

Study of Fluid Flow and Heat Transfer in Rectangular Micro Channel

Sunil Bharati

PG Research Scholar

Department of Thermal Engineering

Agnos College of Technology RKDF University, Bhopal, M.P

Sohail Bux

Head of Department

Department of Thermal Engineering

Agnos College of Technology RKDF University, Bhopal, M.P

Abstract

The computational fluid dynamics (CFD) model equations are solved to predict the hydrodynamic and thermal behaviour of the exchanger. The geometry of the problem and meshing of it have been made in ANSYS 14.0. The models have been solved by ANSYS Fluent 14.0 solver. Water and its Nano fluids with alumina (A_2O_3) are used as the coolant fluid in the micro channel heat sink. The relation between heat transfer coefficient and thermal conductivity of the fluid i.e. $h \propto k$ is proved in the present study. Thus use of Nano fluids has been found beneficial both in laminar and turbulent zone. The result shows that Nano fluids help to increase the heat transfer coefficient by 15% and 12% respectively in laminar and turbulent zone. The entrance length for the fully developed velocities depends on Reynolds number. The temperature rise between outlet and inlet depends on the Reynolds number, Re and Peclet number, Pe . Temperature distribution is found to be independent of radial position even for $Pe \ll 1.0$. The hydrodynamic and thermal behaviour of the system have been studied in terms of velocity, pressure and temperature contours. The velocity contours at the exit show that wall effect penetrates more towards the center and the thickness of the zone with maximum velocity shrinks with increase in Re . The pressure drop across the channel increases with increase in Re . The experimental work done by Lee and Mudawar (2007) has been predicted by the present CFD results. The hydrodynamics and thermal behaviour of a rectangular micro channel are studied here. The variation wall temperature, pressure drop in the channel and the friction factors calculated using ANSYS Fluent can well predict the experimental data. The effect of Re on the behaviour the channel are also studied. Its behaviour also has been analyzed with the help of temperature, pressure and velocity contours.

Keywords: ANSYS Fluent 14.0 solver, Thermal Conductivity, CFD

I. INTRODUCTION

This chapter deals with simulation of single phase fluid flow in a rectangular micro channel embedded in a test module. The experimental setup with test module was developed by Lee and Mudawar (2007) as shown in Fig. Construction of micro channel test module with thermocouple locations is shown in Fig. (Lee and Mudawar, 2007). The micro channel is made of oxygen free copper.

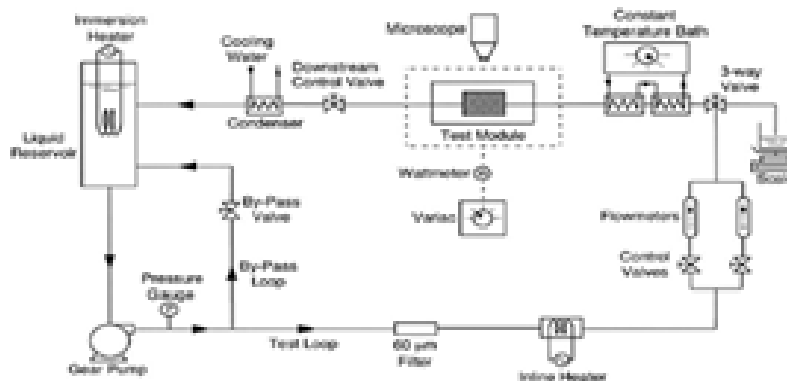


Fig. 1: Experimental Setup of Micro Channel Test Module

II. PROBLEM SPECIFICATION

The experimental work which was performed by Lee and Mudawar, (2007) on the test rig is simulated in the present study. Fluid is flowing through a rectangular micro channel embedded in a test module. There are 21 parallel rectangular micro-channels in the module. The dimension of each micro channel is $215 \mu m$ width, $821 \mu m$ depth and $4.48 cm$ length. The inlet velocity is u (m/s). The micro channel is made of oxygen free copper. The heating is provided by 12 cartridge heaters that are embedded in

underside of test module. The top surface of micro channel is subjected to adiabatic conditions. Operating conditions for the study are as follows: The operating range of Reynolds number based on the hydraulic diameter of the channel, $Re_{Dh} = 140-941$, the power input range to the channel, $Q = 100-300$ W, the inlet temperature of fluid to the channel, $T_{in} = 30$ °C, the range of inlet pressure, $P_{in} = 1.17-1.36$ bar, and the output pressure $P_{out} = 1.12$ bar.

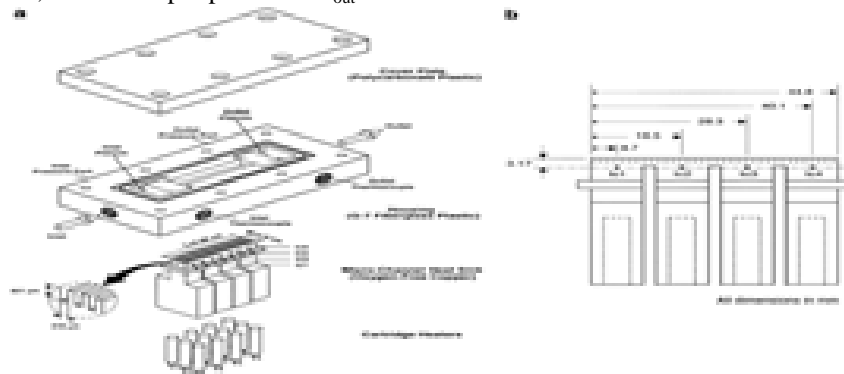


Fig. 2: Construction of Micro Channel Test Module with Thermocouple Locations

III. GEOMETRY OF THE COMPUTATIONAL DOMAIN

In this study, a single micro channel out of 21 channels is considered as computational domain. Figure represents the unit cell (computational domain) of micro channel heat sink with half of surrounding copper walls. Dimension of unit cell micro channel are shown in figure (Lee and Mudawar 2004).

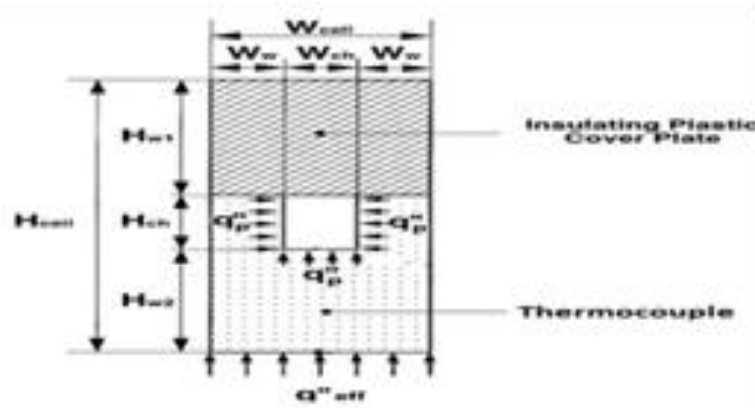


Fig. 3: Computational Domain of Rectangular Micro Channel

IV. MESHING OF THE COMPUTATIONAL DOMAIN

Structured mesh method was used for meshing the geometry as shown in Fig. Nodes were created with element size of 0.0005 in all three dimensional coordinates. Fig. represents three dimensional geometry of rectangular micro channel with structured mesh.

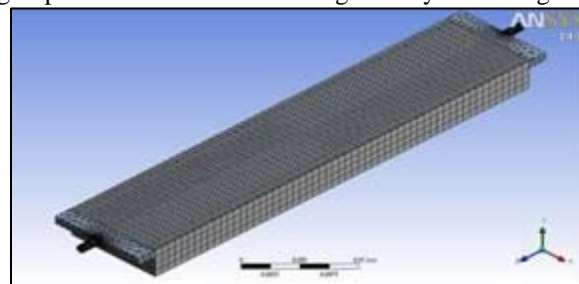


Fig. 4: Three Dimensional Geometry of Rectangular Micro Channel with Structured Mesh

A. Physical Model:

Physical model for simulation of single phase fluid flow is same as discussed in chapter 4.

B. Fluid Properties:

Working fluid properties are same as discussed in Chapter 4. Here copper is considered as solid and its default properties as in ANSYSYS (Fluent), density, heat capacity and thermal conductivity are 8978 kg/m^3 , 381 J/kgK and 387.6 W/mK respectively.

V. SOLUTION METHODS

The specified solver uses a pressure correction based iterative SIMPLE algorithm with 1st order upwind scheme for discretization of convective transport terms. The convergence criteria for all the dependent variables are specified as 0.001. The default values of under-relaxation factor as shown in Table are used in the simulation work.

VI. RESULTS AND DISCUSSIONS

The variation of velocity of water and its Nano fluid (1% alumina and 2% alumina) with axial position (x) at $Re=140$ is shown in Fig.

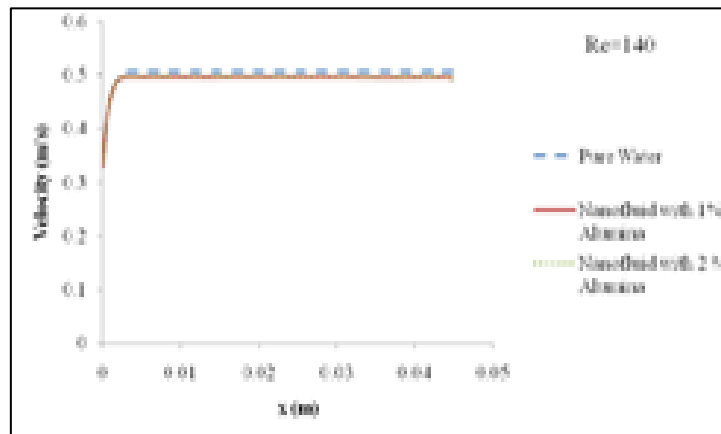


Fig. 5: Velocity profile for water and its Nano fluid at $Re=140$

Velocity is at centerline as indicated in figure. It is observed almost at the entrance velocity of all types of fluids got fully developed. The entrance length of all the fluids also found to be same.

Comparison of the present computed pressure drops and friction factors for water and its nano fluid at different Re 140-941 with experimental results (Lee and Mudawar, 2007) are depicted in Table and respectively. The same is also shown in Fig.

Comparison of computation pressure drop and friction factor of water with experimental values (Lee and Mudawar, 2007)

Comparison of computation pressure drop and friction factor of 1% Alumina with experimental values (Lee and Mudawar, 2007)

Comparison of computation pressure drop and friction factor of 2% Alumina with experimental values (Lee and Mudawar, 2007)

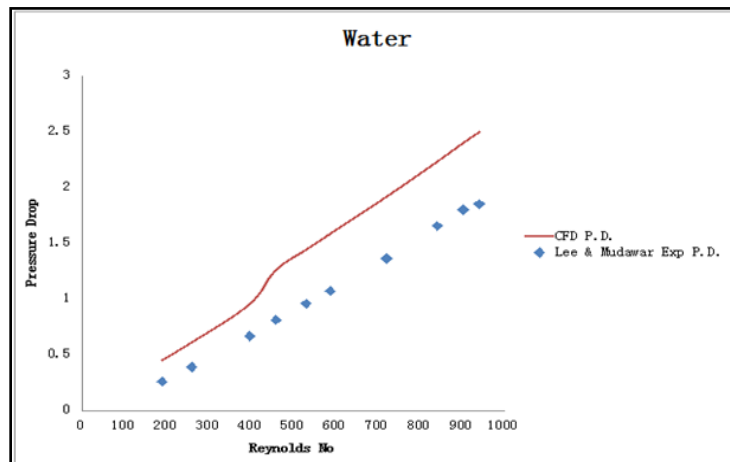
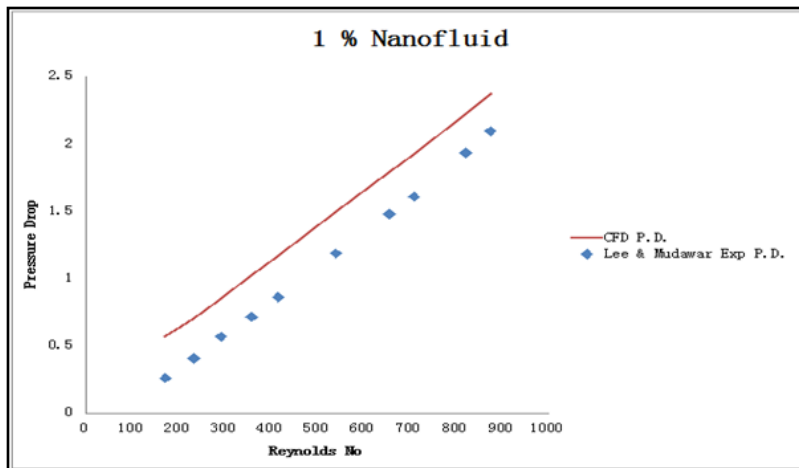
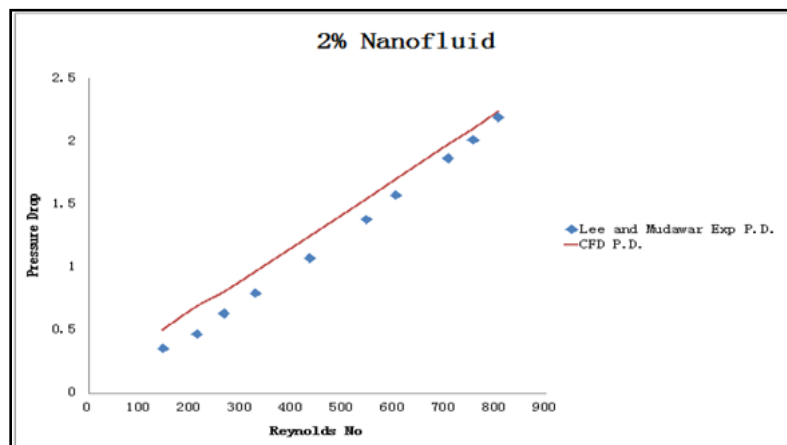


Fig. 6: Variation of computational and experimental pressure drop and friction factor of water with Re . (a) Pressure drop



(a)

Fig. 7: Variation of computational and experimental pressure drop and friction factor of 1% Alumina with Re. (a) Pressure drop

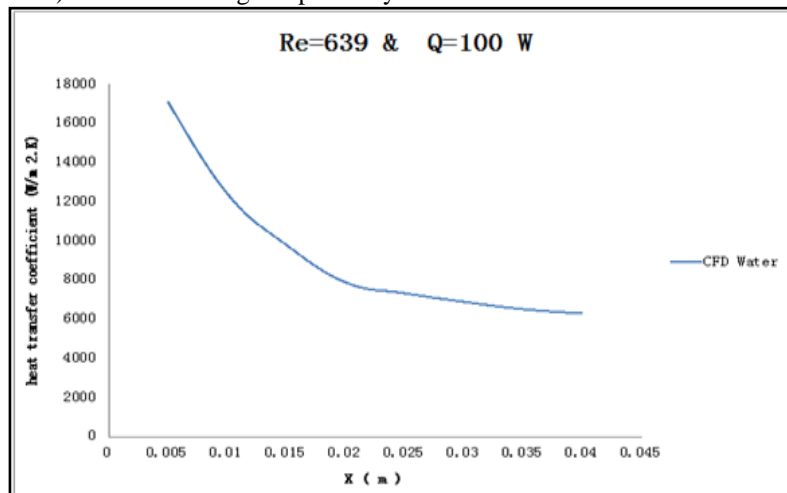


(a)

Fig. 8: Variation of computational and experimental pressure drop and friction factor of 2% Alumina with Re. (a) Pressure drop

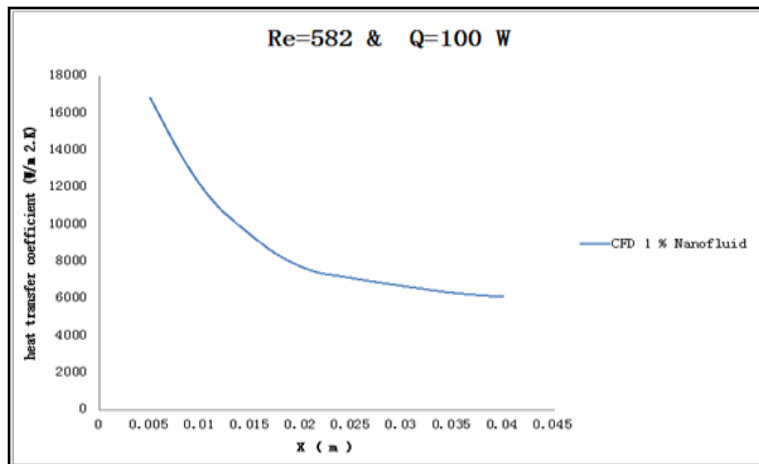
These show that as Reynolds number increases pressure drop increases and friction factor decreases. Here one thing is noticeable that the pressure drop increases with increasing nanoparticle concentration at the same Reynolds number. For example at Reynolds number 200, pure water and its nanofluid with alumina (1% and 2% volume concentration) gives pressure drop across micro channel equal to 0.38 bar, 0.46 bar and 0.61 bar respectively.

Variation of heat transfer coefficient on the bottom wall along micro channel at different power inputs for water and its nanofluids (1% and 2% alumina) are shown in Figs respectively.



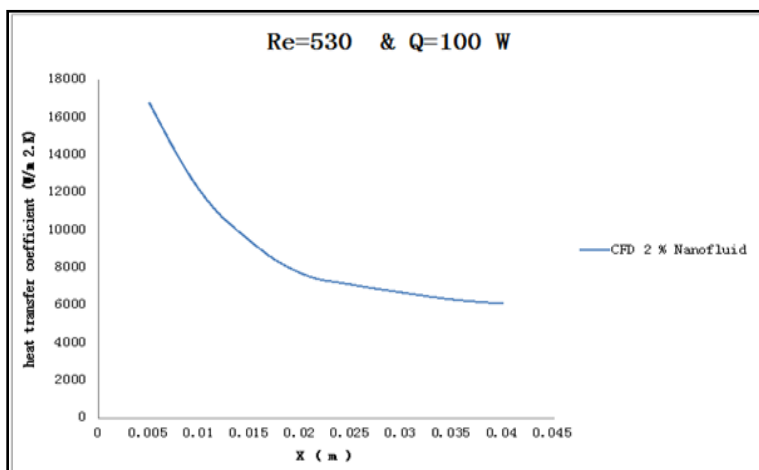
(a)

Fig. 9: Variation of Heat Transfer Coefficient for water along micro channel at different Power inputs (a) 100 W



(a)

Fig. 10: Variation of Heat Transfer Coefficient for Nanofluid with 1% Alumina along micro channel at different Power inputs (a) 100 W



(a)

Fig. 11: Variation of Heat Transfer Coefficient for Nanofluid with 2 % Alumina along micro channel at different Power inputs (a) 100 W

A decreasing trend of the heat coefficient values in the flow direction are observed in all the cases. Higher values of heat transfer coefficient are obtained at entry region of micro channel whereas lower values are obtained at the exit region for all fluid properties. For example heat transfer coefficient at entry region and exit region are 16,837 W/m²K and 5,811 W/m²K respectively at 100 W heat input for pure water. Here one noticeable thing is that increasing the heat flux has a very weak effect on the heat transfer coefficient of pure water and its nanofluids and the observation matches with Lee and Mudawar, 2007 observations.

The variation of wall temperature at different power inputs along micro channel are shown in Figs. using water and its nanofluids as the coolant.

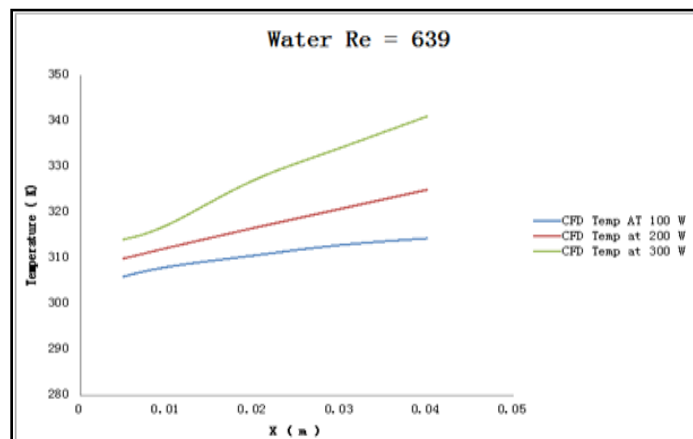


Fig. 12: Variation of wall temperature at different power inputs along micro channel for Water.

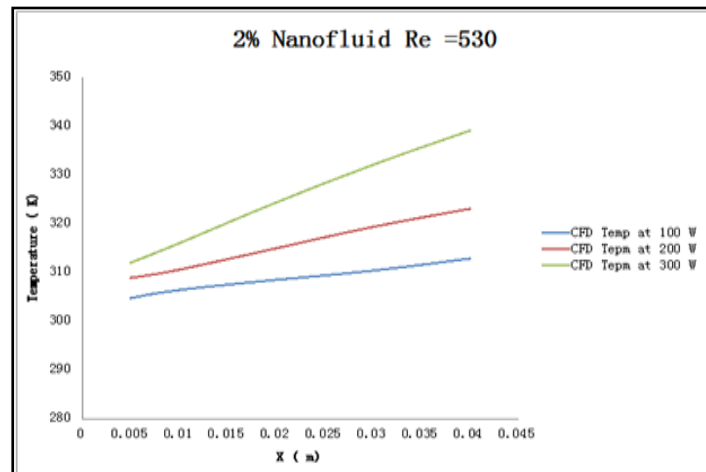
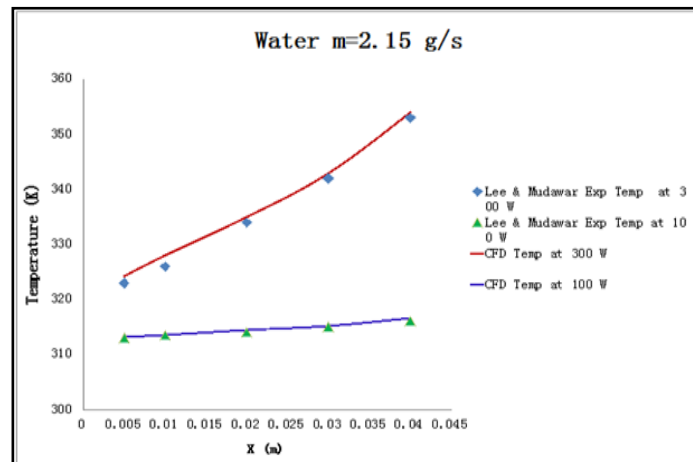


Fig. 13: Variation of wall temperature at different power inputs along micro channel for Nanofluid with 2 % Alumina

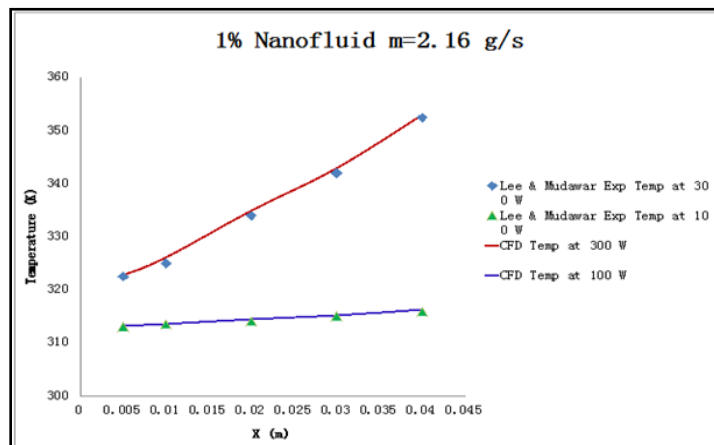
Graph shows that temperature is appeared to increase on the wall in the flow direction in all the cases. The rise in temperature from inlet to the outlet of the channel is found directly proportionate to heat input to the channel.

The computed values of wall temperature in the flow direction using water and its nanofluids are compared with the experimental data (Lee and Mudawar, 2007). These are displayed in Figs. The comparison shows that CFD results can predict well the experimental data.



(a)

Fig. 14: Variation of wall temperature for pure water at mass flow rate of (a) 2.15 g/s



(a)

Fig. 15: Variation of wall temperature for Nanofluid with 1 % Alumina at mass flow rate of (a) 2.16 g/s

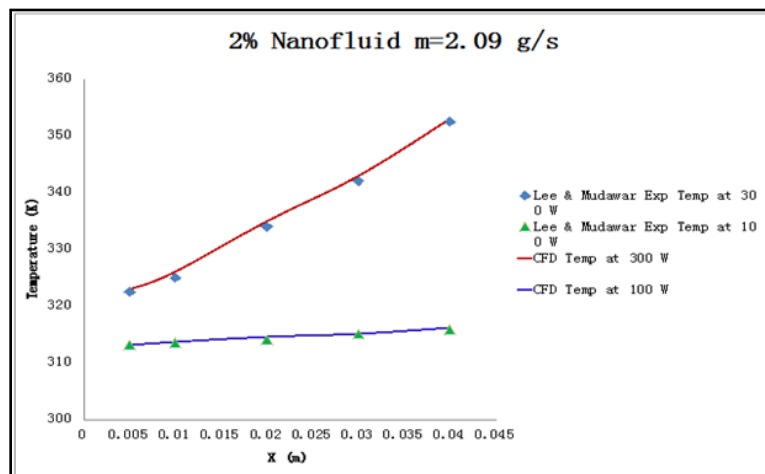


Fig. 16 Variation of wall temperature for Nanofluid with 2 % Alumina at mass flow rate of (a) 2.09 g/s

The hydrodynamic and thermal behavior is also studied in terms of contours of velocity, pressure and temperature on a surface passes through the centerline in the flow direction of the channel.

The plot shows that at lower re, there is distribution of temperature in the transverse direction. But at high Re because of high Pecklet number, the inlet temperature almost reaches to outlet around the centerline and only very nears to the wall some temperature distributions are observed in transverse direction.

VII. CONCLUSION

The theoretical work leads to the following conclusions:

- Pressure drop increases as Reynolds no. increases.
- Increasing nanoparticle concentration increases single- phase pressure drop compared to pure fluids at the same Reynolds number.
- Greater heat transfer coefficient is obtained at micro channel entrance.
- Wall temperature increases from entry region of micro channel to exit region.
- The computational result successfully validated the test data in terms of wall temperature distributions, pressure drop of the channel and friction factor.

REFERENCES

- [1] Allen, P. W. 2007. Experimental and Numerical Investigation of Fluid Flow and Heat Transfer in Microchannels. Msc Thesis, Mechanical Engineering Department, Louisiana State University.
- [2] Al-Nimr, M.A., Maqableh, M., Khadrawi, A.F. and. Ammourah, S.A. 2009. Fully developed thermal behaviors for parallel flow microchannel heat exchanger, International Communications in Heat and Mass Transfer. 36, 385-390.
- [3] ANSYS Fluent 14.0 Theory Guide, November , (2011).
- [4] Bachok, N., Ishak, A. and Pop, I. 2011. Flow and heat transfer over a rotating porous disk in a nanofluid. Physica B. 406, 1767 – 1772.
- [5] Bayraktar, T. and Pidugu, S.B. 2006. Characterization of Liquid Flows in Microfluidic Systems. International Journal of Heat and Mass Transfer. 49, 815-824.
- [6] Bahrami, M and Jovanovich, M. M. 2006. Pressure Drop of Fully Developed Laminar Flow in Microchannels of Arbitrary Cross-Section. Journal of Fluids Engineering. 128, 1036-1044.