IJE TRANSACTIONS B: Applications Vol. 31, No. 5, (May 2018) 812-819

International Journal of Engineering

Journal Homepage: www.ije.ir

Enhancement of the Cooling System Performance of the Proton-exchange Membrane Fuel Cell by Baffle-restricted Coolant Flow Channels

E. Afshari*, S. P. Hosseini, M. Mosharaf-Dehkordi

Department of Mechanical Engineering, Faculty of Engineering, University of Isfahan, Isfahan, Iran

PAPER INFO

Paper history:
Received 01August2017
Received in revised form 24April 2018
Accepted 17January 2018

Keywords:
Proton-exchange Membrane Fuel Cell
Cooling Flow Field
Baffle
Temperature Uniformity
Surface Temperature
Pressure Drop

ABSTRACT

The performance of proton-exchange membrane fuel cell cooling system using coolant flow channels enhanced with baffles was numerically investigated. To do this, the maximum temperature of the cooling plate, temperature uniformity and also pressure drop along the flow channels were compared for different cases associated with number of baffles and their dimensions inside the channels. The governing equations by the finite-volume approach in three dimensions were solved. Numerical results indicate that the baffle-restricted cooling flow channels, generally improved the performance of the fuel cell in such a way that a reduced maximum temperature of the cell and a better temperature uniformity in the cooling plates were determined. As the pressure drop increases by incorporating the baffles inside the coolant flow channels, one needs to compromise between the improvement of cooling system performance and the total pressure drop.

***doi**: 10.5829/ije.2018.31.05b.17*

1. INTRODUCTION[1]

The chemical energy of fuel is directly be converted into the electricity through a device named fuel cell. In the fuel cells, oxygen and fuel are combined to generate electricity along with heat and water as the by-products [1]. Compared to the internal combustion engines, the fuel cells have higher efficiency as they directly convert the chemical energy to electricity, without any combustion process [2]. However, there are several issues associated with the proton-exchange membrane fuel cell (PEMFC) technology, particularly the heat dissipation process. As the PEMFC performance highly depends on its operation temperature; the heat removals and thermal management of the fuel cell are very important [3, 4]. There are several ways for improving thermal performance of mechanical equipment [5-10] and cooling system of a PEM fuel cell, such as heat spreaders, air flow or liquid cooling, and the cooling associated with the phase change such as evaporation[11, 12]. Among these approaches, the water-cooled system is more of interest as it provides a better temperature distribution and needs smaller cooling channels, when compared to the air flow system.

As one of the most important factors of the water-cooled systems, several studies focused on finding an optimized configuration of the flow field geometry. The purpose of the optimization process is to design a flow field capable of maintaining the operating temperature in a certain limit in various voltage domains as well as providing a nearly uniform temperature distribution in the cell. Also, the flow field geometry should have a reasonable pressure drop along the flow channels.

The fuel cell cooling system performance has been assessed and also improved by various researchers for different flow channel configurations [13-16]. Afshari et al. [17, 18] have developed a novel design of water flow in a number of zigzag channels. They have investigated the effect of such configuration on the maximum and average temperature, and the index of uniform temperature of the fuel cell. Also, they investigated heat transfer in cooling plates and the capabilities of parallel, serpentine, and one metal foam porous media field, and compared these models based on the maximum surface temperature, uniformity of temperature and the pressure

*Corresponding Author's Email:e.afshari@eng.ui.ac.ir(E. Afshari)

Please cite this article as: E. Afshari, S. P. Hosseini, M. Mosharaf-Dehkordi, Enhancement of the Cooling System Performance of the Proton-exchange Membrane Fuel Cell by Baffle-restricted coolant flow channels,International Journal of Engineering (IJE),IJE TRANSACTIONS B: Applications Vol. 31, No. 5, (May 2018) 812-819

drop. Liso et al. [19] have focused on modeling a control-based water-cooling system in the fuel cells. The objective was to check the temperature changes during a rapid load variation. Saygili et al. [20] have conducted a research on a 3kW fuel cell with a water-cooled system including circulating channels, pumping system and a radiator integrated with a cooling fan. This research has not been limited to those topics introduced above. Providing a control system on water circulation with energy-efficient pump and a fan has also been considered in this work. An evaporative-cooled closed cycle is another option for cooling the fuel cells which is investigated by Fly and Thring [21]. Castelain et al. [22] have introduced several chaotic geometries to improve the thermal performance of the PEMFCs. They have studied the heat transfer procedure in chaotic geometries, and compared the results with the straight tubes.

Although several designs and various flow fields have been proposed for the coolant circulation in the PEMFC, to the author's knowledge, no effort has been devoted to study the performance of cooling channels enhanced with the baffles in the cooling system of PEMFCs; which is the main objective of the present work. To do this, the effects of number of baffles and their geometrical properties (including their heights and thicknesses) on the cooling system performance were studied, considering the following parameters:

1. Minimum, maximum and average temperature in the cooling plates,
2. Uniform temperature distribution in the cooling plates,
3. The least pressure drop of the cooling fluid.

Based on the mentioned parameters, the suitable values for the number of baffles and their geometrical properties were selected to improve the performance of the cooling system.

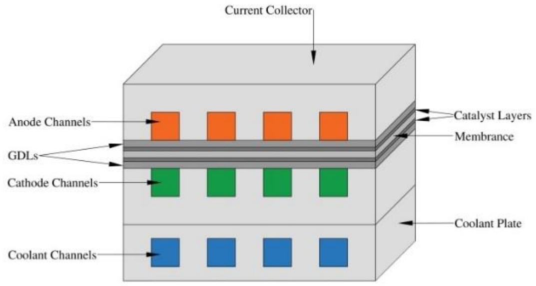
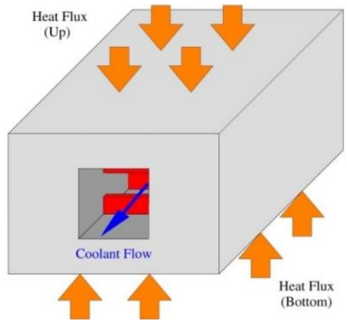
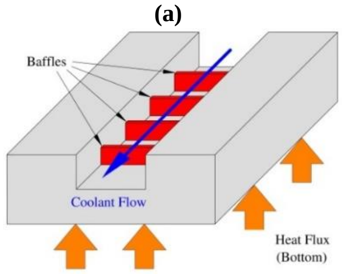


Figure 1. A PEM fuel cell with its major components and cooling channels



(a)



(b)

Figure 2. Geometrical model of the cooling plate **(a)** channel with embedded baffles and associated thermal boundary conditions, **(b)** One half of the coolant channel

2. PHYSICAL, NUMERICAL MODEL AND INPUT PARAMETERS

As a single cell cannot usually provide the required power [23, 24]; A fuel cell stack consisting of a series of cells is used to supply the total required power. The generated heat, as a by-product of the fuel cell, should be taken out from the cell stack [25]. This can be performed by a coolant, like water flow through several cooling channels of each individual cell.

An individual cell with its corresponding cooling channels embedded in the cell is shown in Figure 1. Here, the coolant channels are equipped by a number of baffles increasing the heat transfer rate from the cell (Figures 2a and b). It is assumed that the all baffles inside a coolant channel have the same size. The performance of the cooling plate is examined for both general cases with and without existence of the baffles.

To reduce the computational cost of the simulation process, one half of the symmetric coolant channel is considered (Figure 2b).

The coolant channels characteristics, fluid properties, and the cell operating condition are presented in Table 1. Here, the inlet temperature to the cooling plate is fixed at 40°C. In addition, the generated heat by fuel cell is considered as a heat flux imposed on the cooling plate surface. As the fuel cell operates at steady state condition, the average heat flux value at the surface of each cooling plate, q, is considered, which is given by $q=Q/2A$. Here, Q is the total heat generated by the fuel cell and A is the cooling plate surface area. The calculation for the total heat generation and the average value of heat flux is straightforward. The local heat generation in the fuel cell has usually non-uniform distribution due to the interdependence of the local

TABLE 1. The information on the half geometry cooling channels simulation

Parameter	Values
Channel width	2 mm
Distance between two channels	2 mm
Channel depth	2 mm
Hydraulic diameter of the channel	2 mm
Number of baffles	1 , 2 , 4 , 9 , 11 , 14 , 19
Height of baffles	0.2 , 0.3 ,0.4 , … ,0.8 mm
Thickness of baffles	0.3 ,0.5, 1, 1.2, 1.5 mm
Density of plate	2250 kgm^{-3}
Specific heat of plate	690 $Jkg^{-1}K^{-1}$
Thermal conductivity of plate	24 $Wm^{-1}K^{-1}$
Density of water	992.2 kgm^{-3}
Specific heat of water	4179 $Jkg^{-1}K^{-1}$
Thermal conductivity of water	0.62 $Wm^{-1}K^{-1}$
Thermal flux	5000 Wm^{-2}
Coolant fluid temperature at inlet	313 K
Flow rate at channel inlet	3.234×10^{-4} m^3s^{-1}

current density, temperature, reactant concentration and water content [18]. Assuming constant heat flux at coolant surfaces not only facilitates the simulation process but also provides essential information about cooling system performance in practice. Beside the constant heat flux assumption, the nominal voltage of each cell of the fuel cell is considered to be V_i=0.6 V.

Here, the effect of the number of baffles and also their dimensions, including the height and thickness, on the cooling performance of the cell is investigated independently. For all cases, it is assumed that all baffles have the same dimensions with a constant width equals to the coolant channel width. Based on the described assumptions, the maximum and average temperature, and its standard deviation in the cooling plate were reported and compared for the different cases associated with number and dimension of the baffles inside the cooling channel.

3. GOVERNING EQUATION

Based on the flow velocity and the corresponding Reynolds number (Re=330), the laminar flow is assumed for the fluid flow inside the cooling channel. For a Newtonian incompressible cooling fluid flow, the mass, momentum, and energy conservation equations at steady state condition can be written as:

$$\frac{\partial u_j}{\partial x_j} = 0, \tag{1}$$

$$\rho\left(u_j \frac{\partial u_i}{\partial x_j}\right) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial u_i}{\partial x_j}\right), \tag{2}$$

$$\rho C_p\left(u_j \frac{\partial T}{\partial x_j}\right) = \frac{\partial}{\partial x_j}\left(k \frac{\partial T}{\partial x_j}\right), \tag{3}$$

where, the parameters ρ and μ represent the density and viscosity of the cooling fluid, respectively. In addition, C_p is the specific heat coefficient, and k is the thermal conductivity of cooling fluid. As the nature of the fluid flow is closed to steady state condition, the transient terms has been dropped from Eqs. (1) to (3). In the solid regions, u_j is equal to zero in Eq. (3), only conductive heat transfer is performed [26].

4. NUMERICAL SOLUTION

The governing equations corresponding to the fluid flow and heat transfer in the channels can be numerically solved by a proper CFD method. Here, a computer program is developed based on finite volume method to solve the mass, momentum and energy equations. The domains are discretized into finite sets of computational cells in such a way that a relatively fine mesh has been utilized for regions nearthe baffles inside the coolant channel (Figure 3). The size of grid cells before/after the baffles is at least three/seven times smaller than the width of the baffles. In regions far from the baffles, coarser grid cells are used to reduce the total number of computational cells. The grid independency of the results has been checked for the reported results. Here, the numbers of cells range from 200,000 in the straight channels to 1,700,000 cells in the geometry with the total number of 19 baffles. For instance, a grid used for the case with nine baffles with 0.5mm thickness and height has 1,300,000.

5. RESULTS AND DISCUSSION

5. 1. Validation of Results The numerical results, including pressure difference, Darcy friction factor, and Nusselt number for the straight channels can be compared with the results obtained by the analytical solution. Pressure drop in the fluid can be calculated by following equation:

$$\Delta P = \rho g z + \frac{\rho f L u_m^2}{2D_h}, \tag{4}$$

where, g and z are the gravitational acceleration and the height difference between the inlet and outlet of the channel. The parameter f is Darcy friction factor, and L

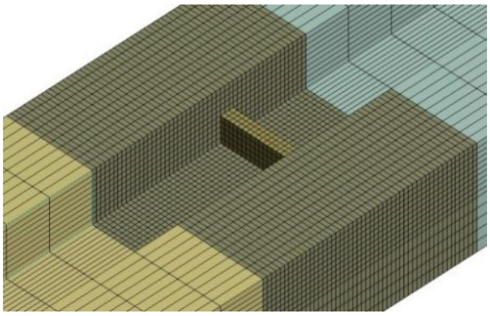


Figure 3. The computational grid of the coolant channel with embedded baffle

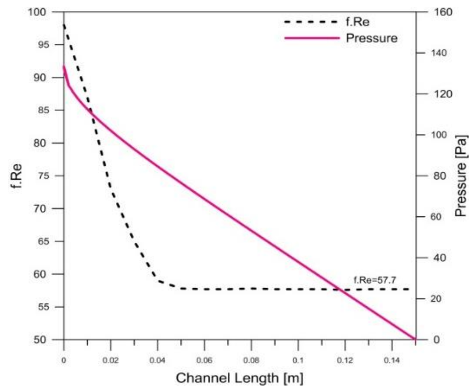


Figure 4. Pressure drop, "f.Re" values along the channel

is the channel length. The terms u_m and D_h represent the average velocity of fluid flow inside the channel and the hydraulic diameter, respectively.

$$u_m = \frac{Q}{A_{ch}}, \tag{5}$$

$$D_h = \frac{4A_{ch}}{P_{ch}}, \tag{6}$$

here, A_{ch} and P_{ch} are the cross section area and perimeter of the cooling channel respectively. In addition, Q represents the volumetric flow rate of cooling fluid. According to the analytical relations, the value of f.Re for a square cross section in a fully developed laminar flow is 56.91 [26]. Where, the Reynolds number, Re, is given by the following relation [21]:

$$Re = \frac{\rho u_m D_h}{\mu}. \tag{7}$$

For fully-developed laminar flow through a square cross section channel, the simulated value of “f.Re” is 57.1, as shown in Figure 4 Therefore, there is a good agreement between the simulated value and the analytical one which is 56.91 [26, 27]. The Nusselt number, Nu, along the channel while the heat flux is constant as shown in Figure 5. It can be seen in this figure that the calculated Nusselt number value in a fully developed region is 3.65 which is very close to analytical Nusselt number of 3.66 for fully-developed laminar flow through a square cross section channel under constant heat flux as the boundary condition [26]. This slight difference between the simulated and analytical values is mainly due to a deviation existed in the assumption for the constant heat flux boundary condition [27]. In fact, the heat flux on the bottom boundary of the channel is not constant due to the heat condition within bipolar plate. Besides the analytical results (Figures 4, 5), the numerical results are validated by comparing to data reported in the literture [18]. Adding baffles inside the flow channels induces co-vortex flow along the channel as shown in Figure 6. This also results of a fluid flow with a higher velocity in the baffled regions, as the flow accelerates and alsocirculates when passing over the top of the baffles.

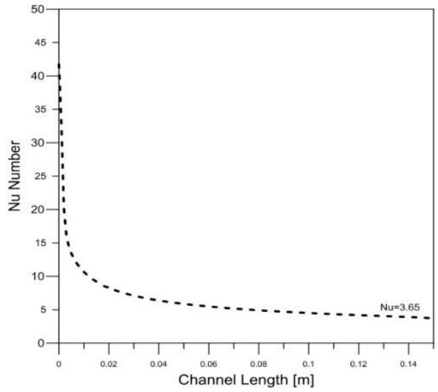


Figure 5. Nusselt number along a parallel channel (the heat flux is constant)

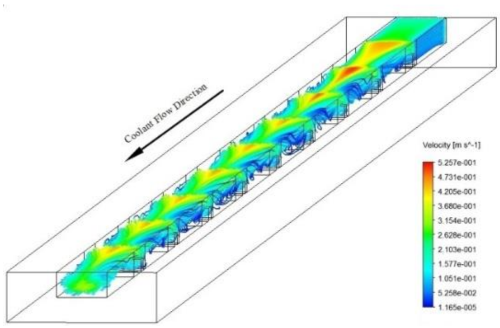


Figure 6. Temperature contour on the surface of the cooling plate in one half of the channel including 9 baffles with 0.5 mm height and thickness

According to Figure 7, the most velocity value is observed just after the baffle inside the channel and close to the cooling channel top wall. Figures 7 and 8 represent the temperature contours inside the flow channel and at the cooling plate of the fuel cell, respectively. The regions behind the baffles with a reverse/recirculating fluid flow provide a better mixing property between the core and the near wall regions of the channels. This mixing flow has a great influence on

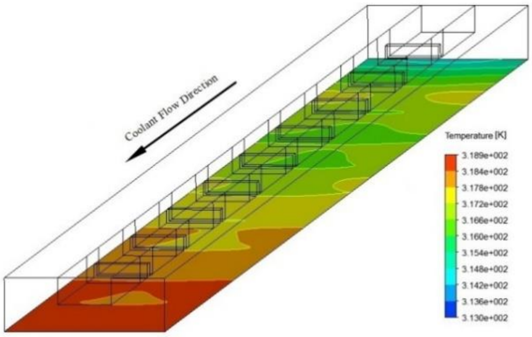


Figure 7. Temperature inside half channel including 9 baffles with 0.5 mm height and thickness

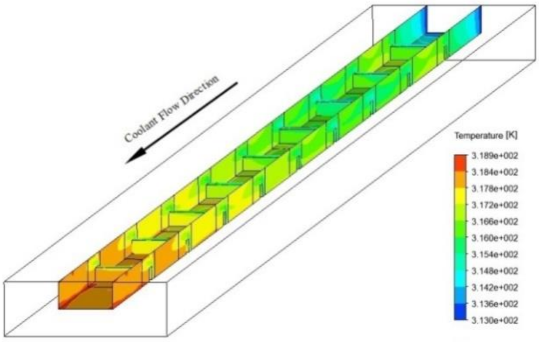


Figure 8. Flow circulation inside channels with baffles

the velocity and temperature distributions in the cooling channel which results in to a higher temperature gradient over the cooling wall.

5. 2. 1. Effects of Number of Baffles The results associated with the straight flow channel enhanced by various numbers of baffles are presented in Table 2. Here, the fixed dimensions of 0.5mm and 1 mm are considered for the height and thickness of the baffles, respectively. The maximum and average temperate surface along the index of temperature uniformity for the cooling plates and the pressure drop along with the axis of the flow channel are given in Table 2. The temperature uniformity index is calculated based on the difference between calculated temperature of different regions and average temperature at the surface of the cooling plate.

According to Table 2, it can be observed that the maximum and also average temperature of cooling plate surface and the index of temperature uniformity are decreased by increasing the number of baffles. It is observed that using more than nine baffles inside the half of the channel not only has a very little effect on the temperature parameters but also it leads to more pressure drop. As a result, the total number of nine baffles is embedded inside the one half of coolant flow channel to compromise between the favorable temperature distribution and reasonable pressure drop (Table 2).

5. 2. 2. Effects of Baffle Height In this section, the effect of height variation of the same nine baffles with constant thickness of 1mm on the fluid flow and temperature distribution inside each channel is investigated. Based on the data given in Table 3, the maximum, average, uniformity index of temperature at the cooling surface decreases and the pressure drop increases by increasing the height of the baffles. For cases with the height of baffles larger than 0.5mm, the baffle height has no sensible effect on the maximum and average temperature and its uniformity index. As a result, the height of baffles is selected 0.5mm. As shown in Figure 9 by a dashed line, the average surface temperature has a reducing trend by increasing the height of the baffles in such a way that the slope of dashed line in the range of 0.2-0.5 mm height is higher than that of 0.6-0.8mm. This demonstrates that the effects of baffle height on temperature profile decreases for dimensions larger than 0.5mm. On the other hand, the pressure difference increases by using baffles with longer height. It can be realized that using baffles with greater than 0.5 mm height leads to significant pressure drop, which is not favorable in terms of required power for the pumping system. Similar to temperature, the favorable baffle height lies between 0.3 to 0.6mm.

5. 2. 3. Effect of Baffle Thickness At this case, the number of baffles and the selected height are specified. Considering nine baffles with the 0.5mm height, the effect of baffles thickness is investigated and numerical results are reported in Table 4. It is clear that employing baffles with smaller thickness results to a less pressure drop, and also an improvement in the temperature distribution.

TABLE 2. Effect of the number of baffles on the temperature, temperature distribution and pressure drop

Number of Baffles	T_{max} (K)	T_{ave} (K)	S_D	ΔP (Pa)
0	322.46	319.70	2.3710	136.66
1	321.80	319.65	0.9685	154.87
2	320.70	319.04	0.9975	179.22
4	319.77	318.04	0.7449	249.66
9	318.98	317.33	0.7695	435.94
11	318.76	317.10	0.6986	518.30
14	318.50	316.80	0.7129	611.02
19	318.16	316.67	0.7933	733.13

TABLE 3. Effect of the baffle height on the temperature, temperature distribution and pressure drop

Height (mm)	T_{max}(K)	T_{ave}(K)	S_D	ΔP(Pa)
0.2	321.93	319.49	1.5824	156.85
0.3	320.55	318.82	1.1041	171.78
0.4	319.36	317.63	0.7287	249.62
0.5	319.06	317.32	0.7695	435.94
0.6	318.34	316.98	0.7077	850.25
0.7	318.18	316.74	0.6562	1780.88
0.8	318.08	316.60	0.6626	4553.41

TABLE 4. Effect of the baffles thickness on the temperature and pressure drop

Thickness (mm)	T_{max} (K)	T_{ave} (K)	S_D	ΔP (Pa)
0.3	318.51	317.23	0.6725	414.17
0.5	318.78	317.30	0.7003	414.34
0.7	319.09	317.33	0.7152	416.54
1.0	319.06	317.32	0.7695	435.94
1.2	319.27	317.41	0.8126	448.43
1.5	319.28	317.62	0.8595	471.24

However, the effect of the thickness of the baffles on the fluid characteristics is comparatively less than those of number of baffles and their height. A thinner baffle, if it could be made and embedded in the channels, may result to better results, due to the higher capability of mixing the layers of the fluid flow. As expressed in Table 4, the appropriate value of the baffle thickness is chosen to be 0.5mm.

6. CONCLUSION

A baffle restricted cooling channel will have a more efficient heat transfer compared to the straight channels. This configuration, in general, reduces the maximum temperature of the cell and results in to a better temperature uniformity in the cooling plates of the PEM fuel cell.

In this study, the effects of using baffles and their dimension on the heat transfer and fluid behavior in cooling channels of a PEM fuel cell were investigated by considering the maximum/average temperature of cooling plate surface, uniformity index of the temperature distribution, and pressure drop. The results of this study would be beneficial in the designing of the PEM fuel cell cooling systems. The main achievements of the present study are summarized below:

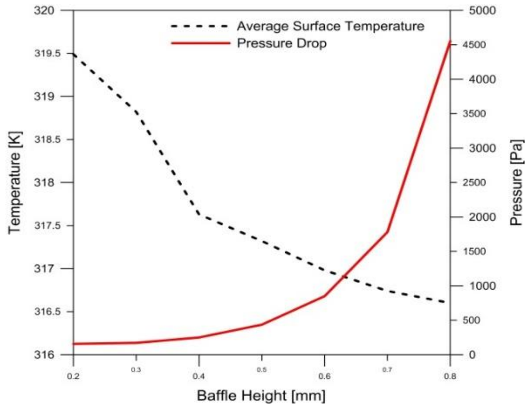


Figure 9. The effect of height of baffles on surface average temperature for cooling plate (surface with heat flux boundary) and pressure loss for coolant flow

1. Increasing the height of baffles may have a negative impact on the maximum temperature of constant heat flux surface and temperature distribution of this surface. Increasing this parameter not only increases the pressure drop but also produce nearly no considerable improvement on the cooling system performance. Therefore, the height of the baffles should be generally restricted to an upper limit, which is here 0.5mm according to the numerical simulation results.
2. Increasing the number of baffles embedded inside the coolant flow channel may generally improve the temperature distribution and its uniformity. However, it can also increase the pressure drop along the channel which is not favorable. Therefore, one has to compromise between the improvement of temperature distribution and pressure drop.
3. A thinner baffle is generally preferable due to its capability to improve temperature distribution with a less pressure drop. Here, baffles with 0.5 mm thickness is proposed for the coolant flow channel.
4. Coolant flow channels can be equipped by a number of baffles to provide a better cooling performance in a PEM fuel cell.

7. REFERENCES

1. Promislow, K. and Wetton, B., "A simple, mathematical model of thermal coupling in fuel cell stacks", ***Journal of power sources***, Vol. 150, (2005), 129-135.
2. Snyder, L.E., "A feasibility study of internal evaporative cooling for proton exchange membrane fuel cells", Texas A&M University, (2006),
3. Anderson, R., Zhang, L., Ding, Y., Blanco, M., Bi, X. and Wilkinson, D.P., "A critical review of two-phase flow in gas flow channels of proton exchange membrane fuel cells", ***Journal of Power Sources***, Vol. 195, No. 15, (2010), 4531-4553.
4. Baek, S.M., Yu, S.H., Nam, J.H. and Kim, C.-J., "A numerical study on uniform cooling of large-scale pemfcs with different

coolant flow field designs", ***Applied Thermal Engineering***, Vol. 31, No. 8-9, (2011), 1427-1434.

5. Hashmi, S.M.H., "Cooling strategies for pem fc stacks", (2010).
6. Hu, P., Cao, G.-Y., Zhu, X.-J. and Hu, M., "Coolant circuit modeling and temperature fuzzy control of proton exchange membrane fuel cells", ***International Journal of Hydrogen Energy***, Vol. 35, No. 17, (2010), 9110-9123.
7. Afshari, E. and Jazayeri, S., "Analyses of heat and water transport interactions in a proton exchange membrane fuel cell", ***Journal of Power Sources***, Vol. 194, No. 1, (2009), 423-432.
8. Sheikholeslami, M., "Numerical investigation of nanofluid free convection under the influence of electric field in a porous enclosure", ***Journal of Molecular Liquids***, Vol. 249, (2018), 1212-1221.
9. Sheikholeslami, M., "Numerical investigation for cuo-h2o nanofluid flow in a porous channel with magnetic field using mesoscopic method", ***Journal of Molecular Liquids***, Vol. 249, (2018), 739-746.
10. Sheikholeslami, M., "Lattice boltzmann method simulation for mhd non-darcy nanofluid free convection", ***Physica B: Condensed Matter***, Vol. 516, (2017), 55-71.
11. Asghari, S., Akhgar, H. and Imani, B.F., "Design of thermal management subsystem for a 5 kw polymer electrolyte membrane fuel cell system", ***Journal of Power Sources***, Vol. 196, No. 6, (2011), 3141-3148.
12. Mohamed, W.A.N.W., Talib, S.F.A., Zakaria, I.A., Mamat, A.M.I. and Daud, W.R.W., "Effect of dynamic load on the temperature profiles and cooling response time of a proton exchange membrane fuel cell", ***Journal of the Energy Institute***, (2017).
13. Chen, F., Gao, Z., Loutfy, R. and Hecht, M., "Analysis of optimal heat transfer in a pem fuel cell cooling plate", ***Fuel Cells***, Vol. 3, No. 4, (2003), 181-188.
14. Choi, J., Kim, Y.-H., Lee, Y., Lee, K.-J. and Kim, Y., "Numerical analysis on the performance of cooling plates in a pefc", ***Journal of mechanical science and technology***, Vol. 22, No. 7, (2008), 1417-1425.
15. Ravishankar, S. and Prakash, K.A., "Numerical studies on thermal performance of novel cooling plate designs in polymer electrolyte membrane fuel cell stacks", ***Applied Thermal Engineering***, Vol. 66, No. 1-2, (2014), 239-251.
16. Gould, B.D., Ramamurti, R., Osland, C.R. and Swider-Lyons, K.E., "Assessing fuel-cell coolant flow fields with numerical models and infrared thermography", ***International Journal of Hydrogen Energy***, Vol. 39, No. 26, (2014), 14061-14070.
17. Afshari, E., Ziaei-Rad, M. and Dehkordi, M.M., "Numerical investigation on a novel zigzag-shaped flow channel design for cooling plates of pem fuel cells", ***Journal of the Energy Institute***, Vol. 90, No. 5, (2017), 752-763.
18. Afshari, E., Ziaei-Rad, M. and Shariati, Z., "A study on using metal foam as coolant fluid distributor in the polymer electrolyte membrane fuel cell", ***International Journal of Hydrogen Energy***, Vol. 41, No. 3, (2016), 1902-1912.
19. Liso, V., Nielsen, M.P., Kær, S.K. and Mortensen, H.H., "Thermal modeling and temperature control of a pem fuel cell system for forklift applications", ***International Journal of Hydrogen Energy***, Vol. 39, No. 16, (2014), 8410-8420.
20. Saygili, Y., Eroglu, I. and Kincal, S., "Model based temperature controller development for water cooled pem fuel cell systems", ***International Journal of Hydrogen Energy***, Vol. 40, No. 1, (2015), 615-622.
21. Fly, A. and Thring, R., "Temperature regulation in an evaporatively cooled proton exchange membrane fuel cell stack", ***International Journal of Hydrogen Energy***, Vol. 40, No. 35, (2015), 11976-11982.
22. Castelain, C., Lasbet, Y., Auvity, B. and Peerhossaini, H., "Experimental study of the thermal performance of chaotic geometries for their use in pem fuel cells", ***International Journal of Thermal Sciences***, Vol. 101, No., (2016), 181-192.
23. Ahmadi, N., Rezazadeh, S. and Mirzaee, I., *Retracted: Numerical study the effect of downward and upward deflected membrane electrode assembly on species distribution in polymer electrolyte membrane fuel cell with experimental validations*. 2015, Elsevier.
24. Frano, B., "Pem fuel cells: Theory and practice", ***Academic, Burlington Google Scholar***, (2005).
25. Hu, G., Li, G., Zheng, Y., Zhang, Z. and Xu, Y., "Optimization and parametric analysis of pemfc based on an agglomerate model for catalyst layer", ***Journal of the Energy Institute***, Vol. 87, No. 2, (2014), 163-174.
26. Bergman, T.L., Incropera, F.P., DeWitt, D.P. and Lavine, A.S., "Fundamentals of heat and mass transfer, John Wiley & Sons, (2011).
27. Bejan, A., "Convection heat transfer, John wiley & sons, (2013).

Enhancement of the Cooling System Performance of the Proton-exchange Membrane Fuel Cell by Baffle-restricted Coolant Flow Channels

E. Afshari, S. P. Hosseini, M. Mosharaf-Dehkordi

Department of Mechanical Engineering, Faculty of Engineering, University of Isfahan, Isfahan, Iran

P A P E R I N F O

Paper history:
Received 01August 2017
Received in revised form 24April 2018
Accepted 17January 2018

Keywords:
Proton-exchange Membrane Fuel Cell
Cooling Flow Field
Baffle
Temperature Uniformity
Surface Temperature
Pressure Drop

چکیده

در این مقاله عملکرد حرارتی و خنک کاری پیل سوختی غشا پلیمری با قرار دادن موانع در کانال‌های خنک‌کاری بررسی شده است. بدین منظور، فاکتورهای ماکزیمم دمای صفحه خنک کاری، شاخص یکنواختی دمای سطح خنک‌کاری و همچنین افت فشار در داخل کانال‌ها بدون مانع و با نصب مانع بررسی و مقایسه شده است. معادلات حاکم در داخل کانال‌های پیل به روش عددی و به کمک روش حجم محدود با مدل‌سازی سه بعدی صفحات خنک‌کاری حل شده‌اند. نتایج عددی نشان می‌دهند که در پیل با کانال‌های همراه با مانع نسبت به کانال‌های بدون مانع، عملکرد حرارتی پیل بهبود یافته است؛ به طوری که ماکزیمم دمای سطح صفحات خنک کاری کاهش یافته و شاخص یکنواختی دما نیز بهبود می‌یابد. افت فشار نیز هر چند با قرار دادن مانع داخل کانال‌ها افزایش می‌یابد؛ اما باید مصالحه‌ای بین بهبود شاخص‌های دمایی و افزایش افت فشار انجام شود

***doi**: 10.5829/ije.2018.31.05b.17*