Cooling channels design analysis with chaotic laminar trajectory for closed cathode air-cooled PEM fuel cells using non-reacting numerical approach

W Mohamed W A N
1
1 Alternative Energy Research Centre, Faculty of Mechanical Engineering
Universiti Teknologi MARA, Malaysia
E-mail: wanajmi@salam.uitm.edu.my

Abstract. The thermal management of Polymer Electrolyte Membrane (PEM) fuel cells contributes directly to the overall power output of the system. For a closed cathode PEM fuel cell design, the use of air as a cooling agent is a non-conventional method due to the large heat load involved, but it offers a great advantage for minimizing the system size. Geometrical aspects of the cooling channels have been identified as the basic parameter for improved cooling performance. Numerical investigation using STAR-CCM computational fluid dynamics platform was applied for non-reacting cooling effectiveness study of various channel geometries for fuel cell application. The aspect ratio of channels and the flow trajectory are the parametric variations. A single cooling plate domain was selected with an applied heat flux of 2400 W/m² while the cooling air are simulated at Reynolds number of 400 that corresponds to normal air flow velocities using standard 6W fans. Three channel designs of similar number of channels (20 channels) are presented here to analyze the effects of having chaotic laminar flow trajectory compared to the usual straight path trajectory. The total heat transfer between the cooling channel walls and coolant were translated into temperature distribution, maximum temperature gradient, average plate temperature and overall cooling effectiveness analyses. The numerical analysis shows that the chaotic flow promotes a 5% to 10% improvement in cooling effectiveness, depending on the single-axis or multi-axis flow paths applied. Plate temperature uniformity is also more realizable using the chaotic flow designs.

1. Introduction
The application of hydrogen-fuelled PEM fuel cells is growing positively from test scales to market research scale as its fast initiation and reliable operation proves to be an attractive factor. A significant effort to model the behaviour of fuel cell operation has been made due to the fact that the electrochemical and thermo fluid reactions occur internally and non-visible for direct analysis. Numerical models have proven highly useful to optimize the design of various components such as the reactant flow channels and the cooling channels.

1 To whom any correspondence should be addressed.
The generation of electrical power within the fuel cell system is always accompanied by a near equivalent amount of heat energy and increases in an exponential behaviour as the electrical power increases. Therefore, continuous active cooling of the cells is required. Thermal management of a PEM fuel cell keeps cell temperatures at levels where excessive membrane drying leading to increased ionic transport resistance is avoided [1]. It also allows near-uniform temperature distribution, stabilizes the power density [2] and prevents propagation of thermal stresses [3].

Air as a coolant requires minimal components and merely increases the overall system size by less than 50% whereas water-cooled systems generally increase the system size by more than 200%. For portable and limited space applications, air-cooled fuel cells are very much desirable. Technically, the design of the coolant flow path is very important in allowing the proper air quantity to flow smoothly and interact with the hot surfaces.

Cooling channels provides an interface for the interaction of the cooling fluid with the hot surface. Conventional air-cooling channels for PEMFC are normally straight path channels integrated between the anode and cathode sides of the bipolar plates. The temperature difference between the anode/cathode and cooling channel surfaces are typically very small due to the high plate conductivity and small distance between them.

Conventional cooling channel designs of a PEM fuel cell have hydraulic diameters larger than 1 mm and are still within the mini-channel definition where the classic theory of fluid behaviour is still valid. For liquid cooling, a variety of cooling channel designs has been proposed such as multi-pass serpentine oblique-fins, coiled, parallel-serpentine, wavy and a novel hybrid parallel-serpentine-oblique-fins design [4]. Lasbet et al. [5] proposed chaotic flow channels to enhance water and surface interactions. From the technical perspective of air-cooling, the chaotic flow channels are more suitable than multi-pass serpentine channels. As the specific heat of air is much lower than water, it has to be allowed to flow in and out of the heated domain swiftly to avoid acting as a heat carrier to the other parts of the cell. At low Reynolds number, the cooling effectiveness could be unsatisfactory as effective cooling would be severely constrained within the inlet region, resulting in large temperature gradients relative to the air outlet region.

Very limited numerical studies on air-cooled closed cathode PEM fuel cell, which utilizes separate fluids as reactants and coolant, are available compared to open cathode thermal investigations. Previously, a non-reacting numerical investigation at stack level was published for a simple iscal case of singular air cooling channel system [6]. Then, Matian et al. [7] investigates three different air cooling channel designs for closed cathode PEM fuel cells in a non-reacting fuel cell numerical domain. The cooling channel designs all are rectangular with a straight flow path but with different aspect ratios. Their main conclusion stated the capability of the cooling plate to balance the temperature variation with a strong inclination towards wider cooling channel designs.

Here, a simplified non-reacting domain is applied to investigate three rectangular cooling channel designs of equal numbers but with variations in aspect ratio and flow path, specifically for closed cathode air-cooled PEM fuel cell application. The computational domain consists of a single cooling plate imbedded with the cooling channels and a constant heat flux was applied on one side of the plate based on the actual heat generation of an experimental fuel cell stack. The results are analyzed from the perspectives of cooling effectiveness, average plate temperature, plate temperature distribution and specific cooling with the purpose for identifying the optimal design for physical development.

The principle question of the analysis was on the cooling effectiveness (Eq. 1) of each cooling plate, calculated from the ratio of the plate cooling rate (total boundary heat transfer) to the thermal power (heat generated) acting on the plate. The cooling effectiveness and average plate temperature are the main optimization parameters for design evaluation.

\[ \varepsilon_c = \frac{Q_c}{P_{th}} \]  

(1)
2. Cooling channel designs
A commercial closed cathode PEM fuel cell stack originally designed for water cooling was chosen as the reference design. The commercial stack is rated at 3 kW power output, consisting of 73 bipolar plates, each 5 mm thick, and the overall dimension of the stack is 365 mm in length x 149.5 mm in width x 23.9 mm in height. Technically, it can be considered as a large-sized fuel cell. Of the total plate height of 239 mm, only 179 mm is available for cooling area designation due to the requirements of the reactant gas ducts and manifolds construction (Figure 1). The length of the channels follows the exact width of the plate at 149.5 mm (refer Table 1).

The number of cooling channels is limited to the width of the channels design. In this work, the number of channels for each design are fixed, \( n = 20 \) channels. Ribs between the channels must be at least 1 mm distance as the ribs assist electron flow in actual operation. Three cooling channel designs are covered in this brief report. The first design is a standard straight flow channel (Ms20). As figure 3, the second design is termed as the square wave mixer channel with 11 bend and single-axis horizontal fluid flow (Ch20-CF). The third design explores a more complex geometry which is the C-shaped channels with 11 bend and two-axis horizontal-vertical flows (Ch20-CV). The Ch20 designs theoretically promotes more fluid-surface interactions as it allows the fluid to be in a more non-linear (chaotic) trajectory even in laminar phase flow.

The practical inlet air velocity is taken at 2 m/s, which translates to a Reynolds number of approximately 400 for the conventional Ms20 straight channel design. This Reynolds number is taken as a reference for the other designs. At a Reynolds number of 400, the coolant air inlet velocity varies for each design due to the differences in hydraulic diameter of the channels. Consequently, the air mass flow rate differs slightly.

![Figure 1. Cooling plate geometry and base dimension.](image)

![Figure 2. Inlet and outlet air surface boundaries.](image)

| Design label | Number of channels | Width (mm) | Height (mm) | Length (mm) | Aspect ratio |
|--------------|--------------------|------------|-------------|-------------|-------------|
| 1. Ms20      | 20                 | 8.0        | 2           | 149.5       | 4           |
| 2. Ch20-CF   | 20                 | 4.0        | 2           | 180.0       | 2           |
| 3. Ch20-CV   | 20                 | 4.0        | 1           | 200.0       | 4           |
3. Computational domain, models and boundaries

The applied commercial software was STAR-CCM. There are only two regions in the work domain which are the plate (graphite) region and the fluid (air) region. The interface layers between the fluid region and plate region was designated as the boundary heat transfer layer where the heat flow rate between the regions, or the cooling rate, would occur and its values captured.

In practice, the nominal (incoming) cooling air velocity range for a closed cathode depends on the fan model and operation setting. Usually, a smooth flow of air is induced into the stack by negative pressure (suction) flow to avoid high fluid entry resistance into the channels. Hence, the typical air velocity range is between 1 m/s to 2 m/s if the fuel cell is assumed to apply a set of standard 6W mini fans. Here, the cooling performance at the highest air velocity is reported where the flow is a fully developed laminar flow.

The projected heat generated is the assumed thermal energy of the fuel cell that the cooling plate needs to remove. Based on the 3 kW stack of reference which is made up of 73 cells, each cell would theoretically generate 41W of thermal energy at 50% conversion efficiency. The cooling plate in this simulation is assumed to handle thermal energy generated from two adjacent cells simultaneously, or applying a thermal safety factor of 2. The heat is applied to one of the plate surfaces with an effective surface area of 0.0342 m². This approach is similar to [8] in their work on a water cooled fuel cell using a similar computational platform. The other sides of the plate were set as adiabatic surfaces to totally reflect the cooling effectiveness of the coolant by internal convection. External forced and free convection as well as radiation cooling effects are neglected as a simplification to the simulated model. Tables 2 and 3 lists the other main boundary conditions of the simulation.

### Table 2. Boundary conditions for bipolar plate.

| Properties / parameters | Values       |
|-------------------------|--------------|
| 1. Material             | Carbon graphite |
| 2. Specific heat        | 710 J/kg.K     |
| 3. Density              | 2240 kg/m³    |
| 4. Thermal conductivity | 20 W/m.K      |
| 5. Projected heat       | 82 W          |
| 6. generated            | 2400 W/m²     |

### Table 3. Boundary conditions for cooling air.

| Properties / parameters | Values       |
|-------------------------|--------------|
| 1. Flow phase and model | Laminar (Re= 250 -750) |
| 2. Flow source – Uniform plane | 50 mm from channel inlets |
| 3. Inlet specifications | 3 m/s, 30°C |
| 4. Outlet specifications – pressure outlet | 1 atm, 30°C |
| 5. Thermal conductivity | Constant    |
| 6. Specific heat, Cp    | Constant     |

In the cooling channels, the air flow field and internal forced convection was obtained by solving the steady state Navier-Stokes equations, which includes the continuity equation, momentum equation and the mass and energy conservation equation. The computational physics models selected are ideal gas, laminar flow model, segregated flow and the segregated solid energy model. Iterations in the
range of 800-1200 are needed to obtain a satisfactory convergence with residuals less than 0.05% than the previous iteration value.

The air inlet and outlet surface boundaries are set as in figure 2. At the inlet, air velocity inlet is specified. In practice, one or more air fans are used to induce the cooling air. Simplification was made here by neglecting the presence of the momentum source (fan blowers), hence eliminating complex transitional flows between the fan area and the channel inlet area. Inlet air flows are assumed uniform with a constant incoming velocity; therefore, the cooling rates would be theoretically optimal at the stated boundary conditions. At the outlet surface, the exit velocity could not be directly specified; so, the main boundary conditions are pressure and the static temperature. The static fluid temperature of both inlet and outlet were set at the normalized ambient condition of 30°C and 1 atm.

4. Results and discussion

The main numerical analyses for a fuel cell cooling plate design are temperature distribution, average plate temperature and the cooling effectiveness. Figure 4 shows the plate temperature distribution of each cooling channel design. The dotted box within the plate indicates the boundary of the fuel cell active area. The general profile for all three cases are similar; lowest plate temperature are found near the inlet (approximately 50°C) and increases in a semi-ellipsoidal profile along the coolant path towards the exit. The near-inlet temperatures are more uniform compared to other parts of the active area. This trend is normally encountered for bulk solids with internal convection effects as the coolant gradually loses its cooling capacity along the flow path. In practice, this steady-state thermal profile is achieved when both the conduction heat flow within the plate and the convection heat transfer from the plate to fluids in contact reaches an asymptotic saturation level.

The straight path design (Ms20 design) leads to a highly heterogeneous temperature profile compared to the channels with chaotic flow trajectory. Within the active area, the maximum
temperature gradient between the inlet and exit is 40°C. The maximum temperature gradient for designs CF and CV are approximately 30°C and 20°C respectively. As the cooling effect is based on a similar Reynolds number for all cases, the differences in nominal coolant velocity of each design have a big impact on the overall temperature distribution. Here, the temperature profile as shown by design CV is most desirable for fuel cell operation as the plate temperature distribution approaches homogeneity with high temperature areas merely located at the edges of the exit section.

Figure 5 displays the combined parametric comparison analysis on cooling effectiveness and averaged plate temperature. These parameters contribute to an inversely proportional effect to one another. At a nominal coolant flow of Reynolds number equals to 400, the CV design reached 90% cooling effectiveness which is the minimal performance criteria for a sustained bulk cooling effect. This in turn translates to a much lower average plate temperature of 64°C which is still in the normal range of operating temperatures for PEM fuel cells.

![Figure 5](image_url)

**Figure 5.** Cooling effectiveness and averaged plate temperature relation at Re 400

The results on cooling effectiveness indicate an improved cooling capability when laminar flows are allowed to have a more interactive trajectory with the surfaces. At similar nominal Reynolds number, the cooling improvement is nearly 10% when the flows are multi-axial while a 5% increase can be expected for single axis chaotic flow. Based on Nusselt number definitions, the heat transfer coefficient of the three cases might not show any significant differences as the Reynolds number is similar. Therefore, the increase in cooling effect is highly due to the contact surface factor in the classic Newton’s law of cooling theory.

The translation of this result to actual fuel cell cooling plate design depends on the plate structural integrity due to the more complex geometry of the CV channel designs. Further study in optimizing the cooling plate thickness (from the base case of 5mm thickness here) while simultaneously applying better cooling channel geometries is a challenge in fuel cell thermal engineering. The main results here show that laminar coolant flows with lower fan power consumption performs better when the flow trajectory that depends on the cooling channel design are built to promote a chaotic flow path throughout the channel.

### 5. Conclusions

The thermal engineering aspect in PEM fuel cell operation is critical for performance reliability and sustainability. The use of air cooling for closed cathode PEM fuel designs is a rare option and was studied here to identify the basic design effects on cooling capability. The three designs proved that introducing a slightly different coolant trajectory within the cooling channels can have positive cooling effects and is useful for size-constrained systems such as for vehicle applications. The results obtained here at a practical Reynolds number of 400 provided a platform for more detail studies in cooling plate design optimization for air-cooled closed cathode PEM fuel cells.
References
[1] Park J and Li X 2006 J. Power Sources 162 444
[2] Didierjean S, Lottin O, Maranzana G and Geneston T 2008 Electrochimica Acta 53 7313
[3] Sadiq al-Baghdadi M A R and Shahad al-Janabi H A K 2007 Proc. of IMechE 2221 Part A: J. Power and Energy
[4] Sasmito A P, Kurnia J C and Mujumdar A S 2012 Energy 44 278
[5] Lasbet Y, Auvity B, Castelain C and Peerhossaini H 2006 J. Power Sources 156 114
[6] W Mohamed W A N, Ismail A A and Atan R 2010 IEEE 2010 Int. Conf. on Sci. and Soc. Research (CSSR), IEEE Xplore Digital Library 91
[7] Matian M, Marquis A and Brandon N 2011 Int J Hyg Energy 36 6051
[8] Yu S H, Sohn S, Nam J H and Kim C-J 2009 J. Power Sources 194 697