Feasible Analysis of Pulsating Heat Pipe Applied to Proton Exchange Membrane Fuel Cell

Abstract—Proton exchange membrane fuel cell (PEMFC), as a new power generation method with high efficiency, cleanliness and no pollution, has broad development prospects, but its thermal management problems need to be solved urgently. This paper uses ANSYS Fluent software, based on the VOF model and PHMPC model, uses the unsteady state method to simulate PEMFC without pulsating heat pipe and PEMFC with pulsating heat pipe, respectively, to study the temperature distribution law inside PEMFC. The results show that the temperature at PEMFC channel is the highest, about 68℃. Compared with PEMFC without pulsating heat pipe, it is found that the temperature of PEMFC channel decreases by about 12℃. Therefore, the pulsating heat pipe can reduce the internal working temperature of PEMFC. The simulation results can provide a theoretical basis for future experiments.

Key words—proton exchange membrane fuel cell, thermal management, temperature distribution, numerical simulation

I. INTRODUCTION

Energy shortage and environmental pollution are two major challenges facing human society. Efficient use of clean energy is an important guarantee for the sustainable development of human society. Among them, hydrogen fuel cells, especially proton exchange membrane fuel cells (PEMFC), are favored because of their high efficiency, cleanliness and pollution-free characteristics. However, the thermal management problem of PEMFC needs to be solved urgently.

Proton exchange membrane fuel cell (PEMFC) is a three-dimensional, multi-phase, multi-scale and dynamic complex system. The flow of reactants and products in the fuel cell and the chemical reaction in the electrode coexist and are coupled with each other. Therefore, it is difficult to obtain the distribution laws of temperature, material concentration and current density inside the battery by experimental means, and the cost is very high. In order to better understand the temperature distribution in PEMFC and optimize the PEMFC thermal management system, the numerical simulation of PEMFC can be used.

II. RESEARCH STATUS OF PEMFC NUMERICAL SIMULATION

Studying the characteristics of fuel cell through numerical simulation can not only reduce the cost of experiment, but also greatly shorten the design cycle of the battery. Therefore, the numerical simulation of the flow, heat and mass transfer process in PEMFC has extremely important engineering and academic significance. Many scholars at home and abroad have studied the numerical simulation of PEMFC.

Wang et al. studied the influence of temperature on the performance of PEM fuel cell through experiments, and used simulation software to simulate and analyze the influence of temperature and gas on the temperature of each component in the cell. The research results have a certain reference value for the design and development of PEM fuel cells.

Hu et al [2] discussed the mathematical model of proton exchange membrane fuel cell in the form of one dimension, two dimensions and three dimensions respectively, and concluded that the mathematical model should develop in the direction of three-dimensional, non isotherm and two-phase flow, and gradually develop the battery stack model. They discussed various humidification methods and the general structure and principle of the cooling system.

Wu et al [3] divided the mathematical model of PEMFC into four scales: proton exchange membrane: electrode; Single cell: stack. The research object, scope and development process of these four scale mathematical models are described respectively.
III. MATHEMATICAL MODEL

The mathematical model is used to describe the heat transfer, mass transfer and chemical reaction processes in PEMFC. The purpose is to theoretically explain the heat transfer, mass transfer and chemical reaction processes in the fuel cell and their effects on the performance of the fuel cell, thus providing theoretical guidance for the layout of the fuel cell structure thermal management system and the selection of the flow channel structure.

The mathematical models used in this numerical simulation include energy conservation equation, fluid transfer equation, momentum conservation equation, component conservation equation, charge conservation equation and net migration flux equation.

1. Energy conservation equation:

\[
\frac{\partial (\rho c_T T)}{\partial t} + \nabla \cdot (\rho c_T V T) = \nabla \cdot (k \nabla T) + \dot{q} + \dot{C}_T
\]

In the formula, \(T\) indicates the temperature, \(\rho\) represents the density and \(c_T\) denotes the specific heat, respectively, and \(k\) and \(\dot{q}\) represent the thermal conductivity of the electrolyte.

2. Fluid transfer equation:

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon V) = 0
\]

In the formula, \(\varepsilon\) and \(V\) indicate the current density and the voltage, respectively.

3. Momentum conservation equation:

\[
\frac{\partial (\rho \mu V)}{\partial t} + \nabla \cdot (\rho \mu V V) = -\nabla \cdot \tau + \nabla q + \mu \nabla \cdot (\mu \nabla V) + \rho G
\]

In the formula, \(\varepsilon\) indicates the porous medium porosity, \(\rho\) and \(\mu\) represent the pressure and the fluid flow velocity vector in the porous medium respectively, \(\tau\) and \(G\) represent the stress tensor and the fluid mass force respectively, and \(k\) and \(\mu\) represent the fluid permeability and fluid viscosity in the porous medium respectively.

4. Charge conservation equation:

\[
\frac{\partial (\rho_n \varepsilon_n)}{\partial t} + \nabla \cdot (\rho_n \varepsilon_n V) = \frac{n_m}{F} - D_n V \varepsilon_n
\]

In the formula, \(C\) represents the specific heat, \(\varepsilon\) is the utilization rate; the internal temperature of the battery is optimal when the channel width is equal to 3mm.

5. Net migration flux equation:

\[
\frac{\partial (\rho_n \varepsilon_n)}{\partial t} + \nabla \cdot (\rho_n \varepsilon_n V) = \nabla \cdot (n_m \varepsilon_n V) + \nabla \cdot (q \varepsilon_n V) + \dot{C}_n
\]

In the formula, \(\varepsilon\) and \(V\) indicate the current density and the voltage, respectively.
B. Numerical calculation method

The calculation method used in this simulation is the Finite control volume method, which discretizes the nonlinear control equation and uses the SIMPLE operation scheme to solve it through iteration.

As PEMFC and pulsating heat pipe are affected by various external factors in the actual work process, the following assumptions are made to reduce the complexity of calculation:

This stimulation applies the following assumptions:
1. The whole device operates under transient conditions;
2. The fuel cell reaction gas is an ideal gas and an incompressible gas; the thermal conductivity of the materials used in the fuel cell is constant;
3. Uniform consumption and heat release are adopted in the whole model reaction area, and natural convection and radiation heat transfer inside the fuel cell and between the cell and the pulsating heat pipe are not considered;
4. The gas in the pulsating heat pipe is ideal gas and saturated steam; All liquids are incompressible.

V. ANALYSIS OF SIMULATION RESULTS

Figure 2 is the current density distribution diagram of PEMFC section. It is observed that the highest current density of PEMFC is 8540 (A/m²) near the transverse and longitudinal flow channels, and the current density gradually decreases from the flow channel outward. This is because PEMFC chemical reaction mainly occurs in the membrane electrode assembly (MEA), while the flow channel is tightly attached to the membrane electrode assembly.

Figure 3 shows the distribution of hydrogen mole concentration at the center of PEMFC catalytic layer, Figure 3(1) shows the distribution of hydrogen mole concentration at the center of anode catalytic layer, and Figure 3(2) shows the distribution of oxygen mole concentration at the center of cathode catalytic layer. The abscissa is the distance from the anode channel outlet. It can be seen from the figure that the molar concentrations of hydrogen and oxygen in the catalytic layer gradually increase from the outlet of the channel to the inlet of the channel along the direction of the channel. This is because the water generated by the chemical reaction accumulates at the end of the channel. The increase of the molar concentration of water in the diffusion layer will affect the diffusion of hydrogen from the channel to the diffusion layer and catalytic layer. With the gradual reduction of the depth of the channel, the molar concentration of oxygen in the catalytic layer also shows a rising trend.

Figure 4 shows the cloud diagram of gas-liquid two-phase volume integral number under the stable operation condition of pulsating heat pipe. From the distribution of flow pattern in the pipe, it can be seen that there is an obvious staggered distribution of gas column and liquid column in the pulsating heat pipe. At this time, the flow pattern is mainly a plunger flow. As the evaporation end continues to receive heat, the wall temperature rises, forming a vaporization core, and the small bubbles generated are heated, expanded, and contacted with the bubbles on the opposite wall to form a gas cold. When the liquid phase at the outermost side of the pulsating heat pipe rises to the condensing end, the liquid cold air plug condenses and exchanges heat, and the bubbles break, causing pressure changes. At the same time, with the evaporation end heating, the liquid film evaporates, resulting in pressure difference between the left and right pipes. When the differential pressure in one side of the pipe is not enough to support the liquid plug to continue rising after offsetting the liquid gravity, flow resistance and capillary resistance, the liquid phase of the working medium starts to stop or run in reverse, pushing the liquid phase on the other side to run towards the condensing end. Finally, a counterclockwise unidirectional cycle is formed. It can be seen that the pulsating heat pipe has been working normally, and there is oscillating flow in the pipe.
Figure 5 and Figure 6 respectively show the temperature distribution of PEMFC without pulsating heat pipe and PEMFC with pulsating heat pipe. It can be seen from the figure that the temperature distribution of PEMFC is generally uniform, and the temperature change is limited to a small range, because natural convection is not considered in this simulation. The highest temperature in both figures appears in the flow channel, the temperature distribution gradually decreases from the inside to the outside, the temperature outside the bipolar plate is the lowest, the temperature change of the cathode catalytic layer is more obvious than that of the anode, and the temperature change under the flow channel is more obvious than that under the shore. The maximum temperature of PEMFC without pulsating heat pipe is 81.42° C, and the maximum temperature of PEMFC with pulsating heat pipe is 68.33° C.

VI. CONCLUSION

Proton exchange membrane fuel cell (PEMFC) has the characteristics of high efficiency, clean, pollution-free, and has broad prospects for development. The thermal management of PEMFC has become an urgent problem in the application research of PEMFC. In this paper, the cooling effect of pulsating heat pipe on PEMFC is studied by numerical simulation method. The results show that the pulsating heat pipe can operate normally in PEMFC and reduce the maximum temperature of PEMFC by about 12° C. Therefore, the application of pulsating heat pipe to PEMFC is feasible, which provides a theoretical basis for the experimental research of PEMFC heat management system in the future.

REFERENCES

[1] Yicheng Wang, Experiment and Numerical Simulation of heat and mass transfer of temperature effect on PEM Fuel Cell performance [D], Shenyang Architectural University.

[2] Mingruo Hu, Xujian Zhu, Anzhong Gu, Shi Yumei, Hydrothermal management of proton exchange membrane fuel cells [J], Battery, 2003 (04): 258-260.

[3] Jinfeng Wu, Ming Hou, Baolian Yi, Huamin Zhang Research and Development of Proton Exchange Membrane Fuel Cell and System Modeling [J], Power Technology, 2004 (01): 56-63.

[4] YING W, YANG T, LEE W, J, Chang Three-dimensional simulation for effects of bed discordance on flow dynamic at Y-shaped open channel confluences [J], Journal of Hydrodynamics, 2007(05): 587-593.

[5] Mingbiao Shao Research on Numerical Simulation Method of Steady State Temperature Field of High Power Fuel Cell [J], Journal of Wuhu Vocational and Technical College, 2020(03): 43-46+58.

[6] Young G, Lee W, Park G, Yang T, Kim C. Effect of channel and rib widths of flow field plates on the performance of a PEMFC [J], International Journal of Hydrogen Energy, 2005(30): 1363-1366.

[7] Lizhong Ma, Shuanglin Shen, Fei Jia, Hongtian Liu, Liejin Guo Numerical Simulation of Water Distribution and Temperature [J], Journal of Engineering Thermoplastics. 2010, 31(12): 2051-2056.