# Effect of cross-sectional geometry in flow behaviour: A steady state computational study

Vijai Kaarthi Visvanathan[1], Karthikeyan Palaniswamy [2], Dineshkumar Ponnaiyan[3], Boopathi Rathinam[4]
{vijaikaarthivexam@gmail.com[1], apk.auto@psgtech.ac .m[2],
dineshponnaiyan@gmail.com[3], Bhoobathi.Rathinam@in.bosch. com[4]}

Department of Automobile Engineering, PSG College of Technology, Coimbatore, India [1,2,3,4]

**Abstract.** In this article, the effect of cross-sectional geometry on the flow behaviour of hydrogen in a polymer electrolyte membrane fuel cell is studied using a steady state computational model. Three geometric shapes: semi circular, triangular and rectangular are considered in this study. The results show that the triangular geometry has the highest velocity and pressure drop in the channel, while the lowest is by the rectangular geometry. The velocity profiles of all three geometries are found to be parabolic in the fully developed region, with the triangular geometry having the centroid at 113rd of the base height and hence the maximum velocity shifts to that line. The turbulent kinetic energy values are found to be highest at the inlet and saturates at fully developed regions. From this analysis, a rectangular section (or) a semicircular section is having uniform flow distribution and less turbulence and is suitable for the fuel cell flow applications.

**Keywords:** Fluid flow, Turbulent kinetic energy, Pressure drop, Velocity distribution, Geometric Effects.

## 1 Introduction

Proton exchange membrane fuel cells are a promising technology for clean and efficient power generation. Essentially, they convert the chemical energy of the reactant gases into electrical energy, without combustion. These fuel cells are useful in a variety of applications like stationary power generation, mobile

One of the key challenges in the development of proton exchange membrane fuel cells is the design of flow channels. The flow channels must be designed to accommodate a high flow rate of gases to the electrodes while minimising the pressure drop and turbulence [1]. The cross-sectional geometry of the flow channel is an important factor that affects the behaviour of the gases. In this article, we investigate the effect of cross sectional geometry on the flow behaviour of the hydrogen

ICSETPSD 2023, November 17-18, Coimbatore, India

DOI 10.4108/eai.17-11-2023.2342862

across three different geometries: semi circular, triangular and rectangular. In this study, we compare the velocities, pressure and turbulent kinetic energy profiles for these three geometries. The effect of cross-sectional geometry on the flow behaviour of the fluids has been extensively studied in the literature. Certain studies investigated the effect of channel geometry on the pressure drop in PEMFCs and established that the pressure drop was inversely proportional to hydraulic diameter [2]. These studies concluded that the pressure drop was higher for the sharp comers as well. Another study investigated the effect of geometry on the mass transport for PEMFCs. They found that the mass transport for the channel with larger aspect ratio was higher [3]. The mass transport was lower at the sharp comers as well. Other studies investigated the effect of geometry on the flow distribution and heat transfer in PEMFCs [4]. They found that the flow distribution was more uniform and heat transfer was effective for channels with larger aspect ratio. On the other hand, the flow uniformity and heat transfer were reduced at the sharp comers as well.

## 2 Methodology

### 2.1 Numerical Details

The CFD simulations were performed using ANSYS Fluent 2023 Student Version. The computational domain was a I 00 mm long channel, with different cross-sectional areas, respectively accommodating to the particular geometry. In this study, three geometrical sections (semi-circular, triangular and rectangular) are considered and the dimensions are tabulated below (Table 1). Meshing is performed using patch conformance with tetrahedrons. The minimum element size is parameterized for the grid independence study. The value of converging velocity is used to perform the mesh number test. The mesh element size is maintained the same for other sections as well. The results of grid independence are tabulated below (Table 2). From this, 1e-4 m is selected as the mesh element size for the study.

**Table 1:** Dimensions of the Geometry

| Serial Number | Name of the Geometry | Length of the channel (mm) | Width of the channel(mm) | Depth of the channel(mm) |
|---|---|---|---|---|
| 1 | Semi-circular | 100 | 2 | 1 |
| 2 | Triangular | 100 | 2 | 1 |
| 3 | Rectangular | 100 | 2 | 1 |

During the solution procedure, the following conservation relations are taken into consideration:

i) Continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \tag{1}$$

**Table 2:** Grid-Independence Test

| Serial Number | Geometry | Minimum mesh element size (m) | Number of elements | Velocity (m/s) |
|---|---|---|---|---|
| 1 | Semi-circular | 1.04e-4 | 8,26,497 | 1.3026 |
| 2 | | 1.02e-4 | 8,39,494 | 1.3026 |
| 3 | | 1.00e-4 | 8,88,531 | 1.3026 |
| 4 | | 0.98e-4 | 9,37,495 | 1.3026 |

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot \left( \rho . u \bigotimes u \right) = -\nabla \rho + \Delta \cdot \tau + \rho g \tag{2}$$

ii) Navier Stokes equation:

SST - kw model is used to capture the turbulence effect for this low Reynolds flow study [5] as advised from literature. The fluid under study is hydrogen. The properties are tabulated in the table below (Table 3). The simulation is run using an iterative solver for studying the steady state characteristics of the system. The pressure and velocity plots were analysed to determine the flow behaviour of hydrogen in the three geometries.

**Table 3:** Properties of hydrogen used for flow simulation

| | |
|---|---|
| Density (kg/$m^3$) | 0.08189 |
| Viscosity (kg/ms) | 8.411 e-6 |

## 2.2 Model Validation

In order to ensure that the model accurately models the physical system of interest, we validate with velocity obtained from analytical calculations.

$$Q = Av \tag{3}$$

$$\frac{\hat{m}}{\rho} = \frac{\pi r^2}{2 \cdot v} \tag{4}$$

v = 1.298 m/s

The average velocity as is estimated by the simulation for a circular mesh is 1.3 m/s. This difference is due the weighted average of velocity taken at the outlet, taking into effect the boundary layer development. Since the problem is quite simple, this model validation is applicable for further studies undertaken in this article.

# 3 RESULTS AND DISCUSSIONS

## 3.1 Effect of geometry on velocity:

The velocity profiles for all the three geometries are shown in figure 1. The triangular geometry has the highest maximum velocity, followed by the semi circular geometry and the rectangular geometry. The maximum velocity of the triangular geometry is 4.4 m/s, compared to 2.6 m/s for the semi-circular geometry and 2.0 m/s for the rectangular geometry.

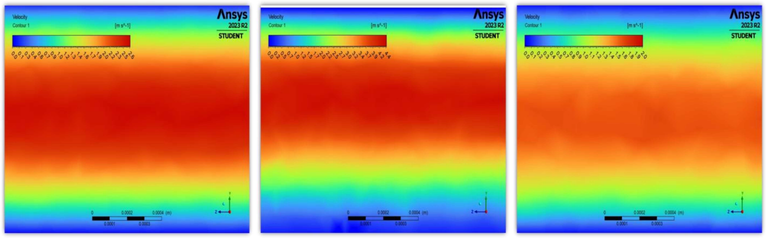

**Fig. 1.** Distribution of velocity as a function of geometry in flow channels at fully developed regions. (a) Semi circular geometry (b) Triangular geometry and (c) Rectangular geometry

The velocity profile along the z-length for fully developed flow is a characteristic feature of fluid flow in channels and pipes. In fully developed flow, the velocity profile remains constant along the z-length, indicating that the flow is no longer influenced by the entrance conditions. The velocity profile for fully developed flow is typically parabolic in shape, with the maximum velocity occurring at the centre of the channel or pipe and decreasing towards the walls. On the entry, due to sudden involvement to boundary layer as fluid starts to enter the channel, turbulent profiles (figure 2) are predominant with respect to the symmetry of the geometry as shown in the figure below. The velocities for semi-circular, triangular and rectangular regions are 1.3 m/s, 2.05 m/s and 1.01 m/s respectively. The figure below (figure 3) illustrates the parabolic velocity profile for the channel at fully developed region. The velocities for semi-circular, triangular and rectangular regions are 2.5 m/s, 4.2 m/s and 1.8 m/s respectively.

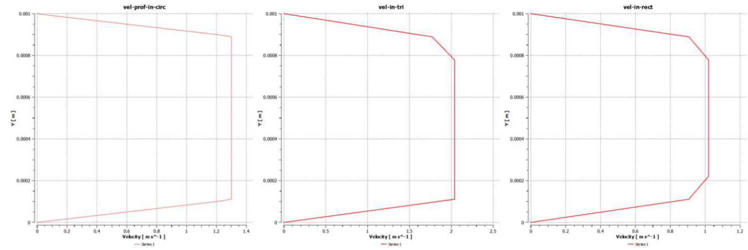

**Fig. 2.** Distribution of velocity at the entry region in flow channels (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

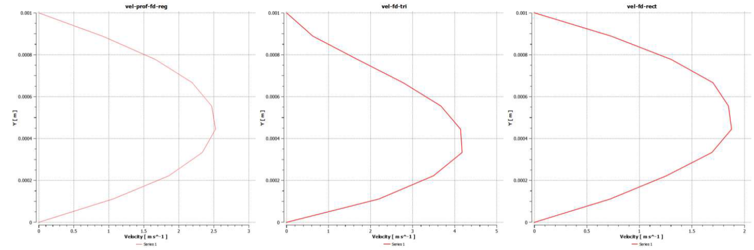

**Fig. 3.** Distribution of velocity at the fully developed region in flow channels (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

The development of the velocity profile from the entrance to the fully developed region is a result of the interaction between inertia and viscous forces. As the fluid enters the channel or pipe, it experiences a sudden change in velocity, which creates a boundary layer near the walls. The boundary layer is a thin layer of fluid where viscous forces are dominant and the velocity is reduced. As the fluid travels downstream, the boundary layer grows thicker and the velocity profile gradually develops until it reaches a fully developed state.

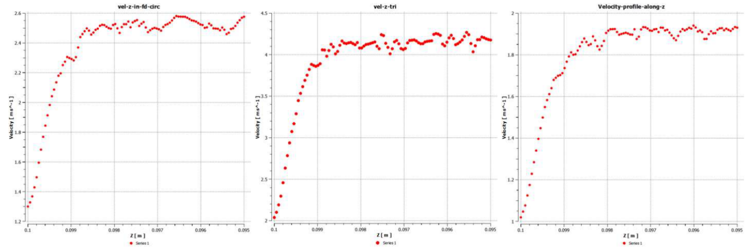

**Fig. 4.** Distribution of velocity along z-direction - Entrance to fully developed region. (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

The length of the entrance region required for the flow to become fully developed depends on the Reynolds number of the flow. The Reynolds number is a dimensionless parameter that characterizes the ratio of inertial forces to viscous forces in a flow. For low Reynolds number flows, the entrance region is relatively short, while for high Reynolds number flows, the entrance region can be quite long. In this case, all three geometries show a similar entrance length of 2 mm, illustrated from the above figure (figure 4).

### 3.2 Effect of geometry on pressure:

Pressure plots are used to visualize the pressure distribution in the fluid flow. Pressure in a flow domain varies with respect to location and the instantaneous velocity. Navier Stokes equations are simultaneously solved for iteration until 1e-3 of residue, for pressure plot generation. The pressure drop in fluid flow relates to the energy loss in fluid due to friction against the walls of the channels. The triangular geometry has the highest pressure drop because it has the largest surface area of contact with the fluid. In addition to the surface area, the shape of the geometry also affects the pressure drop: the sharp edge of a triangular flow geometry creates a high flow separation, thus leading to high pressure drop. The pressure plots for the three geometries are shown in the figure below (figure 5). The triangular geometry has the highest pressure drop, followed by the semi-circular geometry and the rectangular geometry. The pressure drop for the triangular geometry is 67.1 Pa, compared to 23.7 Pa for the semi-circular geometry and 15.4 Pa for the rectangular geometry.

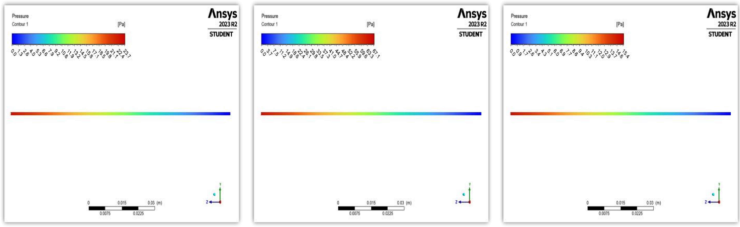

**Fig. 5.** Distribution of pressure drop along the length of the channel (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

### 3.3 Effect of geometry on turbulent kinetic energy:

Turbulent kinetic energy measures the average kinetic energy of the eddies in a turbulent flow. This is a very the walls of the channels. Here, the velocity is reduced, and turbulence is high, as shown in the figure below (figure 6). At the entrance the TKE values for semi circular, triangular, rectangular geometry for the given important factor that affects the flow behaviour, heat transfer and mass transfer in fluid flow. At inlet, the TKE values are at the highest due to sudden change in velocity; this disturbance creates boundary layer near boundary gives 0.0065 $m^2s^{-2}$, 0.016 $m^2s^{-2}$ and 0.004 $m^2s^{-2}$ respectively.

As we move away from the entrance to the fully developed region, the boundary layer thickens, and turbulence intensity reduces. In the fully developed region (figure 7), the flow is no longer influenced by the entrance condition or the parabolic velocity profile. However, the TKE values are still non-zero due to the interaction of flow with eddies. In the fully developed region, the TKE values for semi-circular, triangular, rectangular geometry for the given boundary gives 2e-10 $m^2s^{-2}$, 3.Se-10 $m^2s^{-2}$ and 2.4e-11 $m^2s^{-2}$ respectively.

The triangular geometry has the highest TKE values at both the inlet and the fully developed region because it has the sharpest comers. The sharp comers create areas of high flow separation, which can increase the turbulence intensity and TKE values. The semi-circular and rectangular geometries, on the other hand, have smoother walls and less flow separation, which contributes to their lower TKE values.

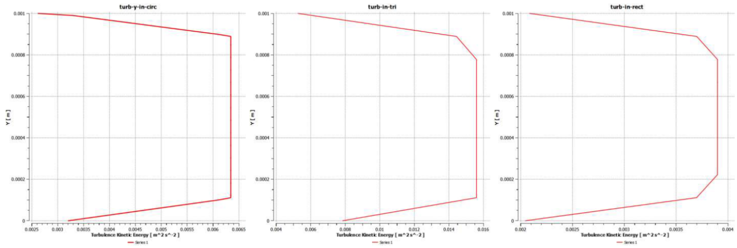

**Fig. 6.** Distribution of turbulent kinetic energy at the inlet of the channel (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

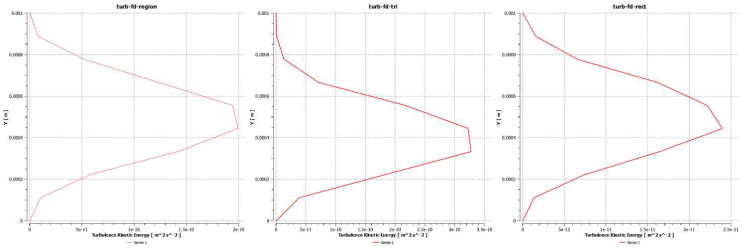

**Fig. 7.** Distribution of turbulent kinetic energy at the fully developed region of the channel (a) Semi-circular geometry (b) Triangular geometry and (c) Rectangular geometry

# 4 CONCLUSIONS

The study's findings demonstrate that the cross-substantial impact on the flow behaviour of hydrogen in a PEMFC. This implies that the shape of the channels can significantly influence how hydrogen flows through the channels, thus affecting the performance of the fuel cell.

## 4.1 Triangular Geometry: Highest Velocity and Pressure Drop

The triangular geometry exhibited the highest maximum velocity and pressure drop among the three geometries examined. This suggests that hydrogen triangular channels compared to semi-circular or rectangular channels. This could be attributed to the triangular geometry's sharp comers, which can induce flow separation and turbulence, leading to increased pressure drop.

## 4.2 Parabolic Velocity Profiles

The velocity profiles for all three geometries were found to be parabolic in the fully developed region. This indicates that the velocity distribution across the channel becomes more uniform as the flow progresses downstream. The triangular geometry, however, exhibited a shift in the maximum velocity towards the centroid, which is located at 1/3 of the base height. This deviation from the typical parabolic profile could be due to the triangular geometry's unique shape and the influence of its comers.

### 4.3 Highest TKE Values at Inlet and Lowest in Fully Developed Region

The TKE values were found to be highest at the inlet and lowest in the fully developed region for all three geometries. This trend indicates that turbulence is most intense at the beginning of the flow path due to the sudden change in velocity as the fluid enters the channel. As the flow progresses downstream, the turbulence intensity decreases, leading to lower TKE values.

### 4.4 Implications for PEMFC Design

The findings of this study have important implications for the design of PEMFCs. The choice of cross sectional geometry for the flow channels can significantly impact the flow behaviour of hydrogen and, consequently, the overall performance of the fuel cell. Triangular channels, while exhibiting higher maximum velocity and pressure drop, could be advantageous in applications where rapid hydrogen transport is crucial. However, for applications requiring lower pressure losses and reduced turbulence, semi circular or rectangular channels might be more suitable.

Overall, this study provides valuable insights into the relationship between cross-sectional geometry and flow behaviour in PEMFCs. By understanding these relationships, engineers can optimize flow channel design to improve PEMFC performance and efficiency.

**Table 4:** LIST OF ABBREVIATIONS

| Abbreviation/ Symbol | Explanation/ Units |
|---|---|
| PEMFC | Polymer/Proton Electrolyte/Exchange Membrane |
| TKE!k | Turbulent Kinetic Energy/ $m^2s^{-2}$ |
| $\rho$ | Density/ $kgm^{-3}$ |
| u | Velocity Tensor/ $ms^{-1}$ |
| $\nabla$ | Divergence |
| g | Acceleration due to gravity/ $ms^{-2}$ |
| p | Pressure/ $Nm^{-2}$ |
| $\tau$ | Time/ s |
| T | Shear Stress/ $Nm^{-2}$ |
| $\otimes$ | Cross Product |

## Acknowledgement

The authors would like to acknowledge the Science and Engineering Research Board (SERB) for funding this research under Grant No. 005159/2021-22. This support is gratefully appreciated.

# References

[1] Arenas LF, Ponce de León C, Walsh FC. Critical Review—The Versatile Plane Parallel Electrode Geometry: An Illustrated Review. Journal of The Electrochemical Society. 2020 Jan;167(2):023504. Available from: `http://dx.doi.org/10.1149/1945-7111/ab64ba`.

[2] Wilberforce Awotwe T, Hassan Z, Ogungbemi E, Ijaodola O, Nisar F, Durrant A, et al. A comprehensive study of the effect of bipolar plate (BP) geometry design on the performance of proton exchange membrane (PEM) fuel cells. Renewable and Sustainable Energy Reviews. 2019 05;111:236-60.

[3] Shen J, Tu Z, Chan SH. Enhancement of mass transfer in a proton exchange membrane fuel cell with blockage in the flow channel. Applied Thermal Engineering. 2019;149:1408-18. Available from: `https://www.sciencedirect.com/science/article/pii/S1359431118365293`.

[4] Yu Y, Zhan Z, He L, Yang X, Wan X, Sui PC, et al. Effects of Distribution Zone Design on Flow Uniformity and Pressure Drop in PEMFC. Journal of the Electrochemical Society. 2021 Sep;168(9):094505.

[5] Asnaghi A, Svennberg U, Bensow R. Evaluation of curvature correction methods for tip vortex prediction in SST kOmega turbulence model framework. International Journal of Heat and Fluid Flow. 2019 02;75:135-52.