BRAC University Journal, vol. VI, no. 2, 2009, pp. 1-10

COMPUTATIONAL FLUID DYNAMICS STUDY OF FLOW BEHAVIOUR IN MEMBRANE MODULE WITH MULTIPLE FILAMENTS

Sharmina Hussain
Department of Mathematics and Natural Sciences
BRAC University, 66 Mohakhali
Dhaka-1212, Bangladesh
email: sharmina@bracu.ac.bd

ABSTRACT

Membrane Separation Technology has added a new dimension in modern separation technology. 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. 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 spacer filled channels. Three different configurations of the cylindrical spacers are investigated with different channel Reynolds number and different meshlength. 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: Membrane fouling , Spacer, Reynolds number, Mesh length, Shear stress.

I. INTRODUCTION

Continuous and automated operation are advantageous factors of Membrane separation technology. Membrane separation technology is gaining increasingly wide spread use because of its adaptability to a large number of application streams. Key markets for membranes include water and wastewater treatment, food and beverage processing, pharmaceuticals and medical products, chemical production, industrial gas production and others. The pharmaceutical and medical, environmental applications and fuel cells are the emerging market of membrane separation technology. Fuel cells, although currently negligible in market value terms, are expected to emerge as a significant outlet for membranes in the next 10 to 20 years. The water and wastewater treatment market continued to be the largest for membranes.

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. D.E. Wiley and her coworkers have investigated different aspects of spacer filled channel flow and mass transfer. A CFD model of laminar incompressible steady state fluid flow and concentration polarization in pressure driven membrane separation is developed in [2] where variable solution properties (i.e. viscosity and diffusion) vary as a function of concentration but the density is kept constant. This model is then extended to perform simulation for flow into both feed and permeate channel simultaneously in [3]. The effects of change in rejection, wall permeation rates and solution properties on velocity and concentration profile are presented for empty channels and channels with eddy promoters. Fluid dynamics and mass transfer in spacer filled two dimensional channel are investigated in [4] and [5]. The effect of mesh length, spacer orientation and filament diameter to the velocity field, mass flux and pressure loss is observed in these two papers. Schwinge *et al.* [6] also have performed an experimental investigation of similar flows for ultrafiltration of dextran solution. An unsteady CFD study has also presented for some of these flows in [7]. The pattern and length of the recirculation region for different arrangement of filaments, for different filament diameter and mesh length is depicted in this work.

This paper describes the CFD model and provides a discussion of hydrodynamics of a spacer filled two dimensional channel. All the model parameters are considered as [8] and results are reproduced by numerical simulations with CFD package FLUENT 6.0.The effects of multiple filaments on the flow pattern is presented. Effects of the types of spacers, channel Reynolds number and mesh length on the flow field, and pressure loss along the channel is discussed based on the numerical simulations of the flow field.

II. PROBLEM DESCRIPTION:

Details of this model have been discussed in [2]. Spacer geometry, generated grid for numerical simulation and flow configurations are also depicted in [2].

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 multiple cylindrical obstructions is investigated to improve fundamental understanding of hydrodynamics (Fig.1). 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. Three arrangements are simulated (): (i) Simulation of spacer touching the same channel wall, (ii) Simulations of spacer cylinders placed in zig-zag manner (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: $\mathrm{Re}_{ch} = d_h u_{ave} / v$ where d_h is the hydraulic diameter for spacer filled flow channel.

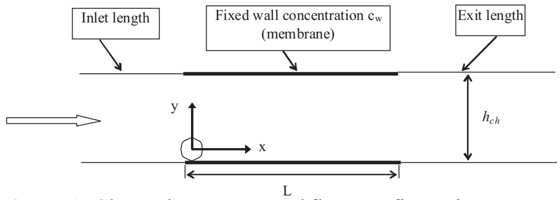


Figure 1: Channel geometry and flow configuration

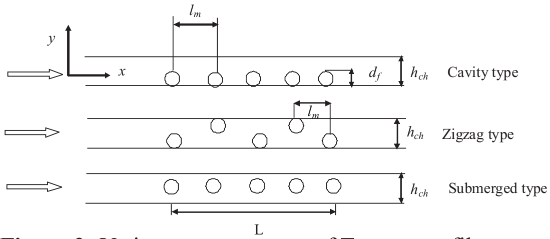


Figure 2: Various arrangements of Transverse filaments

Numerical method and modeling
Simulations were performed over a range of channel Reynold snumbers ($\mathrm{Re}_{ch} = d_h u_{ave} / v$) 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.

III. FLOW PATTERNS IN SPACER FILLED NARROW TWO-DIMENSIONAL CHANNEL:

Effects of the dimensionless channel Reynolds number, Re_{ch} / h_{ch}, dimensionless mesh length, l_m / h_{ch}, on the flow pattern in a narrow channel are examined for different placement of the filaments. For all the cases, the dimensionless filament diameter, d_f / h_{ch} is chosen 0.5.

A. Flow Profile around Multiple filaments placed adjacent to the bottom wall:
To study the effects of the multi filaments cavity spacers on the flow patterns between sequential filaments, simulations are carried out with different channel Reynolds number and different mesh length. Effects of the Reynolds number on the flow pattern

between two sequential filaments is illustrated in Figure: 3 (a,b,c). Contours of the velocity magnitude shows that, as the Reynolds number increases, the recirculation regions, in front and behind the filaments become larger. For Reynolds number 100, a small recirculation region is formed behind the filament. For Reynolds number 100 separate recirculation region is observed between the two sequential filaments. But for higher Reynolds number the recirculation region is merged between two sequential filaments and one large "fully formed" recirculation region is formed. Highest velocity region is observed between the opposite wall and the top of the filament, similar to the single filament adjacent to the bottom wall. As the Reynolds number increases, the tail of the high velocity region above the filaments extends further downstream and it effects the high velocity region for the next downstream filaments. The highest velocity is almost 2.3 times higher than the inlet velocity for Reynolds number 100 and 300 and 2.1 times higher for Reynolds number 200.

The effect of dimensionless mesh length, l_m/h_{ch}, on the flow pattern with filaments of constant diameter 0.5 and Reynolds number 300 is illustrated in Figure: 4 (a,b,c). For each of the cases separate recirculation region is formed between sequential filaments. As the mesh length is increased the size and length of the recirculation region is decreased. For the dimensionless mesh length, l_m/h_{ch} below 6 the separation streamline detaches from the wall. It has shown earlier in Figure 3: (c) that for l_m/h_{ch}=3, one large recirculation region is formed between sequential filaments.

Figure:5 (a,b,c) shows the characteristic distribution of the wall shear stress, C_f, for the cavity spacers. Since for smaller mesh length (l_m/h_{ch}=3 and below) there is a merged larger recirculation region between sequential filaments, the shear stress becomes higher in the recirculation region behind each filament. For higher dimensionless meshlength (l_m/h_{ch} =6,8) there are separate recirculation region and the shear stress is enhanced within each recirculation region on the bottom wall. On the top wall, Shear stress takes the maximum value immediately above the filament and the minimum value is achieved halfway between the filaments.

B. Flow profiles for multiple filament submerged in the channel:

Simulations are carried out with different Reynolds number and different dimensionless mesh lengths to predict the effects of the submerged spacers on the flow pattern between two sequential filaments. Similar to the single filament submerged in the channel, in this case also, recirculation region is formed behind each filaments in the center of the channel. At a mesh length of l_m/h_{ch}=3, stream function and velocity contours are plotted for different Reynolds number in Figure 7 (a,b,c). As Reynolds number increases the size of the recirculation region is also increased. Compare to the cavity spacer, the submerged spacers forms much lower size of recirculation region.

Figure 8: (a,b,c) shows the effect of the dimensionless mesh length on the flow pattern between sequential filaments. From the contour plot of the velocity magnitude with constant Reynolds number 300, it can be seen that, different recirculation zones are formed for each cases. The size of the recirculation region is increased as the mesh length decreases. Since the size of the recirculation region is much less than those formed for cavity spacers, a fully formed recirculation region is observed with much smaller mesh length for submerged spacers. The impact of recirculation region for upstream filament on the downstream one, is much reduced in this case because of the smaller size of the recirculation region. Thus, a "fully formed" recirculation can be formed with smaller mesh length. Figure 6: (d) shows the characteristic distribution of the shear stress on any of the wall between two sequential filaments. Maximum absolute value of the shear stress is achieved immediately above and underneath each filament. Halfway between two sequential filaments, the minimum absolute value of the shear stress is achieved.

From the discussions stated above, it is clear that for both the cavity and submerged spacers, a small Re_{ch} or a large l_m/h_{ch} produces separate recirculation region between sequential filaments. When Re_{ch} is increased at a fixed meshlength or the mesh length decreased at fixed Re_{ch}, a single large recirculation region is formed instead of two separate recirculation region. Compare to the cavity spacer, submerged spacers lead into a large recirculation region between sequential filaments with much higher Reynolds number or a much lower mesh length. Since for the filament placed adjacent to the wall, the length of the recirculation region is higher, the interaction between the separate recirculation region between sequential filaments occur more rapidly even for a higher mesh length.

C: Flow patterns for spacers placed in zigzag manner

Another configuration of the spacer, in which spacers are placed alternately adjacent to the top and bottom wall is also considered. The effects of the Reynolds number, Re_{ch} in the flow pattern between sequential filaments is presented in Figure:7 (a,b,c). Because of the configuration of the spacers, the flow direction is forced to change direction after passing the upstream filament. Like the single filament adjacent to the wall, in this case a small recirculation region is also formed before the filament, a large one is formed behind the filaments. The size and length of the recirculation region increases with Re_{ch}. The highest velocity above each of the filament is almost 2.3 times higher than the inlet velocity, which is similar to the single filament adjacent to the wall. For the zigzag spacers, the effects of the mesh length is studied for constant Reynolds number, Re_{ch}=300 in Figure 8: (a,b,c). The flow shows the similar feature with the single filament adjacent to the bottom wall for large mesh length. As the mesh length decreases, the flow is forced to zigzag around the filament and the tail of the higher velocity region above each filament extends towards the downstream one but with no further increment of the size and length of the recirculation region.

Three local maxima of the absolute value of the wall shear stress are observed in the characteristics distribution of the wall shear stress in Figure 8 (d). The highest absolute value of the wall shear stress occurs immediately above and opposite each filament. Another two small local maxima are achieved in front and behind each filament because of the recirculation region.

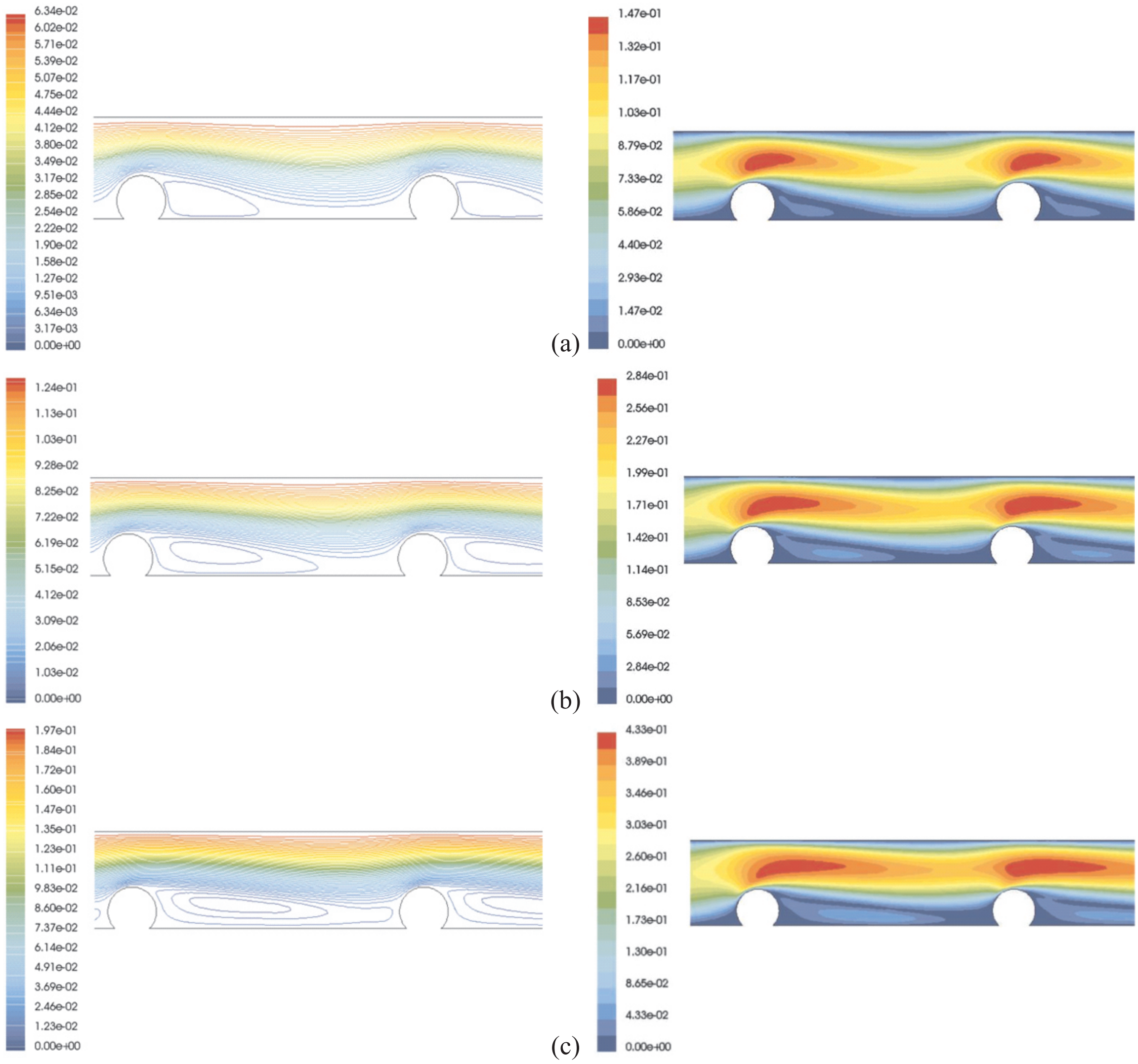
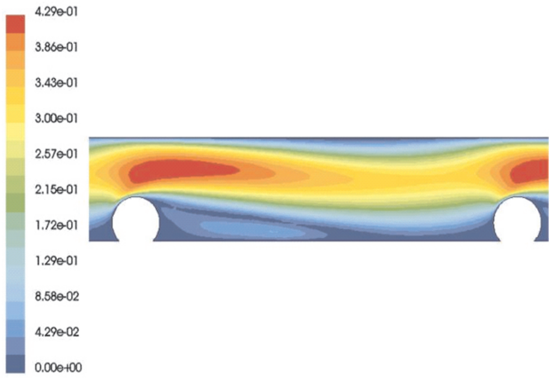
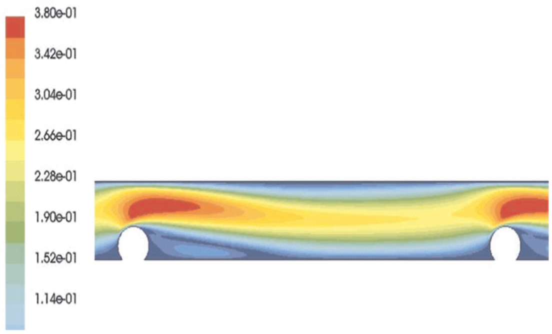


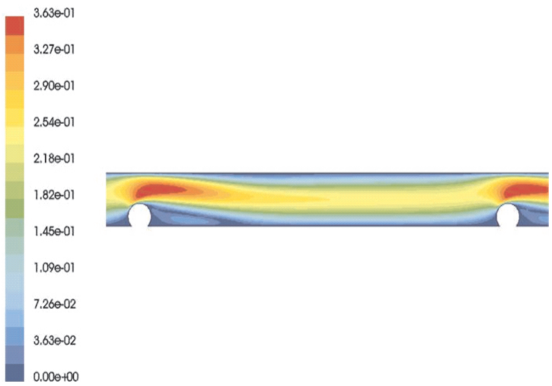
Figure 3: (a,b,c): Velocity contours for the cavity spacers for Reynolds number 100,200, 300 respectively with dimensionless mesh length l_m/h_{ch}=3. Left: streamlines pattern; right: streamwise velocity.



(a)

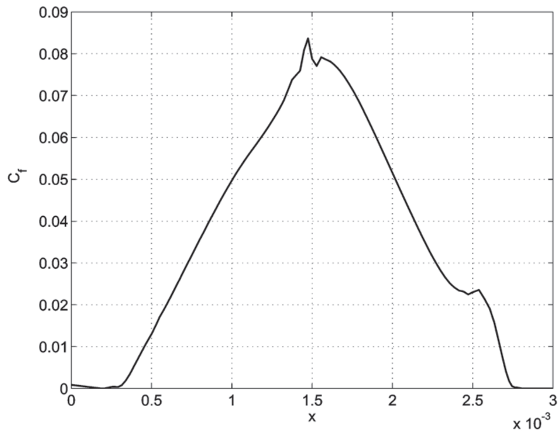


(b)

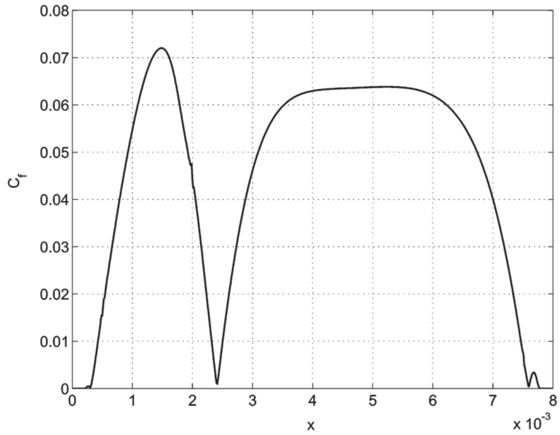


(c)

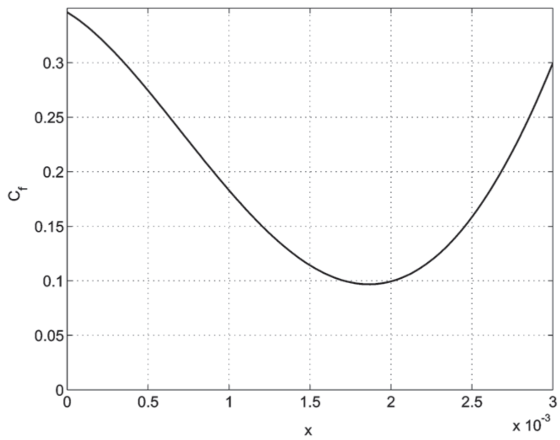
Figure 4: (a,b,c): Contours of the velocity magnitude for cavity spacers with dimensionless meshlength, l_m/h_{ch}= 4,6,8 respectively for Reynolds number Re_{ch}=300.



(a)

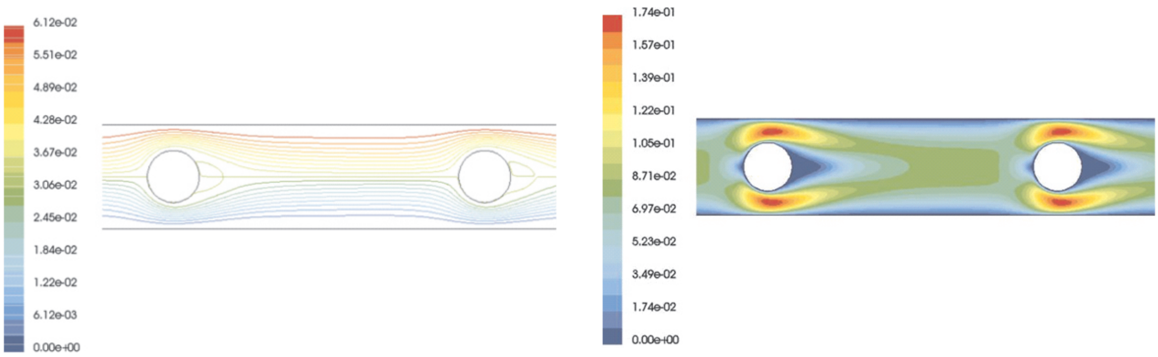


(b)

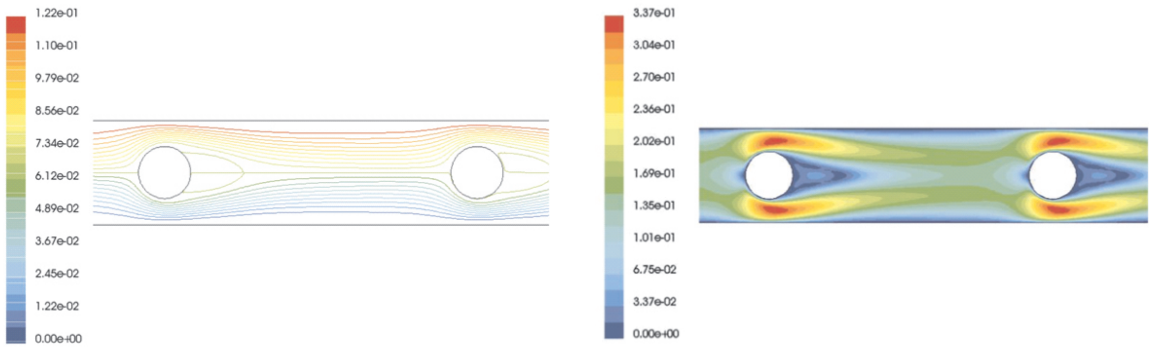


(c)

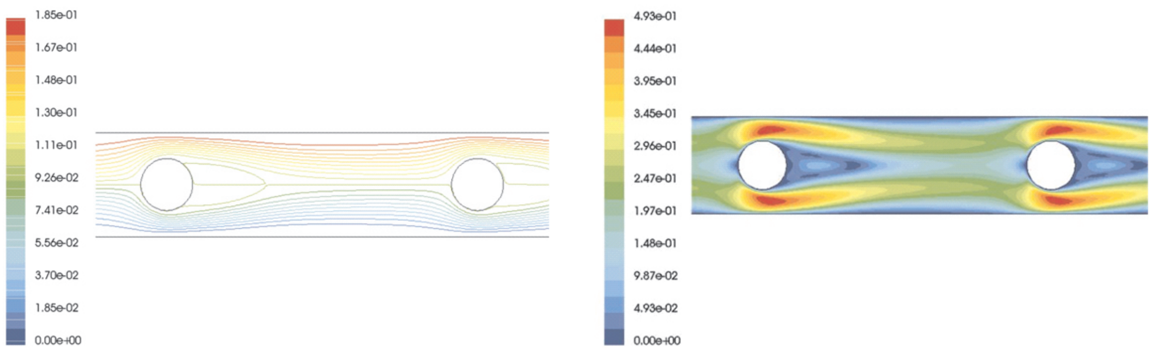
Figure 5: **(a)** Shear stress distribution for the cavity spacer on the bottom wall for l_m/h_{ch}=3, Re_{ch}=300.
(b) Shear stress distribution on the bottom wall for l_m/h_{ch}=6 and Re_{ch}=300.
(c) Shear stress distribution on the top wall between sequential filaments when l_m/h_{ch}=3 and Re_{ch}=300



(a)



(b)



(c)

Figure 6: (a,b,c): Velocity contours for the submerged spacers for Reynolds number 100,200, 300 respectively with dimensionless mesh length, l_m/h_{ch}=3. Left: streamlines pattern; right: streamwise velocity.

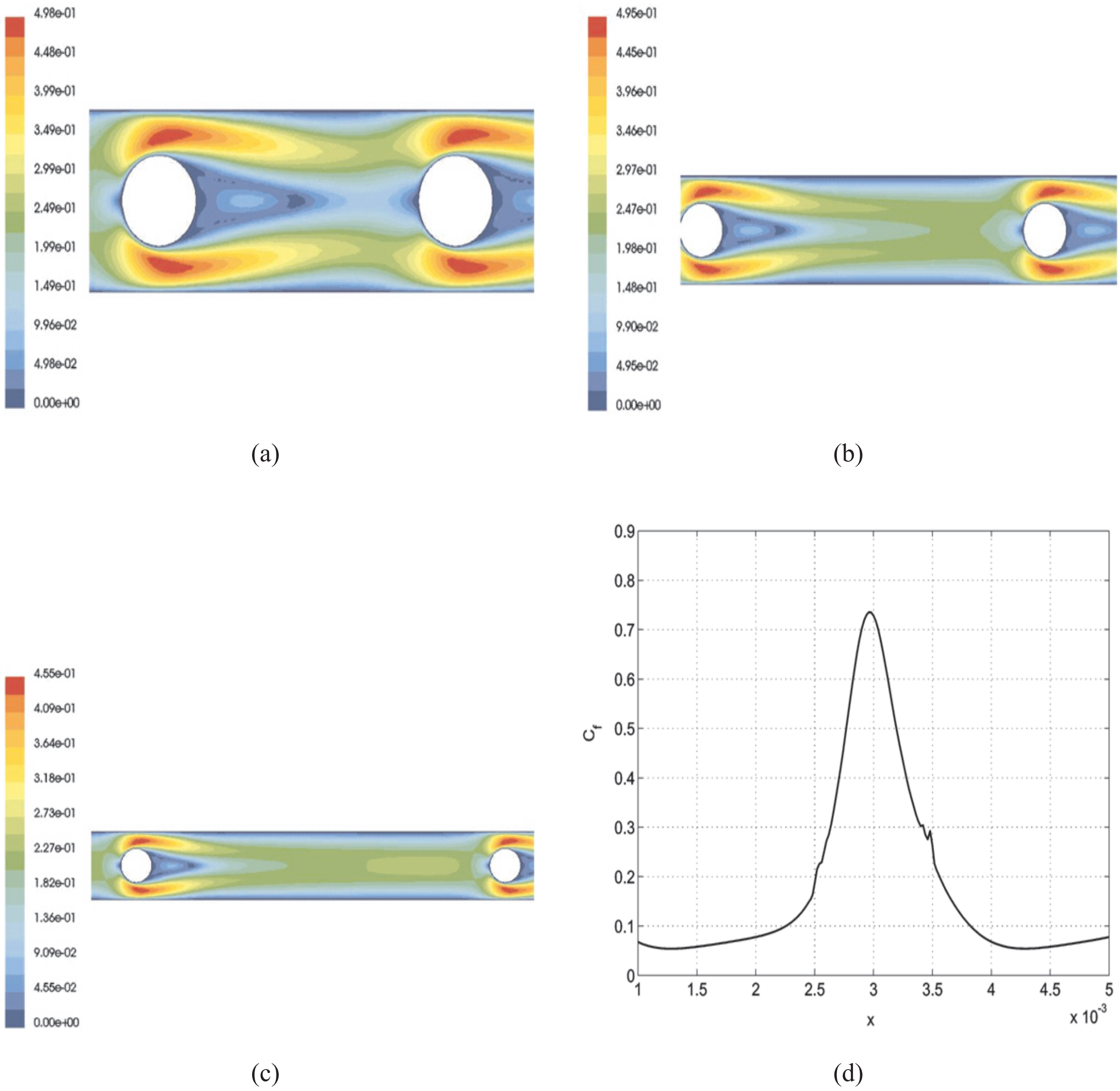
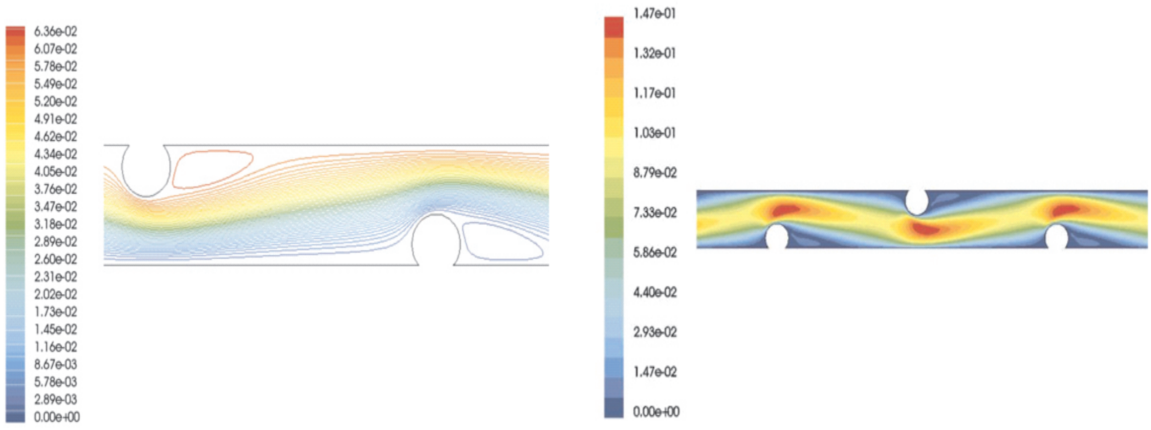
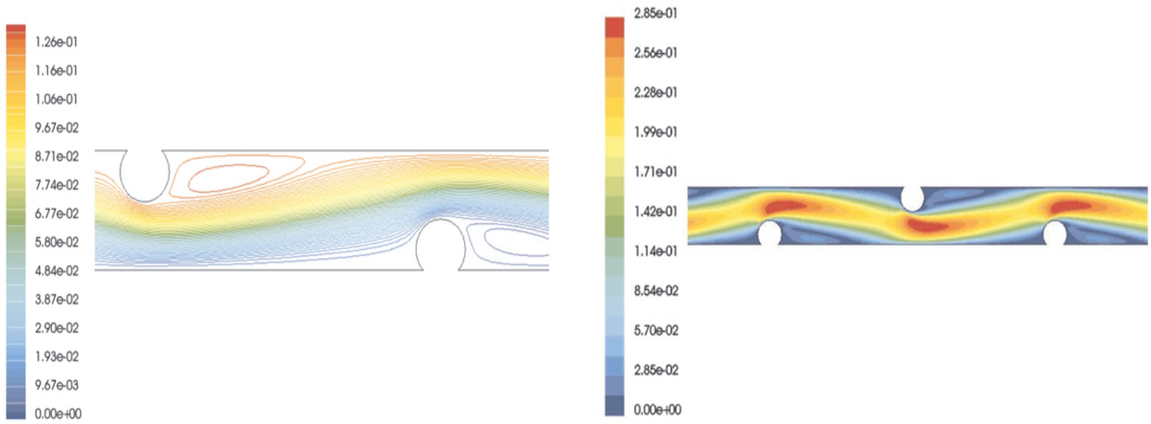


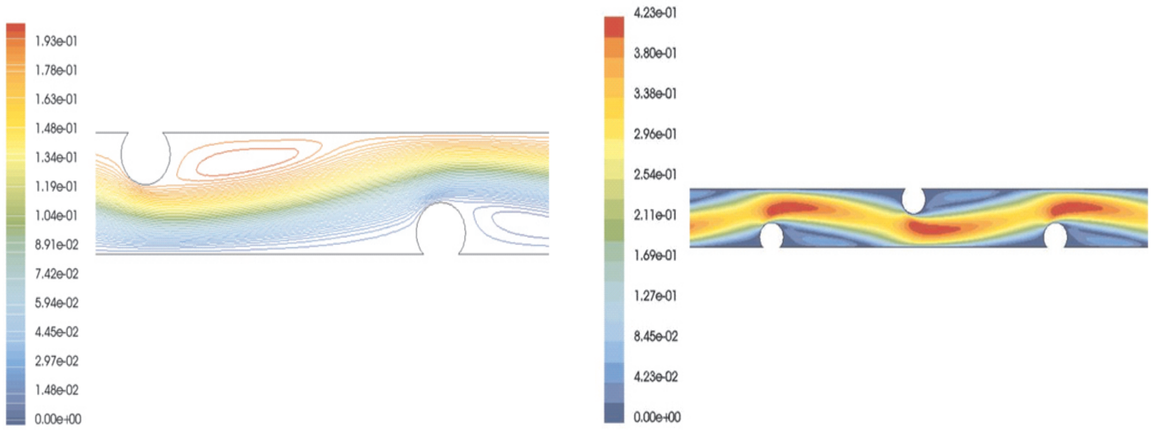
Figure 6: (a,b,c): Contours of the velocity magnitude for submerged spacers with meshlength, l_m/h_{ch}=4,6,8 respectively for Reynolds number Re_{ch}=300.
(d). shear stress distribution for the submerged filament on the top wall between sequential filaments (l_m/h_{ch}=3, Re_{ch}=300).



(a)

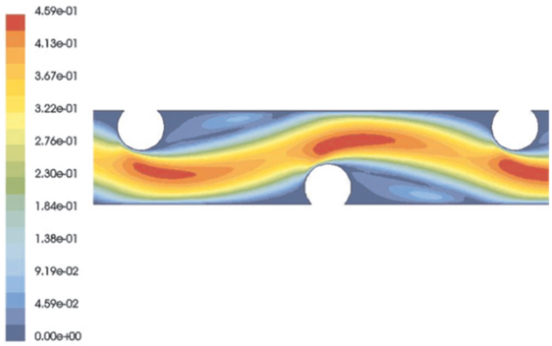


(b)

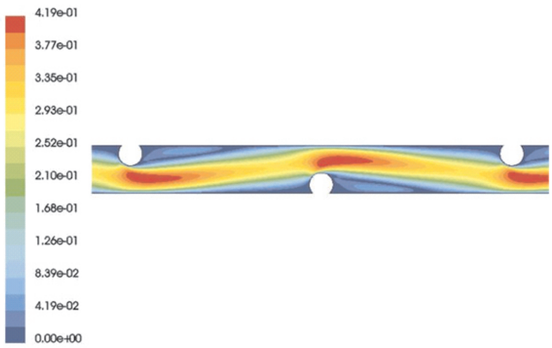


(c)

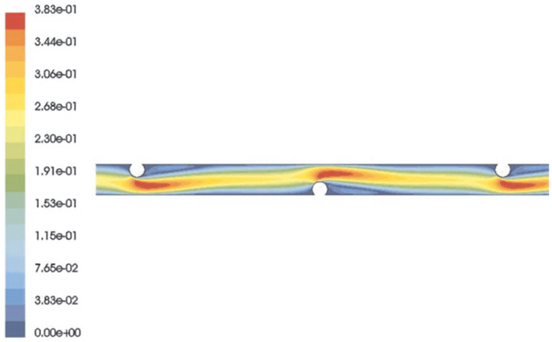
Figure 7: (a,b,c): Velocity contours for the zigzag spacers for Reynolds number 100,200, 300 respectively with dimensionless mesh length, l_m/h_{ch}=3. Left: streamlines pattern; right: streamwise velocity.



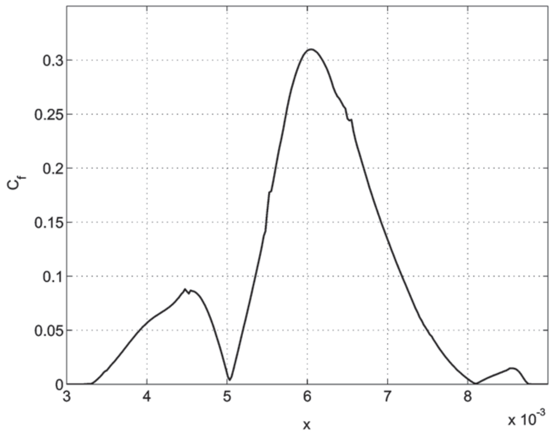
(a)



(b)



(c)



(d)

Figure 8: (a,b,c): Contours of the velocity magnitude for zigzag spacers with meshlength, l_m/h_{ch}=2, 4,6, respectively for Reynolds number Re_{ch}=300.
(**d**). shear stress distribution for the zigzag spacer on the top wall between sequential filaments (l_m/h_{ch}=3, Re_{ch}=300).

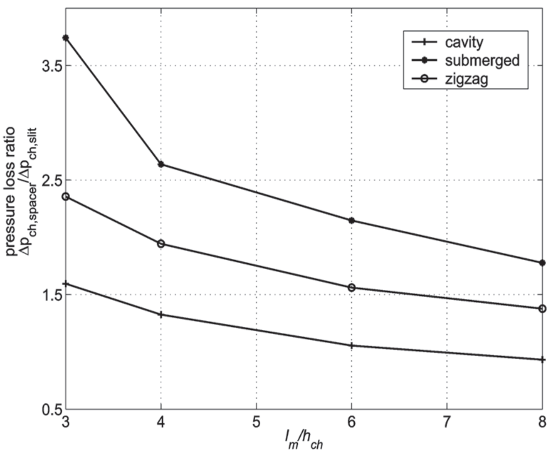


Figure 9: comparison of pressure loss along the channel, at different mesh length for three types of spacers with channel Reynolds number Re_{ch}=200, Sc=100.

IV. CONCLUSION:

This present study is devoted to reveal the importance of hydrodynamics on mass transfer and pressure loss enhancement along the flow channel. CFD simulations of Hydrodynamics in spacer obstructed membrane module has been carried out. 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. 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 three two types of spacers, submerged spacers show the higher wall shear stress. 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 spacers can be broaden to study the flow filled in 3D channel.

V. REFERENCES

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

[2] 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).

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

[4] 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,

[5] 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.

[6] 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.

[7] 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)

[8] Sharmina Hussain. Computational fluid dynamics study of flow behaviour in a single spacer filled membrane module, *BRAC University Journal*, Vol.5, No.1,2008.