviously, in a full size membrane module flow will be fully developed. Consideration of a unit cell offers possibility of resolving small-scale features of flow in greater detail without simulating the complete membrane module. This is especially important when one is interested in studying influence of spacer shapes on resulting fluid dynamics.

It is however essential to understand possible implications of approximating a spacer filled channel by periodic unit cells. It is well known that symmetry of a flow over a single cylinder breaks when Reynolds number increases beyond a critical Reynolds number of 60 [14]. The unit cell approach is not valid for cases where periodic symmetry of flow is absent despite the symmetric and periodic geometry. Fortunately, when cylinders are packed closely together in a regular fashion (as in spacer filled channels), the onset of symmetry breaking unsteady flow is delayed considerably. Hill et al. [17] have shown this for the case of spherical particles. Secondly in spacer filled channels, the fluid is increasingly confined and hence stabilized by neighboring spacers. For a specific Reynolds number, viscous dissipation will be higher at lower porosity, and therefore more effective in damping velocity fluctuations. Finally in spacer filled channels, cylin-