Special Issue for International Conference of Advanced Materials Engineering and Technology (ICAMET 2013), 28-29 November 2013, Bandung Indonesia

Advances in Environmental Biology, 7(12) October Special Issue 2013, Pages: 3649-3653

FSS Microchannel Fluid Flow Profile Investigation at High and Low Re Number

[1]Nor Shakirina Nadzri, [1]Vithyacharan Retnasamy, [1]Zaliman Sauli, [1]Tan Hsio Mei and [2]Hussin Kamarudin

[1]School of Microelectronic Engineering, Universiti Malaysia Perlis (UniMAP), Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.
[2]School of Materials Engineering, Kompleks Pusat Pengajian Jejawi 2,Taman Muhibbah, Universiti Malaysia Perlis, 02600 Jejawi, Arau, Perlis, Malaysia.

A R T I C L E I N F O

Article history:
Received 11 September 2013
Received in revised form 21 November 2013
Accepted 25 November 2013
Available online 5 December 2013

Key words:
Microfluidic;Forward Facing Step;Velocity Profile

A B S T R A C T

The fundamental understanding of dynamic fluid flow behavior in different geometry channel is crucial due to transport phenomena influence on the key design and process control of the microfluidic systems. Recently, the Computational Fluid Dynamics (CFD) technology has received priority to fully understand the performance of the microfluidic design. In this paper, simulation of liquid flow over forward facing step (FFS) microchannel has been explored using CFD-Ansys software. This work focused on velocity profiles for low and high Reynolds (Re) numbers. Different step heights were used as main parameter. The results revealed a parabolic profile across the x-axis channel. Besides that, recirculation zone is detected near the step for Re=500. An increase for step height value contributed to higher fluid flow velocity.

INTRODUCTION

Microfluidics research has been burgeoning tremendously over the past decade. Since its inception, microfluidics contribute enhancement in various application such as biomedical industry, chemical separation process, pharmaceuticals and agriculture field [1,2]. The obvious benefits offered include miniaturization especially reduction of material consumption and improvement of analytical performance for instance shorter response times and better accuracy [3,4]. Fundamental understanding of fluid flow behaviorin microchannel is vital for the design and process control of the microfluidic system [5]. Several publications addressed the investigation of fluid flow characteristics inside different cross-section microchannels have been topicsin recent years such as rectangular, circular, triangular and elliptical [3]. Jiann-Cherng, Jyh-Tong and Ralph (2010) employed curved rectangular microchannelto study the hydrodynamic behavior of the water flow with different rectangular diameters and curvature ratios. Both experimental method and numerical simulation conducted generating similar trend of the pressure drop distribution over mass flow rate. The authors also observed reduction of channel width producing an abrupt increase in the pressure drop.

Z.Sauli, S.Taniselass, T.K Ramasamy andV.Retnasamy investigated velocity profiles for liquid flow in obstructed straight square microchannel with no slip and free slip boundary conditions. The low and high Reynolds numbers were adapted in the simulations to differentiate the velocity magnitude for both boundary conditions [6]. The highest velocity magnitude is noticeable at the bump location for both conditions due to acceleration by some molecules after liquid hitting the front space of obstacle. Slip velocity is observed to occur under no slip condition for low and high Reynolds numbers but not for free slip condition. An experimental technique by N.Fujisawa, Y.Nakamura, F. Matsuura and Y. Sato studied the pressure and velocity distribution in the microchannel with bifurcation and confluence geometry. The flow behavior for both geometries showed opposite characteristics in term of velocity and pressure contour. At the intersection of both channels, the bifurcation flow exhibited decreasing velocity while the velocity for confluence flow was increased. Since the pressure is strongly related to velocity distribution, the pressure contour in the bifurcation flow is observed to reduce abruptly in the upstream channel. Meanwhile, strong opposition expressed by the confluence flow. The interest of this paper is to study the fluid flow velocity profile in a square microchannel with forward facing step (FFS) configuration. Paper written by DS Pearson, PJ Goulart, and B Ganapathisubramani stated that although the flow over FFS configuration is essential for basic of fluid dynamics research and applied engineering design, still very less research done compared to exploration on the backward facing step scope [8]. The simulation environment is conducted based on different step heights with low and high Reynolds numbers.

Methodology:

The three-dimensional computational model created for this work was using Anys-CFD software. The model was meshed until 2560k nodes to generate the most accurate result. Fig. 1 showed the microfluidic design with liquid media using water. Therefore, the the value of density was 997.0479 kg/m$_3$ and viscosity was 8.90 x 10^{-4} kg/m.s at 25^{o}c.The model has been treated as no slip boundary condition with steady-state fluid flow.

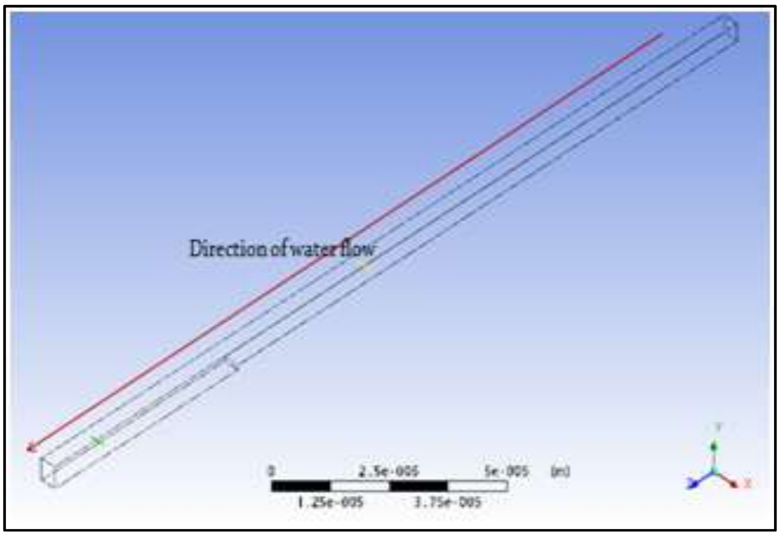

Fig. 1: Method of analysis of velocity before and after step height.

Schematic description of the FFS microchannel was illustrated in Fig. 2. The length of microchannel was 1000 µm and step height located at 550 µm from inlet channel after considering for full flow development. L_i is the distance between step and inflow, L_a is the distance between step and outflow. Channel height and step height were denoted by H and h respectively. Further, the inlet cross section was designed with dimension of 4 µm x 4 µm with three different step heights, h, which were 1 µm, 2µm and 3µm.

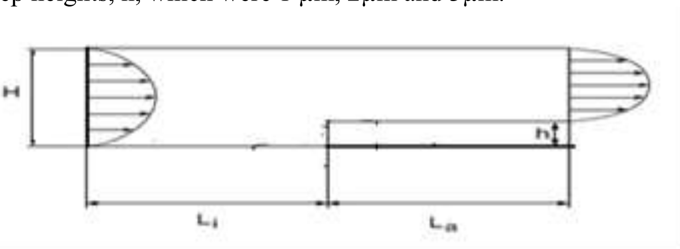

Fig. 2: FFS geometry design.

In this work, the simulations were performed based on low and high Reynolds (Re) numberto investigate the velocity distributions of the water flow at location of 750µm from inlet channel. Table 1 showed detail inlet velocities according to Reynolds numbers used in the simulation.

Table 1: Inlet Velocity For Re 1 And 500.

Re	Inlet velocity (ms^{-1})
1	0.223
500	111.580

RESULT AND DISCUSSION

The velocity value obtained for X-axis at distance of Z = 750 µm was summarized as per Table 2.

Table 2: VELOCITY VALUE AT 0μM, 2 μM AND 4 μM OF X-AXIS FOR RE 1 AND 500.

Step height	Re numbers	0 μm	2 μm	4 μm
1μm	1	0.0092	0.6127	0.0121
1μm	500	35.2383	285.408	35.8842
2μm	1	0.0268	0.8774	0.0235
2μm	500	79.3085	368.981	76.5427
3μm	1	0.0865	1.5599	0.0822
3μm	500	231.691	577.621	226.528

Then, the result was plotted as in Fig. 3 (a-c) for step height 1μm, 2μm and 3μm respectively. Same graph pattern was obtained for all step heights. At low Re, viscous interaction between the wall and the fluid is strong hence produce slow fluid flow regime with no turbulence or vortices. Laminar flow condition is identified for low Re. While for high Re, the liquid experienced higher momentum and causes greater velocity distributions. The flow velocity at the centerline and channel wall was noticeably different. This parabolic flow distribution showed that the velocity of the fluid increase as moving toward the center of the channel.The highest velocity between the three steps height occurred at step height 3μm due to the smallest cross section area after step compared to the 2μm and 1μm step height. The outlet size of the step height 3μm is 4μm x 1μm. When the fluid flowed from bigger geometry area to smaller area, the fluid's kinetic energy increased thus fluid velocity arises. As a summary, the velocity is inversely proportional to the cross section area.

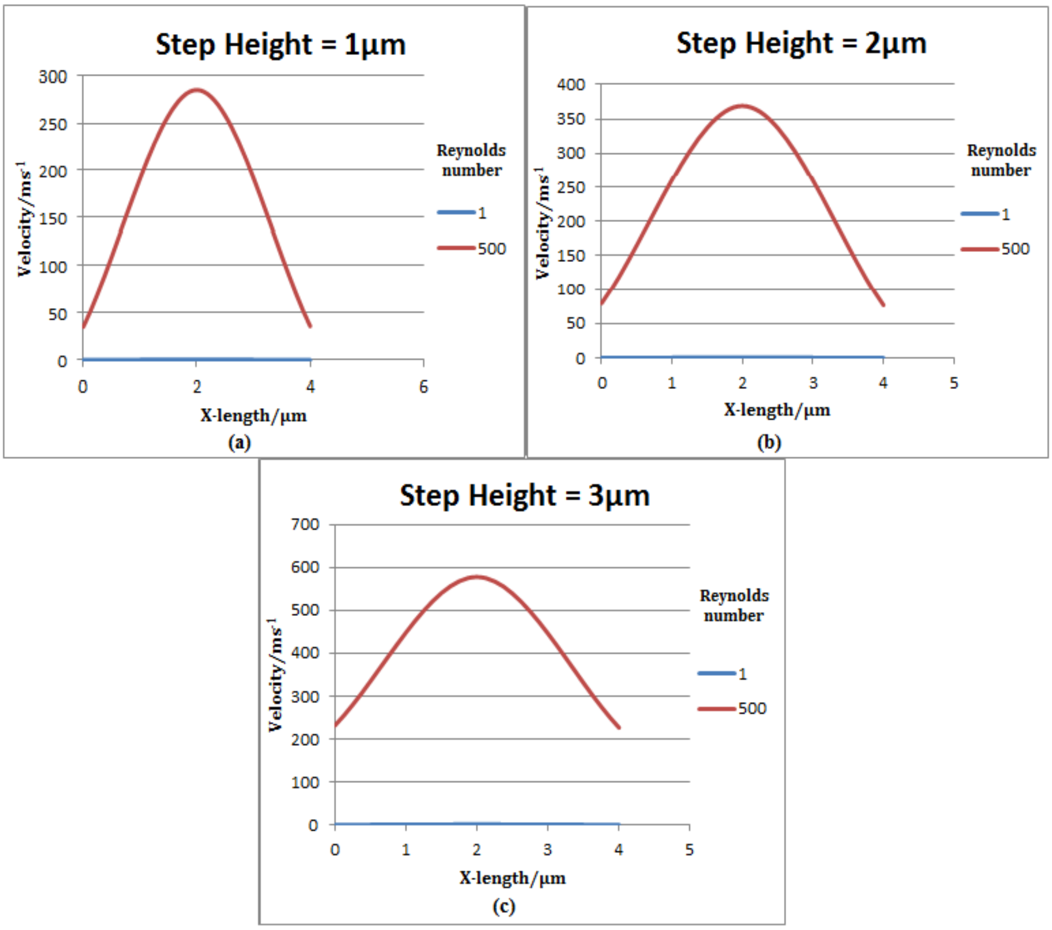

Fig. 3: Velocity profile for Re 1 and 500 at Z=750μm for step height (a)1μm (b)2μm and (c)3μm.

As a comparison, velocity profile for all step heights for Re 500 was illustrated in Fig. 4. It is clearly showed that step height 3μm performed the highest velocity distribution followed by step height 2μm and 1μm. The peak velocity occured at 2μm on X-axis which represented the center of the channel.

Fig. 5(a-c) illustrated velocity streamline analysis for this FFS configuration for both low and high Re. Creeping flow pattern was observed at low Re (Re=1) for all step height as per Fig. 5a(i), b(i) and c(i). The layers of this laminar flow do not mix with neigbouring layers and flow smoothly from inlet towards outlet. The minimum velocity is obviously appeared at the wall region which represented by the blue streamline. On the contrary, rough pattern streamline is viewed for Re=500. Higher inlet velocities naturally cause high kinetic energy for the water in the inlet direction. Therfore, higher Re contributes higher velocity of fluid flow. When some molecules hit the step, collision occurred among some molecules that cause the neighbouring molecules to accelerate. Consequently, high Re may contribute to recirculation regime near the step height.From Fig.5a(ii), b(ii) and c(ii), we noticed that the recirculation at the step height 1µm is small comparing to the step height 2µm and 3µm. Since the cross section area after stepforheight 3µmis the smallest, the highest inlet velocity and kinetic energy were produced. Once reach the wall step height, there have a very high drag force between wall step and fluid flow, so more recirculation zone occur at the step height.

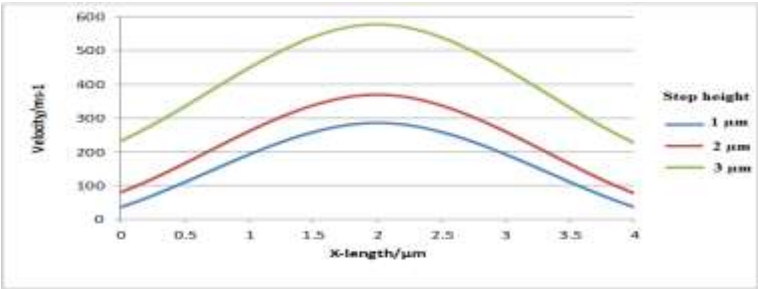

Fig. 4: Velocity profile for step height 1µm, 2µm and 3µm comparison.

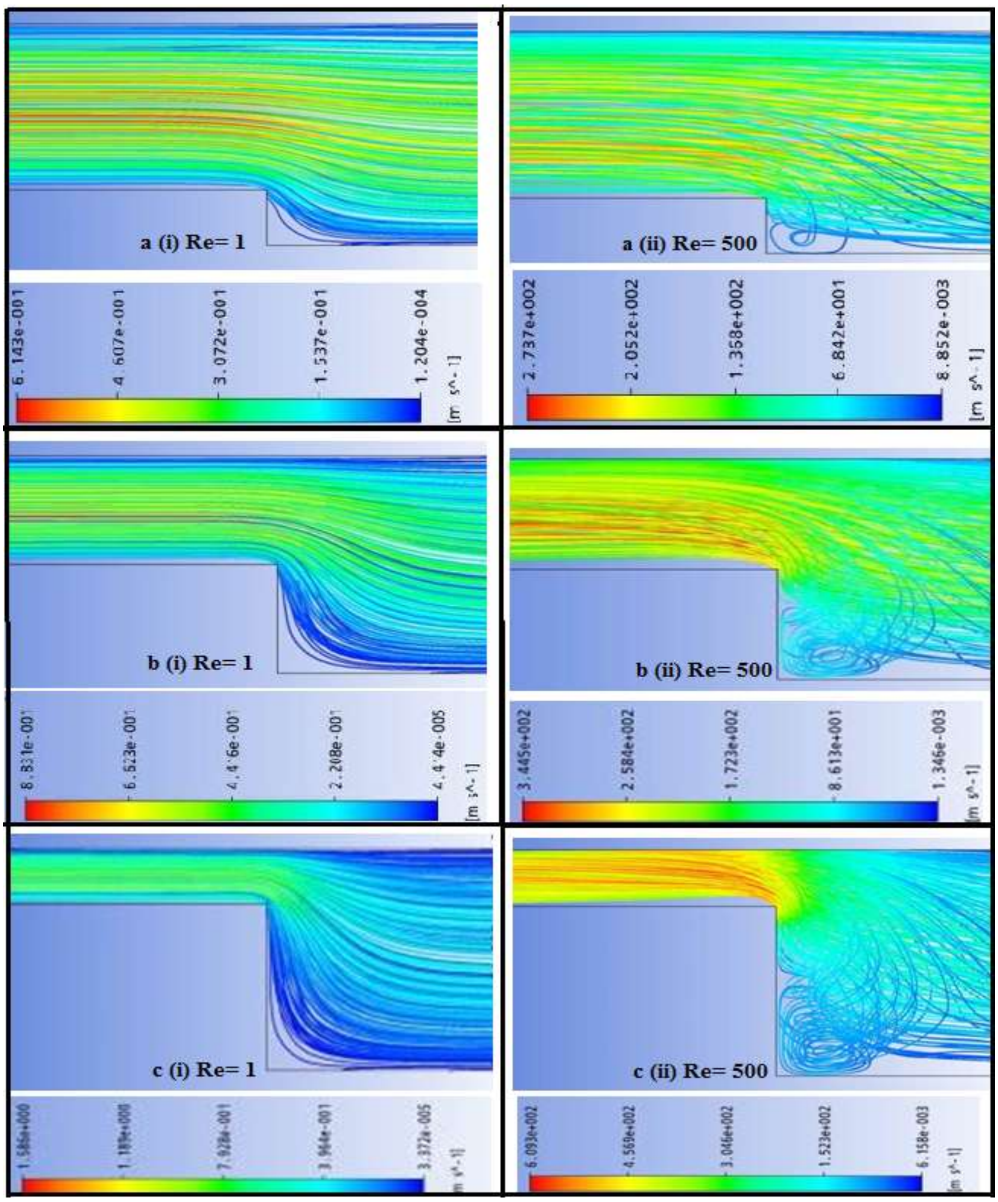

Fig. 5: Velocity streamline for step height (a)1µm (b)2µm and (c)3µm for Re (i)1 and (ii)500.

Conclusion:

The effect of geometrical feature of the microchannel on fluid behavior has been investigated. In this study, the velocity distribution development over forward facing step configuration is analyzed. The maximum of flow velocity is observed at the channel center but approaching zero at the wall region. Laminar flow is observed at low Re. But the turbulent flow may occur at high Re. Higher height of the step, higher speed of the fluid flow generated after the fluid travels through the narrower channel.

ACKNOWLEDGMENT

The authors would like to thank and acknowledge the School of Microelectronic Engineering,Universiti Malaysia Perlis for their support and facility. The authors appreciation are extended to the Ministry of Higher Education for the support given.

REFERENCES

[1] Judy, J., D. Maynes, B.W. Webb, 2002. “Characterization of frictional pressure drop for liquidflows through microchannels,” International Journal of Heat and Mass Transfer 45.

[2] Uda Hashim1, P.N.A. Diyana 2, Tijjani Adam, 2012. “Numerical Simulation of Microfluidic Devices,” IEEE-ICSE2012 Proc.

[3] Tuba Bayraktar and Srikanth B. Pidugu, 2006. “Characterization of liquid flows in microfluidic systems,” International Journal of Heat and Mass Transfer 49.

[4] Nicolas Szita, Karen Polizzi, Nicolas Jaccardand Frank Baganz, 2010. “Microfluidic approaches for systems and synthetic biology.

[5] Jiann-Cherng Chu, Jyh-Tong Teng and Ralph Greif, 2010. “Experimentalandnumericalstudyonthe flow characteristicsncurved rectangularmicrochannels.

[6] Sauli, Z., S. Taniselass, T.K. Ramasamy, V. Retnasamy and P. Poopalan, 2010. “No Slip and Free Slip Boundary Conditions for Liquid Flow in Obstructed StraightMicrochannel,” Second International Conference on Computational Intelligence, Modelling and Simulation.

[7] Fujisawa, N., Y. Nakamura, F. Matsuura and Y. Sato, 2006. “Pressure field evaluation in microchannel junction flows through µPIV Measurement.

[8] Pearson, D.S., P.J. Goulart and B. Ganapathisubramani, 2011. “Investigation of turbulent separation in a forward-facing step flow,” 13th European Turbulence Conference.