Flow field design for high-pressure PEM electrolysis cells

Olesen, Anders Christian; Kær, Søren Knudsen

[Link to publication from Aalborg University](#)

Abstract No (don't use) – Preference: oral

Flow field design for high-pressure PEM electrolysis cells

Anders Christian Olesen and Søren Knudsen Kær

Department of Energy Technology, Aalborg University, Pontoppidanstræde 111, 9220 Aalborg East

aco@et.aau.dk

Summary. To study the electrochemical behaviour of full-scale, flow field designs of high-pressure PEM electrolysis cells, a computational fluid dynamics model has been developed. For a set of specific operating conditions, different flow fields are investigated in terms of their ability to handle gas and heat management.

Abstract. With the increasing interest in producing hydrogen through water electrolysis, the importance of understanding the transport phenomena governing its operation increases. To ensure optimal operating conditions for PEM electrolysis, it is particularly important to understand how the liquid feed-water distributes. Water not only serves as a reactant, it also aids in cooling due to its high specific heat capacity. The movement of liquid water at the anode is difficult to model, since it is highly coupled to the formation of gas bubbles. To capture the complex two-phase flow behaviour that takes place within micro-channels and porous media, our research group has developed an Euler-Euler model in the computational fluid dynamics modelling framework of ANSYS CFX. In addition to two-phase flow, the model accounts for turbulence, species transport in the gas phase, heat transport in all three phases (i.e. solid, gas and liquid), as well as charge transport of electrons and ions. Our recent improvements have focused on the models ability to account for phase change and electrochemistry as well as the modelling of two-phase flow regimes.

For comparison, an interdigitated and parallel channel flow field will be investigated with regard to how well they distribute gas and heat. To obtain a high durability when operating at high current densities, it is essential that liquid water and temperature is evenly distribution over the entire active surface area. In Figure 1, a contour plot of the temperature distribution within an interdigitated flow field is presented as an example. The axis-symmetric simulation shows that the formation of hot spots in the order of 8°C above the inlet temperature can occur for a current density of 3 A/cm^2 and a water stoichiometry of 350. These spots are primarily located underneath the outlet channels. To avoid them, an in-depth design and simulation study is necessary.
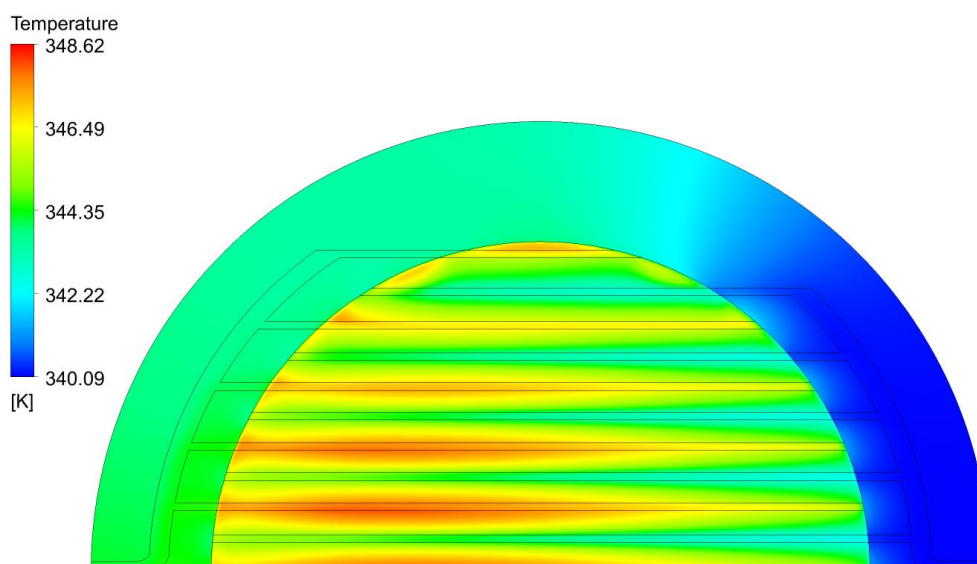

Figure 1: Contour plot of the temperature distribution for an interdigitated flow field at a current density of 3 A/cm^2 and a water stoichiometry of 350.