Simulation of Flow in a Rectangular Channel of a PEM Fuel Cell

A. L. R. Paulino[1], E. Robalinho[2], E. F. Cunha[1], M. Linardi[1], R. R. Passos[3], E. I. Santiago[1]

[1]Instituto de Pesquisas Energéticas e Nucleares – IPEN/CNEN, São Paulo, SP, Brazil

[2]Instituto de Pesquisas Energéticas e Nucleares – IPEN/CNEN, São Paulo, SP, Brazil; and Instituto Federal do Rio Grande do Sul – IFRS, Porto Alegre, RS, Brazil

[3]Universidade Federal do Amazonas - UFAM, Manaus, AM, Brazil

Abstract

Fuel cells are electrochemical devices that convert the chemical energy stored in a fuel (typically hydrogen) directly into electricity. One particularly interesting type of fuel cell is the Proton exchange membrane fuel cell (PEMFC), that operates at low temperatures (50-100 °C), and are being researched for various applications, such as transportation and portable power generation.

The performance of a PEMFC depends on several factors, including the flow inside gas channels. Low gas velocity inside the channel can make water management more difficult, possibly causing water to accumulate and condense in certain regions of the cell, which is detrimental to fuel cell operation. Conversely, high flow velocity is useful to avoid water accumulation and channel clogging, thus maintaining cell performance.

In the present work, the software COMSOL Multiphysics® was used to visualize and describe gas flow inside the channel of a PEMFC operating with humidified hydrogen and oxygen. The model built represents a PEMFC with a single rectangular channel, based on an actual fuel cell under tests on the laboratory (Fig. 1-A). The physics used were Fluid Flow, Mass Transport and Electrochemistry, for the corresponding parts of the cell. Hydrogen and oxygen, both humidified, were fed into the cell, and the system was considered to be isothermal. The mesh built for this model was structured, with all elements consisting on orthogonal hexahedrons (Fig. 1-B).

Variables were measured in a cross section along the channel, at the midpoint of its width (Fig. 2), as this is the symmetry plane, the region that is less influenced by edge effects. Figure 3 shows the distribution of the velocity field in the cross section, along with the velocity profiles in some regions. It was observed that the velocity profiles are parabolic, as expected for laminar flow in closed ducts. Flow occurs also in the porous layer, but its velocity is negligible, particularly when compared with that in the channels, where flow resistance is minimal. It was also noticed that velocity in the anode channel decreases with distance, while it increases in the cathode. This is due to the reactions at the electrodes: hydrogen is consumed in the anode,

causing the flow rate to lower, while in the cathode there is oxygen consumption and water production (in a 1:2 volumetric ratio). The relation between reactions and the variations in flow rates is due to reaction stoichiometry and was observed in Figure 4 at different cell potentials.

It was verified that gas velocity changes along the channel and, decreasing in the anode and increasing in the cathode. This is interesting because, while water is being produced in the cathode, it can be better removed from the cell, as higher prevent water accumulation inside the channels. In the anode, velocity decrease is not a serious issue, because only intake water must be removed.

Reference

1. X.-D. WANG et al. Numerical analysis on performances of polymer electrolyte membrane cells with various cathode flow channel geometries. Int. J. Hydrogen Energy, v. 37 p. 15778-15786 (2012).

2. G. DOUBEK et al. Application of CFD techniques in the modelling and simulation of PBI PEMFC. Fuel Cells, v.11, p. 764-774, (2011)
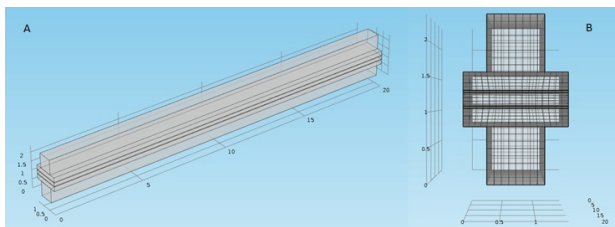
Figures used in the abstract

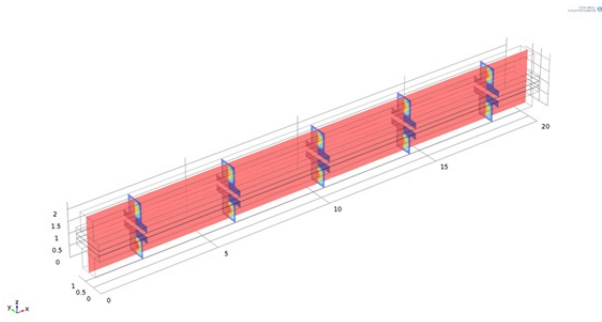**Figure 1**: Geometry and mesh used in the fuel cell model

Figure 2: Midsection zx (red) where variables were calculated
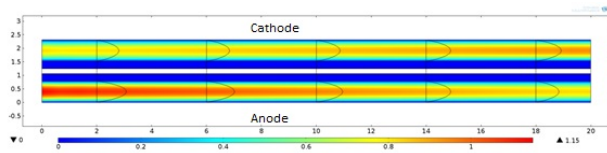

Figure 3: Velocity field and velocity profiles in the midsection of the channel
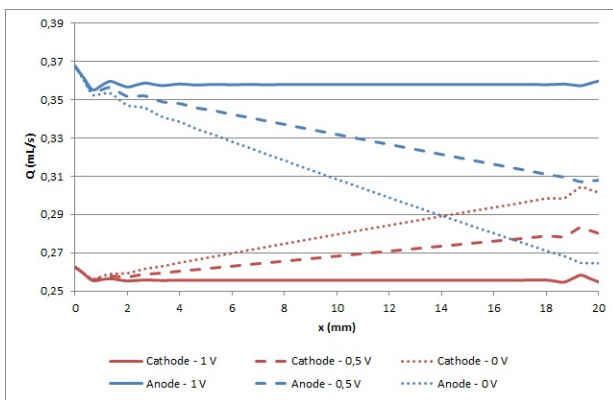

Figure 4: Gas flow along the channel at different cell potentials