The International Journal of Engineering and Science (IJES)
ISSN (e): 2319 – 1813 ISSN (p): 23-19 – 1805
|| Pages || PP 16-21 || 2020 ||

Numerical analysis of heat transfer characteristics of Taylor slug flow through microchannel with obstruction

Akash Patel[a], V. R. K. Raju[b,*]
[a]*Mechanical enginnering department, National Institute of Technology, Warangal,*
Telangana State - 506004, India

ABSTRACT

Taylor slug flow through microchannel has attracted many researchers because of its capability of improving thermal performance of a system. The Global Nusselt number for Taylor flow is always higher than that of simple primary fluid flow through same microchannel. This enhanced Nusselt number can be further enhanced by droplet manipulation techniques which is the aim of present study. In present novel study, analysis of Taylor flow through rectangular microchannel with obstruction in path has been carried out using ANSYS FLUENT 15.0. The Nusselt number for simple primary fluid flow and Taylor flow with and without obstruction through same microchannel is compared. For simple primary fluid flow the velocity variation along radial line is verified against existing equations and found to be in good agreement. From results it is found that heat transfer rate (Global Nusselt number) indeed increases if cylindrical obstruction is to be provided on the path of Taylor flow. To be exact, 76 % increment is observed in Taylor flow with obstruction in path compared to simple primary fluid flow. Peaks in local Nusselt number were observed near cylindrical obstruction region. Due to cylindrical obstruction pressure drop is found to be increased by 196.94 % compared to simple primary fluid flow.

KEYWORDS;-*Taylor slug flow, Obstruction,Heat transfer, microchannel*

I. INTRODUCTION

Numerous researchers have contributed their efforts in extensively studying the flow physics, heat transfer behavior of Taylor flow and convincingly explained its potentiality over single-phase flow (Gupta et. al.[6], Talimi et. al.[13], Muzychka et. al.[11]). They found Nusselt number for Taylor flow is significantly higher than single phase flow through same microchannel. Bandara et.al.[12] analysed the effect of various parameters on heat transfer taking place in case of liquid-liquid(Water-oil) Taylor as well as sliding flow through cylindrical channel and found 200 % increment in Nusselt number. Numerical investigation was done for actual electronic chip cooling by them. The numerical investigation done by Qian et. al. [10]and Shano et. al. [11]lead to conclusion that inlet configuration of microchannel indeed affects the thermal performance. The inlet configuration in present numerical work can be achieved by nozzle practically.

From literature review it was concluded that the analysis of Taylor flow through non-circular microchannel has to be done in detail since only few studies have been reported on it till now. For rectangular microchannel more computational time is required if computational domain is to be taken as three dimensional. But Talimi et al[13] have suggested that the error between two dimensional and three dimensional results will be less if the length of droplet is higher than channel width and even less if contact angle is higher. Thus three dimensional computational domain can be easily reduced to two dimensional computational domain for a given problem. Link et. al.[8] have suggested two droplet manipulation techniques. One of which is to provide obstruction in the path of Taylor flow which is implemented in present study. Chung et al[4] have analyzed the effect of different type of obstruction on hydrodynamics of Taylor flow. They have suggested that heat transfer rate can be increased if such obstructions are provided in path of Taylor flow which is the aim of present study.

Computational domain

Consider two phase flow through a rectangular microchannel of width 100 micrometer with cylindrical obstruction as shown in Fig. 1. The diameter of cylindrical obstruction was taken as 50 µm to make the ratio of diameter of cylinder to width of channel to be 0.5. The distance of cylindrical obstruction from inlet was provided in such a way that obstruction lies in the center of hot region i.e. 2mm.

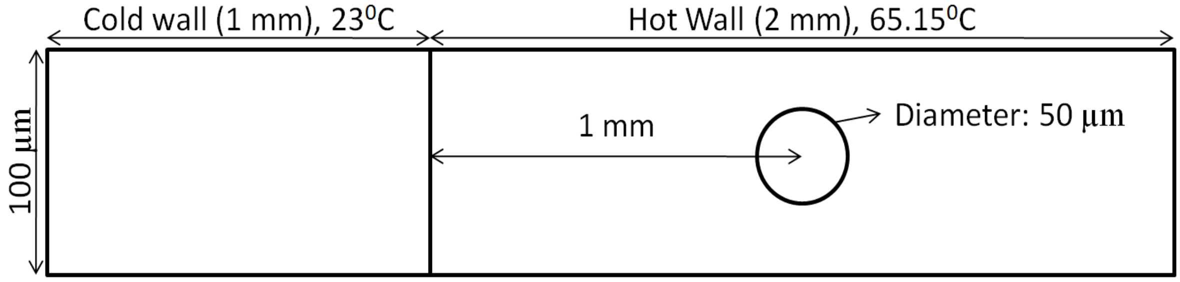


Fig. 1. Computational domain

Materials and boundary conditions

The Primary fluid was taken as water, the secondary fluid was taken as mineral oil and the wall material was taken as silicon. Material of cylinder was also taken as silicon. The properties of all three materials are mentioned in Table 1. The velocity boundary condition at inlet was taken as 0.22 m/s such that capillary number becomes 0.0044. Periodic boundary condition was taken using UDF at inlet such that water and oil enters periodically from middle 90 micrometer portion of inlet. While from other 10 micrometer portion water is allowed to enter. At outlet gauge pressure was taken as zero. The Dirichlet boundary condition was taken for walls as shown in Fig. 1 with reference to Bandara et al[12].

Material	Density [kg/m^3]	Specific heat [J/kg K]	Thermal conductivity [W/m K]	Viscosity [kg/m s]
Water	998	4182	0.6	0.001003
Light mineral oil	838	1670	0.17	0.023
Silicon	2330	710	149	-----------

Table 1: Material Properties

II. METHODOLOGY

To understand the difference in thermal performance of same microchannel with different case, four separate cases were taken (mentioned in Table 2.) and numerically analysed using ANSYS Fluent 15.0. The continuity, momentum and energy equation along with volume fraction function equation given by Hirt et. al.[7] were solved simultaneously in Fluent with transient analysis. The numerical methodology is explained in section 2.2. The results of all four cases were compared in terms of hydrodynamics and thermal point of view.

Mathematical modelling

The governing equations of the Taylor slug flow through microchannel are given by continuity, Navier-Stokes, energy equations as well as volume fraction function equation given by Hirt et. al.[5] as shown in Eqs. (1), (2), (3) and (4) respectively,

$$\frac{\partial \rho}{\partial t}+\nabla\left(\rho V\right)=0 \tag{1}$$

$$\rho\left[\frac{\partial V}{\partial t}+\left(V\cdot\nabla\right)V\right]=-\nabla p+\nabla\cdot\left[\mu\left(\nabla V+\nabla V^{T}\right)\right]+F_{s} \tag{2}$$

$$\frac{\partial\left(\rho E\right)}{\partial t}+\nabla\cdot\left[V\left(\rho E+p\right)\right]=\nabla\left(k\nabla T\right) \tag{3}$$

$$\frac{\partial F}{\partial t}+u\frac{\partial F}{\partial x}+v\frac{\partial F}{\partial y}=0 \tag{4}$$

where, Fs is a force given by continuum surface force(CSF) proposed by Brackbill et al.[3] as shown in Eq. (5)

$$F_{s}=\sigma k\frac{2g\rho\nabla\alpha_{s}}{\rho_{p}+\rho_{s}} \tag{5}$$

The curvature, k, is defined in terms of the divergence of the unit normal,

$$\boxed{k = \nabla \hat{n}} \tag{6}$$

Numerical Methodology

Two dimensional rectangular geometry was created with dimensions shown in Fig. 1. Near wall fine mess treatment as shown in Fig. 2.was done to effectively capture secondary fluid thickness around Taylor droplet. VOF model was used for interface tracking. To initialize the entire computational domain by parabolic boundary condition and to give periodic boundary condition at inlet separate User Defined Function (UDF) code was written and used. The variable time stepping method was chosen. The purpose of that was to maintain global Courant number of 0.25. According to that the time step size was found to be varying in-between 10^{-9} to 10^{-6}.

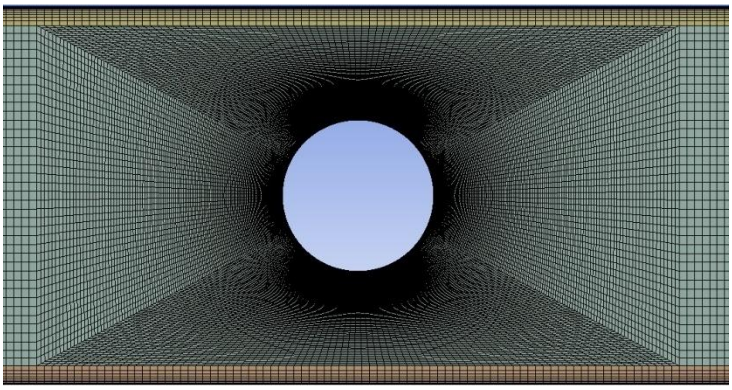
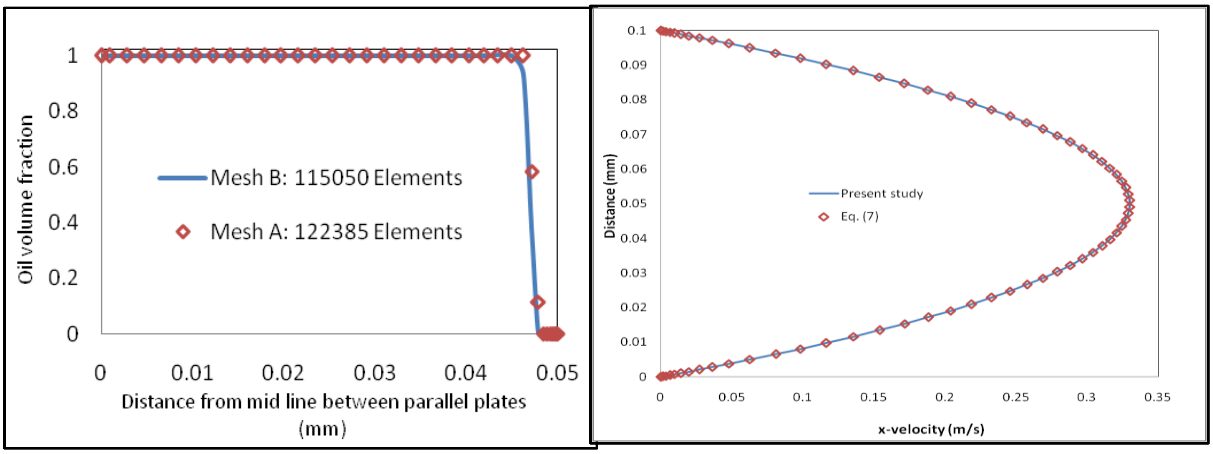


Fig. 2. Geometry and meshing

Grid independence and validation

Meshing of computational domain was done in such a way that the computational domain consists 122385 quadrilateral elements in case of Mesh A and 115050 quadrilateral elements in case of Mesh B. The grid independence test was carried out by comparing the variation of oil volume fraction along a line perpendicular to plates at the Centre of oil droplet in case of Mesh A and Mesh B as shown in Fig. 3(a). Mesh A is taken for further analysis since in Mesh A case oil volume fraction decreases from one to zero rapidly compared to Mesh B which indicates fine interface.



(a) (b)

Fig. 3 (a) variation of volume fraction along a line perpendicular to wall at centre of oil droplet (b) Variation of velocity in fully developed region

The verification of hydrodynamics of simple water flow was done by comparing velocity profile obtained from fully developed region and obtained from equation 7 which is found to be in good agreement.

$$\boxed{u_x = 6 \cdot u_{TP} \cdot \frac{y}{h}\left(1 - \frac{y}{h}\right)} \tag{7}$$

III. RESULT VIEW

To study hydrodynamics and heat transfer behavior of flow, four different type of cases were taken into consideration. These four different cases are mentioned in Table 2. The computational domain and mesh element size was kept same in all four cases except the obstruction part.

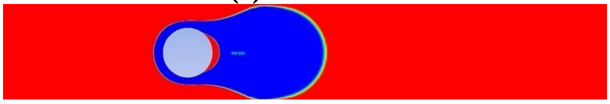
Case 1	Simple primary fluid flow without obstruction
Case 2	Primary fluid flow with obstruction
Case 3	Taylor Flow without obstruction
Case 4	Taylor Flow with obstruction

Table 1. Four different cases which are considered for analysis

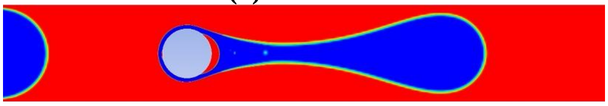
Flow Development

The development of flow with time in case 4 needs a little attention which is shown in Fig 4. From Fig. 4 it is clear that the oil droplet splits into two halves, passes over cylindrical obstruction and merges together once again after obstruction into single droplet. As time passes droplet gets elongated and splits leaving some portion of oil droplet behind stuck to cylinder. This is due to adhesive friction between cylindrical wall and droplet. The more the contact angle is there between wall and secondary fluid the more will be thee friction. In present work contact angle was taken as 160. This type of splitting of oil droplet from cylinder is a result of symmetrical mesh as shown in Fig. 2.

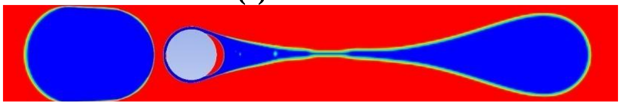
(a) 8.649 ms



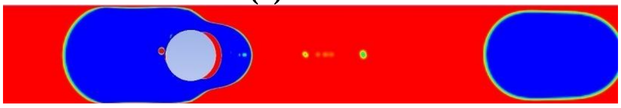
(b) 9.339 ms



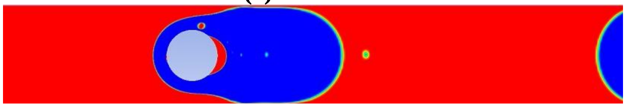
(c) 13.56ms



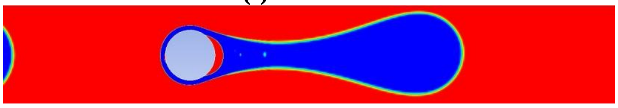
(d) 14 ms



(e) 14.2 ms



(f) 14.6 ms

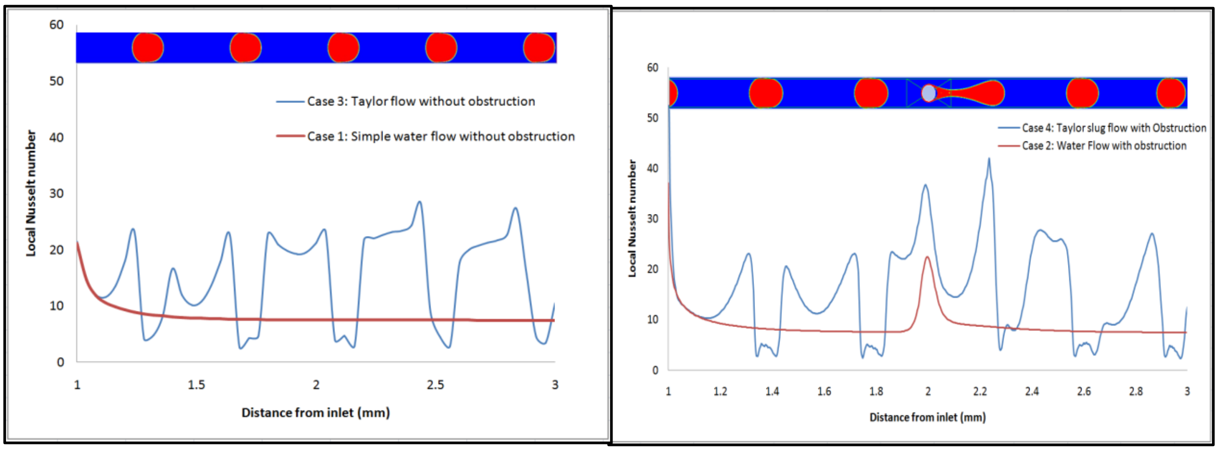


(g) 15.2 ms

Fig. 4. Flow development with time

Heat transfer and pressure drop

Local Nusselt number variations along the length of channel is shown in Fig. 5 (a) and (b), where red color indicates oil droplet. From Local Nusselt number variation in case 3 and 4 it was found that the value of Nusselt number is higher at trailing edge of the droplet then that of leading edge. This is due to the different strength of recirculation zones in contact with trailing and leading edge of droplet. In case three two extra recirculation zones were found near cylinder region. In case four extra recirculation zones at two different locations were found. Because of that peaks in local Nusselt number variation were found at the same locations. In Taylor flow, Taylor droplet shear pass through primary fluid and while doing that it forces primary fluid towards the channel wall at leading side which results in recirculation and better thermal performance. When obstruction is provided such recirculation zones gets disturbed and two new recirculation zones were found in-between obstruction and wall and at the location where thinnest primary fluid film is there. Consequently it results into increment in local Nusselt number.



(a) (b)

Fig. 5. Local Nusselt number variation along the length of channel in (a) case 1 & 3 (b) and case 2 & 4

The variation of static gauge pressure along the line (which is at equidistance from both wall) is shown in Fig. 6. From Fig. 6 it is observed that the pressure shoots up wherever there is secondary fluid region i.e. oil droplets within channel. The reason behind this is surface tension forces. Droplets always tries to minimize the volume occupied by them because of surface tension forces. So pressure inside the droplet increases.

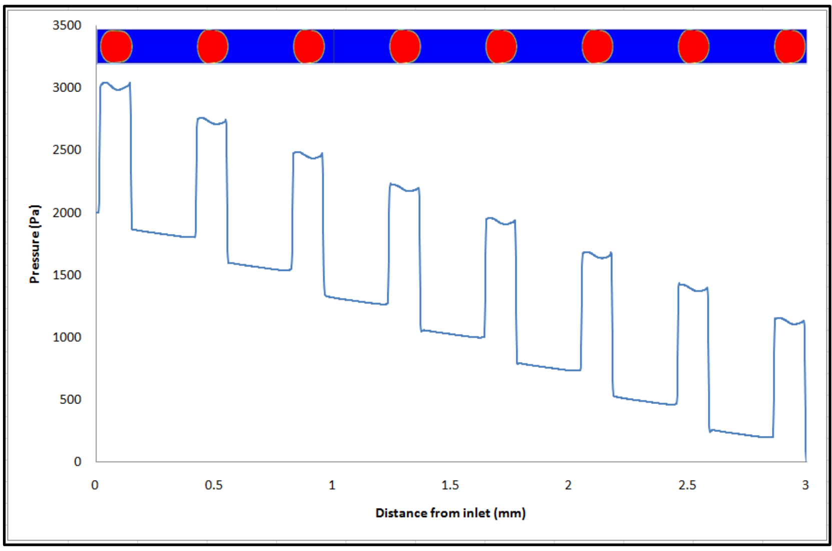


Fig. 6. Gauge Pressure variation along a line equidistance from both walls for case 3

The comparison of Global Nusselt number for all four cases is shown in Table 3. The Global Nusselt number was found to be highest in case 4 as predicted. So the augmentation in Global Nusselt number was observed to be 76 percent approximately compared to simple primary flow through same microchannel. The comparison of pressure drop in all cases is also shown in Table 3.

Particular	Pressure drop(Pa)	Nusselt number
Case 1	833.8204	8.6071
Case 2	1242.0540	9.5317
Case 3	2043.0083	14.7087
Case 4	2475.9529	15.1541

Table 3. Comparison of pressure drop and Nusselt number

IV. CONCLUSION

The heat transfer and hydrodynamic behavior of Taylor flow along with simple primary fluid flow (with and without obstruction) through rectangular microchannel was numerically investigated in present study. The velocity profile in case of simple flow was compared with basic velocity distribution equation and found to be in good agreement. The Nusselt number was found to be increased by 76 percentage in case 4 compared to case 1. It is obvious that the pressure drop increases if there is an obstacle in path. So such microchannel can be used when pressure drop is not a concern. The local Nusselt number was found to be drastically increased near cylindrical obstruction area of channel which is a direct result of additional recirculation zones. So in practical point of view obstruction can be provided in microchannel where the point heat source is located. Such heat sink will be more effective compared to heat sink with simple Taylor flow in terms of heat removal rate.

This work creates an opportunity to investigate both heat transfer and hydrodynamics of flow further with multiple obstructions and/or with different shape and orientation of obstruction.

REFERENCES

[1]. Asadolahi N., Gupta R., Fletcher D. and Haynes B.(2011), "CFD approaches for the simulation of hydrodynamics and heat transfer in Taylor flow", Chemical Engineering Science 66 5575–5584

[2]. Bretherton F.P., 1961. "The motion of long bubbles in tubes", Journal of Fluid Mechanics, 10(2):166 188.

[3]. Brackbill J.U., Kothe D.B. & Zemach C., 1992. "A continuum method for modeling surface tension", Journal of Computational Physics", 100(2):335–354.

[4]. Chung C, Ahn KH and Lee, 2009a "Numerical study on the dynamics of droplet passing through a cylinder obstruction in confined microchannel flow". J Non-Newtonian Fluid Mech 162:38–44 SJ.

[5]. Chung C., Lee M., Char K., Hyun K. and Lee S., " Droplet dynamics passing through obstructions in confined microchannel flow", Microfluid Nanofluid (2010) 9:1151–1163

[6]. Gupta R., Fletcher D.F. and Haynes B.S. "On the CFD modelling of Taylor flow in microchannels", Chemical Engineering Science 64, 2941–2950.

[7]. Hirt C. W. and Nichols B. D. 2009 "Volume of fluid (VOF) method for the dynamics of free boundaries", Journal of computational physics, 39(1), pp. 201-225.

[8]. Link DR, Anna SL, Weitz DA, Stone HA (2004), "Geometrically mediated breakup of drops in microfluidic devices", Phys Rev Lett 92:054503-1–054503-4

[9]. Muzychka Y. S., Walsh, E. & Walsh, P. 2010. "Simple Models for Laminar Thermally Developing Slug Flow in Noncircular Ducts and Channels", J. Heat Transfer, 132:1– 10.

[10]. Qian D., Lawal A. 2006, "Numerical study on gas and liquid slugs for Taylor flow in aT- junction microchannel", Chemical Engineering Science 61, 7609–7625.

[11]. Shao N., Salman, W. Gavriilidis, A. Angeli P., 2008. "CFD simulations of the effect of inlet conditions on Taylor flow formation", International Journal of Heat and Fluid Flow 29, 1603–1611.

[12]. T. Bandara, Sherman CP Cheung & Gary Rosengarten 2015, "Slug flowheat transfer in microchannels:a numerical study", Computational Thermal Sciences, 7 (1): 81–92.

[13]. Talimi V., Muzychka, Y. S., Kocabiyik S., 2011b. "On the validity of two-dimensional heat transfer simulation of moving droplets between parallel plates", In: Proceedings of the 9th International Conference on Nanochannels, Microchannels, and Minichannels, ICNMM9, June 2011, Edmonton, Canada.