Journal of Power Sources 239 (2013) 596–603

Contents lists available at SciVerse ScienceDirect

Journal of Power Sources

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

Investigations on dynamic water transport characteristics in flow field channels using neutron imaging techniques

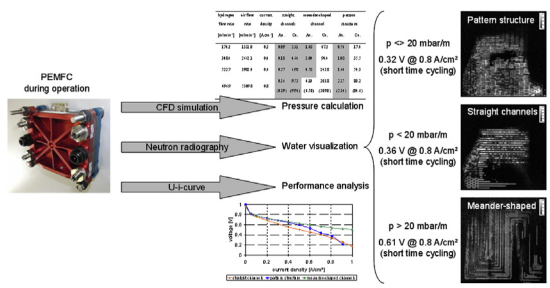
M. Klages [a,*], S. Enz [a], H. Markötter [b], I. Manke [b], N. Kardjilov [b], J. Scholta [a]

[a] Zentrum für Sonnenenergie- und Wasserstoff-Forschung Baden-Württemberg (ZSW), Helmholtzstr. 8, 89081 Ulm, Germany
[b] Helmholtz-Zentrum Berlin für Materialien und Energie GmbH (HZB), Hahn-Meitner-Platz 1, 14109 Berlin, Germany

HIGHLIGHTS

- ► Three basic flow field designs are investigated: pattern structure, meander-shaped channels, straight channels.
- ► Flow fields are compared regarding power and water inventory.
- ► Water thicknesses under channel and under land are separately calculated.
- ► Results of neutron radiography and CFD simulation are combined.
- ► A condensate removal criterion of 20 mbar m^{-1} is proven.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 1 October 2012
Received in revised form
15 January 2013
Accepted 29 January 2013
Available online 4 March 2013

Keywords:
Fuel cell
Water management
Flow field
Pressure drop
Condensate removal
Neutron imaging

ABSTRACT

Handling of water accumulation is still a key issue in fuel cell research. The presented study evaluates the condensate removal capability of three different flow field designs. The designs are compared regarding cell voltage at different current densities using the same operating conditions. The investigated type of meander-shaped channels with a high degree of parallelization shows the best performance with stationary water thickness inside channels throughout the analyzed current densities. To develop and evaluate a condensate removal criterion for fuel cell construction, the pressure drop of each flow field is correlated to the water appearance visualized with neutron radiography. For a further insight, computational fluid dynamics simulation is used to calculate pressure drops present inside the characteristic regions of each flow field. Thus, a characteristic design limit of 20 mbar m^{-1} for meander-shaped channels is proven to ensure the absence of channel blockage. The meander-shaped channels show specific pressure drops around this limit depending on water production and gas supply. The two other analyzed flow fields suffer from higher channel filling rates: the investigated straight channels with less parallelization fill up with time, while the pattern-structured flow field demonstrates gravity as an additional influence on condensate removal.

1. Introduction

Low temperature polymer electrolyte membrane fuel cells (PEMFCs) offer a great potential for a broad range of applications because of their technical maturity and durability [1–3]. Nonetheless, one of the most important challenges for commercialization is an appropriate water management in order to achieve reliable performance and high power output [4]. Good membrane proton conductivity needs a high amount of water present inside the fuel cell, whereas too much water results in flooding of gas diffusion layers (GDLs) and blocking of channels [5] leading to lower performance and degradation effects [6].

* Corresponding author. Tel.: +49 731 9530 209; fax: +49 731 9530 666.
E-mail address: merle.klages@zsw-bw.de (M. Klages).

0378-7753/$ – see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.jpowsour.2013.01.196

In literature, there is more than one parameter to evaluate condensate removal capability. Some work was done correlating droplet movement to the Reynolds number of a channel at particular air flow rates in order to predict water removal [7]. Other dimensionless values, like Laplace number or Weber number, can be used to evaluate multi-phase flow [8]. The driving force of the water movement inside the flow field channels is the pressure difference between gas inlet and outlet. Thus, the pressure drop between inlet and outlet is a powerful parameter to optimize fuel cell design. Chen and Zhou correlated the time dependency of pressure drops to a voltage decrease because of flooding. They propose to measure the frequency in pressure change for the diagnosis in fuel cell operation [9]. Spernjak et al. investigated the pressure of a parallel and a single-serpentine flow field to evaluate the water accumulation. The parallel flow field showed a low pressure difference in combination with an unstable performance, whereas the single-serpentine flow field showed overall a four times lower water content and higher performance [10].

One developmental issue that remained open so far is the combination of low pressure drop on one hand and good condensate removal capability on the other hand. In previous research activities, the condensate removal capability of flow field channels was investigated both ex-situ for single straight channels [6] and in-situ for parallel or serpentine flow fields [11]. The ex-situ investigations were performed using a graphite composite material covered and sealed by a transparent endplate. Thus, the influence of channel geometry on the water transport ability could be seen. Since this procedure is not possible, if the channel is covered with a GDL, other methods have to be used to characterize the water removal characteristics. In Ref. [11], in-situ condensate removal behavior was investigated by indirect methods like pressure drop changes and gas utilization curves.

Before performing utilization curves, an overall condensate removal criterion using a simulation can be calculated. A simulation of the flow field before the actual manufacturing is often used to predict fuel cell performance simultaneously reducing development costs. There are different custom-designed tools and different models. The software used within the presented study is ANSYS® FLUENT®. Further studies using a three-dimensional simulation are described e.g. in Refs. [12–14].

In the past, imaging methods based on X-rays and neutrons were frequently used for the visualization of water distribution in fuel cells under operation [15–17,37]. The purpose of the presented study is to extend recent fuel cell investigations by in-situ diagnostics during fuel cell operation which can be performed using neutron radiography. Several investigations using neutron radiography and tomography on water transport in flow fields proved the applicability of this method [18–20]. In this study a detailed examination of the condensate removal behavior with respect to pressure drop is performed.

2. Experimental

Three basic structures in flow field design with different magnitudes of pressure drop were investigated regarding power and condensate removal: pattern flow field, meander-shaped and straight channels. Each channel had a rectangular cross section. Fig. 2 shows the design sheets of the flow fields. The same design was used on both anode and cathode side in order to evaluate both sides of a fuel cell regarding condensate removal capability at the same time and in the same way. As the fuel cells were operated in counter-wise gas direction, anode and cathode flow fields were horizontally flipped. The meanders and the pattern structure had a high degree of parallelization. The cross-sectional area for these flow fields was 0.36 mm^2 and 1.2 mm^2 for the parallel flow field. The width of the straight channels was more than doubled compared to the other two designs. This flow field had 40 straight channels. Table 1 summarizes the characteristics of the investigated flow field designs used within this study. The specified designs were investigated with neutron radiography (2.1) and a simulation was performed to get a further insight (2.2).

2.1. Neutron radiography procedure

From previous studies neutron radiography turned out to be very suitable to detect and quantify water clusters while allowing an investigation of the whole active area of 100 cm^2 [21–23]. Neutron imaging was carried out at the CONRAD/V7 facility of the Helmholtz-Zentrum Berlin (BER II research reactor). Experimental details of the setups for the neutron imaging technique used for this study can be found in Refs. [24–26]. To analyze the dynamics of water evolution and transport, the exposure time and the readout time were minimized down to 3 s and 2 s, respectively. As time and spatial resolution conflict, the radiographs were taken with 100 μm per pixel using a ^{6}LiZnS scintillator with a thickness of 200 μm. As the water inside the gas channels is the main object of investigation, this pixel size is sufficient. With a longer exposure time, a spatial resolution down to 25 μm per pixel is possible in neutron imaging [22]. The water content of a fuel cell is highly influenced by the amount of water production through the electrochemical reaction as well as the condensate removal capability. To analyze the dynamic water characteristics inside the flow field, current density and thus gas flow rate at constant utilization was varied with short holding times of 3 min. Within this time, current density was varied within the range of 0.2 A cm^{-2} and 0.8 A cm^{-2}. These current densities correspond to a high and a low voltage level of the investigated fuel cells. The operating conditions are shown in Table 2. All fuel cells contained the same GDL (Sigracet® SGL 10BC) as well as the same membrane electrode assembly (GORE™ PRIMEA® 5761).

2.2. Parameters for CFD simulation

Neutron radiography can be used to identify critical flow rates causing blockage of channels as well as a further investigation of the shape of water accumulations, like slug or droplet formation. The different shapes are described e.g. in Ref. [27]. For the development and optimization of fuel cells neutron radiography in combination with computational fluid dynamics (CFD) simulation are tools of growing interest. Simulation can be additionally consulted if operating parameters cannot be measured, like pressure distributions throughout single channels, or are within tolerance of measurement instruments. On the other hand, neutron radiography helps to verify simulation results predicting e.g. the condensation of water. Within this study, for mechanisms that cannot be studied under real in-situ conditions ANSYS® FLUENT® (version 13.0) was used. For the performed half cell simulations, anode as well as cathode geometry consisting of flow field, GDL

Table 1
Characteristics of the flow field designs.

	Meander	Straight	Pattern
Land width	0.8 mm	1.0 mm	5.0/1.0 mm
Channel width	0.6 mm	1.5 mm	0.6 mm
Channel depth	0.6 mm	0.8 mm	0.6 mm
Number of channels	23	40	63 horizontal, 10 vertical
Cross-sectional area	0.36 mm^2	1.2 mm^2	0.36 mm^2 horizontal, 0.6 mm^2 vertical
Specific channel length	301.7 mm	61 mm	61 mm