Int. J. Electrochem. Sci., 13 **(2018)** 2080 – 2089

International Journal of
ELECTROCHEMICAL
SCIENCE
www.electrochemsci.org

Short Communication

Finite element analysis of PEMFC assembling based on ANSYS

Guilin Hu[*], *Xiaojun Wu, Yange Suo, Yuzhen Xia, Yousheng Xu, Zhiguo Zhang*

School of Mechanical and Automotive Engineering, Zhejiang University of Science and Technology, Hangzhou, 310023, P. R. China
[*]E-mail: enehgl@163.com

doi: 10.20964/2018.02.60

Received: 17 October 2017 / *Accepted:* 5 December 2017 / *Published:* 28 December 2017

Proton exchange membrane fuel cells (PEMFC) are composed of end plates, flow fields, the membrane electrode assembly (MEA) and sealing components. Before assembling a fuel cell stack, obtaining the distribution of stress and deformation of PEMFC components by simulating the assembly process can enhance cell performance and reduce the risk of gas leakage, as well as decrease production costs. To provide a good reference datum for the actual assembly, a real fuel cell stack was used in research. Finite element analysis of PEMFC assembling, based on software ANSYS, was conducted to better learn the stress and deformation during assembly. The effect of varying the clamping loads during assembly was investigated for different forces (300N, 400N, 500N and 600N) and for different displacements (0.1 mm, 0.2 mm and 0.3 mm). For the single cell, a load of 400N and 0.2 mm displacement are appropriate. They provide good reference datum for real assembly process. Further, the distribution of stress and deformation in the core component MEA of single cell and 10-cell stack under different loads were obtained, providing good reference datum for the actual stack assembly. Results from stack simulation indicate that the distribution of stress and deformation in different cell is similar in trend, with slight differences.

Keywords: Proton exchange membrane fuel cell; assembly; simulation; ANSYS

FULL TEXT