NRC Publications Archive Archives des publications du CNRC

Wall proximity effects on flow over a simple membrane spacer

Mahdavifar, A.; Pollard, A.; Pharoah, J. G.; Beale, S. B.

This publication could be one of several versions: author's original, accepted manuscript or the publisher's version. /
La version de cette publication peut être l'une des suivantes : la version prépublication de l'auteur, la version
acceptée du manuscrit ou la version de l'éditeur.

For the publisher's version, please access the DOI link below. / Pour consulter la version de l'éditeur, utilisez le lien
DOI ci-dessous.

Publisher's version / Version de l'éditeur:

<https://doi.org/10.1016/j.compfluid.2013.08.017>

Computers & Fluids, 88, pp. 180-188, 2013-12

NRC Publications Record / Notice d'Archives des publications de CNRC:

<https://nrc-publications.canada.ca/eng/view/object/?id=6176792d-055a-43ec-8f36-cf7a0d61823d>

<https://publications-cnrc.canada.ca/fra/voir/objet/?id=6176792d-055a-43ec-8f36-cf7a0d61823d>

Access and use of this website and the material on it are subject to the Terms and Conditions set forth at

<https://nrc-publications.canada.ca/eng/copyright>

READ THESE TERMS AND CONDITIONS CAREFULLY BEFORE USING THIS WEBSITE.

L'accès à ce site Web et l'utilisation de son contenu sont assujettis aux conditions présentées dans le site

<https://publications-cnrc.canada.ca/fra/droits>

LISEZ CES CONDITIONS ATTENTIVEMENT AVANT D'UTILISER CE SITE WEB.

Questions? Contact the NRC Publications Archive team at

PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca. If you wish to email the authors directly, please see the
first page of the publication for their contact information.

Vous avez des questions? Nous pouvons vous aider. Pour communiquer directement avec un auteur, consultez la
première page de la revue dans laquelle son article a été publié afin de trouver ses coordonnées. Si vous n'arrivez
pas à les repérer, communiquez avec nous à PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca.

Elsevier Editorial System(tm) for Computers and Fluids
Manuscript Draft

Manuscript Number:

Title: Wall Proximity Effects on Flow over a Simple Membrane Spacer

Article Type: Original Research Paper

Keywords: Membranes, DNS, Wall Proximity

Corresponding Author: Prof. Andrew Pollard, Ph.D., P.Eng., FASME

Corresponding Author's Institution: Queen's University at Kingston

First Author: Andrew Pollard, Ph.D., P.Eng., FASME

Order of Authors: Andrew Pollard, Ph.D., P.Eng., FASME; Alireza Mahdavifar; Jon Pharoah

Abstract: Spacer-filled channels are employed in membrane modules in many industrial applications where feed-flow spacers (employed to separate membrane sheets and create flow channels) tend to enhance mass transport characteristics (and possibly) mitigate fouling and concentration polarization phenomena. In this work direct numerical simulation was performed for the flow in spacer-filled channels to obtain a better understanding of fluid flow phenomena in these assemblies. Reynolds numbers of 300, 500 and 800 were considered. The effect of spacer location was also studied for three different configurations: spacer at the centre of the channel, off-centre, and attached to the wall. Instantaneous velocity fields and flow structures such as boundary layer separation on the walls and on the cylinder, eddies on the walls, recirculation regions and vortex shedding were investigated. A Fourier analysis was carried out on the time series velocity data. Using this analysis the Strouhal number was calculated and the development of the flow towards a broader turbulent state at higher Reynolds number was captured. Other statistical characteristics such as time-averaged velocities and wall shear rates are obtained and discussed. The average pressure loss was calculated for the channels and found to be highest for spacer at the centre of the channel and lowest for spacer attached to the wall.

Please find attached an original paper “Wall Proximity Effects on Flow over a Simple Membrane Spacer”, submitted for publication in Computers and Fluids. This paper has not been submitted for publication elsewhere.

The paper uses direct numerical simulation to explore the flow physics of a cylindrical spacer element located at various positions in a channel flow. The cylinder is oriented perpendicular to the mean flow. Instantaneous velocity fields and flow structures such as boundary layer separation on the walls and on the cylinder, eddies on the walls, recirculation regions and vortex shedding were investigated.

On behalf of my co-authors, we thank you for considering this paper for publication.

Highlights:

The effect of spacer location was also studied for three different configurations: spacer at the centre of the channel, off-centre, and attached to the wall for three different Re $300 < Re < 800$.

Instantaneous velocity fields and flow structures such as boundary layer separation on the walls and on the cylinder, eddies on the walls, recirculation regions and vortex shedding were investigated.

A Fourier analysis was carried out on the time series velocity data. Using this analysis the Strouhal number was calculated and the development of the flow towards a broader turbulent state at higher Reynolds number was captured.

Other statistical characteristics such as time-averaged velocities and wall shear rates are obtained and discussed.

The average pressure loss was calculated for the channels and found to be highest for spacer at the centre of the channel and lowest for spacer attached to the wall.

Wall Proximity Effects on Flow over a Simple Membrane Spacer

A. Mahdavifar^a, A. Pollard^{a,*}, J.G. Pharoah^a, S.B. Beale^b

^a*Department of Mechanical and Materials Engineering, Queen's University at Kingston,
Kingston, ON, Canada K7L 3N6*

^b*National Research Council, M-2, ON K1A 0R6*

Abstract

Spacer-filled channels are employed in membrane modules in many industrial applications where feed-flow spacers (employed to separate membrane sheets and create flow channels) tend to enhance mass transport characteristics (and possibly) mitigate fouling and concentration polarization phenomena. In this work direct numerical simulation was performed for the flow in spacer-filled channels to obtain a better understanding of fluid flow phenomena in these assemblies. Reynolds numbers of 300, 500 and 800 were considered. The effect of spacer location was also studied for three different configurations: spacer at the centre of the channel, off-centre, and attached to the wall. Instantaneous velocity fields and flow structures such as boundary layer separation on the walls and on the cylinder, eddies on the walls, recirculation regions and vortex shedding were investigated. A Fourier analysis was carried out on the time series velocity data. Using this analysis the Strouhal number was calculated and the development of the flow towards a broader turbulent state at higher Reynolds number was captured. Other statistical characteristics such as time-averaged velocities and wall shear rates are obtained and discussed. The average pressure loss was calculated for the channels and found to be highest for spacer at the centre of the channel and lowest for spacer attached to the wall.

1. Introduction

Membrane technology is employed in separation, filtration and electricity production. The separation of species in a solution is accomplished by applying driving forces such as gradients in pressure, temperature and electrical potential. In pressure driven membrane separation, solute particles pass across a semi-permeable wall and the feed flow is separated into permeate and retentate. Microfiltration (MF); ultrafiltration (UF); reverse osmosis (RO); gas separation; electrodialysis and pervaporation are different types of such processes. Also

*Corresponding author

Email address: pollard@me.queensu.ca (A. Pollard)

blood oxygenators, artificial kidneys, and controlled release of drugs are other examples of membrane processes [1].

Microfiltration, ultrafiltration, reverse osmosis, and conventional filtration differ in the average pore size of the membrane material. Microfiltration and ultrafiltration are similar in that the mode of separation is molecular. In reverse osmosis membranes, the membrane pores are so small that they are within the range of thermal motion of the polymer chains that form the membrane. The mechanism of transport through these membranes is explained using the solution-diffusion model. According to this model, solutes permeate the membrane by dissolving in the membrane material and diffusing along a concentration gradient. Separation occurs because of the difference in solubilities of different solutes in the membrane.

The three commonly used configurations in membrane modules are capillary-fibre, hollow-fine fibre and spiral-wound modules. Plate-and-frame and tubular types are also used, however due to the high manufacturing cost, their application is usually limited to small-scales. Spiral wound modules (SWM) and hollow fibre modules are the most common types in industry. SWMs are often preferred due to a good balance between permeation rate, ease of operation and packing density [2].

In membrane separation processes, a fluid mixture passes across the feed side of the membrane, and part of that turns to the permeate which is enriched in one of the components of the mixture. In this process, unless the solutions are well mixed, concentration gradients form in the fluids on both sides of the membrane because the feed mixture components permeate at different rates. This phenomenon is called concentration polarization. A layer of solution adjacent to the membrane wall becomes enriched in the permeating component on the permeate side and diluted in the permeating component on the feed side. Thus, the flux decreases because the concentration difference across the membrane permeating component is reduced. In addition, membrane fouling is a process by which solute or particles deposit onto a membrane surface or into membrane pores so that the performance is degraded by higher resistance to the solvent.

In membrane modules, spacers are employed to prevent adjacent membrane walls from contacting and sticking together. The spacers may also promote mixing and instabilities by disrupting fluid flow in the module channels. This may decrease concentration polarization and fouling tendencies in SWMs. However, the existence of spacers in the modules also increases the pressure drop along the feed channel and therefore increases the unit operation costs. Some researchers and SWM producers try to optimize the performance of modules with new spacer designs. Such spacers for spiral-wound modules have the form of net-like arrangement of filaments aligned parallel with, transverse to or at an angle to the module axis.

The wide range of membrane module applications in industry has motivated researchers to study flow in spacer-filled channels using numerical analysis and/or experimental methods to study the hydrodynamics and heat and mass transfer processes so as to improve the performance of the modules, see for example [3].

Yuan et al. [4] indicated that for duct flow and heat transfer, entrance effects are shorter than for flows without the disturbances. Therefore the heat and fluid flow become fully developed after a few repetitions of the periodic section, depending on the Reynolds and Peclet numbers. A membrane module channel may consist of hundreds of repeating sections, the analysis of a single section can give a clear picture of the transport phenomena of the module. This is done by simulating a "unit cell" with periodic boundary conditions.

Patankar et al. [5] proposed a method for obtaining a periodic temperature and velocity field, this method performs an iterative solution of the flow field by decomposing the pressure into a linear and a periodic component and reapplying the outlet flow variables and their gradients at the inlet of the unit cell. In this method constant fluid properties must be assumed.

The proper choice of boundary condition for simulation of membrane walls was considered in some studies. Geraldes and Afonso [6] showed that the permeate velocity through the membrane walls is some orders of magnitude smaller than the velocity of the bulk fluid in the channel for nano-filtration and RO processes, therefore for simulations the velocity can be assumed to be zero on the membrane walls where chemical reactions are not involved.

To optimize spacer design, Schwinge et al. [7, 8, 9] performed two-dimensional numerical simulations of flow and mass transfer in narrow spacer-filled channels for SWMs. The simulations were carried out for laminar, steady and unsteady flow in the range $Re = 90 - 2000$ (based on the hydraulic diameter and the bulk velocity). They changed the distance between the spacers and the diameter of the spacers for three different configurations namely cavity, zigzag and submerged. The characteristics of the flow in terms of wall shear stress, mass transfer and pressure drop were investigated for each configuration.

Koutsou et al. [10] performed calculations for laminar unsteady flow around an array of cylindrical spacers in a channel, employing periodic boundary conditions. They performed two dimensional simulations for $174 \leq Re_h \leq 633$ and investigated the effect of Reynolds number on flow transition to chaotic states, pressure drop and wall shear stress.

While two-dimensional analyses have provided insight into the interactions between the fluid flow and mass transfer, SWMs in practical applications are 3D. To illustrate this, Iwatsu et al. [11, 12] simulated and analyzed the flow in a 3D cavity with a moving top wall, this work demonstrated the differences that exist between 2D and 3D flows. It was found that at low Reynolds numbers the flow on the vertical symmetry plane is similar to the flow obtained from 2D simulation. However, for Reynolds numbers above 800 (based on the velocity of the moving wall and the cavity height), the 3D simulation cannot be extrapolated from the characteristics of the 2D flow and the differences are more important.

Kumar and Karode [13] performed the first 3D simulation of steady laminar flow for $(225 \leq Re \leq 2225)$, based on bulk velocity and hydraulic diameter of the channel of fluid flow around commercial spacers in membrane modules. A unit cell was considered of 2.01mm height, 25mm wide and 35mm long to evaluate the effectiveness of spacers. Pressure drop and shear stress on the membrane

surfaces were used as measures of this effectiveness. The spatial resolution of the simulation was low and mass transfer was not considered in their study. It was determined that the bulk of the fluid followed a zig-zag path through spacers that possessed a large inter-spacer distance to spacer diameter ratios.

Ranade and Kumar [14] employed a periodic unit cell to simulate rectangular and curvilinear spacer-filled channels. It was noted that the curvature of the channels, unlike the shape of the spacer, did not affect the flow behaviour significantly. However, Li and Tung [15] found different performance on the inner and outer walls in a curvilinear spacer-filled channel and therefore recommended using spacer diameters along the inner walls that were larger than those on the outer walls so as to mitigate the unequal shear stress distribution on two membrane walls.

Koutsou et al. [16] employed direct numerical simulation with periodic boundary conditions to investigate three dimensional spacers in the steady and the unsteady flow regimes. The flow attack angle, thickness and spacing between spacers were equal for top and bottom spacer for all cases. Flows with Reynolds numbers between 35 and 300 (based on cylinder diameter and bulk velocity) were considered. Unlike two-dimensional flow, in three dimensions the flow does not move in closed streamlines but in spiral paths along the spacers. They found a central free vortex and the interaction of this vortex with the vortices adjacent to the spacers created some closed recirculation regions. Transition to unsteady flow at Reynolds number within the range from 35 to 45 were reported.

In addition to numerical work, researchers have employed experimental methods to investigate the flow in spacer-filled channels. Kang and Chang [17] reported that the critical Reynolds number at which flow becomes unsteady varies with the geometric characteristics of the system. As Re is increased the flow regime changes from steady laminar with recirculation to a vortex shedding regime.

Belfort and Guter [18] investigated the effects of spacers on fluid dynamics and mass transfer. They recommended that an efficient spacer must generate as few regions of stagnant flow as possible and must enhance mixing without significantly increasing the pressure drop.

While in the current work mass transfer is not considered, it is useful to review briefly the relevant literature that included this additional transport phenomena. Kim et al. [19] found that the use of spacers in a channel causes the flow to recirculate in regions close to the concentration boundary layer and therefore can increase the mass flux from the walls. Other researchers [20, 21] concluded that the interaction of the wake behind the spacer with the boundary layer on the membrane surface with the concomitant increase in viscous friction on the walls improves the overall mass transfer in membrane modules. Da Costa et al. [22] found that by employing spacers, mass transfer may be enhanced between 3 to 5 times compared to an empty channel but that pressure losses also increased between 5 to 160 times. It was also found that by varying the characteristic angle of the spacer the mass transfer enhancement is improved significantly. Gimmelshtein and Semiat [23] used two dimensional

particle image velocimetry to investigate the flow inside spacer-filled membrane channels. They reported a laminar flow with only small fluctuations.

In an effort to optimize spacer design, Li et al. [24] varied the spacer diameter and the angle of attack in spacer-filled channels. They investigated flows in the range of $Re = 100-500$ (based on the channel height and on the average inlet velocity). They found that the effect of the entry region in the channels can be neglected because the flow becomes fully developed after passing five spacers.

According to this literature review, flow separation, vortex shedding and unsteadiness are features of flow in membrane filled channels. Many of the previous numerical studies in the field of spacer-filled channels were limited to two-dimensional simulations and/or low Reynolds numbers where the flow regime is laminar [1, 2, 3]. Importantly, most of the previous works were limited to two dimensional simulations and/or low Reynolds numbers and information on turbulent flow structures in spacer-filled channels is not readily available. The goal of the current work is to obtain a better understanding of transport phenomena and flow structures in membrane elements at Reynolds numbers where turbulence effects are more important. This study is carried out by means of direct numerical simulation.

2. Numerical Methods

The flow in spacer-filled channels in general is complex, three dimensional and the flow features depend on the spacer geometry. The fluid dynamics is described by the Navier-Stokes equations. By decomposing the instantaneous velocity into a mean and fluctuating part, the Reynolds averaged Navier-Stokes equations can be obtained for turbulent flow. These equations are not closed and additional assumptions must be employed to close the equations. To obtain the complete details about the flow, the Navier-Stokes equations must be solved numerically without using any closure assumptions [25]; this method is called Direct Numerical Simulation (DNS).

Previous work [26] has reported that in many practical membrane applications neither density nor gravity variation have a significant effect on the fluid flow. Thus, in this work the effect of gravity is neglected and the fluid properties are considered to be constant.

The governing equations of the unsteady, incompressible flow of a Newtonian fluid are the mass continuity and Navier-Stokes equations, which are given by

$$\nabla \cdot \vec{U} = 0 \quad (1)$$

$$\frac{\partial \vec{U}}{\partial t} + \vec{U} \cdot \nabla \vec{U} = -\frac{1}{\rho} \nabla P + \nu \nabla^2 \vec{U} \quad (2)$$

where $\vec{U}$ is the velocity vector, P is pressure, t is time and ν is the kinematic viscosity of the fluid.

To solve the system of equations, the open source code, OpenFOAM, was employed to carry out the numerical calculations. The governing equations are discretized in space using a second-order central differencing scheme and are advanced in time using a second-order Crank-Nicholson scheme [27].

The feed channel cross-section is considered to be planar since the channel height is small compared to its radius of curvature as normally found in SWM. Figure 1 is a schematic of a channel filled with several spacers. In this geometry an array of cylinders is inserted in a plane-channel of thickness H . The cylinders of diameter D , are separated equally by a distance L , and are located in the channel of width W . The coordinate system is aligned with the channel such that the x axis is in the streamwise direction, the y -axis is normal to the wall and the z -axis is in the spanwise direction. The corresponding velocity components are u , v , and w . In both x and z directions the channel is taken as being infinite in extent.

In commercial membranes, spacers are usually adjacent to the walls; however, there are situations where spacers may be located at the channel centre. The channel geometry is composed of repeated elements as shown in Figure 1. In practical applications each module may consist of many hundreds of such elements, and the entrance length is shorter than for an empty channel [4]. Therefore the flow can be assumed to be fully-developed and periodic. This means the computational domain may be limited to a "unit cell" that contains a single spacer as presented in Figure 2. Calculations of pressure drop, wall shear stress, and other important parameters were made using this unit cell. This reduces the size of the computational domain and the computation cost. The dimensions of the unit cell are presented in Figure 2. The length of the domain in the streamwise direction is $6D$, which is the pitch between two successive spacers. This distance was chosen to be in the range of typical industrial spacers [9]. In practice, two layers of spacers keep the channel walls apart; therefore, the channel height was chosen to be twice the diameter of the cylinder. In other words the spacer blockage at the minimum cross-section is 50%. In addition, the channel width in the spanwise direction is $4D$.

To study the effect of cylinder location on the flow, three different geometries as presented in Figure 3, were considered. For all three cases the geometrical parameters are identical apart from the distance of the spacer from the walls. The cylinder diameter, bulk flow velocity and the fluid density were chosen to be unity; thus, a specific Reynolds number for each simulation is achieved by prescribing the value of viscosity.

To study the effect of turbulence in membrane modules, Reynolds numbers of 300, 500 and 800 were chosen. Generally, these Reynolds numbers would be considered to be in the laminar flow regime; they are in fact relatively high in the range of operation of membrane modules for reverse osmosis process [28].

In the streamwise (x) direction the flow is assumed to be periodic with periodicity L . Therefore, for the velocity

$$\vec{U}_{x+L,y,z,t} = \vec{U}_{x,y,z,t} \quad (3)$$

The mean pressure gradient in the flow direction is presumed constant or time invariant and is obtained by $\Delta P/L$ where ΔP is the pressure loss for the length L of the channel. Using this mean pressure gradient, the pressure term is decomposed into two parts [5]

$$P_{(x,y,z,t)} = \left(\frac{\Delta P}{L}\right)x + \hat{P}_{(x,y,z,t)} \quad (4)$$

where $\hat{P}_{(x,y,z,t)}$ is periodic, *i.e.*

$$\hat{P}_{(x+L,y,z,t)} = \hat{P}_{(x,y,z,t)} \quad (5)$$

This decomposition is applied to the Navier-Stokes equation and a cyclic boundary condition was implemented for the periodic part of pressure. Then the constant mean pressure gradient $\Delta P/L$ is treated as a body force *i.e.* a volumetric source term, and the cyclic condition is applied to $\hat{P}$ as per Eqn. (3). At each time step the mean pressure gradient is prescribed a constant value but the value of that is corrected to keep the bulk velocity U at a fixed value.

The bulk velocity is obtained using

$$U_b = \frac{\int_A u \cdot dA}{A} \quad (6)$$

where A is the area of the channel cross-section normal to the flow direction and u is the velocity component in the streamwise x direction.

Simulations start from a uniform velocity and pressure over the domain. The flow is driven by the applied body force the magnitude of which is adjusted at each time step in order to maintain the bulk velocity at a fixed value. During the simulations, solutions are integrated in time until the flow reaches a statistically steady state, *i.e.* the values of velocity, *etc.* become steady. For the problem at hand it took about 10^5 time steps to get to a statistically steady state, and then calculations were performed for another 5×10^5 time steps during which flow statistics were accumulated. To prevent divergence of the solution, it was required to keep the maximum Courant number as low as 0.2 during the simulations. In this problem the Courant number at a point in the flow domain is defined as $C_O = (|\vec{U}| \cdot \Delta t) / \delta V^{1/2}$ where $|\vec{U}|$ is the magnitude of velocity vector at that point, Δt is the time step and δV is the volume of the computational cell at that point. In the spanwise z -direction, a cyclic boundary condition is employed, which means the velocity and pressure fields are identical on the boundary surfaces that are normal to the cylinder axes.

The cylinder surfaces were treated as no-slip walls, where all velocity components and the concentration gradient normal to the surface are set to zero ($u = v = w = 0$).

A normalization procedure was employed to get dimensionless results that can be applied to other cases with similar characteristics. This procedure may be also applied to experimental data. The cylinder diameter, D , which is half of the channel thickness H , is chosen as the characteristic length, and the channel

	Surface normal	Streamwise	Spanwise
Non-cylinder walls	0.69	4.2	3.2
Cylinder surface	0.98	4.6	4.6

Table 1: Grid spacing in wall units for the grids located on the cylinder surface and on the walls (calculated for the highest Re considered in this work)

bulk velocity U_b as the characteristic velocity, then the characteristic time will be D/U_b . The Reynolds number for this problem is defined as $U_b D/\nu$. Note that different definitions of Reynolds number have been used in the literature, for example using the hydraulic diameter of the channel as the characteristic length or by considering the porosity of the channel due to the presence of the spacers [31].

The commercial package "Gambit" was used to generate the computational grids. Fluid dynamics computations, for practical applications such as spacer-filled channels, may require unstructured grids that can faithfully represent complex geometries while minimizing the number of grid points and hence computation time. OpenFOAM treats all grids as unstructured. Therefore all grids used in this study are unstructured. As presented in Figure 4, in order to capture boundary layers and high shear zones, the computational grid was chosen to be finer near the cylinder and the walls. The first row is located on the walls with the distance $y^+ \leq 1$ from the wall. The distribution of grid points in the spanwise direction is uniform. Near the walls hexahedral elements were used, whereas in the rest of the flow field the grid was comprised of pentahedral (wedge-shaped) elements. By numerical experimentation, it was found that a grid containing approximately 2,500,000 compute cells was adequate for the range of Reynolds numbers covered in this study. Table 2 presents the grid spacing in wall units for the grids located on the cylinder surface and on the walls. These data were obtained for the highest Reynolds number considered in this work ($Re = 800$).

Each simulation was performed on 64 processors (Intel Xeon 2.86 GHz). Computations are parallelized using OpenMPI, a high performance message passing interface (MPI) library, moreover CPUs communicate via a InfiniBand interconnection.

3. Presentation and Discussion of Results

Time resolved data from DNS can provide insights into many details and features of the flow. Figure 5 presents the contours of instantaneous dimensionless velocity at $Re=800$ for the three different configurations considered in this work. The velocity vector fields corresponding to these velocity contours are presented in Figure 6. The flow approaching the cylinder has been already "conditioned" by the upstream cylinders (by application of streamwise periodic boundary condition). There is a stagnation point in front of the cylinder, since the cylinder pitch spacing is greater than its diameter, where the velocity is

low and pressure increases due to stagnation. The location of the stagnation point oscillates due to upstream flow oscillations; however, it moves slightly to a higher location for the cylinder case near the bottom wall. The flow adjusts as it passes the gaps between the cylinder and the walls where the velocity is high and passage height is small, as presented in Figure 5. For all cases there is a closed recirculation region downstream of the cylinder. On the cylinder, the flow separates at the top and bottom points. Vortex shedding from these points is the major cause of the large scale flow oscillations observed in the channel at this Re. A closed recirculation region is observed behind the cylinder in Figure 6. After the cylinder the flow passage diverges and this causes separation of the boundary layer from the walls and formation of spanwise vortices on the walls. The vortices move downstream from the cylinder until the flow reattaches to the wall close to the next cylinder. The location of the separation point on the wall is influenced by the vortex shedding from the cylinder. It is hypothesized that this wall-region recirculation affects the mass transfer from the walls; previous work on membranes [10, 28] supports this hypothesis. As the cylinder approaches the bottom wall, the size of recirculation region on the top wall decreases.

The second invariant of the velocity gradient tensor (denoted as Q) is often used for the visualization of turbulent flow. Isosurfaces of Q are used to identify turbulent coherent structures. For incompressible flow Q is defined [32] as

$$Q = -\frac{1}{2}(S_{ij}S_{ij} - \Omega_{ij}\Omega_{ij}) \quad (7)$$

where $S_{ij} = 1/2(\partial u_i/\partial x_j + \partial u_j/\partial x_i)$ and $\Omega_{ij} = 1/2(\partial u_i/\partial x_j - \partial u_j/\partial x_i)$ are the symmetric and anti-symmetric components of the velocity gradient tensor. An isosurface of the Q value for flow with a cylinder in the channel at the centre at Re=800 is presented in Figure 7. In this figure some horseshoe-like structures are observed at the front of the cylinder. Also, a system of elongated vortex structures are observed in the wake region behind the cylinder, and the structures maintain their general shape as they flow downstream until they reach the next cylinder and form the horseshoe structures around that cylinder.

The instantaneous velocity components were averaged over time; after about 10^5 time steps the simulation had reached a statistically steady state, the velocity was then further averaged over another 5×10^5 time steps. Figure 8 presents the contours of average velocity in the channel. For a cylinder at the channel centre there is a low velocity region behind the cylinder associated with a closed recirculation region. There is also a region associated with velocity values at the front of the cylinder that are well below the free stream magnitude due to stagnation. In the gap between the cylinder and wall, the flow velocity becomes about twice the bulk velocity of the flow. The contours of average velocity indicate that as the cylinder approaches the bottom wall, the flow in the channel is divided into a high velocity region above the cylinder and a low velocity region at the bottom half channel.

Turbulence kinetic energy (TKE) is the mean kinetic energy per unit mass

of the broad spectrum of eddy sizes in a turbulent flow. The turbulence kinetic energy is characterized by the root-mean-square (RMS) of velocity fluctuations. Figure 9 presents contours of turbulence kinetic energy fields for simulations at $Re=800$. For a cylinder at the centre of the channel, the contours of TKE are symmetric with respect to the channel median plane, as expected. In this case the maximum fluctuations occur downstream of the cylinder at the centre of the channel. These high fluctuation regions are due to vortex shedding from the top and bottom surface of the cylinder. Also note the fluctuations are a minimum in the gaps between cylinder and walls where the flow is accelerated which tends to damp cross stream turbulent fluctuations. When the cylinder is located in an off-centre location, the region of maximum fluctuations is related to the vortex shedding from the top surface of the cylinder since the vortex shedding from the bottom surface is close to the wall and interacts with the boundary layer. For a spacer attached to the wall the region of maximum fluctuation is located in a shear layer between the faster moving flow at the upper part of the channel and the slower moving flow in the cavity formed between two successive cylinders.

To obtain further information about the dynamics of the flow in spacer-filled channels, a Fourier analysis of the time-dependent velocity was carried out at a point in the middle of the channel between successive spacers. The velocity component in the y direction (wall normal) was used for this analysis [28]. The instantaneous velocity was recorded at that point for each time step. These data were then subjected to a fast Fourier transform (FFT) [33] and the frequency spectra associated with the wall-normal velocity component were obtained by calculating the square of the absolute value of the Fourier coefficients. The time series for this variable is provided in Figure 10 for the spacer located at the centre of channel and $Re = 800$. The flow is chaotic and does not follow a fully time-repetitive pattern. This confirms the results found by Fimbres-Weihs *et al.*[28]. As the Reynolds number was increased, the oscillations display higher frequencies, which is synonymous with a broadening of the scales of turbulent length scales.

Figure 11 presents the frequency spectra of the wall normal velocity component (v) for three Reynolds numbers with the cylinder in the centre of the channel. The straight line has a slope of $-5/3$, which represents the Kolmogorov scaling theory of the inertial range in turbulence [34]. The frequency spectra were obtained using dimensionless time and velocity and therefore energy and frequency are dimensionless in the figures. For all flow Reynolds numbers, there is a pronounced energy peak around dimensionless frequency of 0.5, which corresponds to the observed vortex shedding frequency and the magnitude of which diminishes as Re increases. The changes in slope around the $-5/3$ range with increased Reynolds number suggests that the flow is developing towards a broader turbulent state.

The Strouhal number is a dimensionless number describing oscillating flow characteristics. It is defined here as

$$St = \frac{\omega D}{U_o} \quad (8)$$

where ω is the frequency of vortex shedding, D is the characteristic length cylinder diameter and U_o is the reference bulk mean velocity.

Previous experimental [35] and numerical results [36] suggest that the Strouhal number for cross-flow over single unconfined circular cylinders increases slightly with Reynolds number and stabilizes at about 0.21 for cylinder Reynolds numbers over 500. Strouhal number values of 0.22 to 0.24 for the flows at Reynolds numbers of 300 to 800 are obtained using twice the bulk velocity as the reference velocity (which is, in fact, the average velocity of the fluid in the gap between the filament and the opposite wall) and the cylinder diameter as the length scale [28].

Figure 12 presents the frequency spectra as a function of the spacer position for the case of $Re=800$. The velocity is obtained at a point in the middle of the channel for all three cases. In this figure, the energy content at the higher frequencies is very similar for all configurations; at low frequencies, it is apparent that the energy content increases inversely proportional to distance from the wall. As the spacer approaches the bottom wall, the point where these spectra are obtained reflects a stronger influence of the larger eddies generated from the cylinder top surface. Therefore, the energy spectra obtained displays higher energy content at lower frequencies for spacers either attached or close to the wall.

The wall shear rate, denoted by γ , is defined as the magnitude of the velocity gradient at the wall. The shear rates are normalized using D and U . The value of the local wall shear rate can affect the local mass transfer coefficient in membrane elements [28]. Figure 13 presents the distribution of the wall shear rate along the channel for the case spacer is at the centre. For this case shear rate values are identical on the top and bottom walls due to symmetry and therefore a single curve is presented for each Reynolds number. As the flow approaches the cylinder from upstream it accelerates around the cylinder and the shear rate increases on the walls. The maximum shear rate on the walls occurs near the constrictions above and below the cylinder. As the Reynolds number is increased this maximum is slightly shifted upstream. There are low shear rate regions downstream of the cylinder due to boundary layer separation. As expected the shear rate increases with Reynolds number but the locations of maximum and minimum shear rate do not change significantly.

Figure 14 presents the distribution of shear rate on the walls when the spacer located off-centre in the channel. The shear rate on the top and bottom wall is no longer the same. The local maximum below the cylinder is about twice the local maxima above the cylinder. This is due to the small gap below the cylinder, which makes the boundary layer thinner and therefore a higher velocity gradient occurs on the wall below the cylinder. However for the rest of the channel the shear rate on the top wall is higher. This is because according to Figure 4 the average velocity is higher near the top wall upstream and downstream of the cylinder.

The shear rate for the case of the spacer attached to the bottom wall is presented in Figure 15. A local minimum upstream of the cylinder corresponds to the separation point on the bottom wall in front of the cylinder. There is also

a region of low shear rate right behind the cylinder, which is due to a low speed recirculation region. As for the other geometries there is a maximum shear rate above the cylinder on the top wall because the flow passage is narrow in this region.

A high pressure drop in spacer-filled channels is an additional energy cost required to achieve improved mass transfer and is considered an important factor in the design and optimization of membrane modules. Figure 16 presents the average pressure drop per unit length of the channel $\Delta P/L$, non-dimensionalized by $\rho U^2/D$, as a function of the Reynolds number for three different geometries considered in this work. As the Reynolds number increases, the average non-dimensional pressure drop decreases. In the case where the spacer is located at the channel centre, the pressure drop has the highest magnitude, whereas with the spacer at the bottom wall the pressure drop decreases. This is because when a spacer is attached or close to the bottom wall there is a bigger gap between the spacer and the top wall and the fluid can more easily flow through this region.

4. Conclusion

From this study, it can be concluded that the fluid flow in spacer-filled channels is highly affected by geometry characteristics with the onset of turbulence at lower Re than in plane ducts. By employing periodic boundary conditions in the streamwise direction and cyclic boundary condition in spanwise direction the computational domain is restricted to a unit cell containing one cylinder; this makes for significant savings in the computational cost. The instantaneous velocity fields reveal that in a spacer-filled channel, several vortices form at the top and bottom wall behind the cylinder and are convected downstream toward the next spacers. Due to the acceleration of the fluid above each spacer there is a region where the flow turbulence is damped and the local shear rate is significantly increased in this region. The convection of the shed vortices in the channel causes local changes in wall shear stress and may decrease concentration polarisation and fouling in membrane systems. Comparison of the flow structures and the local shear rate results suggest that flow acceleration around the spacers close to the wall and the vortices near the wall would have an important effect on mass transfer enhancement. Vortices near the membrane walls would tend to enhance mass transfer by mixing the high concentration fluid on the wall with the low concentration bulk fluid. At Reynolds numbers investigated in this work, the main component of pressure loss for the geometries analyzed is form drag. Therefore, greater energy savings can be achieved by reducing this type of drag. However the vortices located downstream of the spacers increase form drag, but they also tend to enhance mass transfer by renewing the boundary layer fluid through flow separation and reattachment. A spacer design for optimal mass transfer should primarily promote the mixing such that the fluid close to the membrane wall is frequently renewed. To reduce the mass transfer boundary layer thickness, wall shear stress perpendicular to the bulk flow should also be enhanced. The understanding obtained in this study is expected

to contribute to the optimization studies of practical membrane modules, where three-dimensional net-type spacers and more realistic geometries are applied.

Acknowledgements

This work is supported by Queen's University, Natural Sciences and Engineering Research Council of Canada and Sun Microsystems. The simulations were carried out on SHARCNET clusters. SHARCNET is part of seven consortia of Canadian academic institutions operated under the aegis of HPC network of Compute Canada.

Figure 1: Schematic of the channel (flow direction is left to right)

Figure 2: Geometry of the unit cell

Figure 3: Three different geometries considered in this work (a) spacer at the centre (b) spacer in an off-centre location (c) spacer attached to the bottom wall

Figure 4: Grid structure close to the cylinder surface and the wall

Figure 5: Contours of dimensionless instantaneous velocity for cylinder displacements from the bottom wall at $Re=800$

Figure 6: Instantaneous velocity vector fields for cylinder displacements from the bottom wall at $Re=800$

(a)

(b)

Figure 7: An isosurface of the second invariant of the velocity gradient tensor (Q -value) at $Re=800$ (spacer at the centre of the channel, flow from left to right), (a) top view (b) 3D isometric view of the channel

Figure 8: The average velocity fields at $Re=800$ for different configurations of the cylinder location

Figure 9: Contours of dimensionless turbulence kinetic energy at $Re=800$ for the three cylinder positions.

Figure 10: Time series of the v velocity component (normal to the walls) for spacers located at the centre and $Re=800$

Figure 11: Frequency spectra of wall normal velocity at three Reynolds numbers (spacer in centre of channel)

Figure 12: Frequency spectra of wall normal velocity for different geometries at $Re=800$

Figure 13: Non-dimensional wall shear rate for spacer at the centre

Figure 14: Non-dimensional shear rate at the top and bottom walls for spacer located off-centre in the channel.

Figure 15: Non-dimensional shear rate on the top and bottom walls (spacer attached to the bottom wall.)

Figure 16: The average pressure drop in the channel versus Reynolds number.

References

- [1] R. W. Baker, Membrane Technology and Applications. New York: J. Wiley, 2004.
- [2] S. P. Nunes and K. V. Peinemann, Membrane Technology in the Chemical Industry. Weinheim, Germany: Wiley-VCH, 2006.
- [3] J. Schwinge, P. R. Neal, D. E. Wiley, D. F. Fletcher and A. G. Fane, "Spiral wound modules and spacers review and analysis," J. Membr. Sci., vol. 242, pp. 129-153, 10, 2004.
- [4] Z. Yuan, W. Tao and Q. Wang, "Numerical prediction for laminar forced convection heat transfer in parallel-plate channels with streamwise-periodic rod disturbances," 1998.
- [5] S. V. Patankar, C. H. Liu and E. M. Sparrow, "Fully-developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area," ASME J. Heat Transfer, vol. 99, pp. 180-187, 1977.
- [6] V. Geraldes and M. D. Afonso, "Generalized Mass-Transfer Correction Factor for Nanofiltration and Reverse Osmosis," AIChE J., vol. 52, pp. 3353-3362, October, 2006.

- [7] J. Schwinge, D. Wiley and D. Fletcher, "Simulation of unsteady flow and vortex shedding for narrow spacer-filled channels," *Ind Eng Chem Res*, vol. 42, pp. 4962-4977, 2003.
- [8] J. Schwinge, D. E. Wiley and D. F. Fletcher, "Simulation of the Flow around Spacer Filaments between Narrow Channel Walls. 1. Hydrodynamics," *Ind Eng Chem Res*, vol. 41, pp. 2977-2987, June 12, 2002.
- [9] J. Schwinge, D. E. Wiley and D. F. Fletcher, "Simulation of the Flow around Spacer Filaments between Channel Walls. 2. Mass-Transfer Enhancement," *Ind Eng Chem Res*, vol. 41, pp. 4879-4888, September 18, 2002.
- [10] C. P. Koutsou, S. G. Yiantsios and A. J. Karabelas, "Numerical simulation of the flow in a plane-channel containing a periodic array of cylindrical turbulence promoters," 2004.
- [11] R. Iwatsu, K. Ishii, T. Kawamura, K. Kuwahara and J. M. Hyun, "Numerical simulation of three-dimensional flow structure in a driven cavity," *Fluid Dyn. Res.*, vol. 5, pp. December, 1989.
- [12] R. Iwatsu, Jae Min Hyun and K. Kuwahara, "Analyses of three-dimensional flow calculations in a driven cavity," *Fluid Dyn. Res.*, vol. 6, pp. 91-102, 07, 1990.
- [13] S. K. Karode and A. Kumar, "Flow visualization through spacer filled channels by computational fluid dynamics I. Pressure drop and shear rate calculations for flat sheet geometry," 2001.
- [14] V. V. Ranade and A. Kumar, "Fluid dynamics of spacer filled rectangular and curvilinear channels," *J. Membr. Sci.*, vol. 271, pp. 1-15, 03, 2006.
- [15] Y. Li and K. Tung, "The effect of curvature of a spacer-filled channel on fluid flow in spiral-wound membrane modules," *J. Membr. Sci.*, vol. 319, pp. 286-297, 07, 2008.
- [16] C. P. Koutsou, S. G. Yiantsios and A. J. Karabelas, "Direct numerical simulation of flow in spacer-filled channels: Effect of spacer geometrical characteristics," *J. Membr. Sci.*, vol. 291, pp. 53-69, 03, 2007.
- [17] I. S. Kang and H. N. Chang, "The effect of turbulence promoters on mass transfer numerical analysis and flow visualization," *Int. J. Heat Mass Transfer*, vol. 25, pp. August, 1982.
- [18] G. Belfort and G. A. Guter, "An experimental study of electrodialysis hydrodynamics," *Desalination*, vol. 10, pp. 221-262, 05, 1972.
- [19] D. H. Kim, I. H. Kim and H. N. Chang, "Experimental Study of Mass Transfer Around a Turbulence Promoter by the Limiting Current Method," *Int. J. Heat Mass Transfer*, vol. 26, pp. 1007-1016, 1983.

- [20] D. G. Thomas, "Forced convection mass transfer: Part II. Effect of wires located near the edge of the laminar boundary layer on the rate of forced convection from a flat plate," *AICHE J.*, vol. 11, pp. September, 1965.
- [21] A. R. Da Costa and A. G. Fane, "Net-type spacers: effect of configuration on fluid flow path and ultrafiltration flux," *Ind Eng Chem Res*, vol. 33, pp. 1845-1851, July, 1994.
- [22] A. R. Da Costa, A. G. Fane, C. J. D. Fell and A. C. M. Franken, "Optimal channel spacer design for ultrafiltration," *J. Membr. Sci.*, vol. 62, pp. 275-291, 1991.
- [23] M. Gimmelshtein and R. Semiat, "Investigation of flow next to membrane walls," *J. Membr. Sci.*, vol. 264, pp. 137-150, 11, 2005.
- [24] F. Li, W. Meindersma, A. B. De Haan and T. Reith, "Experimental validation of CFD mass transfer simulations in flat channels with nonwoven net spacers," *J. Membr. Sci.*, vol. 232, pp. 19-30, 03, 2004.
- [25] P. Moin and K. Mahesh, "Direct numerical simulation: a tool in turbulence research," *Annu. Rev. Fluid. Mech.*, vol. 30, pp. 539-578, 1998.
- [26] D. E. Wiley and D. F. Fletcher, "Techniques for computational fluid dynamics modelling of flow in membrane channels," *J. Membr. Sci.*, vol. 211, pp. 127-137, 01, 2003.
- [27] J. D. Anderson, *Computational Fluid Dynamics: The Basics with Applications*. McGraw-Hill, 1995.
- [28] G. A. Weihs, D. E. Wiley and D. F. Fletcher, "Unsteady Flows with Mass Transfer in Narrow Zigzag Spacer-Filled Channels: A Numerical Study," *Ind Eng Chem Res*, vol. 45, pp. 6594-6603, September 13, 2006.
- [29] S. B. Beale, "Use of Streamwise Periodic Boundary Conditions for Problems in Heat and Mass Transfer," *ASME J. Heat Transfer*, vol. 129, pp. 601-605, 2007.
- [30] N. V. Ndinisa, D. E. Wiley and D. F. Fletcher, "Computational fluid dynamics simulations of Taylor bubbles in tubular membranes, model validation and application to laminar flow system," *Chem. Eng. Res. Des.*, vol. 83, pp. 40-49, 2005.
- [31] G. Shock and A. Miquel, "Mass transfer and pressure loss in spiral wound modules," *Desalination*, pp. 339-352, 1987.
- [32] M. S. Chong, A. E. Perry and B. J. Cantwell, "A general classification of three dimensional flow fields," *Phys. Fluids*, pp. 765-777, 1990.
- [33] W. H. Press, *Numerical Recipes in FORTRAN*. Cambridge [England]: Cambridge University Press, 1992.

- [34] S. B. Pope, Turbulent Flows. Cambridge University Press, 2000.
- [35] C. H. K. Williamson, "Defining a universal and continuous Strouhal-Reynolds number relationship for the laminar vortex shedding of a circular cylinder," Phys. Fluids, vol. 31, pp. 2742, 1988.
- [36] M. Ozgoren, "Flow structure in the downstream of square and circular cylinders," Flow Meas. Instrum., vol. 17, pp. 225-235, 2006.