Design Improvement of a Bench-scale Nanofiltration Device by CFD Study

B. Balannec[*,1,2], F. Cortès-Juan[3] and T. Renouard[1,2]
[1] Université de Rennes 1-ENSCR-CNRS, France, [2]Université européenne de Bretagne, France,
[3] Universidad Politécnica de Valencia, Spain
* Ecole Nationale Supérieure de Chimie de Rennes, UMR 6226, Avenue du Général Leclerc, CS 50837, 35708 Rennes Cedex 7, France, beatrice.balannec@univ-rennes1.fr

Abstract: CFD simulation was carried out to describe the complex 3D flow in a flat circular nanofiltration cell. Several design modifications were simulated in order to improve the flow distribution. The study focused mainly on the influence of the feed chamber thickness, the number and location of the inlet/outlet pipes and the modification of the cell top surface.
The CFD studies show that the velocity and the shear stress can be increased and that the flow distribution can be improved by lowering recirculation zones. Sweeping of the membrane surface and velocity fluctuations, obtained with sequential inlets and by adding grooves on the top of the cell, help to reduce polarization concentration phenomena. A new designed cell was built including most of the suggested modifications: the first filtration results show a sharp enhancement of the nanofiltration flux.

Keywords: CFD, filtration cell, design, flux enhancement.

1. Introduction

Feasibility studies in the field of membrane processes are generally first carried out with bench-scale membrane filtration cells. These small laboratory cells also allow evaluating the influence of operational parameters and the membrane performances. Nevertheless, for the accurate interpretation of filtration experiments, the knowledge of the flow field in the filtration cell is required. This flow is usually considered to be unidirectional with uniform velocity, but this assumption of similar flow conditions is far from reality in most filtration cells [1]. Furthermore, hydrodynamics in the cell (pressure, velocity and shear stress) affects filtration performances, membrane fouling and polarization phenomena. Smart design of the cell geometry may improve the flow distribution and consequently the membrane filtration performances by increasing shear stress and reducing fouling.

CFD simulation was first carried out to describe the complex 3D flow in a flat circular nanofiltration cell. The aim of the study was then to simulate the velocity field with several design modifications in order to improve the flow distribution.

1.1 Nanofiltration cell description

The original cell used for nanofiltration experiments and modeled by CFD (Figure 1) is a flat circular cell with a feed chamber thickness of 5 mm and a diameter of 35 mm. The inlet and outlet pipes diameter is 0.3 mm. The nanofiltration membrane (on the bottom of the cell) is supported on a grooved structure that collects the filtrate flowing through the membrane. Experimentally, it was observed that the filtrate flow was negligible compared to the feed flow (< 1 %). Therefore, it was considered that it does not affect significantly the flow distribution and consequently the effect of permeation was not included in the model.

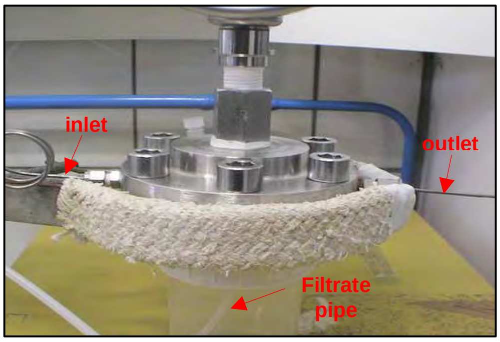

Figure 1. Nanofiltration cell.

2. Use of COMSOL Multiphysics

2.1 Governing Equations

Specifically, we have assumed a steady, isothermal flow of an incompressible Newtonian fluid. In the inlet/outlet pipes the Reynolds number was in the range 100-800 and in feed chamber of the cell it was lower than 400. So, the laminar model (Navier-Stokes and continuity equations (1, 2)) was solved.

$$-\nabla \cdot \eta\left(\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}\right) + \rho(\mathbf{u} \cdot \nabla)\mathbf{u} + \nabla p = 0$$

$$\nabla \cdot \mathbf{u} = 0$$

2.2 Meshing

For this 3D geometry, the low diameter of the inlet/outlet pipes and the small thickness of the feed chamber compared to the cell diameter lead to study different meshing strategies in order to minimize the memory requirements. A free meshing was finally performed for the feed chamber and an extruded meshing for the inlet/outlet pipes which required around 20000 elements.

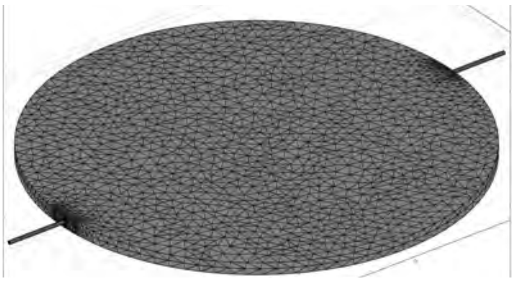

Figure 2. Example of Meshing.

2.3 Physics and boundaries

Fluid properties were considered at 25 °C: constant density of 800 kg/m^3 and viscosity of 0.6 10^{-3} Pa·s.
A laminar inflow boundary condition was given. For the outflow, a pressure boundary condition was set and the pressure was referenced to zero for computational purposes (as only pressure gradients appear in the equations).

At all wall boundaries, even at the membrane surface on the bottom of the cell (insofar as the permeation effect was not taken into account), the no-slip condition was imposed.

2.4 Transient mode

Transient modeling was made in order to simulate the flow in a filtration cell with 2 inlets, each of them being connected to a piston of the pump. The imposed velocities for each inlet versus time were given in figure 3.

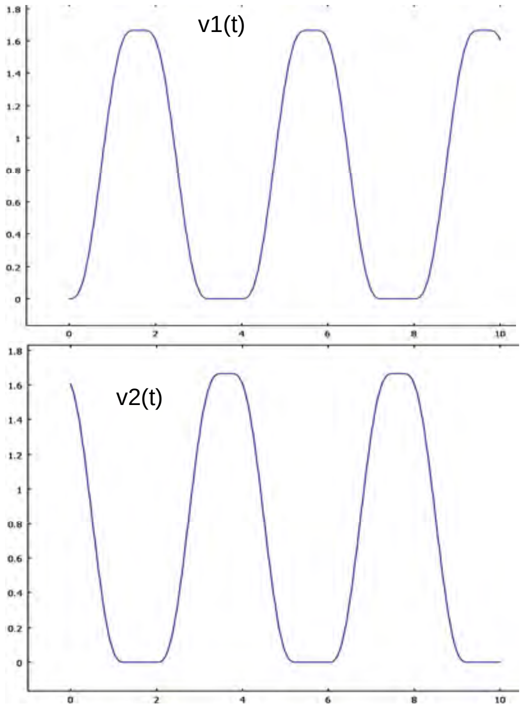

Figure 3. Inlet velocities (m/s) versus time (s) for 2 sequential inlets.

3. Results and discussion

The CFD simulation carried out for the original nanofiltration cell with one inlet and one outlet (Figure 2) confirmed the very bad flow distribution insofar as a jet (by-pass) was created between the inlet and the outlet separating two large recirculation areas of low velocities (Figure 4). This bad hydrodynamics can explain the low performances obtained with this filtration cell especially in terms of filtration fluxes.

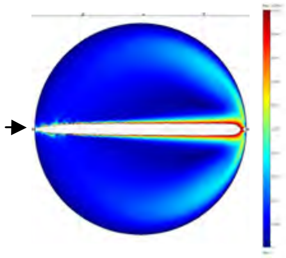

Figure 4. Velocity field (m/s) in the original cell.

A decrease of the feed chamber thickness was first suggested to increase the velocities and consequently the shear stress on the membrane surface. The CFD simulation made with a thickness of 1 mm showed the expected increase of the velocities in the recirculation areas.

To improve the flow distribution, several simulations were carried out by changing the number and the layout of the inlet/outlet pipes. The increase of the number of inlet and/or outlet pipes leads to a decrease of the low velocity areas (Figure 5). Nevertheless, to avoid too much complexity, a design of a cell with two inlets and one outlet seems to be a good compromise.

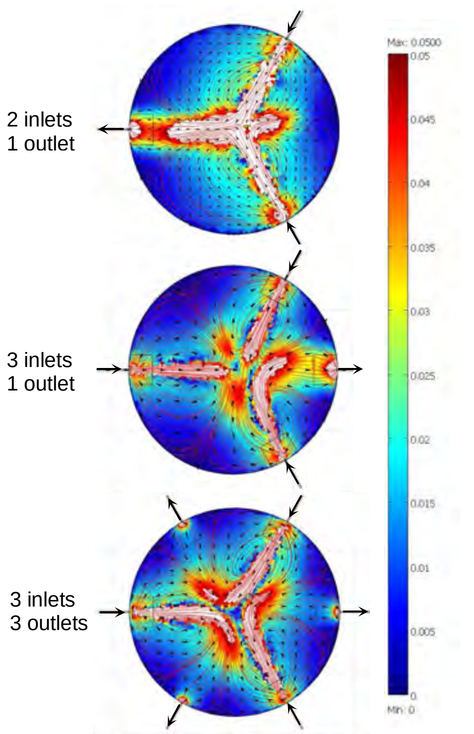

Figure 5. Velocity field (m/s) for different number and layout of inlet/outlet pipes.

Insofar as the pump connected to the cell has two pistons, one can suggest connecting each piston to one of the two inlets. Simulation in transient mode was made considering two sequential inlets, with the inlet velocities given in Figure 3. The results showed important velocity fluctuations with time (in direction and intensity) and consequently sweeping of the membrane surface which should hinder polarization layer development (Figure 6).

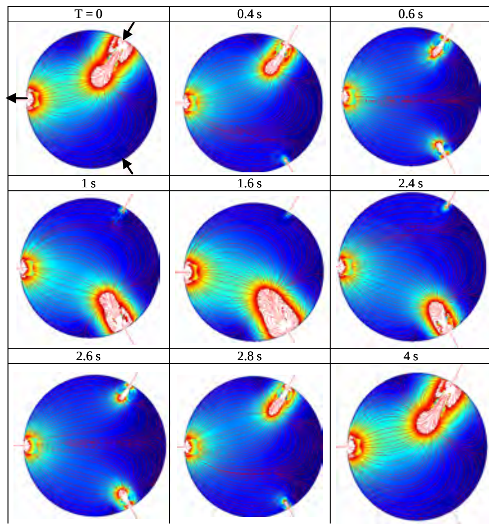

Figure 6. Velocity field and streamlines in transient mode with 2 sequential inlets and 1 outlet.

Addition of grooves on the top of the filtration cell was also proposed with the aim to add local velocity fluctuations. Simulation was carried out with the grooves perpendicular to the outlet pipe. The goal of the 2D simulation was to optimize the size and the frequency of the grooves. The 3D simulation with the grooves strengthened the 2D results insofar as intense velocity fluctuations appeared in the cell (Figure 7). As a consequence, the related shear stress fluctuations at the membrane surface should greatly disrupt the development of polarization layer. Similar results were described by Liu et al. [2] for baffle-filled membrane tubes.

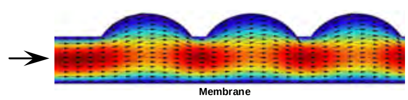

Figure 7. Velocity distribution in the feed chamber of the cell with addition of grooves (2D simulation).

On the basis of this CFD study, a new cell was further designed and manufactured taking into account most of the suggested modifications (Figure 8): the feed chamber thickness was decreased from 5 mm to 0.5 mm, two inlets and one outlet were added (each inlet connected to a piston of the pump). The first filtration results with this new designed cell showed a sharp enhancement of the nanofiltration flux which was increased by a factor 2.5, probably due to an increase of the wall shear stress and of the velocity fluctuations leading to a decrease of the polarization concentration effect.

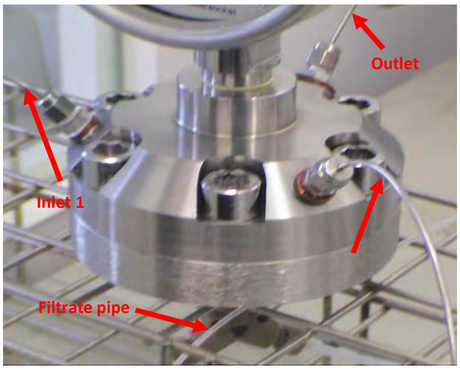

Figure 8. New designed nanofiltration cell

4. Conclusions

The CFD simulations of the flow in the nanofiltration cell with several design modifications showed that the velocity and the wall shear stress can be sharply increased and that the size of the recirculation areas can be lowered. The two sequential inlets and the addition of grooves on the top of the cell permitted to obtain intense velocity fluctuations and as a consequence wall shear stress fluctuations which should greatly hinder the development of the polarization layer. As expected, with the new designed cell the filtration performances were improved and the nanofiltration fluxes were dramatically enhanced.

5. References

1. D. Delaunay, M. Rabiller-Baudry, M. Gozálvez-Zafrilla, B. Balannec, M. Frappart, L. Paugam, Mapping of protein fouling by FTIR-ATR as experimental tool to study membrane fouling and fluid velocity profile in various geometries and validation by CFD simulation, *Chem. Eng. Process.*, **47**, 1106-1117 (2008)

2. Y.Liu, G. He, X. Liu, G.Xiao, B. Li, CFD simulations of turbulent flow in baffle-filled membrane tubes, *Sep. Pur. Technol.*, **67**, 14-20 (2009)