BRAC University Journal, vol. V, no. 1, 2008, pp. 1-8

COMPUTATIONAL FLUID DYNAMICS STUDY OF FLOW BEHAVIOUR IN A SINGLE SPACER FILLED MEMBRANE MODULE

Sharmina Hussain
Department of Mathematics and Natural Sciences
BRAC University, 66 Mohakhali
Dhaka – 1212, Bangladesh

ABSTRACT

Membrane separation processes have added a new dimension to the traditional processes. The recent novel development in membrane processes have made it possible to use various different purposes at economical rates with additional flexibility and improved efficiencies. Regarding the membrane separation processes, concentration polarization is one of the biggest problem. It causes reduction of permeate flux and deterioration of permeate quality. Accumulation of rejected species can be suppressed by creating back mixing from the membrane to the bulk of the liquid. Various hydrodynamic approaches have been developed for that purpose. Feed spacers can provide higher shear rates at the membrane surface, which, promote the mixing between the bulk of the fluid and the fluid element adjacent to the membrane surface. As a result concentration polarization and membrane fouling can be reduced. The present work is devoted to investigate the hydrodynamics in two dimensional single spacer filled channels. Two different configurations of the cylindrical spacers are investigated with different channel Reynolds number. Different size and shape of formation of recirculation region, upstream and downstream of the spacers are closely observed. This recirculation regions have an important role in enhancing the mass transfer in the reattachment region.

Key words: Concentration polarization, Spacer, Reynolds number, Hydraulic diameter, Shear stress.

I. INTRODUCTION

“Membrane separation processes” is understood to mean such methods in which the separation or chemical reactions are based on different membrane permeability for individual components of a solution. A general principle of membrane technologies is a selective transport of system components through a membrane by the action of a driving force. The transported mass particles can be ions, molecules, colloids, etc. The specific characteristics of membrane separation processes is the use of a selective, semi permeable membrane as a basic separation element. Processed raw material (feed) is supplied to be in a contact with the active layer of the membrane, some components pass through , and other ones are blocked by the membrane. The membrane can be defined as an imperfect mass barrier between systems with which it is in contact, and allows a preferred transport of one component from the inlet flow into the product. The main advantages of membrane separation processes comprise their unique separation capacity, possible separation of substances without phase conversion, usually at ambient temperature. Membrane separation processes also have a relatively low energy consumption, actual energy requirements depend on the character of the separated mixture and the membrane material. Continuous and automated operation are also advantageous factors. Membrane separation technology is gaining increasingly wide spread use because of its adaptability to a large number of application streams. For example, cold sterilization of beverage and pharmaceuticals, cell harvesting, metal recovery as colloidal oxides or hydroxide, desalination of brackish water, removal of micropollutents etc. are some of the industrial applications of membrane separation processes. Membranes are currently a component of multi-well plates used for the synthesis and screening of novel molecules in biotechnology.

The membrane can be cast as flat sheets, tubes and fine hollow fibers. Different types of membrane modules are available for accommodating such shapes and structures. The techno-economic factors for the selection, design and operation of membrane modules include cost of supporting materials and enclosure (pressure vessel), power consumption in pumping, easier way to clean and replacement. The last decade of membrane and module development has reduced significantly the effects of physical compaction and has brought forth spiral membrane modules capable of operating at pressure in excess of 55.2 bar [1]. In membrane processes accumulation of rejected species near the feed side of the membrane (concentration polarization) causes reduction of permeate flux and deterioration of permeate quality. Accumulation of rejected species can be suppressed by creating back mixing from the membrane to the bulk of the liquid. Various hydrodynamic approaches have been developed for that purpose. Since membrane performance depends strongly on fluid transport, much experiments and analytical work has been devoted to the hydrodynamics aspects of membrane processes.

II. Fluid Mechanics and Numerical modeling of Membrane flows:

Berman [2] studied the two dimensional steady state incompressible flow with constant transpiration velocity in a channel with rectangular cross section assuming the equal permeability for both channel walls. Solutions are carried out for small Reynolds number by perturbation method. Poiseuille flow is used as the base flow and the wall Reynolds number, Re_w, is the perturbation parameter. These solutions are valid only for very small transpiration velocity. Friedman and Gillis [3] considered the same type of flow through a straight circular pipe. They solved the complete Navier Stokes equations without any approximation. Numerical results were obtained by Finite difference method. The main constraints of Berman's solutions are removed in this work.

Among numerical models of membrane flows, the most illustrative ones are of Pellerin *et. al*. [4], and Pharoah *et.al*. [5,6]. In [4], the formulation is composed of fundamental transport equations integrated with the k-ε turbulence model. A convection diffusion equation is included to take into account the concentration field of a solute under the prevailing hydrodynamics and Darcy's law is used to prescribe the transpiration velocity. Though the simulations are not applied in a real processes, this model sets out a frame work for the modeling of membrane flows, because of its adaptability to complex geometries, the option to account for turbulent flows, and the use of a pressure related boundary condition for the transmembrane permeate flux. Pharoah's [5,6] model also followed a similar procedure in order to consider the mass transfer but he also considered the incomplete solute rejection and solved the full three dimensional Navier Stokes equations, and introduced two formulations for the membrane: a porous wall model and a computationally efficient source term model.

Combined effects of centrifugal and coriolis acceleration to reduce concentration polarization and membrane fouling is studied by J. Pharoah *et. al.*[6]. A 3-D numerical model of flow channel with permeable membrane surface is developed in this work. This numerical model is used to investigate the effect of system rotation on reverse osmosis membrane separation. Experimental investigation is done to find the role of Coriolis acceleration in inducing secondary flows within the cross flow feed stream in [7].

A brief review of the role of fluid mechanics in membrane filtration till 1989 is presented in [8]. The idea of introducing pulsation or centrifugal instability for example, the effect of unsteadiness due to oscillating pressure gradient, oscillating mass transfer, 'furrowed' flow channel and centrifugal acceleration, to disturb the mass boundary is investigated by some researchers, in alleviating concentration polarization and fouling and all of these results are summarized in this paper.

The subject of the investigation of Andeen [9] is to use the centrifugal acceleration for reducing concentration polarization. Experimental data are presented by using commercial hollow- thin fiber membranes in this paper. The model is simplified by ignoring the effects of desalted water concentration, back pressure, and non uniformity of the membrane surface. It is also assumed that only the brine concentration changes along the brine path way and that the concentration increases uniformly at the membrane.

III. Problem Description:

A. Spacer Geometry:

In Figure 1, a net type spacer is schematically represented with relevant geometric parameters. The mesh configuration of net-type spacers is usually a rhombus, parallelogram or square with a two level structure where the cross –filaments are bonded on top of the others. Some spacers are also asymmetric consisting of thicker filaments overlaying thinner ones.

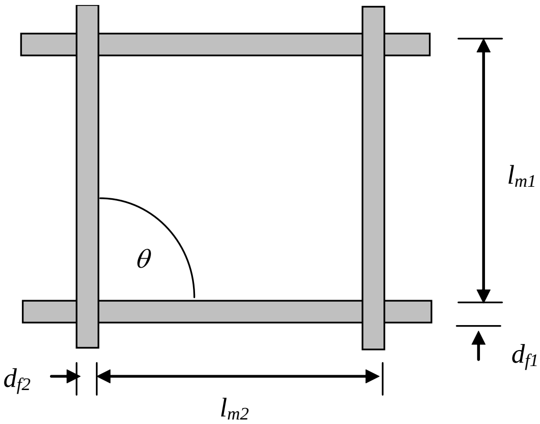

Figure 1 Definition of spacer dimensions. [19]

Tone of the most important geometric characteristics of spacers is the void volume fraction , defined as eqn. (1).

$$\varepsilon = \frac{V_{tot} - V_{sp}}{V_{tot}} = \text{Void volume/total volume} \quad (1)$$

Da. Costa. *et. al*. [19] incorporated the mesh size, diameter of filaments and angle between filaments, and obtained the following expression for ε.

$$\varepsilon = 1 - \frac{\pi\left(d_{f1}^2.l_{m2} + d_{f2}^2 l_{m1}\right)}{4.l_{m2}.l_{m1}.h_{sp}.\sin(\theta)} \quad (2)$$

In eqn . (2), θ, is the angle between two filaments of the spacer facing the channel axis, h_{sp} is the height of the channel and the other parameters are defined in Fig 1. The hydraulic diameter d_h , for a spacer filled channel is given by, d_h=4(cross section)/wetted perimeter, which obtained,

$$d_h = \frac{4\varepsilon}{\left(\frac{2}{h_{sp}}\right) + (1-\varepsilon)S_{vsp}} \quad (3)$$

The specific surface area of the spacers, S_{vsp}, is the ratio of the surface area of an element of the spacer, S_{sp}, divided by the volume of that element V_{sp} :

$$S_{vsp} = \frac{S_{sp}}{V_{sp}} \quad (4)$$

Where S_{sp} is given as:

$$S_{sp} = 0.5\left(2\pi d_{f1} l_{m2} + 2\pi d_{f2} l_{m1}\right) \quad (5)$$

And the volume of that element, V_{sp}, is:

$$V_{sp} = \frac{\pi}{4}\left(d_{f1}^2 l_{m2} + d_{f2}^2 l_{m1}\right) \quad (6)$$

For a rhomboid type mesh with single sized filaments:

$$S_{sp} = \frac{1}{2} \times 4\left(\pi d_f l_m\right) = 2\pi d_f l_m \quad \text{and} \quad S_{vsp} = \frac{4}{d_f} \quad (7)$$

B. Channel Geometry:

Spacers come in various forms that are comprised of a net-like arrangement of filaments aligned parallel, transverse or at an angle to the module axis. Different diameters, various mesh lengths are observed in commercially available spacers. In the present work, flows in narrow channels with single cylindrical obstructions is investigated to improve fundamental understanding of hydrodynamics (Fig.2). Basic understanding and observations can be extended to study the flow filled of multiply filled spacers channel. In order to elucidate the likely extent of flow disturbance, the CFD simulations focus on the effect of cylindrical obstructions positioned normal to the flow. This corresponds to transverse filaments, and allows two-dimensional simulations. Two arrangements are simulated:(i) Simulation of spacer touching the same channel wall, (iii) Simulations of spacer cylinders immersed in the channel. The computational domain is set up with a channel entrance length of 10 times the filament diameter, while the exit length is at least twice the entrance length to avoid any effects of the channel exit on eddy formation behind the filaments. The inlet velocity is specified uniform and as normal to the channel entrance and the flow is fully developed before reaching the first upstream filament. All simulated channel configurations had an identical channel height. Parametric studies of the channel Reynolds number Re_{ch} are performed. The channel Reynolds number Re_{ch} is defined as: $\mathrm{Re}_{ch} = d_h u_{ave} / \nu$ where d_h is the hydraulic diameter for spacer filled flow channel.

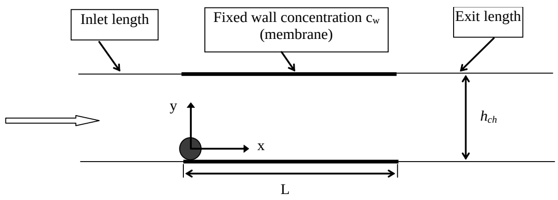

Figure 2. Channel geometry and flow configuration

IV. NUMERICAL METHOD AND MODELING

Simulations were performed over a range of channel Reynolds numbers ($\mathrm{Re}_{ch} = d_h u_{ave} / \nu$) 100-400. The fluid is taken as water at a temperature of 293 K and it is assumed to be incompressible and isothermal and to have constant fluid properties. The general Navier-Stokes equations governing the flow field are solved by using a commercial CFD package FLUENT 6.0. The code utilizes a collocated finite volume formulations, and second order upwind differencing scheme is used in all simulations. The equations are solved by using the SIMPLE algorithm with an iterative line-by-line matrix solver. For low Reynolds numbers the flow remains stationary and no vortex shedding is produced. Convergence is achieved in less than 2000 iterations. An unstructured grid with sufficiently fine resolution is chosen to ensure that the results are independent of grid size. The number of cells generated depended on the number of filaments used for the simulation. Typical grids generated for single filament placed adjacent to the bottom wall and at the center of the channel are shown in Fig.3.

V. Flow patterns in spacer filled narrow two-dimensional channel:

A. Flow profile around a single filament touching to the bottom wall:

For steady case, the channel Reynolds number, Re_{ch} is taken from 150-400 and the dimensionless filament diameter, d_f/h_{ch} is taken as 0.5 for all simulations. The velocity contours and the stream function of the part of the channel for different Reynolds number for a single filament placed at the bottom wall of the channel is shown in Figure 4 (a,b,c). For each case, recirculation regions are observed upstream and downstream of the filament. A very small recirculation region is formed in front of the filament and behind the filament a larger recirculation region is formed. The size of the recirculation region increases with the Reynolds number. For Reynolds number 400 a large recirculation region can be observed and develops more rapidly than for Reynolds number 150 and 200. The highest velocity region is between the filament and the wall because of the flow constriction. In this region the velocity is almost 2.3 times higher than the inlet velocity and the tail of the highest velocity region becomes longer for higher Reynolds number. Because of the higher velocity between the top of the filament and the opposite wall, a higher shear stress region develops on the opposite wall.

B. Flow profile around a single filament submerged in the center of the channel:

Figure.5 (a,b,c) shows the stream function and the contours of the velocity for the part of the channel with a single filament placed at the center of the channel. The dimensionless filament diameter and the Reynolds number are kept the same as those used for simulating the flow field for a filament placed at the bottom wall. Contours of the velocity magnitude show that, recirculation region is formed behind the filament. In this case also, the length of the recirculation region is increased as Reynolds number increases. The length of the recirculation region is smaller than the recirculation region for the filament placed at the bottom wall. This illustrates the effect of the interaction between the wall and the filament adjacent to it on the formation of the recirculation region. For this configuration, velocity is also increased between the filament and the wall and the highest velocity is almost 2.7 times higher than the inlet velocity. The maximum wall shear stress on both top and bottom wall is much higher compared to the case of the filament on the bottom wall. The values of the maximum calculated shear stress coefficient, $C_f=\tau_w/(1/2\rho U_{inlet})^2$ on top and bottom wall for single filament adjacent and submerged filament in the center of the channel and for simple poiseuille flow are given in Table 1. From the table it can be observed that, in the obstructed channel the shear stress co-efficient is much higher than the simple fully developed laminar flow (poiseuille flow).

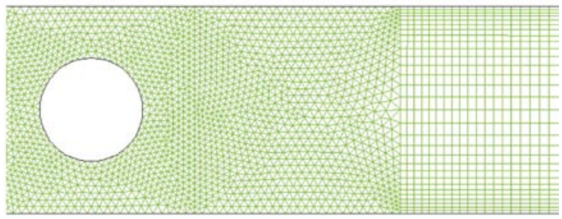

Figure 3 Typical grid distribution around a transverse filament placed adjacent to the wall and at the center of the channel respectively.

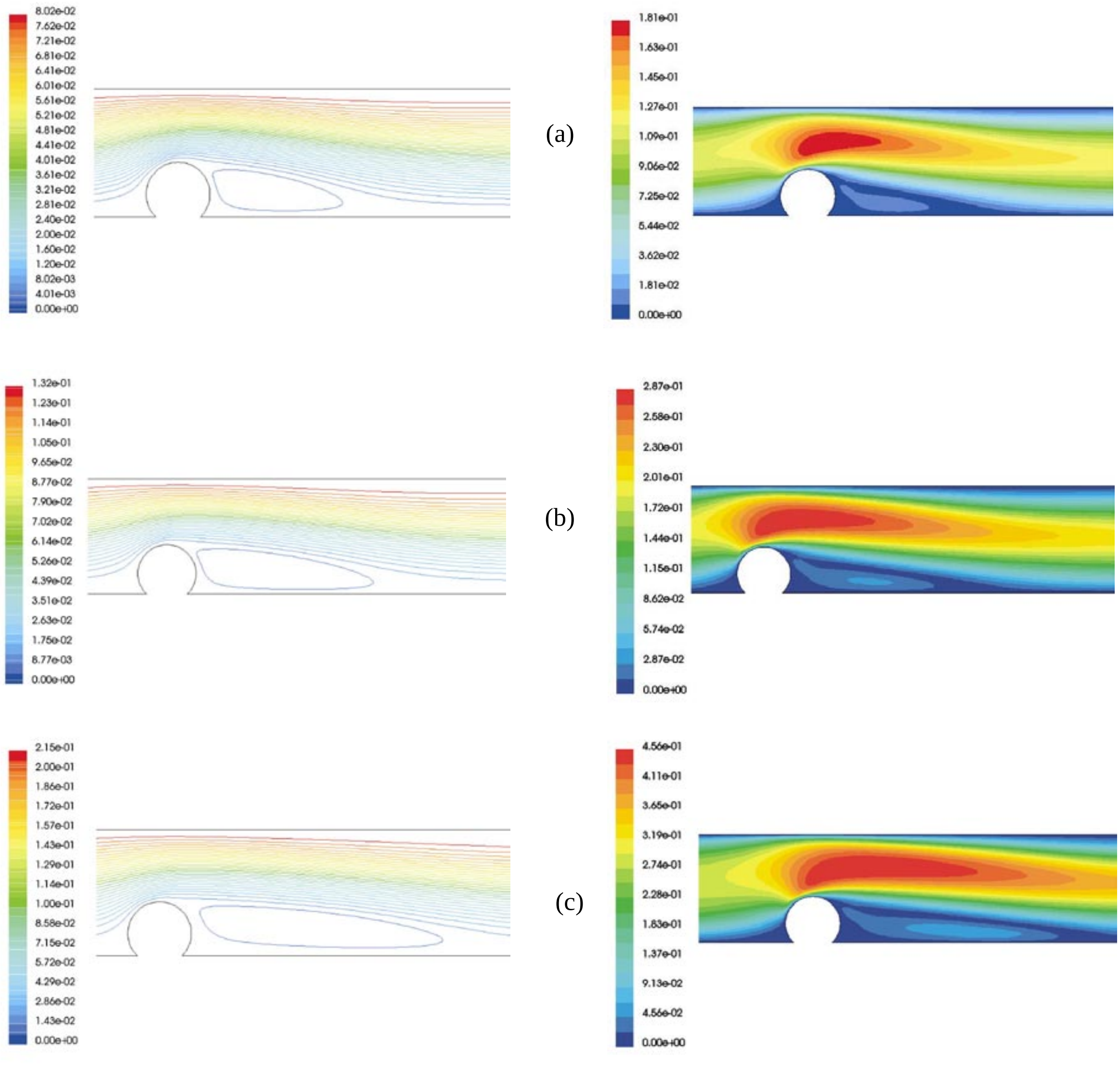

Figure 4 (a,b,c) : Flow around a single filament adjacent to the bottom wall with Reynolds number, Re_{ch}= 150,200 and 400 respectively .Left: streamlines pattern; right: streamwise velocity.

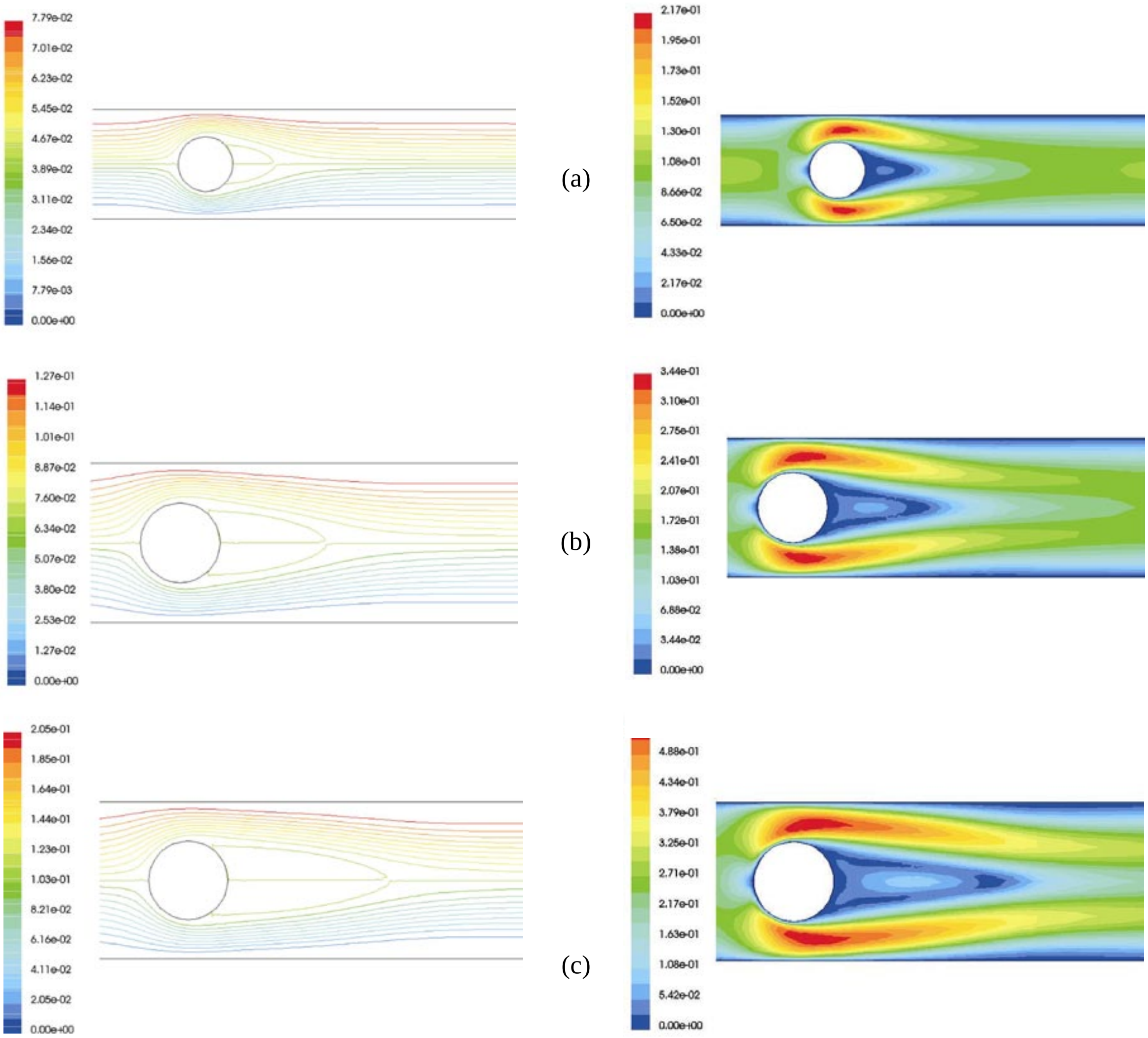

Figure 5 (a,b,c): Flow around a single filament submerged in the center of the channel for Re_{ch} = 150,200 and 400 respectively. Left: streamlines pattern; right: streamwise

Table 1 Values for the C_f ($=\tau_w/(1/2\rho Uinlet^2)$) on both top and bottom wall for single filament placed at the bottom wall and submerged in the center of the channel.

Re_{ch}	Position of the filament	Max. absolute value of the wall shear stress coefficient on the bottom wall	Max. absolute value of the wall shear stress coefficient on the top wall	Max. absolute value of the wall shear stress coefficient for Poiseuille flow on top or bottom wall
150	Touching the bottom wall	.24699	.592782	0.082214
150	submerged	1.547822	1.547822	0.082214
200	Touching the bottom wall	.151675	.66358	0.063479
200	submerged	1.4219602	1.4219602	0.063479
400	Touching the bottom wall	.0710289	.343306	0.03533
400	submerged	.63926012	.63926012	0.03533

VI. CONCLUSION

This study presents CFD simulations of Hydrodynamics in spacer obstructed membrane module. Laminar flow is considered Calculations are carried out for two dimensional channel geometry. Clear understanding of the fluid flow in two dimensional channel can help to select appropriate conditions and parameters for three dimensional simulations. Reduction of fouling and concentration polarization by modification of hydrodynamic condition is very important for enhancing the membrane module performances. Investigation of the flow pattern in a spacer filled channel have revealed the importance of hydrodynamics on mass transfer and pressure loss enhancement along the flow channel. By using spacers, the performance of a membrane module can be improved because spacers enhance the wall shear stress and promote the formation of eddy. As a result the concentration boundary layer thickness at the wall is reduced. The size and shape of the formation of the recirculation region, upstream and downstream of the spacers depends on the channel Reynolds number, the configuration of the spacers and some other parameters. This recirculation regions have an important role in enhancing mass transfer in the reattachment region. An effective spacer should provide higher mass transfer rate from the membrane wall to the bulk flow in order to reduce the concentration polarization with maintaining lower pressure loss along the channel. Among the two types of spacers, submerged spacers show the higher wall shear stress and mass transfer enhancement. The presence of spacers in the channel results in local concentration maximum which exceeds the corresponding concentration in the case of empty channel. From the analysis of hydrodynamics it can be observed a complex correlation among the flow pattern, wall shear stress, pressure loss along the channel being dependent on the channel Reynolds number. These careful observations of flow field, obstructed by single spacer can be broaden to study the flow filled where multiple spacers are placed in different configurations.

VII. REFERENCES

[1] G. Srikanth, Membrane separation Processes-Technology and Business opportunities, http://www.tifac.org.in/news/memb.htm, 31st January, 2004.

[2] A.S. Berman. Laminar flow in channels with porous wall. *J. appl. Phys*., **14**(9), 1953.

[3] M. Friedman and J. Gillis. Viscous flow in a pipe with absorbing walls.. *Journal of Applied Mechanics*, **34**, 819-822, 1967.

[4] E. Pellerin, E. Michelitsch, K. Daracovich, S.Lin and C.M. Tam. Turbulent Transport in membrane modules by CFD simulation in two dimensions. *J. Of Membrane science*., **100**,139-153,1995.

[5] J.G. Pharoah. Computational modeling of Centrifugal membrane and density separation. Master’s Thesis, University of Victoria, 1997.

[6] J.G. Pharoah, N.Djilali, and G.W. Vickers. Fluid mechanics and mass transport in centrifugal membrane separation.. *Journal of membrane science*, 176:277-289, 2000

[7] A. Bergan, T.M. Fyles, D.S. Lycon, and G.W. Vickers. Flux enhancement in reverse osmosis using centrifugal membrane separation. *Journal of Membrane science*, 176:257-266, 2000.

[8] G. Belfort. Fluid mechanics in membrane filtration: Recent developments. *Journal of Membrane science*, **40**,123-147, 1989.

[9] G.B. Andeen, Effects of acceleration on Reverse osmosis desalination. *Desalination*, **36**,265-275, 1981.

[10] J.G. Pharoah. Fluid dynamics and Mass Transport in Rotating Channels with application of Centrifugal Membrane Separation. Ph.D Thesis, University of Victoria, 2002.

[11] M.Mulder. *Basic Principles of Membrane Technology*. Kluwer. Academic Publishers, 1996.

[12] S.K. Karode, A.Kumar, Flow visualization through spacer filled channels by computational fluid dynamics I. Pressure drop and shear rate calculations for flat sheet geometry, *Journal of Membrane science*, **193,** 69-84, (2001).

[13] Dianne.E. Wiley, D.F.Fletcher, Techniques for computational fluid dynamics modeling of flow in membrane channels, *Journal of Membrane science*, **211,** 127-137, (2003).

[14] Dianne.E. Wiley, D.F.Fletcher, Computational fluid dynamics modeling of flow and permeation for pressure-driven membrane processes, *Desaination,* **145**, 183-186, (2002).

[15] J. Schwinge, D.E.Wiley, D.F.Fletcher, Simulation of the flow around spacer filaments between narrow channel Walls.1. Hydrodynamics*. Ind. Eng. Chem. Res*. **41**, 2977-2987, 2002,

[16] J. Schwinge, D.E.Wiley, D.F.Fletcher, Simulation of the flow around spacer filaments between channel Walls.2. Mass transfer. *Ind. Eng. Chem. Res*. **41**, 4879-4888, 2002.

[17] J. Schwinge, D.E.Wiley, A.G. Fane, Novel spacer design improves spiral wound module performance, *Proc. 6th World Congress of Chemical Engineering*, Melbourne, Australia, 2001.

[18] J. Schwinge, D.E.Wiley and D.F. Fletcher, A CFD study of unsteady flow in narrow spacer –filled channels for spiral-wound membrane modules, *Desalination*, 195-201, (2002).

[19] A.R. Da.Costa, A.G.fane, D.E.Wiley, Spacer characterization and pressure drop modeling in spacer-filled channels for ultrafltration. *Journal of membrane science*, **87**, 79-98, 1994.

[20] Schlichting, H., Boundary layer theory, McGraw-Hill Book Co. Inc., New York, (1960).