NUMERICAL ANALYSIS OF PRESSURE DROP AND VARIATION OF NUSSELT NUMBER IN MICROCHANNEL WHEN SUBJECTED TO UNIFORM HEAT FLUX

[1]Kashif Faridi, [2]Firoz Akhtar, [3]Md Shabbeer Ansari

[1]Assistant Professor, [2]Assistant Professor, [3]Assistant Professor
[1]Department of Mechanical Engineering,
[1]Maulana Azad College of Engineering and Technology, Aryabhatta Knowledge University, Patna, India.

Abstract : The different parameters like pressure drop, Nusselt number, Reynolds number, and fluid flow inside the heated microchannels were calculated analytically by using formulae given in the theory, these results were then validated by comparing the predicted local thermal resistances with CFD models using fluent. This work studies the CFD simulation of micro channel flow , pressure drop and single-phase flow friction factor (Po=fRe) of liquid subjected to heat from the bottom of rectangular micro channel and heat conduction in the solids. The single phase models are used for solving the problems. Simulations were performed on the rectangular channel. Width of bottom wall of channel was 210µm; outer side wall height is 810µm, length of channel was 25mm, the thickness of side wall as well as bottom wall was 60µm.The dimension of fluid element inside the channels were, height 750µm width 150µm and length 25mm. The top surface was adiabatic while the two sides were designated symmetric boundary conditions. A uniform heat flux of 240000W/m2 was applied at the bottom surface. The substrate thickness included in the model was chosen to be 60µm, the heat flux in the substrate can be safely assumed to be uniform because of its high thermal conductivity. Nusselt numbers were calculated for the microchannel model for Re no. ranging 131 to 275. For calculating Nusselt numbers and pressure drop wall temperature at different section of microchannels iso-surfaces were created at different section of fluid element perpendicular to the length of microchannel and contact region of wall with fluid.

Keyword: Micro channel, Reynolds number, Nusselt number, Heat transfer, Pressure drop, Fluid flow, Friction factor.

1.0. Introduction

The micro-channels include a number of different types of structure including conduits, valves, and chambers. The micro-channels define small flow paths that may include reagent location sites, constrictions, valves, pumps, and mixing chambers. Such structures may utilize capillarity as the mechanism for moving the applied fluid medium with the movement being by detailed channel design and control of the contact angle. Alternatively or in addition a flow wave could be induced by pumping, depression of a bladder/bellows or vibration (including shaking). The driving force could also be achieved by an osmotic process that utilizes a semi-permeable membrane between medium introduction and reagent location. The need of economically viable energy efficient heat transfer system is increasing day by day. In recent years, advances in fabrication technologies have led to a vast array of miniaturized devices in many fields. Miniaturized electronic components, fuel cell, biomedical equipment, aerospace thermal system, compact heat exchanger and refrigeration system, are a few examples. Successful implementation of flows in these miniaturized devices necessitates understanding of the underlying transport processes. For the purpose of designing mini-channels flow based devices utilizing single-phase flow friction factor (Po) and heat transfer coefficient (Nu).

2. Literature Survey

Lee et al [1] experimentally investigate to explore the validity of classical correlations based on conventional sized channels to show the thermal behavior of single-phase flow in rectangular micro channels. The micro channels were ranged in width from 194 µm to 534 µm , the channel depth being five times the width in each case. Channels was made of copper and contained ten micro channels in parallel. He conducted experiment with deionized water, the Reynolds number ranging from 300 to 3500. In his experimental result are in good agreement with analytical predictions obtained based on a classical, continuum approach (an average variation of 5%). Hence conventional analysis method can be used in predicting heat transfer behavior in micro channels of the dimensions considered in their study. But, the entrance and boundary conditions applied in this experiment need to be carefully correlate with the analytical approaches.

Shen et al [2] experimentally investigate the single phase convective heat transfer in a compact heat sink consisting of 26 rectangular microchannels of 300 µm width and 800 µm depth. The relative roughness is estimated to be 4–6%. They used deionized water as working fluid . They performed experiment with the Reynolds number ranging from 162–1257, the temperatures of liquid at inlet were 30ºC, 50ºC and 70ºC and the heating of 140–450 W was applied. They observed that the friction factors and local average Nusselt numbers abnormally deviate from those of conventional theories, may be due to to the surface roughness. The hydraulically developed and thermally developing flow here shows decrease in Nusselt numbers with the non-dimensional axial distance. The temperature dependent fluid physical properties also affects the heat transfer behavior marginally. They also correlate the friction factors and the Nusselt numbers based on their experimental result.

Gamrat et al [3] presents both three and two-dimensional numerical analysis of convective heat transfer in microchannels. The aspect ratio of rectangular microchannel were very high. Two channel spacings, namely 1mm and 0.3mm (0.1mm), were used for three-dimensional (two-dimensional) numerical model, respectively. They took water as flowing fluid . The experiments were conducted for Reynolds number ranging from 200 to 3000. Conduction/convection coupling effects and thermal entrance effects were considered in the test. Finally, they compair between measured and computed heat flux and temperature fields in their work. The numerical analysis did not show any major effect on heat transfer in microchannel heat sink down to the smallest size considered (0.1mm) but the experimental result show this effect.

Park et al. [4] measured the two-phase pressure drop and heat transfer coefficients of FC-72(C6F14). They performed experiment in two multi-ported rectangular microchannels whose hydraulic diameters were 61μm and 278μm.They supplied heat flux from 0.6 to 45.1 kW/m^2 and mass flux between 188 to 1539 kg/m^2s. They employed DC power supply to heat directly the heating load to the test section. They observed that pressure drop increases with increasing vapor quality and mass flux. The pressure drop was affected marginally by heat flux under identical mass flux and vapor quality conditions, pointing that most of the pressure drop was due to friction. It was seen as the vapor quality exceeded 0.2–0.4, the heat transfer coefficient decreased with increasing vapor quality because of early dry-out of liquid film on the channel surface.

Tu and Hrnjak [5] performed friction factor experiments on smooth microchannels, and they found that their results were well predicted by laminar theory. However, one of the channels that they tested had a significant surface roughness, and a friction factor increase and transition Reynolds number decrease to 1570 was found in their experiment.

J. Li and G.P. Peterson [6] numerically simulated a forced convection heat transfer occurring in silicon based micro channel heat sinks has been conducted using a three-dimensional conjugate heat transfer model (2D fluid flow and 3D heat transfer) consisting of a 10 mm long rectangular micro channel, 57 μm wide and 180 μm deep and hydraulic diameter 86 μm. The impact of thermo physical properties of the fluid and the geometric parameters of the channel on the flow and the heat transfer, were noted by temperature dependent thermo physical property method. Their results illustrate that these properties of the liquid greatly affect both the flow heat transfer in the micro channel Fluid flow and heat transfer experiments were conducted by Phillips[37] on a copper microchannel heat exchanger (MHE). He sets a experimental method of appling a constant surface temperature to the MHE . The friction factor results from his experiments deviated only marginally with theoretical correlations. The experimental Nusselt number result are in good agreement with theory in the transition/turbulent regime, but the experimental results show a higher Nusselt number in the laminar regime than calculated by theoretical correlations. He used CFD model to simulate numerically the fluid flow in the microchannels.

3. Computational Modeling and Simulations

In this study the single phase models is used for solving the problems. This CFD model calculate one transport equation for the momentum and one for continuity for single phase, and then energy equations to study the thermal and hydrodynamic behavior of the system. Theory is taken from the ANSYS Fluent 14.0.

In the present work, a numerical model was formulated to solve for the 3D conjugate heat transfer in the microchannel heat sink, responsible for both convection in the channel and conduction in the substrate. The top surface was adiabatic while the two sides were designated symmetric boundary conditions. A uniform heat flux was applied at the bottom surface The substrate thickness included in the model was chosen to be 60μm, since the heat flux from the bottom of the microchannel can be safely assumed to be relatively uniform due to its high thermal conductivity. At the inlet, velocity profile was a fully developed but thermal profile is developing is modeled for simulations, while a uniform inlet velocity profile was used for simultaneously developing flow simulations. An outflow boundary condition was specified at the exit in this case. The coolant (water) was treated as being incompressible and Newtonian, with constant thermo physical properties over the range of temperatures considered (307-320K). Only flow rates in the laminar regime were considered. The commercial software package FLUENT was used for the computations.

3.1 Basic Pressure Drop Relation s

Considering the equilibrium of a fluid element of length d*x* in a pipe having diameter *D*. The force in pipe due to pressure difference d*p* is balanced by the frictional force due to shear stress τ_w at the wall.

$$\left(\frac{\pi}{4}D^2\right)dp = (\pi D dx)\,\tau_w \quad (1)$$

The relation between pressure gradient and the wall shear stress is given by the following equation

$$\frac{dp}{dx} = \frac{4\tau_w}{D} \quad (2)$$

For Newtonian fluids, the wall shear stress τ_w is expressed in terms of the velocity gradient at the wall:

$$\tau_w = \mu \left.\frac{du}{dy}\right|_w \quad (3)$$

μ is the dynamic viscosity. The Fanning friction factor *f is* used in heat transfer is given by following relation:

$$f = \frac{\tau_w}{(½)\rho u_m^2} \quad (4)$$

Where u_m is the mean flow velocity in the channel.

3.2 Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (5)$$

3.3 Momentum Equation

Let (*x*, *y*, *z*) be the orthogonal components of the body force field in the Cartesian coordinate system; then

x-Momentum

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \frac{dp}{dx} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \quad (6)$$

y-Momentum

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \frac{dp}{dx} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \quad (7)$$

z-Momentum

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\frac{dp}{dx}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right) \quad (8)$$

3.4 Energy Equation

$$\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{1}{\alpha}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right) \quad (9)$$

All of these equations are coupled, so a computational code should be used to solve these equations simultaneously. Here the commercial code Fluent14 was used to solve these equations.

4. Material Properties

Pure water is used as working fluid.
Properties of water at 40^0 C (Incropera and DeWitt) [8].
ρ=991.8 kg/m3; μ=655×10^{-6} N-s/m2; c_p =4179 J/kg-K; k =0.632W/m-K; Pr=4.33
where symbols have their usual meaning.

5. Geometry

Geometry is created in design modeler due to symmetry of microchannels single channel was modeled. The width of bottom wall of channel was 210μm; outer side wall height is 810μm, length of channel was 25mm, the thickness of side wall as well as bottom wall was 60μm.The dimension of fluid element inside the channels was height 750μm width 150μm length 25mm.The figure of microchannels in design modeler were as follows.

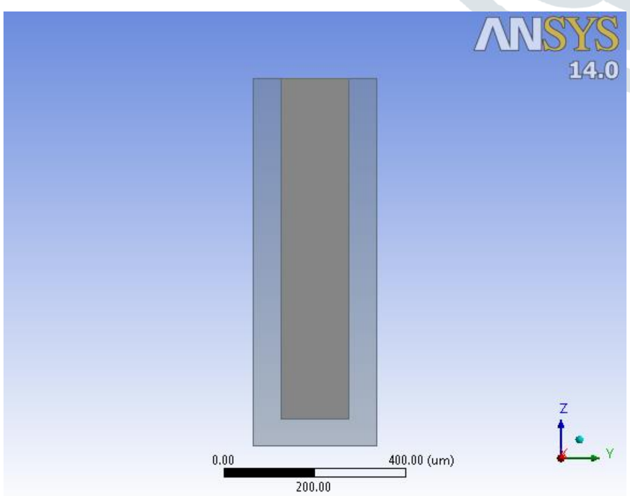

Figure. 1. View of microchannel model at inlet and outlet

To solve the governing equations and obtain a flow field and temperature field, the commercial code Fluent was used. A constant heat flux boundary condition was imposed on the bottom wall of the microchannels in this case. Water with properties measured at 313K was used as the working fluid. In this case laminar model was used. Reynolds number was below 300. Fluent is used for obtaining a converged result in this Reynolds number range.

A no slip boundary condition was assigned for the wall surfaces; Knudsen number was very low in the case of incompressible flow, where velocity components in y and z direction were set to zero. A uniform velocity was specified at inlet in x direction and a constant inlet temperature was assigned at the channel inlet. At the exit, out flow condition was applied. The simulations were done for five cases having same flux at bottom wall for different Reynolds numbers. These Reynolds numbers were 131, 150, 175, 200, and 225. In this case pressure-velocity coupling method was applied scheme was simple. In numerical discretization gradient was set to least squares cell pressure was set to standard momentum and energy were set to second order upwind. Solution initialization was hybrid initialization.

Table 1
Under Relaxation factor.

Factor	Value
Pressure	0.3
Density	1.0
Body force	1.0
Momentum	0.7
Energy	1.0

6. Results On The Basis Of Theory

Here copper is taken as substrate whose thermal conductivity is 380W/m-k and water as working fluid. The copper microchannel is 10mm wide and 37 channels of width 150μm depth 750μm and spacing 120μm are taken and analytically solved from the classical theory formulae.

7. Simulation Result

Each case was run using second order upwind schemes for each governing equation. Nusselt numbers were calculated for the microchannel model for Re ranging 131 to 275. For calculating Nusselt numbers, pressure difference, heat flux, wall temperature at different section of microchannels iso-surfaces were created at different section of fluid element perpendicular to the length of microchannel and contact region of wall with fluidThe Nusselt number was calculated at particular section as;

$$Nu=\frac{qD_h}{k(T_{w(avg)}-T_{f(avg)})} \quad (10)$$

The variation of Nusselt number with the length of microchannes for different Reynolds number is given below.

Table 2: CFD and theoretical result of fRe for different Reynolds number.

Re	*f*Re(theoritical)	fRe(simulation)
131.65	19.07	19.1
150	19.07	19.2
175	19.07	19.25
200	19.07	19.4
225	19.07	19.57

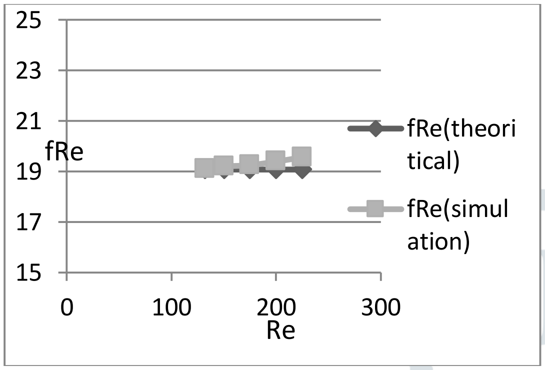

Figure 2.Comparision of simulation and analytical analysis of *f*Re versus Re of microchannels at different Reynolds number at 240000W/m^2 heat flux

Single-phase flow friction factor (Po = fRe) are in good agreement with analytical correlation. There is very little deviation hence CFD modeling can be used to study the behavior of flow in microchannel.

Table 3:CFD result, Nusselt number along the length of microchannels at different Reynolds number, keeping heat flux constant in each case (240000W/m^2).

X	Nu at Re=131	Nu at Re=150	Nu at Re=175	Nu at Re=200	Nu at Re=225
0	38.66	39.582	40.6547	41.64	42.45
0.25	9.879	10.334	10.941	11.521	12.057
0.5	8.178	8.516	8.945	9.31	9.71
2.5	6.814	6.865	7.011	7.174	7.29
5	6.527	6.587	6.675	6.734	6.83
7.5	6.314	6.3454	6.379	6.448	6.47
10	6.465	6.518	6.582	6.63	6.697
12.5	6.468	6.509	6.548	6.61	6.63
15	6.542	6.606	6.663	6.71	6.77
17.5	6.293	6.324	6.373	6.43	6.46
20	6.3224	6.322	6.3896	6.44	6.47
22.5	6.118	6.142	6.191	6.25	6.285
24	6.205	6.203	6.26	6.305	6.37
24.8	6.0765	6.092	6.141	6.21	6.24
25	6.001	6.004	6.09	6.1	6.19

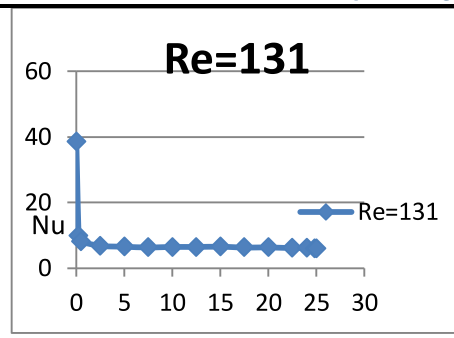

Figure 3. CFD results variation of Nusselt along micro channel at Re=131 and heat flux 240000W/m^2

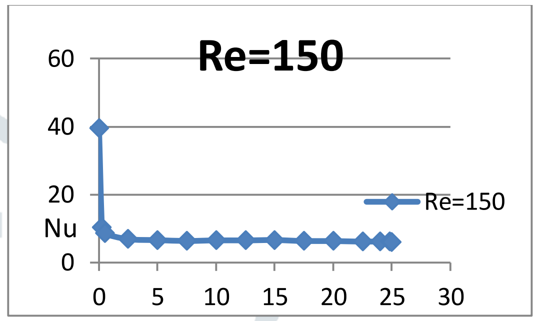

Figure 4. CFD results variation of Nusselt along the length of micro channel at Re=150 and heat flux 240000W/m^2

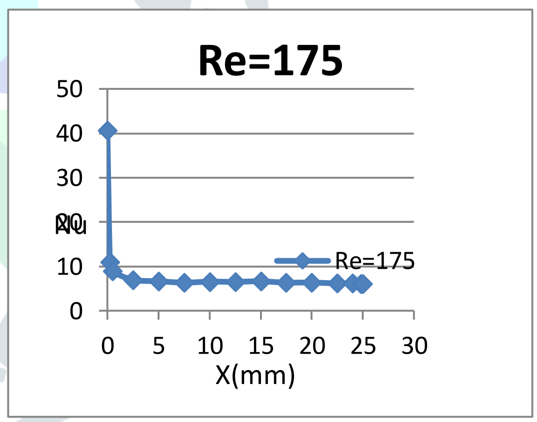

Figure 5. CFD results variation of Nusselt along the length of micro channel at Re=175 and heat flux 240000W/m^2

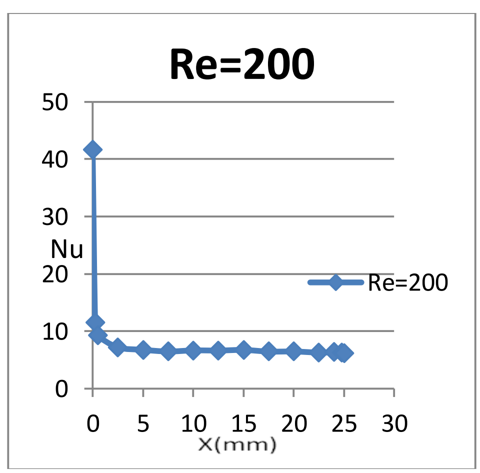

Figure 6. CFD results variation of Nusselt along the length of micro channel at Re=200 and heat flux 240000W/m^2

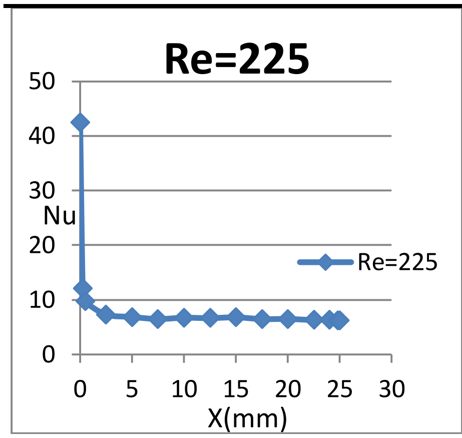

Figure 7. CFD results variation of Nusselt along the length of micro channel at Re=225 and heat flux 240000W/m^2

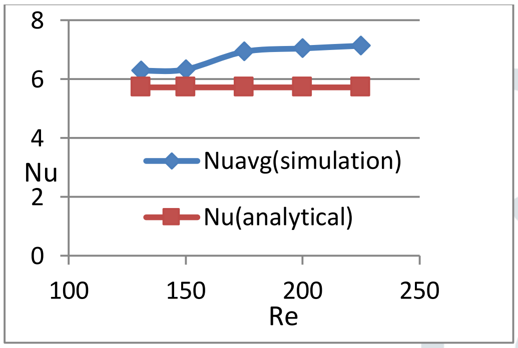

Figure 8.Comparision of simulation and theoretical analysis of average Nusselt number with the increase in Reynolds number at heat flux 240000W/m^2

Table 4. Pressure drop at different Reynolds number in microchannels

Re	Δp(simulation) Pascal	Δp (analytical) Pascal
131	3488.9	3488
150	3989	3959.59
175	4678.7	4619.52
200	5372.67	5276.14
225	6074.03	5939.65

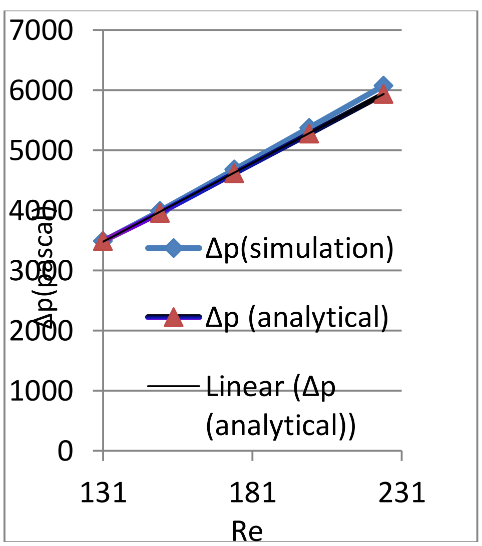

Figure 9. Comparison of pressure difference at inlet and outlet of channels for different Reynolds number between analytical and CFD simulation.

It can be seen that simulation pressure difference between inlet and outlet of microchannels is more than the analytical result and difference increases with the increase in Reynolds number. But these difference is marginal as shown in figure above.Hence CFD modeling can be used safely.

8. Pressure Contour:

The pressure contours are depicted in figure 10. It show that pressure decreases linearly in the flow direction. This is also confirmed by classical theory

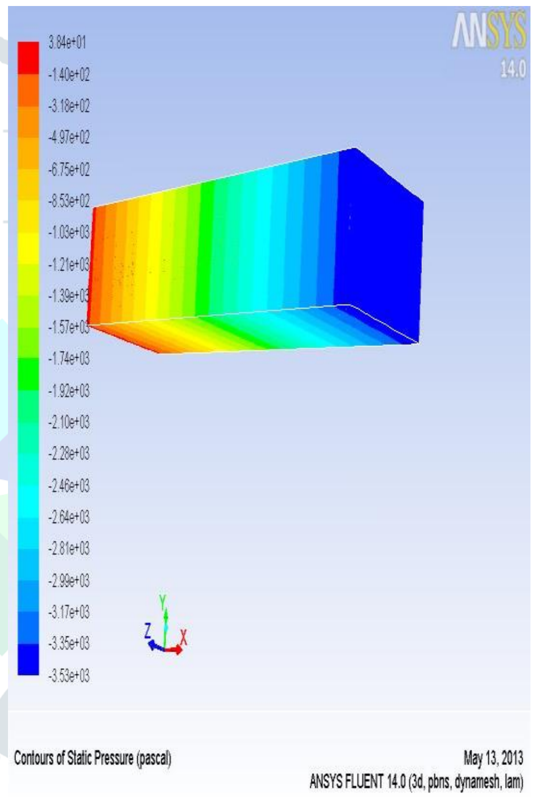

Figure10: 3D view of pressure contour of water at Re=131

9. Conclusion and Future Work

9.1 Conclusion

It is observed with the increase in pressure drop, flow velocity as well as Reynolds number Nusselt number increases in consistence with Reynolds number analogy.The result shows average Nusselt number increases with increase in Reynolds number contrary to constant value of conventional theory.The result shows that there was increase in fRe with increase in Reynolds number contradictory to constant value of conventional theory.

9.2Future Scope Of The Work

Modeling and Simulation of two phase flow in micro channel.Analysis of the boiling characteristics of nanofluids using CFD models.

References

[1] Poh-Seng Lee, Suresh V. Garimella , *Dong Liu ,Investigation of heat transfer in rectangular microchannels,* International Journal of Heat and Mass Transfer, 48 (2005) 1688–1704

[2] S. Shen, J.L. Xu , J.J. Zhou, Y. Chen,Flow, *heat transfer in microchannels with rough wall surface,* Energy Conversion and Management 47 (2006) 1311–1325

[3] Gabriel Gamrat , Michel Favre-Marinet , Dariusz Asendrych , *Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular microchannels,* International Journal of Heat and Mass Transfer 48 (2005) 2943–2954

[4] Chang Yong Park , Yonghee Jang , Bosung Kim , Yongchan Kim, *Flow boiling heat transfer coefficients and pressure drop of FC-72 in microchannels,* International Journal of Multiphase Flow 39 (2012) 45–54

[5] Tu, X. and Hrnjak, P., *Experimental investigation of single-phase flow pressure drop through rectangular microchannels*, International Conference on Microchannels and Minichannels, ASME Publications, 2003, Vol. 1, no. 1028, p. 257–267.

[6] J. Li and G.P. Peterson, *Experimental measurements of fluid flow and heat transfer in microchannel cooling passages in a chip substrate, Proceedings of the ASME International Electronics Packaging Conference in Binghamton,* NY, USA, ASME publications, 1993, Vol. 4-2, p. 685–692.

[7] Wynn Allen Phillips, Jr. *Experimental and Numerical Investigation of Fluid Flow and Heat Transfer in Microchannels,* MS Thesis, B.S., Louisiana State University, August 2008.

[8] Incropera, F. P. and DeWitt, D. P., *Fundamentals of Heat and Mass Transfer*, 5th Edn., New York: JohnWiley and Sons, 2002.