Excerpt from the Proceedings of the COMSOL Conference 2010 Boston

Optimizing Bio-Inspired Flow Channel Design on Bipolar Plates of PEM Fuel Cells

James A. Peitzmeier*[1], Steven Kapturowski[2] and Xia Wang[3]
[1]Michigan Technological University, [2]University at Buffalo, [3]Oakland University
*Corresponding author: Building 20 Room 933, Attn: James Peitzmeier, 1400 Townsend Drive, Houghton, MI 49931-1295 Email address: japeitzm@mtu.edu

Abstract: The flow channel design on bipolar plates affects a proton exchange membrane (PEM) fuel cell performance by influencing reactant distribution and water removal in an operating fuel cell. The fuel cell performance can be improved by varying the type, size, or arrangement of channels. Two bio-inspired designs have been proposed by a research group at Oakland University, which results in improvement on the fuel cell performance. The objective of this research is to optimize the existing bio-inspired designs to further improve the fuel cell performance by the use of gates in the channel design to control the flow distribution. Comsol Multiphysics software is used to model the flow field constrained by the Navier Stokes and Brinkman's Equations. An iterative solver was developed between Comsol and Matlab to find the optimal configuration of gate heights required. An analytical model is also developed which will be used to verify the numerical results.

Keywords: PEM, Fuel Cells, Flow Channel, Bipolar Plates, Optimization

1. Introduction

In a fuel cell the reactant gasses, commonly hydrogen and oxygen, must be transported to the proton exchange membrane (PEM) through a system of channels combined with a thin porous layer called the gas diffusion layer (GDL). The geometry of the channels affects the pressure loss between inlet and outlet, water removal and dispersion of the reactant. A poor design may lead to limited flow for some regions of the PEM limiting local current density. Different flow patterns have been designed on bipolar plates to evenly distribute the reactant and remove the water product. For example, two bio-inspired flow patterns were proposed by a group at Oakland University [1]. It proved that these two new designs result in better fuel cell performance than the traditional flow patterns such as single serpentine or interdigitated channels. However, the proposed bio-inspired designs were not optimized. The focus of this paper is to optimize the bio-inspired flow channel pattern by the use of gates in the channel design to control the flow distribution. This concept is similar to the use of gates in injection mold design. Two approaches will be used to determine the correct gate height. Optimizations will be done by an iterative CFD solver, using Comsol Multiphysics and Matlab, which will compare current and desired flow distributions and an analytical model with gate heights as the un-known variable.

2. Improved Geometry with Gates

Two bio-inspired flow channel designs [1] were investigated. Figure 1 shows a section of leaf design with gates, and Figure 2 shows a section of lung design with gates. Both geometries are symmetric about the diagonal. The fuel or air passes from inlet channel to outlet channel through the GDL.

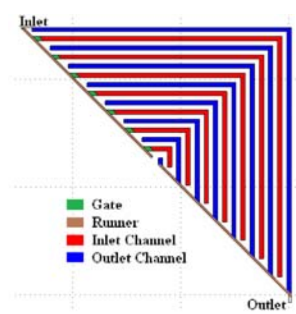

Figure 1: Flow Channel Geometry Leaf Design

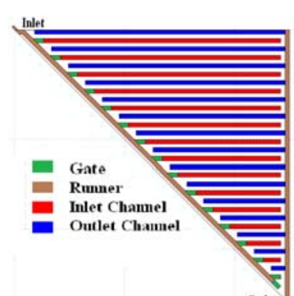

Figure 2: Flow Channel Geometry Leaf Design

3. Analytical Model

The purpose of this model is to calculate the required gate height to create a well dispersed flow. This example is for the leaf design but has also been applied to the lung design. To establish a volumetric flow ($\dot{\forall}_n$) through any given inlet channel it will be assumed that $\dot{\forall}_n$ is proportional to the length of the channel. That is any given length of inlet channel in the geometry will lose the same volume of fluid as any other section of the same length. This is represented by

$$\frac{\dot{\forall}_1}{L_1} = \frac{\dot{\forall}_2}{L_2} = \frac{\dot{\forall}_3}{L_3} = \cdots = \frac{\dot{\forall}_n}{L_n} \tag{1}$$

$\dot{\forall}_n$ is solved by applying the boundary condition at the inlet so that the summation of all $\dot{\forall}_n$ is equal to the inlet volumetric flow rate. To establish a volumetric flow ($\dot{V}o_n$) through any given outlet channel a variable $Ratio_n$ is introduced.

$$\dot{\forall}o_n = Ratio_n * \dot{\forall}_n + (1 - Ratio_{n-1}) * \dot{\forall}_{n-1} \tag{2}$$

$Ratio_n$ represents the portion of the inlet channel that is moving away from the center of the bipolar plate. $Ratio_n$ is found by setting the resistance to flow for each path leading either neighboring outlet channel equal to each other.

$$(\overline{\Delta P_{GDL}} + \overline{\Delta P_{Channel}} + \Delta P_{Minor\ Losses} + \Delta P_{single\ runner\ section})_{moving\ away\ from\ center} = (\overline{\Delta P_{GDL}} + \overline{\Delta P_{Channel}} + \Delta P_{Minor\ Loses})_{moving\ towards\ center} \tag{3}$$

The main section of the model calculates the pressure drop for individual paths that a differentiable volume of fluid may take from inlet to outlet. By equating the pressure loss between different paths gate height can be solved for.

$$\overline{\Delta P_{GDL}} + \overline{\Delta P_{Channel}} + \Delta P_{Minor\ Loses} + \Delta P_{Runners} + \Delta P_{Gate} = \Delta P_{Total} \tag{4}$$

The above equation will be repeated for each channel. Minor losses are from the 45 degree branch off the runner and the 90 degree elbow in the channel. Minor losses are determined as

$$\Delta P_{Minor\ Loses} = \frac{k_l * \rho * V^2}{2} \tag{5}$$

where k_l = .4 for the 45 degree branch and 1.1 for the 90 degree bend [2]. In this case V is taken to be either the velocity of the runner before the branch or the velocity in the channel depending on which location is being considered. The velocity in the runner is the volumetric velocity of channel plus all subsequent channels divided by the channel area. This produces the following 2 equations

$$\Delta P_{minor45} = k_{l45} * \frac{\rho}{2} * \left(\frac{\dot{\forall}_n + \dot{\forall}_{n+1} + \cdots + \dot{\forall}_7}{A}\right)^2$$

$$\Delta P_{minor90} = k_{l90} * \frac{\rho}{2} * \left(\frac{\dot{\forall}_n + \dot{\forall}o_n}{2*A}\right)^2 \tag{6}$$

Since the flow rate in the runner drops as each channel branches off each section of the runner will have a unique term. Pressure drop over a length of square pipe can be found using moody's friction factor and Reynolds number

$$\Delta P = f * \frac{L}{D_h} * \frac{\rho * V^2}{2} \tag{7}$$

$$f = \frac{c}{Re_d}$$

where c= 56.9 for square tube [2].

Combining and replacing V with volumetric flow and area. L is replaced with 2*step where step is the length between inlet channels, multiplied by 2 for since there is a runner on either end of the channel. At every branching point 2*$\dot{\forall}_n$ will enter or exit the runner as there is one channel on each side of the runner.

$$\Delta P_{runners} = \frac{c*\mu*step}{{D_h}^2} * \frac{2*(\dot{\forall}_n + \dot{\forall}_{n+1} + \cdots + \dot{\forall}_7)}{A} + \frac{c*\mu*step}{{D_h}^2} * \frac{2*(\dot{\forall}_{n-1} + \dot{\forall}_n + \cdots + \dot{\forall}_7)}{A} + \cdots + \frac{c*\mu*step}{{D_h}^2} * \frac{2*(\dot{\forall}_2 + \cdots + \dot{\forall}_7)}{A} \tag{8}$$

Applying the same equation to the gate as the runners

$$\Delta P_{gate} = \frac{c*\mu*L_{gate}}{2*{D_h}^2} * \frac{\dot{\forall}_n}{A_{gate\ n}} \tag{9}$$

where D_h is the hydraulic diameter. It is assumed that the volumetric flow rate in the channel will decrease linearly. This assumption will be shown to be invalid but is a close approximation. For any given path that crosses over from inlet to outlet channel at a point Cr meters from the start of the channel and stops at the end of the channel E meters down the path

$$\Delta P_{channel} = \int_0^{Cr} \frac{c*\mu*\left(\frac{\dot{\forall}_n}{A}\right)*\left(1-\frac{x}{E}\right)}{2*{D_h}^2} dx + \int_{Cr}^{E} \frac{c*\mu*\left(\frac{\dot{\forall}_{on}}{A}\right)*\left(\frac{x}{E}\right)}{2*{D_h}^2} dx \quad (10)$$

It can be shown from the above equation that the pressure drop due to the channel will vary from about ½ to ¾ of what a similar length single channel would cause, due to changes in the term Cr. To compensate for this difference in pressure loss down the channel base on where the fluid crosses, the difference in pressure across the GDL must vary as well producing a non linear change in velocity in the channel. As long as $\overline{\Delta P_{GDL}} \gg \overline{\Delta P_{Channel}}$, a linear change in velocity can be considered a good approximation. $\overline{\Delta P_{channel}}$ is then found

$$\overline{\Delta P_{channel}} = \frac{\int_0^E \Delta P_{channel}(Cr)\, dCr}{E} \quad (11)$$

Since velocity in the inlet and outlet channels are assumed to change linearly the $\overline{\Delta P_{GDL}}$ term can be found using a single velocity value and Darcy's law.

$$\overline{\Delta P_{GDL}} = \frac{\mu*\epsilon*Ratio_n*\dot{\forall}_n*\mathrm{L}}{\mathrm{k}*\mathrm{A_{interface}}} \quad (12)$$

where μ is the dynamic viscosity, ϵ is the porosity, k is the permeability, L is the length between channels and $A_{interface}$ is the area of the path of the flow. $A_{interface}$ is approximated by a height less than the thickness of the GDL times the length of the channels. This height term is discussed in the results section.

Each term discussed can now be substituted into equation (4) for n=1 through 7 creating 8 un-knowns, $A_{gate\,1-7}$ and ΔP_{Total}, and 7 equations. The eighth equation will be to set one gate area to be equal to the channel area. This should be the expected smallest gate. If any other gate is selected at least one given solution set will not include a positive or real value.

The model however does not account for flow through the GDL around the gate. The reason to neglect this term is that the pressure loss across the gate is much less than the pressure loss of the same volumetric flow through the section of GDL above the gate. This relation is shown below

$$\Delta P_{gate} \ll \frac{\mu*\epsilon*Ratio_n*\dot{\forall}_n*L_{gate}}{\mathrm{k}*\mathrm{Height_{GDL}}*\mathrm{Width_{channel}}} \quad (13)$$

For a large gate which is 94% the height of the channel and .0016 mm long the GDL is 3.5 times more resistive than the gate.

3.1 Analytical Model - Results

The calculated gate heights for the leaf and lung designs can be seen in Tables 1 and 2.

Table 1: Gate Heights Generated For the Leaf Design ($\mathrm{Height_{GDL}}$ = .2mm)

Gate number	Height (mm)
1	0. 76618
2	0. 672088
3	0. 610084
4	0. 623121
5	0. 693148
6	0. 755002
7	0

Table 2: Gate Heights Generated for the Lung Design

Gate number	Height (mm)	Gate number	Height (mm)
1	0. 784371	8	0. 712703
2	0. 734364	9	0. 785712
3	0. 657552	10	0. 831316
4	0. 496687	11	0. 863745
5	0	12	0. 889100
6	0. 145231	13	0. 910619
7	0. 568091	14	0. 930255
		15	0. 957208

Although the flow through the GDL is modeled as a straight shot between the inlet and the outlet the actual flow will curve as it goes up and over the wall between the inlet and outlet channel.

This leads to the $\mathrm{Height_{GDL}}$ term to be an estimated term representing the height of a porous media section with constant velocity flow and equivalent resistance to the non constant velocity (in space) curved flow through the GDL. In testing the $\mathrm{Height_{GDL}}$ term was estimated at .7 mm, .35 mm and .2 mm. When $\mathrm{Height_{GDL}} = .2$ mm the flow rates through the gate most closely matched the target flow rates. The flow rates for each gate and the flow rates for an un-gated design are compared to flow rates that would be proportional to the channel lengths in Figure 3.

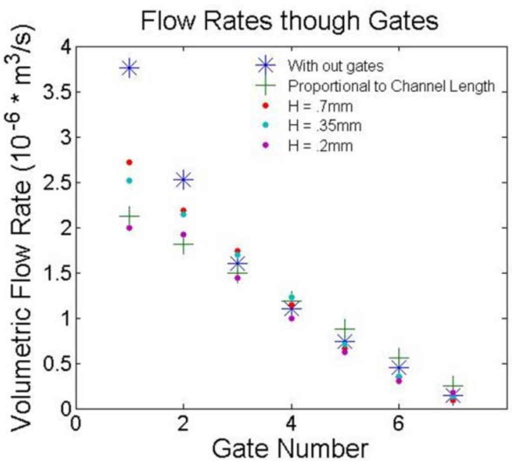

Figure 3: Comparison of volumetric flow rates for different values of $\mathrm{Height_{GDL}}$

Despite having appropriate flow rates at the gates, the $\mathrm{Height_{GDL}} = .2$ mm design does not show much improved performance as compared to the un-gated design when velocity towards the PEM in the GDL is considered. This is shown in Figure 4 where the gated design is on top and the un-gated design is below.

The model does not account for flow from runners directly to channels. In Figure 4, hot spots are located at the ends of the channels near the runners. This un-accounted loss of flow could cause in accuricy in the analytical model.

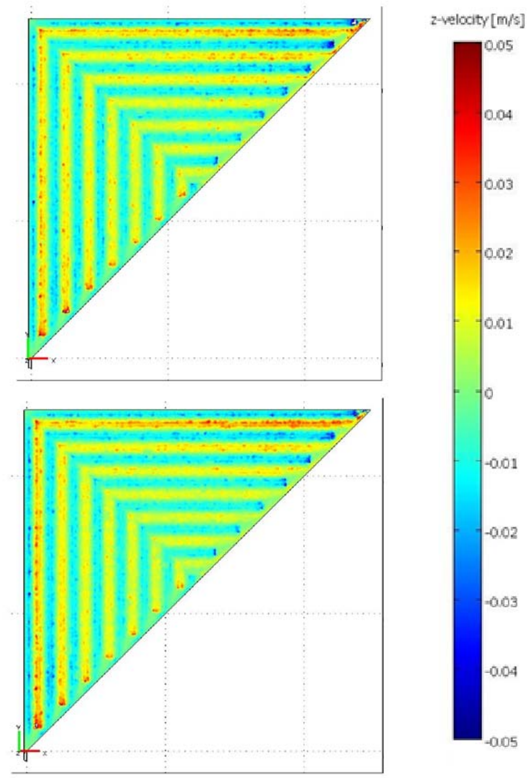

Figure 4: Comparison of $\mathrm{Height_{GDL}} = .2$ mm and un-gated designs

4. Numerical Simulation

4.1 Comsol Model Setup

Comsol Multiphysics 3.5a was selected to model one half of each design split down a symmetric diagonal. Incompressible Navier Stokes equations were used along with Brinkman equations for flow through the channel and porous GDL. A constant velocity of 20.75 m/s was given at the flow inlet and an atmospheric pressure of 0 Pa was prescribed at the flow outlet. The pink walls in Figure 5 below represent the surfaces defined with symmetric boundary conditions. The fluid is air at 75 °C. The dimensions of the computational domain and GDL properties are defined in Table 3.

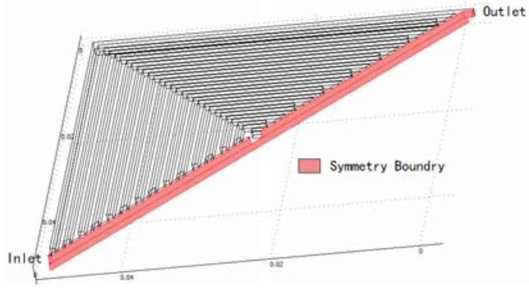

Figure 5: The highlighted surfaces are given a symmetric boundary condition.

Table 3: Geometry Dimensions and GDL Properties

GDL Porosity	.78
GDL Permeability	$5*10^{-12}$ m^2
GDL thickness	.001016 m
Channel height	.001016 m
Channel width	.0007876 m
Wall thickness	.0008 m

4.2 Mesh Independence Study

A test model, Figure 6, has been created to establish mesh independence without requiring the processor time and memory of running the full model. All walls are set to a no slip condition. The inlet is a constant velocity of 10 m/s. The outlets are 0 Pa back pressure with no viscous stress. All dimensions and properties in Table 3 are maintained.

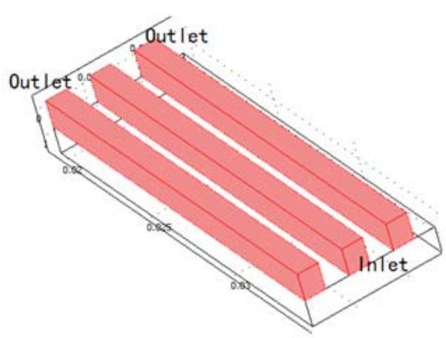

Figure 6: Model for mesh independence study

In Figure 7, mesh A is the finest with 4 elements across the channel and 8 elements in the height. Mesh E is the coarsest with 2 elements in the width and height. Each model has approximately half as many elements as the previous model. Elements twice as large are used in the GDL as gradients are expected to be lower there. Meshes A, B and C tend to be within .5 to 1% in pressure along the center of the inlet and outlet channels. The average Z velocities in the GDL are within 7% for meshes A, B and C and double to triple that for meshes D and E.

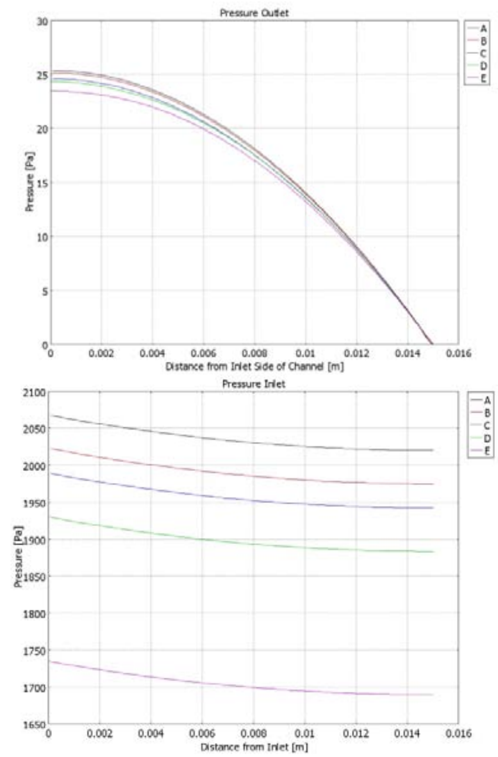

Figure 7: Pressure results for mesh independence study

4.3 Comsol/Matlab Optimization

Comsol Multiphysics was used to generate a Matlab script of the fluid flow through the channels, gates and GDL. This script was modified to adjust gate heights depending on results from a previous solution. When Comsol creates a script file, with some exceptions, it typically saves all operations the user has performed. To maintain simplicity in the script, complex geometries as well as all those imported from CAD programs are stored in a separate binary file. Any geometry that needs to be modified, in this case the gates dimensions, was created in Comsol as the last modification to the geometry shortly before creating the script file.

The iterative CFD solver program is operated by solving the flow field initially without gates, sending the results to Matlab which then compares the difference in pressure between corresponding inlet and outlet channels by boundary integrations, then normalized by the boundary areas to find the average pressure along each channel. Each inlet channel is compared to one neighboring outlet channel which is either towards the center for the leaf design or towards the outlet for the lung design. Matlab will then modify the gate heights by the linear function of the pressure drop percentage deviation from the mean:

$$Height_i^{k+1} = Height_i^k + r\frac{\Delta P_i^k - \overline{\Delta P}^k}{\overline{\Delta P}^k} \qquad (14)$$

where r is the remaining channel height above the gate. A nonlinear function was tested and initially expedited convergence. However, as the program approached a solution stability became an issue more than a simple linear relation. Any gate that is tall enough to produce a very fine gap between the gate and GDL requires many very fine elements that may cause memory problems. If this is the case the gate's height is backed away until the error is resolved. If topology has changed, the final step is to make appropriate changes to boundary conditions before initiating Comsol to solve for the new flow field.

The program was run on a Linux server with 32 GB ram and 8 Xeon 5430 processors. Run times were typically 7 min per iteration.

4.4 Comsol/Matlab Optimization Results

The iteration program requires as input the maximum number of desired iterations and the desired tolerance of difference in pressure deviations from the mean. Placing a limit on the number of iterations is found to be necessary since for very small tolerances the program will not converge. The limit has been found to be a tolerance of a 6 Pa standard deviation where the program will converge within reasonably little time. Figure 8, shows the standard deviation of the difference in pressure illustrating the incremental improvement through the iteration process.

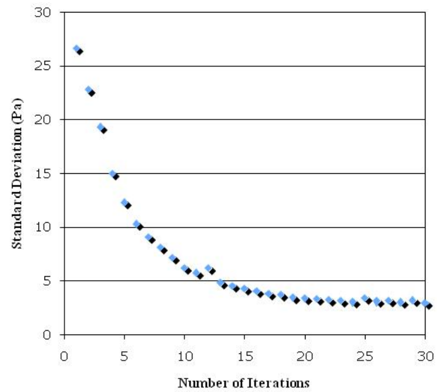

Figure 8: Standard Deviation of Difference in Pressure Vs. Iteration Number

The gate heights from the iterative solver for the lung design can be seen in Table 4 and compared to the analytical solution in Figure 9.

Table 4: Gate Heights Generated For the Lung Design

Gate number	Height (mm)	Gate number	Height (mm)
1	0. 7481	8	0. 0876
2	0. 5001	9	0. 6468
3	0. 124	10	0. 86
4	0	11	0.9076
5	0	12	0.9414
6	0	13	0.9646
7	0	14	0. 9862

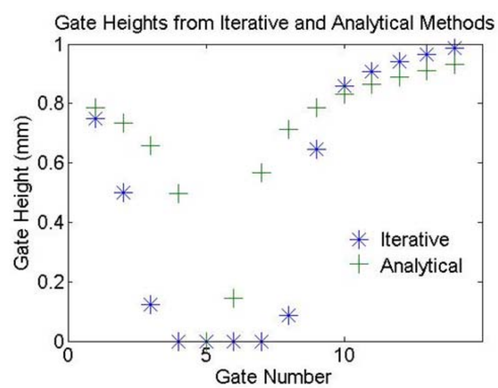

Figure 9: Comparison of Gate Heights Produced by Iterative and Analytical methods

The pressure differences across the inlet and outlet channels indicate diffusion rate to the GDL. Figure 10 illustrates that the iterative methods produces more uniform pressure drop deviations, where a pressure drop deviation is the difference in pressure between an inlet and outlet channel divided by the average difference in pressure between inlet and outlet channels.

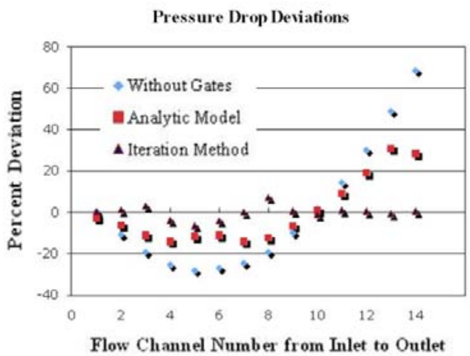

Figure 10: Comparison of pressure deviations produced by Iterative and Analytical Methods

The iterative solution is shown to have limited improvement on the un-gated design when compared by velocity towards the PEM in the GDL. Figure 11 shows the un-gated design on top and a gated design from the iterative method on the bottom. As in the leaf design, hot spots are located at the ends of the inlet and outlet channels due to flow from channels directly to runners.

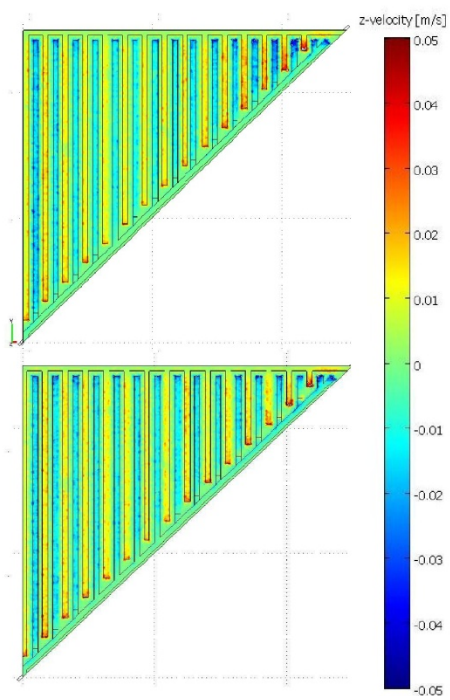

Figure 11: Comparison of velocities towards PEM in GDL for iterative method (bottom) and un-gated design (top)

5. Conclusions

The work has developed an iterative solver through Comsol and Matlab to optimize the existing bio-inspired flow channel design on bipolar plates of fuel cells. A gate was added to each channel to evenly distribute the reactant to the reaction site. The gate concept's ability to modify and improve the flow to the PEM looks promising. The results that have been obtained provide evidence for the capabilities of the developed iteration program to modify the difference in pressure across inlet and outlet channel pairs. The analytical model has been shown to have limited use currently but may become useful when flow from runners to channels is accounted for. The effects on larger cells with many additional channels would also be an interesting direction of study.

6. References

1. J. Kloess, X. Wang, J. Liu, Z. Shi, and L. Guessous, Investigation of bio-inspired flow channel designs for bipolar plates in proton exchange membrane fuel cells, *J. Power Sources*, **188**, 132-140 (2009)
2. B. Munson, D. Young, and T. Okiishi, Fundamentals of Fluid Mechanics, USA (2006)

7. Acknowledgements

This research was supported by the National Science Foundation International Research Experience for Students (IRES) Program and the award number is #0853572.