Journal of Membrane Science 343 (2009) 16–33

Contents lists available at ScienceDirect

Journal of Membrane Science

journal homepage: www.elsevier.com/locate/memsci

# Feed spacer mesh angle: 3D modeling, simulation and optimization based on unsteady hydrodynamic in spiral wound membrane channel

K.K. Lau[a,*], M.Z. Abu Bakar[b], A.L. Ahmad[b], T. Murugesan[a]

[a] Department of Chemical Engineering, Universiti Teknologi PETRONAS, Bandar Seri Iskandar, 31750 Tronoh, Perak, Malaysia
[b] School of Chemical Engineering, Engineering Campus, Universiti Sains Malaysia, 14300 Nibong Tebal, Penang, Malaysia

ARTICLE INFO

*Article history:*
Received 24 January 2009
Received in revised form 30 June 2009
Accepted 2 July 2009
Available online 9 July 2009

*Keywords:*
Spacer mesh angle
Feed spacer
Spiral wound membrane
Concentration polarization
Unsteady hydrodynamics

ABSTRACT

Basic knowledge on the hydrodynamics in the spacer-filled spiral wound membrane (SWM) channel is vital for the understanding of the formation of concentration polarization at the membrane interface. In the present study, a 3D laminar transient hydrodynamics modeling approach was used to study and optimize the spacer mesh angle for the SWM feed spacer. Based on the simulated results, the optimal spacer mesh angle that yields the minimum effective concentration polarization factor, was found to be $\alpha120\beta30$. Under this optimal mesh angles, spacer $\alpha120\beta30$ also demonstrated the highest magnitude of unsteady hydrodynamics (which adjacent to the membrane wall) at a moderate degree of pressure loss.


## 1. Introduction

Computational Fluid Dynamics (CFD) modeling and simulation techniques have been widely applied to predict the hydrodynamics in the spacer-filled spiral wound membrane (SWM) feed channel [1]. The modeling of hydrodynamics in the spacer-filled SWM channel using CFD is crucial for predicting the formation of concentration polarization/fouling at the membrane interface. Three types of approaches, namely steady-state laminar, transient and time-averaged turbulent modeling have been applied to model the hydrodynamics in the spacer-filled SWM channel.

Steady-state laminar techniques involve the solution of the Navier–Stokes and solute balance equations to model the concentration polarization in the membrane channel under steady-state conditions. Geraldes et al. applied this technique to study the hydrodynamics and the membrane transport mechanism in the empty nanofiltration membrane channel [2]. Later, the correlation was modified with the incorporation of intrinsic true rejection parameter in the correlation [3]. Besides, membrane channel filled with ladder type spacer were discretized in 2D and laminar condition to predict the concentration polarization profile by employing CFD mathematical modeling method [4–6]. Wiley et al. conducted a fundamental study on the empty membrane channel using a 2D membrane model through CFX [7,8]. Quantitative study on the effects of the spacer upon concentration polarization was carried out by Ma et al. using a 2D streamline upwind Petrov/Galerkin (SUPG) finite element model [9,10].

Based on the conclusion made from related works [11–14], it is very clear that the laminar modeling approach is insufficient to model the actual hydrodynamic and concentration polarization in the spiral wound feed channel, even though under low Reynolds number ($Re_f < 400$). As an alternative to the laminar modeling approach, time-averaged turbulent modeling has been applied to model the unsteady hydrodynamics in the spacer-filled membrane channel with lower computational resources. Karode and Kumar conducted turbulent CFD simulations for fluid flow through the rectangular channels filled with several commercially available spacers in membrane modules [15]. Schwinge et al. also simulated hydrodynamics and mass transfer phenomena under steady flow [16]. Cao et al. simulated the velocity profile and turbulent kinetic energy distributions in the spacer-filled channel [14], whereas Ranade et al. conducted 3D turbulent simulations to study the curvilinear structure of the spacer feed channel using 'Unit Cell' approach [13].

With the objective to portrait the actual unsteady hydrodynamics in a greater detail, unsteady transient simulation in the confined spacer-filled membrane channel have been conducted by several researchers. Koutsou et al. have applied periodical transient simulation technique to study the unsteady hydrodynamics in the spacer-filled membrane channel [12]. Consequent research on the development and separation of boundary layers, vortex

* Corresponding author. Tel.: +60 5 3687589; fax: +60 5 3656176.
*E-mail address:* laukokkeong@petronas.com.my (K.K. Lau).

0376-7388/$ – see front matter 
doi:10.1016/j.memsci.2009.07.001

formation, high shear regions and recirculation zones have been conducted [17]. Commercial CFD simulations codes (CFX) were employed by Schwinge et al. to simulate the unsteady hydrodynamics phenomenon in the 2D spacer-filled membrane channel [11,18]. Consequent work has been extended to unsteady hydrodynamics with mass transfer for spacer-filled membrane channel [19]. Based on their results, the transition to unsteady flow for multiple filament spacers was found to occur at $Re_{ch} > 400$ for cavity, submerged and zigzag spacers. Li et al. investigated the relation between Sherwood and power numbers using 3D CFD transient models for low Reynolds numbers in a non-woven spacer-filled channel [20,21], and the simulated optimum ratio were validated experimentally [22].

Even though the transient simulation inevitably offers better solution for visualization of the actual unsteady hydrodynamics in the spacer-filled membrane channel, the above mentioned turbulent and transient CFD studies are limited to the assumption of membrane as an impermeable wall. Impermeable wall studies involve the setting of a fixed solute concentration or solute flux at the membrane wall, which results in a concentration gradient at this boundary with the bulk flow. This approach neglected the mass transport across the membrane and led to an incorrect assessment of the concentration polarization phenomenon. In order to overcome this limitation, our previous study has been conducted to investigate the unsteady hydrodynamics and membrane mass transport properties in SWM feed channel which filled with filaments of different geometries [23]. As a consequent work, one of the important design parameters for feed spacer; mesh angle, has been studied and optimized under present work. Integrated CFD simulation approach (with the consideration of membrane mass transport properties) will be applied to simulate the transient unsteady hydrodynamics in the spacer-filled SWM feed channel.

## 2. Methodology

### 2.1. Governing equation

The fluid flowing in the spacer-filled membrane channel presents at unsteady condition at a relatively low feed Reynolds number [11,12]. The fluctuation of velocity and pressure in the unsteady hydrodynamics causes the velocity and pressure varying rapidly with time and space. Hence, in the present work, a laminar transient simulation method (with the governing equations proposed by Ahmad and Lau [23]) was used to solve the unsteady three-dimensional hydrodynamics in spacer-filled membrane channel domain with consideration of solute concentration variation.

### 2.2. Simulation approach

The present work involves the Periodic Unit Cell Simulation (PUCS) to simulate the unsteady hydrodynamics in the SWM channels. The usage of unit cell simulation offered the possibility of resolving small-scale features of flow in greater detail without simulating the complete membrane module [13]. Time-averaged quantities obtained from periodic cell modeling could be taken as representative of actual spacer-filled channels [12]. The disadvantage of using PUCS with commercial simulator was the treatment of membrane as non-porous wall. Typically, to maintain the overall continuity, the membrane was treated as non-porous wall in PUCS. Hence, the solution of governing equations with the integration of membrane permeation properties was impossible to be carried out with PUCS. Since the permeation flux ($J_v$) was significantly small when compared with the inlet velocity ($V_{in}$) ($J_v < 0.1\%$ $V_{in}$ for nanofiltration and reverse osmosis), the velocity profile in the PUCS would be relatively identical with velocity profile in the periodic simulation that integrated with permeation properties.

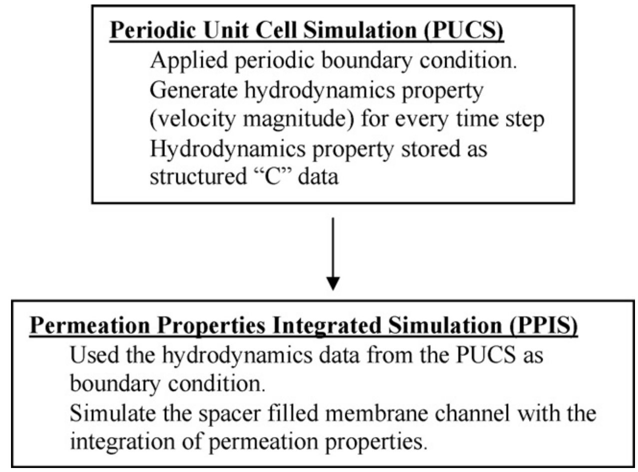


**Fig. 1.** Periodic Unit Cell Simulation (PUCS) and Permeation Properties Integrated Simulation (PPIS).

Thus, PUCS was used in present study to simulate the hydrodynamics in the spacer-filled membrane channel. Since the unsteady hydrodynamics in spacer-filled membrane channel fluctuated with time and space, these fluctuating quantities (velocity profile) were simulated using PUCS and written in "C" language structured data. These hydrodynamics' structured data were later applied in Permeation Properties Integrated Simulation (PPIS) to simulate the hydrodynamics and the permeation properties in the spacer-filled membrane channel (Fig. 1).

Under PPIS, the hydrodynamics data from PUCS were used in all the boundary conditions except for membrane and non-permeable spacer wall in a single computational cell. Since the PPIS used all the hydrodynamics data (velocity profile) from the PUCS (which assumed membrane as non-porous wall), the imbalanced mass due to the existence of permeation flux should be compensated in order to maintain the overall mass balance of the system. In the present study, the imbalanced mass was compensated by the additional of mass in the flow inlet (Fig. 2). The additional of mass were calculated based on averaged velocity of the imbalanced permeation mass flux ($J_v = \rho A\overline{U}_{im}$) under the PPIS. The averaged velocity would change with the variation of permeation flux computed through the integrated CFD model. The details of the boundary conditions for PPIS are discussed below.

### 2.3. Boundary conditions

#### 2.3.1. Boundary conditions for Periodic Unit Cell Simulation (PUCS)

Periodic boundary conditions (1–4) were defined for the flow inlet and outlet boundaries in the PUCS (Fig. 3). In periodic boundary conditions, the total mass flow rate and temperature of the feed

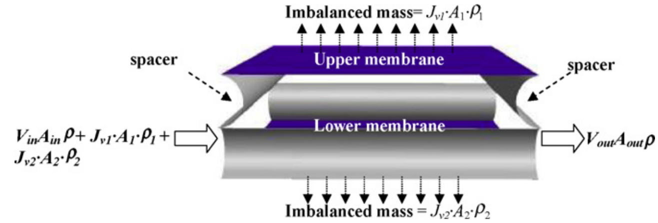


**Fig. 2.** Compensation of imbalanced mass in Permeation Properties Integrated Simulation (PPIS).