COMPUTATIONAL FLUID DYNAMICS (CFD) APPROACH FOR CHARACTERISING AND IMPROVING FLUID FLOW FOR MEMBRANE FILTRATION TECHNOLOGIES AND SUCCESSFUL SCALE-UP

Mohd Shawkat Hussain, Department of Biochemical Engineering, UCL, UK
mohd.hussain.10@ucl.ac.uk
Yuhong Zhou, Department of Biochemical Engineering, UCL, UK

Key Words: Computational fluid dynamics (CFD), screened channels, USD membrane filtration device, scale-up, design optimisation

There is increasing interest in using computational fluid dynamics (CFD) techniques to obtain better understanding of fluid dynamics of various unit operations, both upstream and downstream bioprocessing, and to use simulated results for process modelling, system characterisation and design optimisation. Membrane filtration technologies and in particular, tangential flow filtration (TFF), have made use of CFD analysis to gain better insight into, for instance, velocity and concentration profiles within channels, optimising spacer geometry and configurations to maximise mass transfer for different TFF setups. The applications and benefits for TFF processes are immense since large volumes of liquid are normally required, both for cleaning and processing, and thus even small increments in efficiency can result in significant cost reductions. In this study, a novel Ultra Scale-Down (USD) membrane filtration device and spacer-filled channels in TFF cassettes were modelled using COMSOL Multiphysics. 2D-axisymmetrical and 3D models were developed, respectively, to simulate the effect of geometry, design and hydrodynamic conditions on flow, wall shear rates and pressure profiles. Laminar flow and Shear Stress Transport (SST) formulations were used and parametric sweeps employed for different geometrical and design parameters. The geometrical configuration, such as clearance of the disc, cone angle and disc diameter, and the resulting hydrodynamics within the system was seen to play a vital role in wall shear rate distributions for the USD device. The change could be attributed to the introduction of secondary recirculating flows and vortices that allow greater degree of axial and radial mixing, as seen from the CFD results. Optimal disc type, namely the axial flow impellers such as pitch-blade turbine and the hydrofoil, and positioning was also identified based on parameters such as wall shear rate distribution along the radial axis of the membrane and maximum shear stress:average wall shear stress ratio. Flow within the screened channel was observed to be very controlled and periodic, especially in regions close to and around the cylindrical fibers of the screen, and an overall accelerated flow within the channel compared to the inlet velocity was observed from the CFD simulations. The aim was to use CFD as a guiding tool to optimise the design of the USD device such that it mimics the average wall shear rate and profiles observed within the screened channels, in order to establish successful and accurate scale-up between the USD device and TFF process, at scale. Experimental validation for the CFD results are also presented, along with other potential applications of the CFD studies carried out, such as modelling flux behaviour in the channel, cake formation and pressure drops.

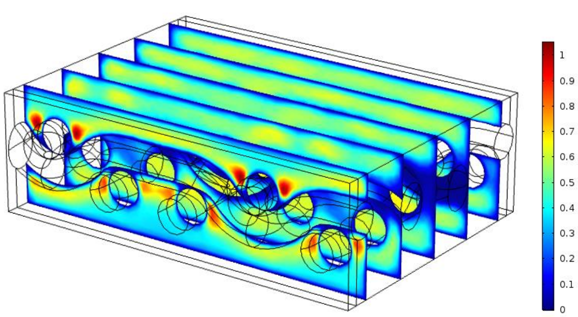

Figure 1 – Velocity magnitude surface plot for a V-screen channel (flow is from left to right), feed flux of 5 LMM, 1cP dynamic viscosity.

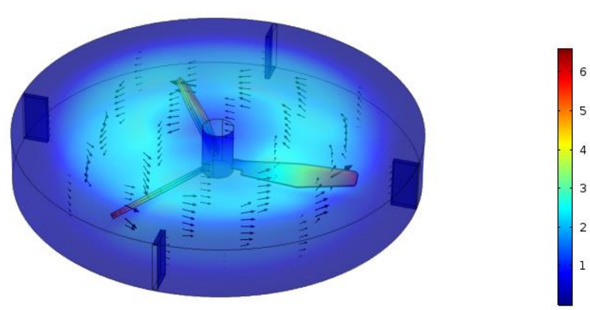

Figure 2 – Volume and arrow volume plot for velocity magnitude and velocity fields respectively, for the USD membrane filtration device, with modified disc (PBT). Simulation conditions were 4000 RPM, dynamic viscosity of 1cP.