Renewable Energy 93 (2016) 269–279

Contents lists available at ScienceDirect

Renewable Energy

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

The effect of anode bed geometry on the hydraulic behaviour of PEM fuel cells

E.G. Karvelas [a, *], D.G. Koubogiannis [b], A. Hatziapostolou [b], I.E. Sarris [b, 1]

[a] Department of Civil Engineering, University of Thessaly, Pedion Areos, Gr-38334 Volos, Greece
[b] Department of Energy Technology Engineering, Technological & Educational Institute of Athens, Agiou Spiridonos 17, 12210 Athens, Greece

ARTICLE INFO

Article history:
Received 23 July 2015
Received in revised form
26 February 2016
Accepted 27 February 2016
Available online 8 March 2016

Keywords:
Fuel cell geometry
Pressure loss
Residence time
CFD

ABSTRACT

The influence of the anode bed geometry on the hydraulic behaviour of PEM fuel cells is assessed. Three basic geometrical patterns are studied, namely the interdigitated, the parallel and the serpentine ones, in their original, as well as in modified forms of them, in which their angles have been smoothed. Issues concerning the anode flow field of a fuel cell, the influence of the Reynolds number on pressure drop, the mass flowrate distribution along the anode bed channels and the residence time of the fluid inside the fuel cell are investigated. All different geometries are studied by means of 3*D* numerical flow simulations. The results indicate that the pressure drop, flowrate nonuniformity in bed channels and residence time increase as the flow Reynolds number increases. The effect of geometry smoothing on the results is also assessed.

1. Introduction

Today, fuel cells are being developed to power passenger vehicles, commercial buildings, dwellings and even small devices such as laptop computers. Fuel cell systems can be extremely efficient over a large range of sizes (from 1 kW to hundreds of megawatts). Some of them can achieve overall efficiencies of 80% or more when heat production is combined with power generation. Fuel cell systems integrated with hydrogen production and storage can provide fuel for vehicles, energy for heating, cooling and electricity to power human communities. Apart from being clean, these systems offer a unique opportunity for energy independence, highly reliable energy services and economic benefits.

In order to improve fuel cell design and operation, it is necessary to learn more about the mechanisms that can cause performance degradation to it, such as for example, the losses due to mass transfer limitations or fluid dynamics characteristics [1–3] like total pressure drop. The latter significantly affects the power consumption of the fuel cell. The hydraulic behaviour of Proton Exchange Membrane (PEM) fuel cells in terms of pressure loss is complex, because the cell is prone to flooding due to excess liquid water formation. The emergence of the water droplets or water columns in the flow channels influences the gas flow (sometimes shutting it down), making the hydraulic behaviour of the flow channels different from that in the single-phase gas flow. In addition, the pressure drop measurement can be considered and exploited as an in situ diagnosis method to characterize the degree of the flooding in the flow channels.

Studies on the pressure drop characteristics of the fuel cell flow field are quite scarce in the literature. The hydrodynamic behaviour of different anode flow field designs of liquid-fed Direct Methanol (DM) fuel cells was investigated by Krewer et al. [4]. They predicted numerically the transient concentration and the fluid residence time in the cell by means of CFD. After simulating the flow field for various different flowrates, their investigations indicated that there was no change of the flow pattern within a certain flowrate interval. In another approach, a pressure drop model based on the homogeneous two-phase flow theory and mass conservation equation was presented by Argyropoulos et al. [5,6]. In their work, the hydraulic behaviour of an experimental large cell was described and its operational parameters such as flowrate, current density, temperature gradient and pressure losses in the anode and cathode sides of the cell were evaluated. The model was appropriately designed to estimate flow and pumping requirements, sizing of tubes and auxiliary equipment, as well as to calculate the flow distribution through the fuel cell stack manifolds. On the other

* Corresponding author.
E-mail addresses: karvelas@civ.uth.gr (E.G. Karvelas), dkoubog@teiath.gr (D.G. Koubogiannis), ahatzi@teiath.gr (A. Hatziapostolou), sarris@teiath.gr (I.E. Sarris).
[1] URL:http://fluids.et.teiath.gr.

http://dx.doi.org/10.1016/j.renene.2016.02.079
0960-1481/

hand, Yang et al. [7] studied and reported the pressure drop behaviour in the anode flow field of DM fuel cells and investigated two-phase flow pressure drop behaviour in the anode flow field of an in-house fabricated DM fuel cell [8]. Considering the current density as a parameter, they measured the corresponding pressure drop between the inlet and the outlet of the flow channel. Their experimental results showed that, at low current densities, the pressure drop increases with the increase of the current density. However, after reaching a peak at certain current density, the pressure drop starts to decrease for greater values of this parameter. The performance of the fuel cell depends on temperature, methanol concentration and flow rates. When the flow rate of the methanol solution becames sufficiently high, the pressure drop does not depend on the current density. Pressure loss and concentration distribution in the anode have been numerically analysed by Hyun et al. [9]. Parallel, serpentine, parallel serpentine and zigzag type flow passages were used in their analyses. The fuel cell performance was found to be proportional to its pressure drop for all the types of flow passages. However, the best resistance against the concentration polarization of the fuel cell was exhibited by the zigzag type.

Another important flow characteristic is the residence time of the species that are transported by the fluid, which must be sufficiently high in order to react with the catalyst. The optimum bed design should ensure low mixture velocity in the flow compartment and create the conditions for flow circulations to evolve and thus, increase residence times, in an as much as possible homogeneous way to provide a balanced use of the catalyst [10].

The aim of the present work is to investigate the performance of a PEM fuel cell, focussing mainly on the flow field distribution in its anode compartment. Such a study is very helpful for the characterization and the improvement of the dynamic behaviour of the fuel cell. For this purpose, numerical simulations by means of a CFD model using open source software are presented. Distributions of various flow quantities such as velocity field, mass flowrate, pressure distribution and residence time are calculated and discussed in order to comparatively assess the performance of the fuel cell for different geometries. Three different geometrical patterns are implemented for the flow passages of a typical anode fuel cell. In addition, modifications of the original geometrical shapes are adopted, according to which their angles are eliminated and the flow paths are smoothed. In each case, the hydraulic performance of the anode bed flow field is assessed in terms of pressure drop, mass flowrate distribution and residence time. The numerical simulations rely on the single phase incompressible Navier–Stokes equations. In what follows, the aforementioned different geometries are described in detail in Section 2. The numerical simulations by the CFD model are described in Section 3. The results are presented and discussed in Section 4, while Section 5 summarizes conclusions.

2. Description of anode bed geometry

Fuel cells contain dozens or even hundreds of fine channels to distribute the fuel flow over the surface of the cell. The shape, size and pattern of the flow channels can significantly affect its performance. Although a wide variety of flow patterns are employed by research groups and developers, most of them are based on the following three basic archetypes:

1. The parallel geometry: This pattern consists of vertical channels and two horizontal ducts that connect the vertical channels (Fig. 1 (a)). In this configuration, the flow evenly enters each straight channel and exits through the outlet. A significant advantage of the this geometry is the low overall pressure drop between the inlet and the outlet. On the other hand, when the width of the flow passage is relatively large, the flow distribution among the channels may not be uniform. This causes water build-up in certain channel areas, leading to greater mass transfer losses and to corresponding current density decrease. Several fuel cells developers employ this channel type in their portable fuel cell systems [11].
2. The interdigitated geometry: This pattern consists of several horizontal and vertical ducts (Fig. 1(c)). The short length of each duct and the large number of ducts create a great flow complexity. This geometry has the advantage that any pockets of impure gases will be shifted by the swirling processes of the probably unsteady flow of gas through the system. The interdigitated design promotes forced convection of the reactant gases through the gas diffusion layer. Recent research efforts show that this design provides far better management of the liquid phase, leading to improved mass transport [11]. Although the forced convection through the gas diffusion layer leads to significant pressure drop losses, there is evidence that this major disadvantage may be partially overcomed by employing extremely small spacing between the ducts.
3. The serpentine geometry: This is the most common pattern found in fuel cell prototypes (Fig. 1(e)). It consists of one continuous channel that proceeds through a series of alternating 180° turns. This single-serpentine configuration forces all of the reactants through a single, long channel. The main advantage of this pattern lies in the water removal capability, since only one flow path exists, so the liquid is forced to exit the channel. However, in fuel cells of a large area, the serpentine design may lead to a large pressure drop [11].

In order to predict the fuel cell performance in each of the above geometry cases, a number of CFD simulations were performed, corresponding to various inlet Reynolds numbers. Pressure drop values, flowrate distribution among the bed channels and residence times were computed. In addition, in order to assess the effect of the geometry to the fuel cell perfomance, the three basic patterns described before were modified and smoothed by removing their sharp angles and replacing them by smooth curves, as shown in Fig. 1 (b), 1 (d) and 1 (f). The dimensions involved in each geometry are presented in Table 1. In all the above geometries, the inlet and outlet ducts are small circular pipes of diameter d and of height $3d$. The orientation of these two pipes is perpendicular to the plane of the fuel cell anode. In all the parallel and interdigitated cases, the flow inlet is at the tube located at the down-right corner and the flow exit at the tube located at the upper-left corner, while in the case of the serpentine one, the inlet is from the left and the exit to the right (Fig. 1 (e,f)).

3. Numerical simulations

All the numerical simulations to be presented herein were carried out using the open source CFD software OpenFOAM [12] for the solution of the incompressible Navier–Stokes single-phase flow equations in nondimensional form. In each different geometry, an unstructured computational grid of about 4 million cells was generated. All the grids were of mixed-type, consisting of tetrahedral and hexahedral elements. The latter are used at the vicinity of the walls to resolve the viscous boundary layers. A representative view of the grid on the half-height symmetry plane of the anode bed, focused at a junction of the interdigitated case is shown in Fig. 2. The computing time per case was approximately 120 h executing the software in parallel, in 32 cores of a multiprocessor distributed memory computing system using MPI. To enable this, in each case the grid was appropriately divided in 32 subdomains (by