INTERNATIONAL JOURNAL OF HYDROGEN ENERGY 38 (2013) 12879–12885

Available online at www.sciencedirect.com

SciVerse ScienceDirect

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

Effect of wettability on water removal from the gas diffusion layer surface in a novel proton exchange membrane fuel cell flow channel

Yanzhou Qin [a,b], *Xianguo Li* [a,b,*], *Qing Du* [a], *Yan Yin* [a], *Kui Jiao* [a]

[a] *State Key Laboratory of Engines, Tianjin University, Tianjin, China*
[b] *Department of Mechanical and Mechatronics Engineering, University of Waterloo, Waterloo, ON, Canada*

ARTICLE INFO

Article history:
Received 4 January 2013
Received in revised form
25 May 2013
Accepted 28 May 2013
Available online 26 July 2013

Keywords:
Proton exchange membrane fuel cell
Water management
Water removal
Water dynamics
Channel flow

ABSTRACT

Effective water removal from the proton exchange membrane fuel cell (PEMFC) surface exposed to the flow channel is critical to the operation and water management in PEMFCs. In this study, the water removal process is investigated numerically for a novel flow channel formed by inserting a hydrophilic needle in the conventional PEMFC flow channel, and the effect of the surface wettability of the membrane electrode assembly (MEA) and the inserted needle on the water removal process is studied. The results show that the liquid water can be more effectively removed from the MEA surface for larger MEA surface contact angles and smaller needle surface contact angles. The pressure drop for the flow in the channel is also examined and it is seen to be indicative of the liquid water flow and transport in the flow channel, suggesting that pressure drop is a useful parameter for the investigation of water transport and dynamics in the flow channel.

1. Introduction

Proton exchange membrane fuel cell (PEMFC) has been recognized as a promising zero-emission power source for portable, mobile and stationary applications [1]. Design of the gas flow channel in bipolar plates is one of the main approaches for improving PEMFC performance [2]. The geometry and the pattern of the gas flow channel significantly influence the reactant transport, water management, thermal management and utilization of the cell. They are especially important for the water management because the product water in the cathode need effective removal by the reactant stream in the gas flow channel having appropriate design.

Considerable effort has been made in the past several decades for the flow channel design in PEMFC. West and Fuller [3] investigated the effect of the rib spacing of the fuel cell channels using a 2D model and showed that the width of the ribs strongly affected the water content in the membrane. Baschuk and Li [4] studied the effect of the length of the gas flow channel on the current production and found that the fuel cell had better performance with shorter channel length. Perng et al. [5] found that a transverse installation of a rectangular cylinder at the axis of the flow channel could effectively enhance the local cell performance. Wu et al. [6] studied the effect of the rectangular cylinder number on the cell performance, and they found the optimal cylinder number considering both cell performance and reasonable pressure drop associated with the gas flow in the channel. In a recent study [7], a cylindrical needle with small diameter was inserted in the middle of the flow channel that improved the water

* *Corresponding author*. Department of Mechanical and Mechatronics Engineering, University of Waterloo, Waterloo, ON N2L 3G1, Canada. Tel.: +1 519 888 4567; fax: +1 519 888 4358.
E-mail addresses: x6li@uwaterloo.ca, xianguo.li@uwaterloo.ca (X. Li).
0360-3199/$ – see front matter Copyright © 2013, Hydrogen Energy Publications, LLC. Published by Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijhydene.2013.05.157

removal in the channel; the volume-of-fluid (VOF) multiphase approach used in earlier studies [8–11] was adopted. It was shown that the hydrophilic needle inserted in the flow channel facilitated water removal from the MEA surface. The effect of the size and length of the needle on the water removal process was also investigated and the optimal design of the needle in the channel was obtained.

In this study, the objective is to investigate the effect of the surface wettability of the MEA and the inserted needle in the flow channel, respectively; and to study the wettability effect on the water transport and removal process from the MEA surface exposed to the novel flow channel design having a hydrophilic needle inserted in the channel. The numerical investigation also uses the VOF method to track the motion of the liquid–gas interface.

2. Model formulation

2.1. Computational domain

A typical single straight flow channel with a square cross section is considered as the computational domain in this study, as shown in Fig. 1. The flow channel is 50 mm long with the cross section of 1 mm × 1 mm. To facilitate the liquid water removal from the MEA surface, a small cylindrical needle is inserted in the middle of the flow channel, normal to the bottom surface of the channel, 4 mm away from the channel inlet. The needle inserted has a diameter of 0.1 mm and a length of 0.7 mm, these are optimal dimensions determined in the previous study [7].

2.2. Governing equations

The air flow in the channel is considered laminar ideal gas flow under isothermal condition and there is no phase change between the two phases since the air is regarded to be fully humidified. Then the equations governing the air–water two-phase flow in the channel are the continuity and momentum equation:

Continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho \vec{v}) = 0 \tag{1}$$

Momentum equation

$$\frac{\partial(\rho \vec{v})}{\partial t} + \nabla\cdot(\rho \vec{v}\cdot\vec{v}) = -\nabla P + \mu\nabla\cdot\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right) + \rho\vec{g} + \vec{F}_s \tag{2}$$

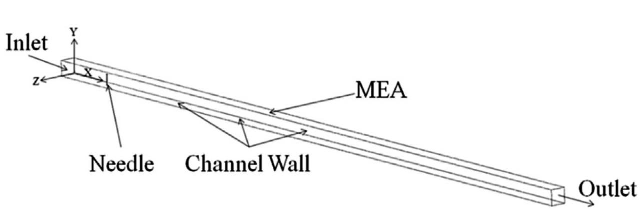

Fig. 1 – Computational domain and the coordinate system considered in the present study.

where ρ (kg m^{-3}) is the density of the air–water mixture, $\vec{v}$ (m s^{-1}) is the velocity vector, P (Pa) is the static pressure, μ (kg m^{-1} s^{-1}) is the viscosity of the mixture, $\vec{g}$ (m s^{-2}) is the gravity, and $\vec{F}_s$ (N m^{-3}) is a source term, which represents the surface tension effect and only acts on the interface between the two phases.

2.3. Volume-of-fluid (VOF) method

The interface between the multiphase fluids is tracked by the VOF method. The volume fraction of one (or more) of the phases in the control volume is resolved by the solution to a continuity equation for the phases expressed as follows:

$$\frac{\partial f_1}{\partial t} + \nabla\cdot(f_1\vec{v}) = 0 \tag{3}$$

where f_1 is the volume fraction of the fluid 1, taken as the liquid water in this study. The volume fraction of the air, f_2, can be obtained based on the relationship shown below:

$$f_1 + f_2 = 1 \tag{4}$$

All the properties are computed in a volume fraction weighted-average manner, e.g.,

$$\mu = \mu_1(1 - f_2) + \mu_2 f_2 \tag{5}$$

for the viscosity. The surface tension effect is considered to be an external force as a source term in the momentum equation shown earlier, and can be written as for the two-phase flow [12]:

$$\vec{F}_s = \sigma\frac{\rho\kappa\nabla f_2}{0.5(\rho_1 + \rho_2)} \tag{6}$$

where σ (N m^{-1}) is the surface tension coefficient between the air and water, and κ is the surface curvature at the interface between the two phases [12]:

$$\kappa = \nabla\cdot\vec{n} = \nabla\left(\vec{n}_w\cos(\theta) + \vec{t}_w\sin(\theta)\right) \tag{7}$$

where $\vec{n}$ is the unit vector normal to the interface between the two phases near the wall, $\vec{n}_w$ is the unit vector normal to the wall, $\vec{t}_w$ is the unit vector tangential to the wall, and θ is the static contact angle at the walls.

2.4. Boundary conditions

The boundary conditions for the governing equations are

$$\text{At the inlet:}\quad v_x = v_{x,0} = \text{const.},\ \ v_y = v_z = 0. \tag{8}$$

$$\text{At the outlet:}\quad \frac{\partial v_x}{\partial x} = 0,\ \ v_y = v_z = 0, P_{out} = \text{const.} \tag{9}$$

$$\text{At the walls:}\quad v_x = v_y = v_z = 0. \tag{10}$$

where v_x, v_y and v_z are, respectively, the velocity component in the x, y, z directions, which are defined in Fig. 1; the velocity at the channel inlet $v_{x,0}$ and the pressure at the channel outlet P_{out} are specified as the flow conditions for the specific flow to be analyzed.