Study of Fluid Dynamics and Heat Transfer in MEMS Structures

Mr. Satyaprakash Narayan Das[1] and Dr. Gouranga Bose[2]
[1]Centurion Institute of Technology, Centurion University, India [2]Institute of Technical education and Research, Sikshya O Anusandhan University,India
*Plot no:230, Patrapada-Bhagawanpur, Bhubaneswar,India, satyaiter174@gmail.com

Abstract: This paper describes the characteristics of MEMS micro channel and various issues of its designing. Here the major parameters are pressure drop and heat transfer rate. Various structures are modeled and optimized to get a minimum pressure drop and maximum heat transfer rate. The simulation results provide the characterization for Temperature, Mass flow rate, Pressure drop and Reynolds number. Here the material used is silicon and fluid used is water. The dimension of the model designed is 10mm*8mm*1mm. and the diameters of the channels are 200µm, 500µm, 1000µm.

Using COMSOL Multiphysics the various structures of micro channels were modeled. In this paper we have modeled for cubical cylinder, circular cylinder as well as staggered fin structure. Then choosing proper boundary conditions for inlet pressure, outlet pressure and temperature, the models are simulated.

The result we obtained are for 200µm, 500µm, 1000µm diameter structures by taking cubical cylinder, circular cylinder and staggered fin comes as per our desired result. The results are compared for different structures and diameters. But when the diameter is decreased below 200µm we did not get the desired result which is the limitation of this model. Till now the conclusion we get is if we take staggered fin shape structure the result is optimized in term of pressure drop and heat transfer rate.

Keywords: micro channel, staggered fin, pressure drop, heat transfer rate

1. Introduction

The idea of micro channel was first proposed by Tuckerman and Pease around 20 years back. Due to advancement of recent technology the use fullness of micro channel structure is increasing day by day. This micro channels can be used in various field such as, On-chip electronics cooling structure, Micro channel mixers which can be useful in chemical industries as well as biomedical field, Localized cooling by using micro nozzle, etc. As electronic components get smaller and heat transfer requirements increase, air becomes a less efficient coolant. Liquid cooling provides a means in which thermal resistance can be reduced dramatically

In this paper fluid dynamics and heat transfer in micro channel is well documented for various diameters. Major challenge in this micro channel model is to increase the heat transfer rate and to minimize the pressure drop across the channels. It is very costly and difficult to manufacture the micro channel (<1000µm) structure and analyze its performance. Therefore here the modeling and simulation is done by using COMSOL Multiphysics software.

2. Analysis of heat transfer and fluid dynamics in Micro channel

Here heat transfer, pressure drop and Reynolds number are studied for single phase fluid flow. In this model it is assumed that heat transfer occurred mainly due to convection process as it is the interaction between liquid and solid surface. Convection heat transfer may be classified according to the nature of the flow for free or natural convection the flow is induced by buoyancy forces, which arise from density differences caused by temperature variations in the fluid. For a forced convection, the flow is caused by external means, such a fan, a pump, or pressure difference.

The heat transfer by convection is described by the Newton's law of cooling and is given by:

$$Q=hA(T_W-T_\infty)$$

Where:

Q = Heat transfer rate (W)
h = Heat transfer coefficient (W/m2.K)
Tw = Wall temperature (K)
T∞= Free stream fluid temperature (K)

Other important parameters which affect the single phase convections are Hydraulic diameter and Reynolds number.

The hydraulic diameter, D_h, is commonly used when dealing with non-circular pipes, holes or ducts.

Hydraulic Diameter = 4 x cross sectional area of flow / wetted perimeter.

Mathematically it is given by:

$$D_h = \frac{4A}{p}$$

The Reynolds no is given by:

$$\mathrm{Re} = \frac{\rho v L}{\mu}$$

Where:

- **V=** mean velocity of the object relative to the fluid (SI units: m/s)
- **L=** characteristic linear dimension, (travelled length of the fluid; hydraulic diameter when dealing with river systems) (m)
- **μ=** is the dynamic viscosity of the fluid (Pa·s or N·s/m² or kg/(m·s))
- **ρ=** is the density of the fluid (kg/m³)

Pressure drop is the result of frictional forces on the fluid as it flows through the tube. The frictional forces are caused by a resistance to flow. The main determinants of resistance to fluid flow are fluid velocity through the pipe and fluid viscosity. A piping network containing a high relative roughness rating as well as many pipe fittings and joints, tube convergence, divergence, turns, surface roughness and other physical properties will affect the pressure drop. High flow velocities and / or high fluid viscosities result in a larger pressure drop across a channel. The pressure drop inside the channel is given by:

$$\Delta p = \frac{2 f L \rho v^2}{D_h}$$

Where:

- **V:** mean flow velocity
- **L:** flow length
- **ρ:** fluid density
- **f:** friction factor depends upon aspect ratio

3. Use of COMSOL Multyphysics

It is very costly and difficult to manufacture the micro channel structure and analyze it's performance. Therefore here the modeling and simulation is done by using COMSOL Multi-physics software and different parameters are studied.

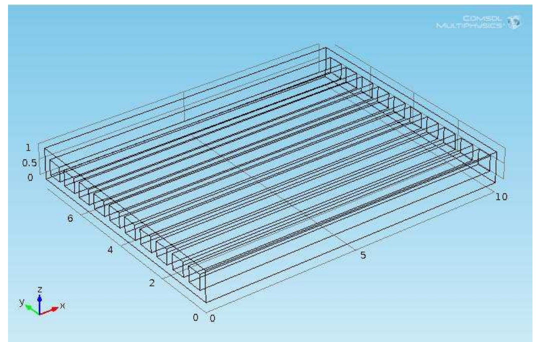

Figure 1. Structure of Square shaped micro channel

The dimension of each model is kept constant, ie, 10mm×8mm×1mm. only the shape (square, circular and staggered fin) and the radius (200µm, 500µm, 1000µm) of the channel are varied. In all the model material taken is silicon and fluid used is water. In each model the inlet flow velocity is fixed at 0.01m/s. All the outer boundaries are thermally insulated except the lower boundary where the temperature is fixed at 393.15K and other temperatures are kept at 293.15K. Inlet water temperature is 293.15k In the outlet the pressure is kept constant for every model at 1.5atm. Each model are simulated and assumed results are obtained.

4. Results

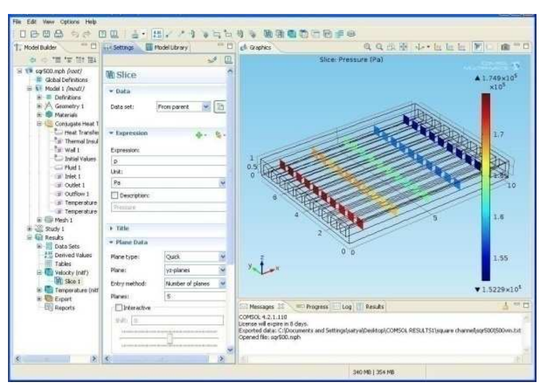

Figure 2. 3D plot of Pressure variation in 200µm square channel

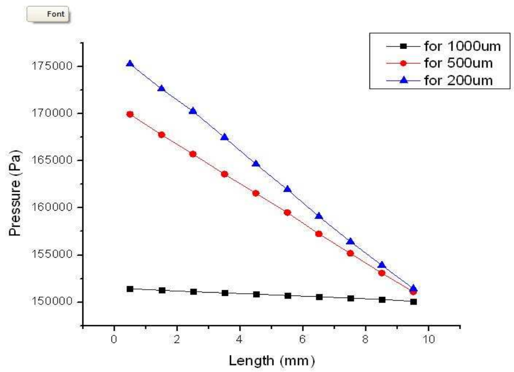

Figure 3. Variation of pressure due to variation D_h in square channel along the length

The figure-2 and figure-3 shows that the pressure drop in increases as the diameter of the channels decreases so higher amount of pumping force is required in case of smaller radius and hence the stress upon the material also increases.

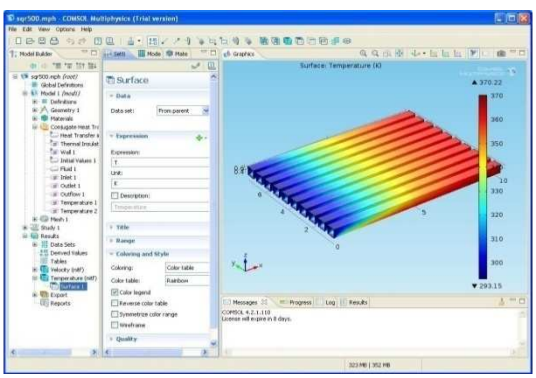

Figure 4. 3D plot of Temperature variation in 500µm square channel

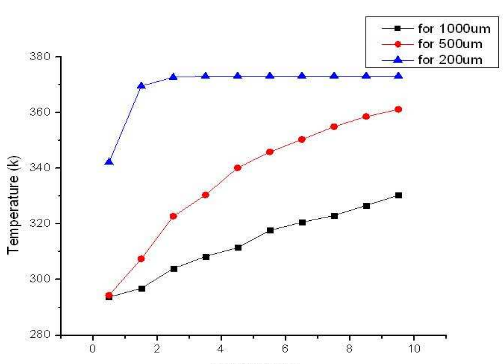

Figure 5. Variation of temperature due to variation D_h in square channel along the length

The figure-4 and figure-5 shows the temperature variation plot along the x-axis. As the radius of the channel decreases the temperature of the liquid increases rapidly. This happens due to decrease in mass flow rate.

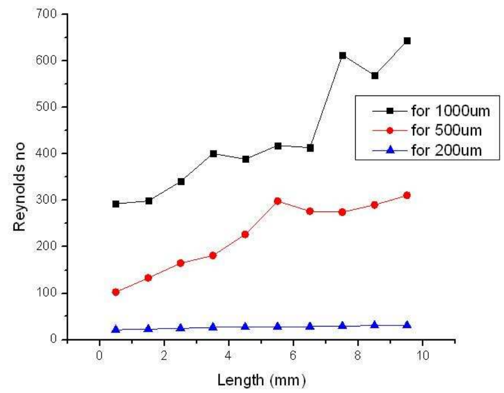

Figure 6. Variation of Reynolds number in square shape chennal

The figure-6 shows that as the radius of the channel decreases the laminar flow of the liquid also increases and turbulence decreases. Here the small variation of the Reynolds number for 500µm and 1000µm is due to small friction factor between the two phases.

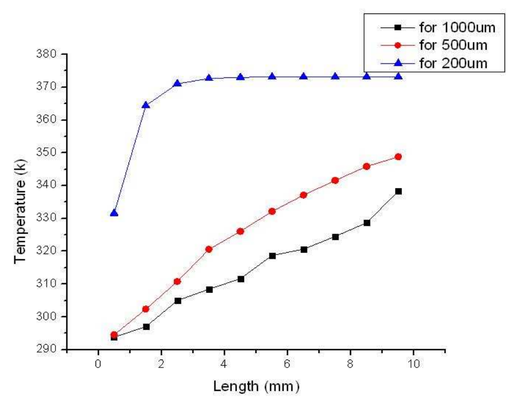

Figure 7. Variation of temperature due to variation D_h in circular channel along the length

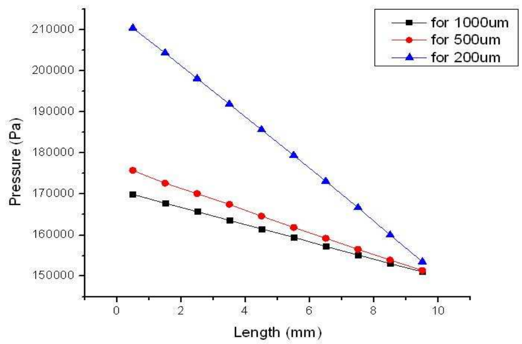

Figure 8. Variation of pressure due to variation D_h in circular channel along the length

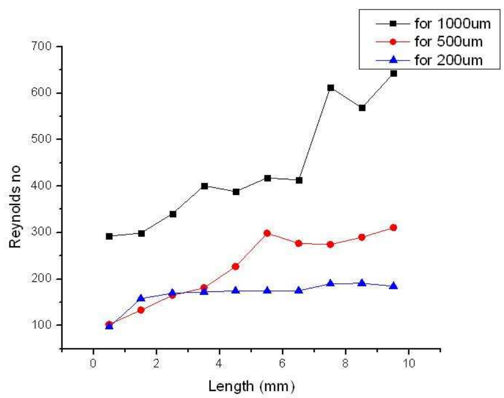

Figure 9. Variation of Reynolds number in circular shape channel

The figure-7, figure-8 and figure-9 shows the similar kind of graph as incase of square channels.

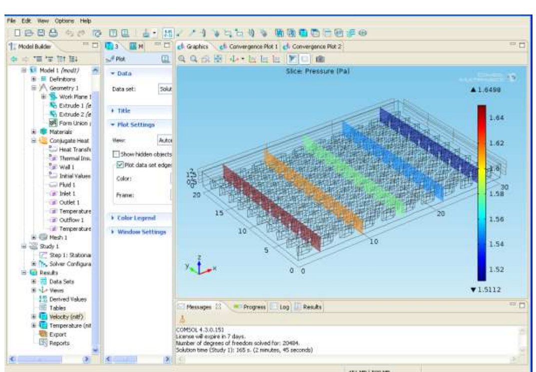

Figure 10. Variation of pressure in staggered fin model

From figure-10 it is clear that the pressure drop is minimized in staggered fin model.

5. Conclusion

From the above results it is concluded that Pressure drop is inversely proportional to the diameter of the channel. Flow become more laminar as the diameter of the channel decreases As the radius decreases the mass flow rate decreases hence the efficiency of the cooling also decreases.

6. References

1. Colgan, E. G., Furman, B., Gaynes, M., Graham, W., LaBianca, N., Magerlein, J. H., Polastre, R. J., Rothwell, M. B., Bezama, R. J., Choudhary, R., Martson, K., Toy, H., Wakil, J., Zitz, J., and Schmidt, R., A Practical Implementation of Silicon Microchannel Coolers for High Power Chips, Invited Paper presented at IEEE Semi-Therm 21, San Jose, March 15–17, pp. 1–7, 2005.
2. Esch, B. van, Kemenade, E. van, Procestechnische Constructies 4B660,Technische Universiteit Eindhoven, maart 2002.
3. Kandlikar, S. G., High Flux Heat Removal with Microchannels – A roadmap of Challenges and Opportunities, Heat Transfer Engineering, vol. 26, no.8, pp. 5- 14, 2005.
4. Kakac, S., Shah, R.K. and Aung, W., Handbook of single-phase convective heat transfer, John Wiley & Sons, New York, 1987.
5. Kandlikar, S. G., and Grande, W. J., Evaluation of Single Phase Flow in Microchannels for High Flux Chip Cooling Thermohydraulic Performance Enhancement and Fabrication Technology, Heat Transfer Engineering, vol. 25, no. 8, pp. 5–16, 2004.
6. Kandlikar, S. G., and Upadhye, H. R., Extending the Heat Flux Limit with Enhanced Microchannels in Direct Single-Phase Cooling of Computer Chips, Invited Paper presented at IEEE Semi-Therm 21, San Jose, March 15–17, pp. 8–15, 2005