Impact Factor (SJIF): 3.632

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 3, Issue 5, May-2016

DESIGN AND ANALYSIS OF MINICHANNEL HEAT EXCHANGER

Keval patel[1], Darshan Gajdhar[2], Devendra patel[3]

[1]Mechanical Engineering, DJMIT College
[2]Darshan Gajdhar Asst. Professor, Mech. Engg. Dept, DJMIT, College
[3]Devendra Patel Asst. Professor, Mech. Engg. Dept, DJMIT, College

Abstract------*In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. Electronic device are in heavy demand for computer processor applications and generate large amount of heat. These high power device can be cooled off very effectively by either liquid or gas coolant flowing through micro or mini channels. Continuous research work is ongoing for developing high speed processor which generates high amount of heat. Cooling of such particular systems requires high amount of mass flow rate and compactness is also required. , a mini channel heat exchanger is designed with assuming inlet and outlet of hot temperature, inlet of cold water temperature and also the mass flow rates of cold and hot water. CFD technique will be used to optimize model and best available model is to be selected for manufacturing.*

Keywords*- Heat transfer ,mass flow rate, pressure drop, cfd*

"I. INTRODUCTION"

A Heat Exchanger is a device built for efficient heat transfer from one medium to another, whether the media are separated by a solid wall so that why they never mix, or the media are in direct contact. They are widely used in space heating, refrigeration, air conditioning, power Plants, chemical plants, petrochemical plants, petroleum refineries, and natural gas processing. One common example of a heat exchanger is the radiator in a car, in which a hot engine-cooling fluid, like antifreeze, transfers heat to air flowing through the radiate.

In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak .Such exchangers are referred to as direct transfer type. In contrast, exchangers in which there is intermittent heat exchange between the hot and cold fluids via thermal energy storage and release through the exchanger surface or matrix are referred to as indirect transfer type, or simply regenerators. Such exchangers usually have fluid leakage from one fluid stream to the other, due to pressure differences and matrix rotation/valve switching.

Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single or multi component fluid streams. In other applications, the objective may be to recover or reject heat, or sterilize, pasteurize, fractionate, distill, concentrate, crystallize, or control a process fluid. Common examples of heat exchangers are shell and tube exchangers, automobile radiators, condensers, evaporators, air pre-heaters, and cooling towers.

" II.DESIGN"

Fluid flow inside channels is at the heart of many natural and man-made systems. Heat and mass transfer is accomplished across the channel walls in biological systems, such as the brain, lungs, kidneys, intestines, blood vessels, etc., as well as in many man-made systems. A channel serves to accomplish two objectives: (i) bring a fluid into intimate contact with the channel walls and (ii) bring fresh fluid to the walls and remove fluid away from the walls as the transport process is accomplished. The rate of the transport process depends on the surface area, which varies with the diameter D for a circular tube, whereas the flow rate depends on the cross-sectional area, which varies linearly with D2. Thus, the tube surface area to volume ratio varies as 1/D. Clearly, as the diameter decreases, surface area to volume ratio increases. In the human body, two of the most efficient heat and mass transfer processes occur inside the lung and the kidney, with the flow channels approaching capillary dimensions of around 4μm.

2.1 DESIGN PROCEDURE

ε - NTU approach is used for designing of this counter-flow heat exchanger. The purpose of this heat exchanger is to reduce the temperature of incoming hot fluid and deliver it at a lower temperature. So inlet and outlet temperature of hot fluid and inlet temperature of cold fluid is taken as reference. Moreover since cold water is available in abundance Cold-Hot-Cold configuration is adopted so that hot water gets cooled faster. The section of this configuration is shown in Figure 2.1. Cross-section dimensions of each channel are taken 1mm ×1mm. Other design parameters are given in Table. 2.1

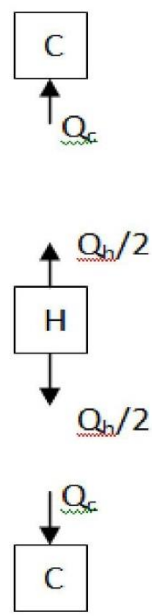
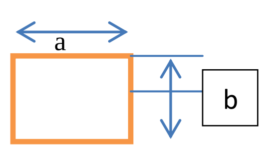


Figure2.1C-H-Carrangement

	HOTFLUID	COLDFLUID
Inlet Temperature(˚C)	T_{hi} = 80	T_{ci}=25
Outlet Temperature(˚C)	T_{ho}=70	-
Total mass flow rate(kg/s)	m_{ht}= 02	m_{ct}=0.2
Number of channels	n_t =60	n_c= 120

Table2.1 Design parameters

- Cross-sectional dimensions of channel:
  Width a = 1 mm
  Depth b = 1 mm
- Cross-sectional area,
  Ac= 1 mm2
- Perimeter



P= 2 (a + b) = 4 mm

- Hydro-dynamic diameter of channel,
  dh = $\frac{4A_c}{p}$ = 1 mm
- Mass flow rate through each hot channel
  $m_h = \frac{\dot{m}_{ht}}{n_t}$ =0.0033 kg/s
- Mass flow rate through each cold channel
  $m_c = \frac{\dot{m}_{ct}}{n_t}$ =0.00166 kg/s

- Cross sectional dimensions of header
  Width a = 6mm
  Depth b = 1mm

2.1.1 COLD CHANNEL CONDITIONS:

- ρ=998.25kg/m^3
- μ=890.059x10^{-6} Pas
- Pr = 6.129
- Reynold number
  $Re = \frac{\rho\, u_m\, d_h}{\mu}$
  Writing this equation in the form of mass flow rate,
  $Re = \frac{d_h \dot{m}_c}{\mu A_c}$
  Re = 1872.39
  Flow through cold fluid channels is laminar.

- Thermal entry length,Z_c
- Z_c = 0.1 Re Prd_h
  Z_c =1147.6 mm

2.1.2 HOT CHANNEL CONDITIONS

- ρ=991.83kg/m^3
- μ=354.374x10^{-6} Pas
- Pr=2.2194

  $Re = \frac{d_h \dot{m}_h}{\mu A_c}$
  Re =9405.76
  Flow through hot fluid channels is turbulent.
- For turbulent flow, if ratio of L to d_h comes out to be less than 10 thermal entry effect is neglected. Here, as d_h is only 1.5 mm and heat exchanger is designed for 10 C temperature drop length of channels is expected to be more than 10 mm, thermal entry effect is neglected.

2.1.3 HEAT TRANSFER CALCULATIONS

Consider a small element of heat exchanger of length 1 mm from the entry of thecold fluid. For one module of C-H-C configuration length of hot and cold fluid channels is 1 mm only so properties of fluid are considered to be constant for this length. These constant properties are calculated at known temperature of both fluids (i.e. for cold fluid, temperature of fluid entering in the segment and for hot fluid, temperature of fluid leaving the segment).

ε-NTU approach is used to calculate cold fluid outlet temperature and hot fluid inlet temperature for first segment.

For hot fluid

- Nusselt number, using equation.

  $Nu = Nu_{Gn}(1+F)$

  $Nu_h = 9.76$

- Convective heat transfer coefficient equation.

  $h = \frac{Nu\,k}{dh}$

  $h_h = 5860.243$ W/m^2 k.

For cold fluid

- The non-dimensionalized length x* is given by

  $X^* = \frac{x/d_h}{RePr}$

  $x_c^* = 0.000087$

The local heat transfer in developing region of a circular tube is given by the following equations

$Nu_x = 4.363 + 8.68(10^3 x^*)^{-0.506} e^{-41x^*}$

- $Nu_c = 25.288$
- Convective heat transfer coefficient

  $h = \frac{Nu\,k}{dh}$

  $h_c = 15172.8$ W/m^2 k.

overall heat transfer coefficient

$\frac{1}{u} = \frac{1}{h_h} + \frac{1}{k} + \frac{1}{h_c}$

U =4154.89W/m^{2}k

$C_c = \dot{m}_c c_p = 6.9487$ w/k

$C_h = \dot{m}_h c_p = 13.8138$ w/k

$C_{min} = C_c$ and $C_{max} = C_h$

$C = C_{min}/C_{max} = 0.5$

Number of transfer units

$NTU = (U*A)/C_{min} = 0.00059$

For counter flow,

Effectiveness

$\varepsilon = \frac{1-\exp[-(1-C)NTU]}{1-C\exp[-(1-C)NTU]} = 0.0005897$

$\varepsilon = \frac{\text{available heat transfer}}{\text{maximum possible heat transfer}} = \frac{Thi-Tho}{2C(Thi-Tci)}$

T_{hi} = **70.069839 °C**

$Q_h = 2\, Q_c$

$C_h(T_{ho}-T_{hi}) = 2C_c(T_{co} - T_{ci})$

T_{co}=25.06 ‘C

The outlet temperature of cold fluid for first segment is the inlet temperature for the next cold segment and inlet temperature of hot fluid for first segment is the outlet temperature for the next hot segment. This iteration goes on till 80˚C temperature of hot water at inlet is obtained.

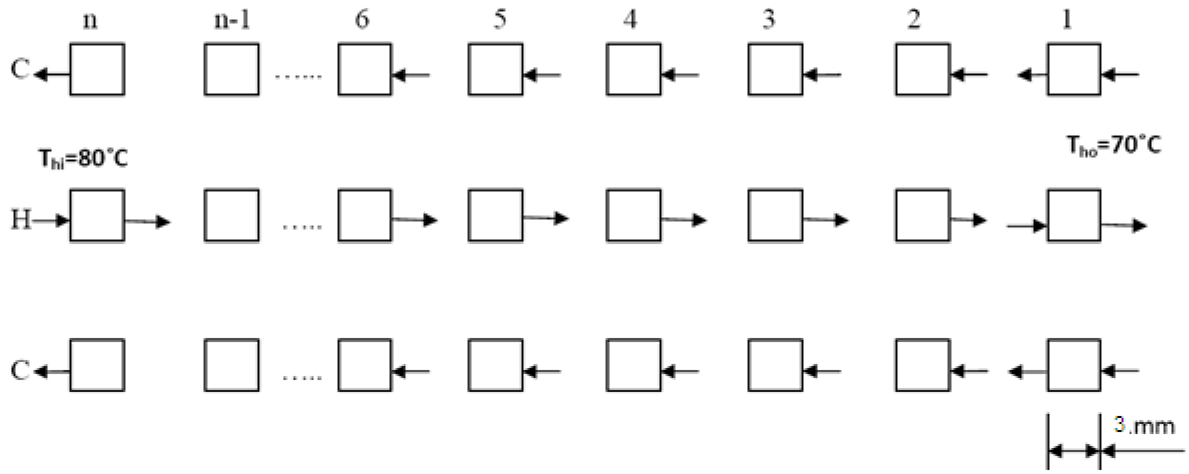


Figure 2.2 Schematic Representation Of Iterative Calculation (C-H-C Config.)

The program terminates when desired hot inlet temperature is achieved. Thus the total length of the heat exchanger is obtained.

2.2 PRESSURE DROP CALCULATIONS:

Dimensions of headers: Header of rectangular cross-section is used which provides flow to all the channels in one plate (i.e.11 channels).

width = 6 mm

height = 1 mm

Dimension of nozzles: Standard coupling of 1” diameter is used to provide flow to headers

2.2.1 For The Flow Of Hot Fluid

Combined pressure drop from the nozzle to header at inlet and from header to nozzle at outlet :

Mean velocity of flow in header , Um= m/ρA

where A is the cross-section area of header. Here, width of header is 6 mm and depth is 1 mm.

U_m = 11.111 m/ s

Area ratio $= \frac{area\ of\ header}{area\ of\ nozzle}$

= 0.0079

Reynolds number of flow through nozzle

$$Re = \frac{\rho\, um\, dh}{\mu}$$

Where $D_h = \frac{2*Ah*Bh}{(Ah+Bh)} = \frac{2*6*1}{(6+1)}$ =1.71 mm

Where$d_h$is hydraulic diameter of header,

Re = 53302.64

From,Appendix B, value of contraction coefficient, Kc and expansion coefficient, Keare 0.5 and 1 respectively.

Combined pressure drop from the nozzle to header at inlet and from header to nozzle at outlet,

$$\Delta p1 = \frac{\rho\, Um\, (Kc + Ke)}{2}$$

$\Delta p1$= 0.92 bar

Combined pressure drop from header to minichannels at inlet and from minichannels to header at outlet :

Mean velocity of flow in header,

U_m= m/ρA

=3.33 m/s

Area ratio =$\frac{area\ of\ header}{area\ of\ nozzle}$

=0.1666

Reynolds number of flow through channels

Re =$\frac{\rho\, um\, dh}{\mu}$

=9405.76

From the graph value of contraction coefficient, K_c and expansio$K_e$are 0.5 and 0.68 respectively.

Considering losses at 90' bend to be 1.2,

Combined pressure drop from the header to channels at inlet and from channels to header at outlet

$$\Delta P2 = ((\frac{\rho u_{m^2}}{2}\left(\frac{A\,channel}{A\,header}\right)^2 * K90) + Kc + Ke)$$

ΔP2 = 0.067bar

Pressure drop in the channels:

Re* =$\frac{\rho\, Um\, Dh\, (\left(\frac{2}{3}\right)+\left(\frac{11}{24}\right)-\left(\frac{1}{\alpha}\right)(2-\left(\frac{1}{\alpha}\right))}{\mu}$

=6330.57

f=0.0791 Re ^-0.25

=0.0088

$$\Delta P3 = \frac{2\, f\, Re\, \mu\, \dot{m}}{Dh^2\, Ac\, \rho}$$

$\Delta P3$ =0.68 bar

Total pressure drop of hot fluid in the heat exchanger

$\Delta P = \Delta P_1 + \Delta P_2 + \Delta P_3$

= 1.667 bar

2.2.2 For The Flow Of Cold Fluid

Combined pressure drop from the nozzle to header at inlet and from header to nozzle at outlet :

Mean velocity of flow in header , Um=ṁ/ρA

where A is the cross-section area of header. Here, width of header is 6 mm and depth is 1 mm.

$U_m = 5.555$

Area ratio $=\frac{area\ of\ header}{area\ of\ nozzle}$

= 0.0079

Reynolds number of flow through nozzle

$$Re = \frac{\rho\ um\ dh}{\mu}$$

Where Dh = $\frac{2*Ah*Bh}{(Ah+Bh)} = \frac{2*6*1}{(6+1)}$ =1.71 mm

Where$d_h$is hydraulic diameter of header,

Re = 10671.65

From,Appendix B, value of contraction coefficient, Kc and expansion coefficient, Keare 0.49 and 1 respectively.

Combined pressure drop from the nozzle to header at inlet and from header to nozzle at outlet,

$$\Delta p1 = \frac{\rho\ Um\ (Kc + Ke)}{2}$$

$$\Delta p1 = 0.23\text{bar}$$

Combined pressure drop from header to minichannels at inlet and from minichannels to header at outlet :

Mean velocity of flow in header,

U_m= m/ρA

=1.666 m/s

Area ratio $=\frac{area\ of\ header}{area\ of\ nozzle}$

=0.166

Reynolds number of flow through channels

$$Re = \frac{\rho\ um\ dh}{\mu}$$

=1872.39

From the graph value of contraction coefficient, K_c and expansion $K_e$are 1 and 0.6 respectively.

Considering losses at 90’ bend to be 1.2, Combined pressure drop from the header to channels at inlet and from channels to header at outlet

$$\Delta P2 = ((\frac{\rho u_{m^2}}{2}\left(\frac{A\,channel}{A\,header}\right)^2 * K90) + Kc + Ke)$$

ΔP2 =0.17bar

Pressure drop in the channels:

Re* $=\frac{\rho\ Um\ Dh\ (\left(\frac{2}{3}\right)+\left(\frac{11}{24}\right)-\left(\frac{1}{\alpha}\right)(2-\left(\frac{1}{\alpha}\right))}{\mu}$

=957.47

f=0.0791 Re ^-0.25

=0.0142

$$\Delta P3 = \frac{2\ f\ Re\ \mu\ \dot{m}}{Dh^2\ Ac\ \rho}$$

$\Delta P3$ =0.162461bar

Total pressure drop of hot fluid in the heat exchanger

$\Delta P = \Delta P_1 + \Delta P_2 + \Delta P_3$

= 0.05765+ 0.001496+ 0.01838

= 0.412bar

III.” CFD MODELLING AND SIMULATION”

3.1 OVERVIEW OF CFD:

CFD is a numerical technique which relies on solving fundamental equation of fluid motion to get the flow field. The most general set of equation that describe Newtonian fluid flow are well known Navier-Stokes equations. Due to their complexity (they are three dimensional, transient, non-linear, partial differential equations) Navier-Stokes equations have not been solved analytical to date. On the other hand numerical procedures have been developed in the past few decades that enable obtaining approximate solution of Navier-Stokes equations for a wide variety of flow problems. Navier-Stokes equation describe equally well both laminar and turbulent flow, however, obtaining solution for fully turbulent flow fields is still a complex task involving vast quantities of data and long computational times. The technique that is used to achieve this called direct numerical simulation (DNS) of the Navier-Stokes equations and practically feasible only for flow at low Reynolds numbers through simple geometries (e.g. flat channel, straight pipe with or without small obstacles). A variety of approximate techniques have been developed to simulate highly turbulent flow. The turbulence can be approximate in a number of ways and a method which has gained much popularity in the past few decades is the so called k-ε model combined with the wall function approach, which was utilized in this study.

3.2 NUMERICAL SIMULATION OF MINICHANNEL HEAT EXCHANGER

Detailed knowledge of the flow, temperature and turbulence fields within a heat exchanger helps towards the design of reliable and efficient units. This information can be obtained by experimental testing or by numerical computations. Experimental testing is usually expensive and time consuming. In addition, flow visualization and detailed turbulence measurements in heat exchangers are difficult to perform. A well validated numerical model can serve as a cost-effective research tool for problem solving and the design of mini channel heat exchangers

3.3 CREATING THE GEOMETRY AND MESH

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in the Meshing application or any of the other geometry/mesh creation tools. The basic steps involve:

1. Defining the geometry of the region of interest
2. Creating regions of fluid flow, solid regions and surface boundary names
3. Setting properties for the mesh

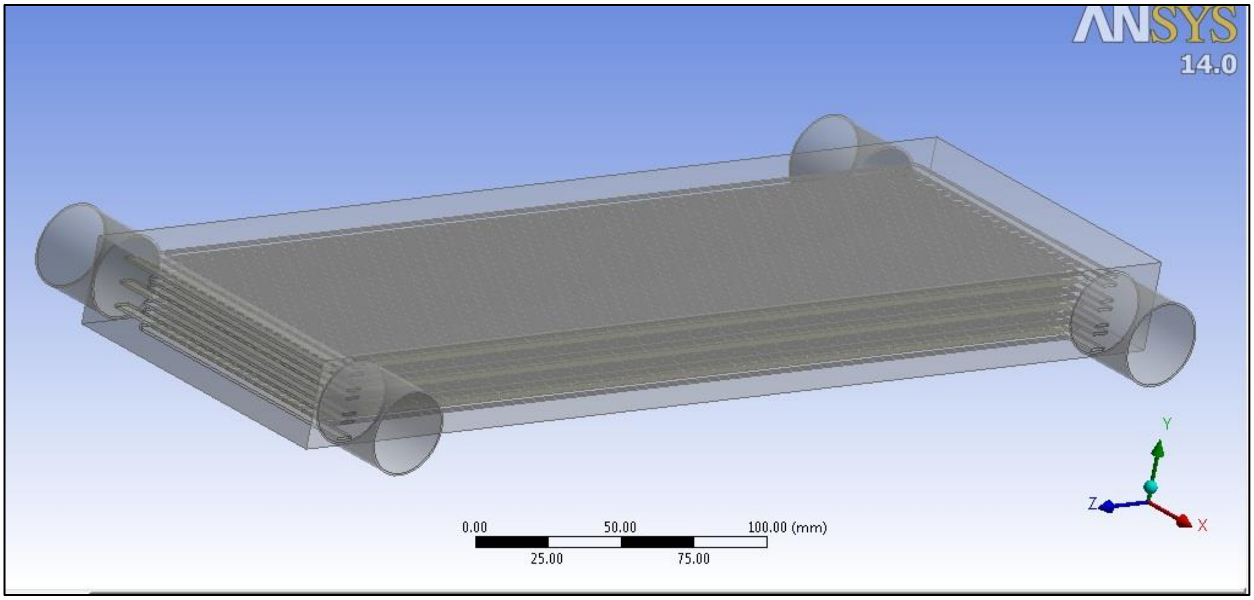


Figure 3.1 Computational Model of mini channel heat exchanger

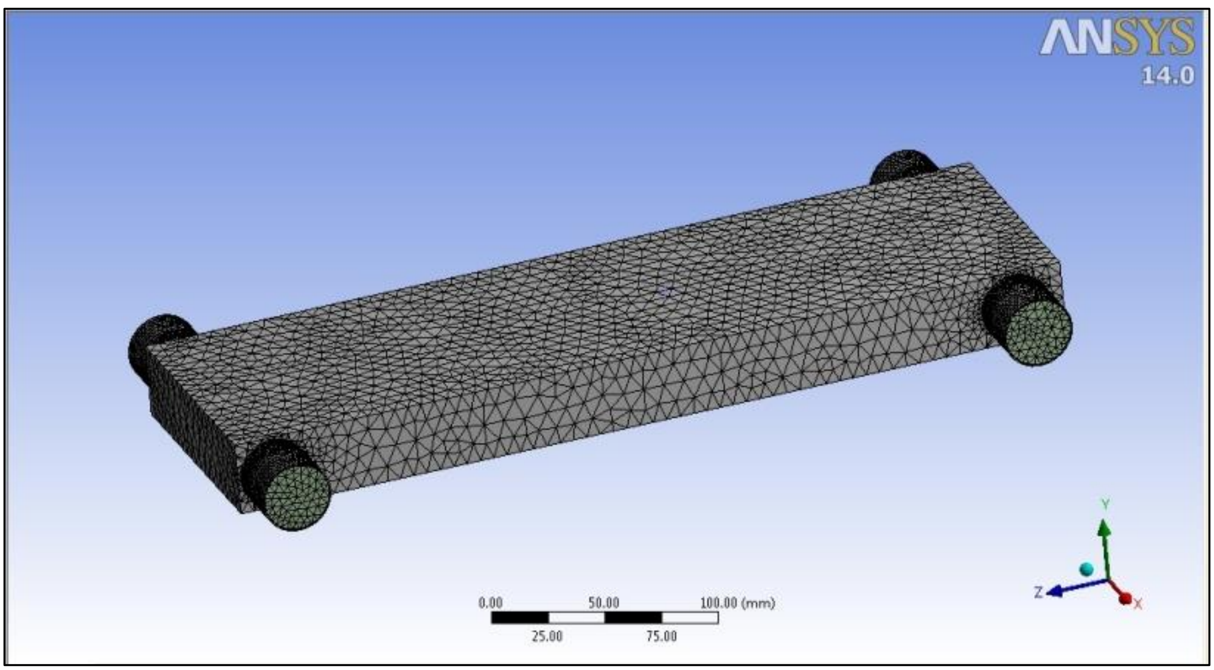


Figure 3.2 Mesh Diagram of mini channel heat exchanger

Hydraulic diameter (mm)	1
No of channels hot side	20
No of channels cold side	20
No of hot plate	3
No of cold plate	6
Width of channel **(**mm)	1
Depth of channel (mm)	1
Header width (mm)	6
Header depth (mm)	1

Table 3.1 Design Parameters and Fixed Geometric Parameters

3.4 BOUNDARY CONDITIONS:

3.4.1 INLET BOUNDARY CONDITION

The inlet boundary condition was "mass flow rate inlet". The mass flow rate and temperature values were specified at the inlet of channel domain. The direction of the inlet mass flow rate was normal to surface. Percentage turbulence intensity and hydraulic diameter were also specified while solving the turbulence equation. The value of the hydraulic diameter was specified as per the equation. Turbulence intensity of 1% was generally considered to be low, while 10% was considered to be high. The values of turbulence intensity were estimated based on an empirical correlation provided in Ansys CFX for fully developed duct flow, which were in the range of 4% to 6%. These values were in agreement with the range specified by Ansys CFX for complex flows.

3.4.2 Outlet Boundary Condition

At the outlet, the "pressure outlet" boundary condition was specified as a constant value equal to zero gauge pressure. Back flow total temperature, percentage turbulence intensity and hydraulic diameter were also specified.

3.4.3 THE HEAT EXCHANGE WALL SURFACES

Now boundary conditions are presented. Non-slip boundary condition is applied on the inner wall of the channel and all solid surfaces within the computational domain. The standard wall function method is used to simulate the flow in the near-wall region. The temperatures of channel are set as constant and their values are taken from the average wall temperature determined in the experiments. The channel of heat exchanger is set as adiabatic. Heat conduction of channel in heat exchanger is considered by using plate conduction in thin-walls model in ANSYS CFX.

3.5 SOLUTION PROCEDURE

The computational domains are meshed with unstructured Tet/Hybrid grids. They are meshed by the CFX software, a geometric modeling and grid generation tool used with ANSYS CFX. In order to ensure the accuracy of numerical results, a careful check for the grid independence of the numerical solutions is conducted. Four different grid systems are meshed for CH-STHX and the final cell numbers for the studied cases are 1823224.

The computational processes carried out using ANSYS CFX , a commercial Computational Fluid Dynamics (CFD) code which is based on the finite volume method. The governing equations were iteratively solved by using SIMPLE pressure-velocity coupled algorithm. This numerical approach stores scalar variables are also required for the convection terms and their values are gained from interpolation.. As the convergence criterion, the sum of the normalized absolute residual in each control volume for energy variables (T,k,ε) are less than 10^{-5}. All computations are preformed on the personal computer with 4 GB RAM and IntelR $Core^{TM}$ 2.40 GHz CPU. Each simulation took 48-50 hours to converge.

IV.” RESULT”

Numerical simulation is performe on the ANSYS CFX with fluid (water) in channel side along with mass flow rate and maintaining the same inlet temperature. Water is used as working fluid on channel plate. The numerical simulation is performed with three dimensional, steady state and turbulent flow system, segregated solver and standard k-ε model are employed and energy equation is included. For flow analysis here Ansys CFX is used as a post processor.

3.4.1 NUMERICAL ANALYSIS

As shown below, different contours of temperature is shown of channel. These contours are plotted at the mid plane of the model of channel type heat exchanger.

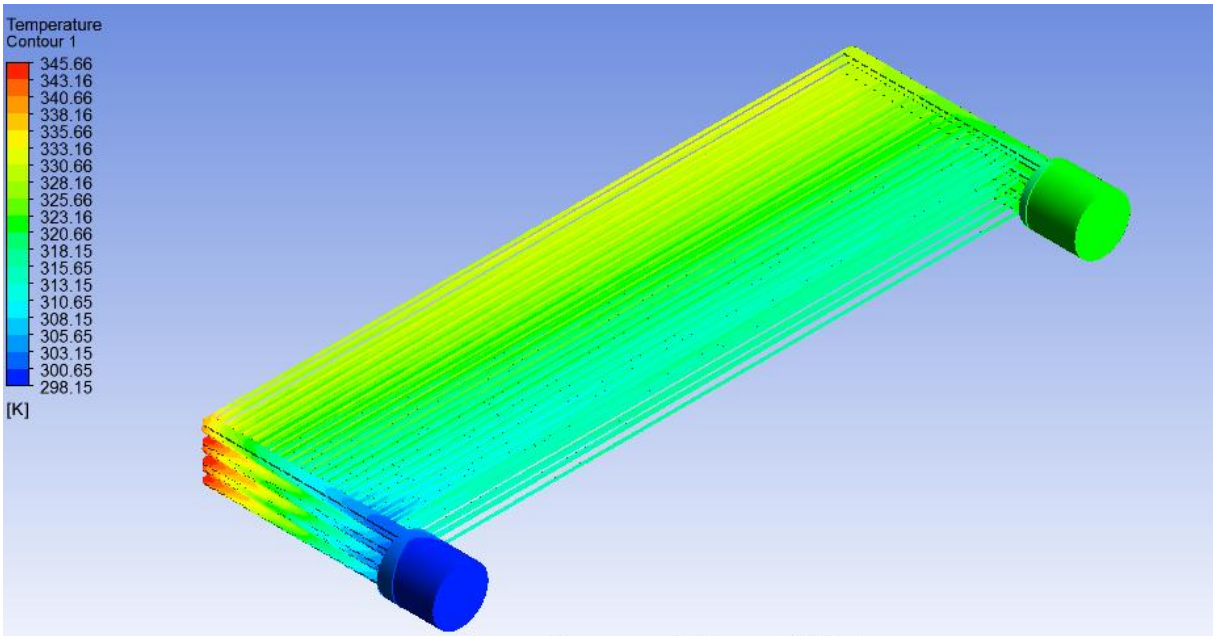


Figure 4.1 Temperature contour of cold fluid

As shown in above Figure 4.1 cold fluid temperature increase up to 321 k at mass flow rate 0.02 kg/s cold side.

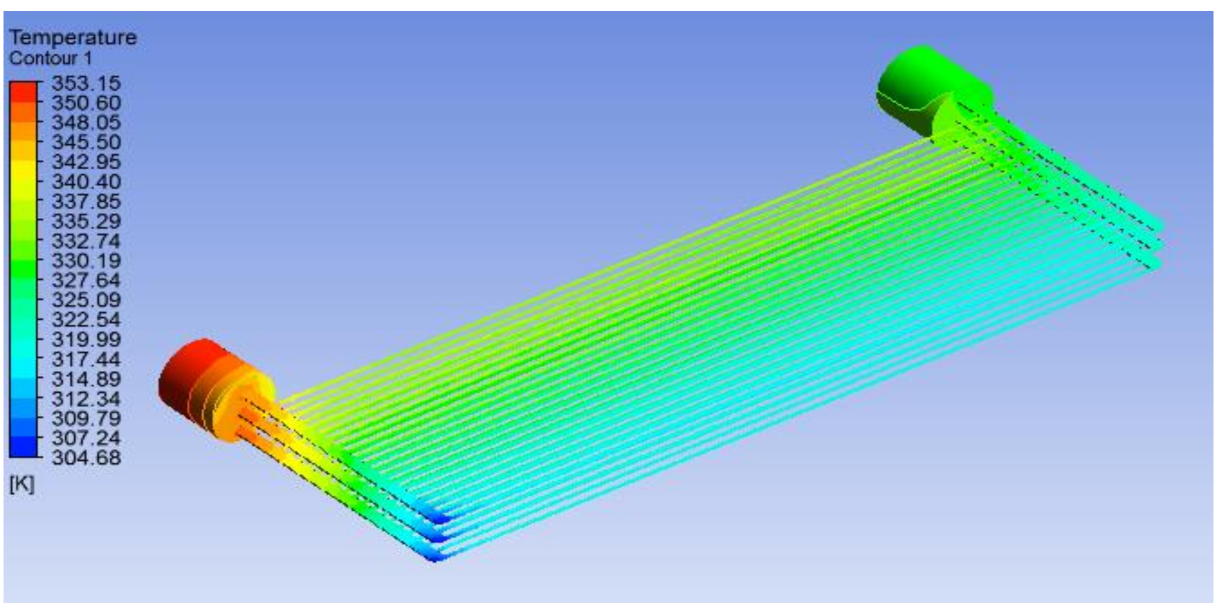


Figure 4.2Temperature contour of hot fluid

As shown in Figure 4.2 hot temperature decrease from 353.15 k to 330 k. and mass flow rate from hot side is 0.02 kg/s.

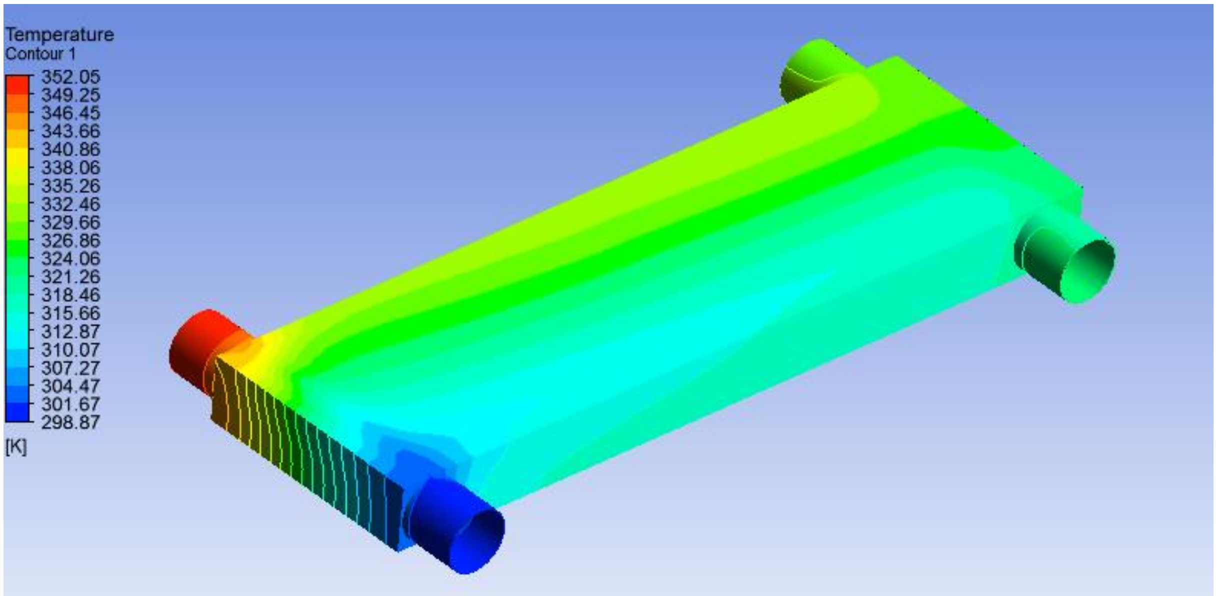


Figure 4.3 Temperature contour on aluminum cover

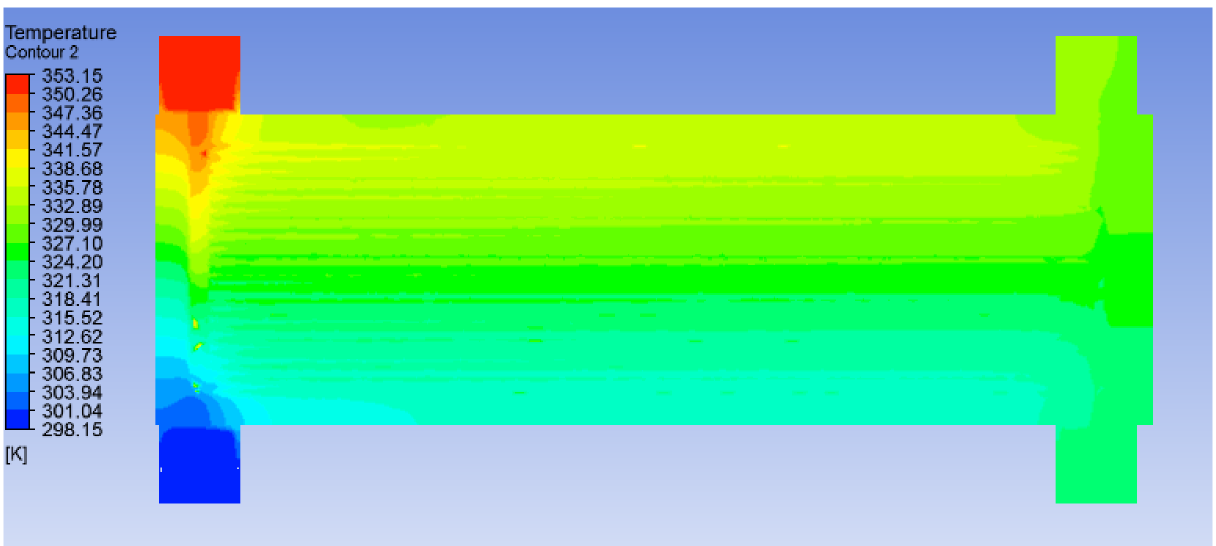


Figure 4.4 Temperature contour on mid plane of heat exchanger

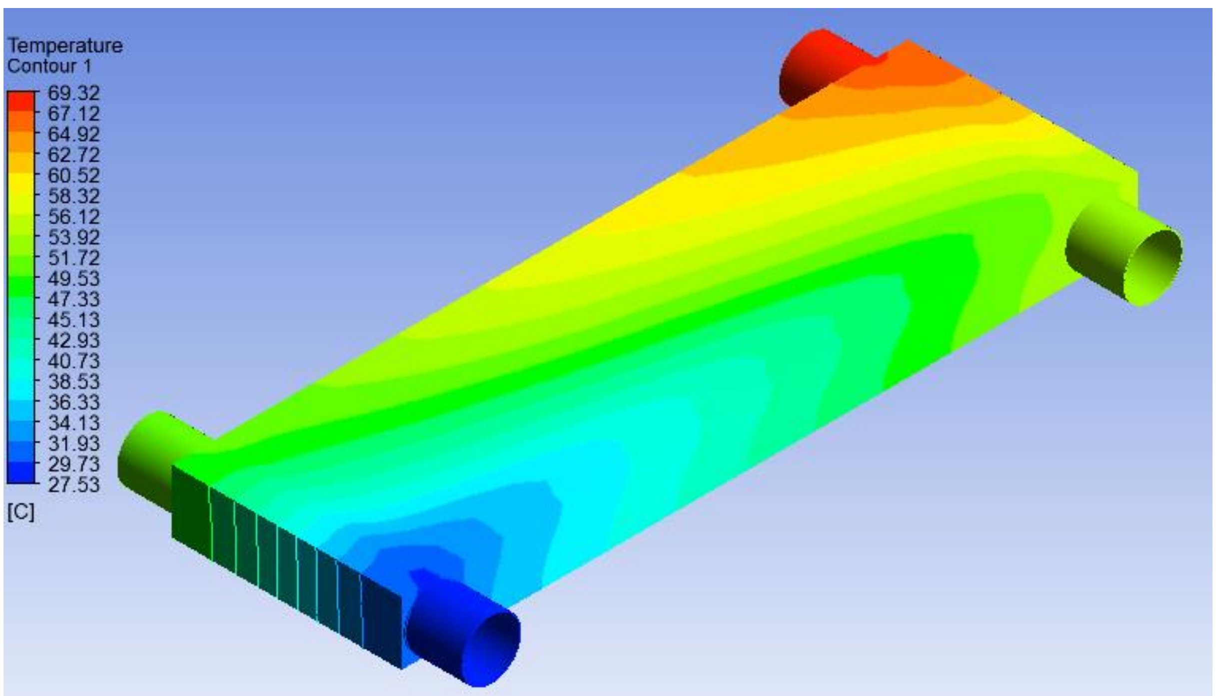
Figure 4.4 shows the temperature distribution on the aluminum cover and for CFD analysis, we consider its outer boundary as adiabatic wall i.e. fully insulated wall.

Result table of parallel flow mini channel heat exchanger:

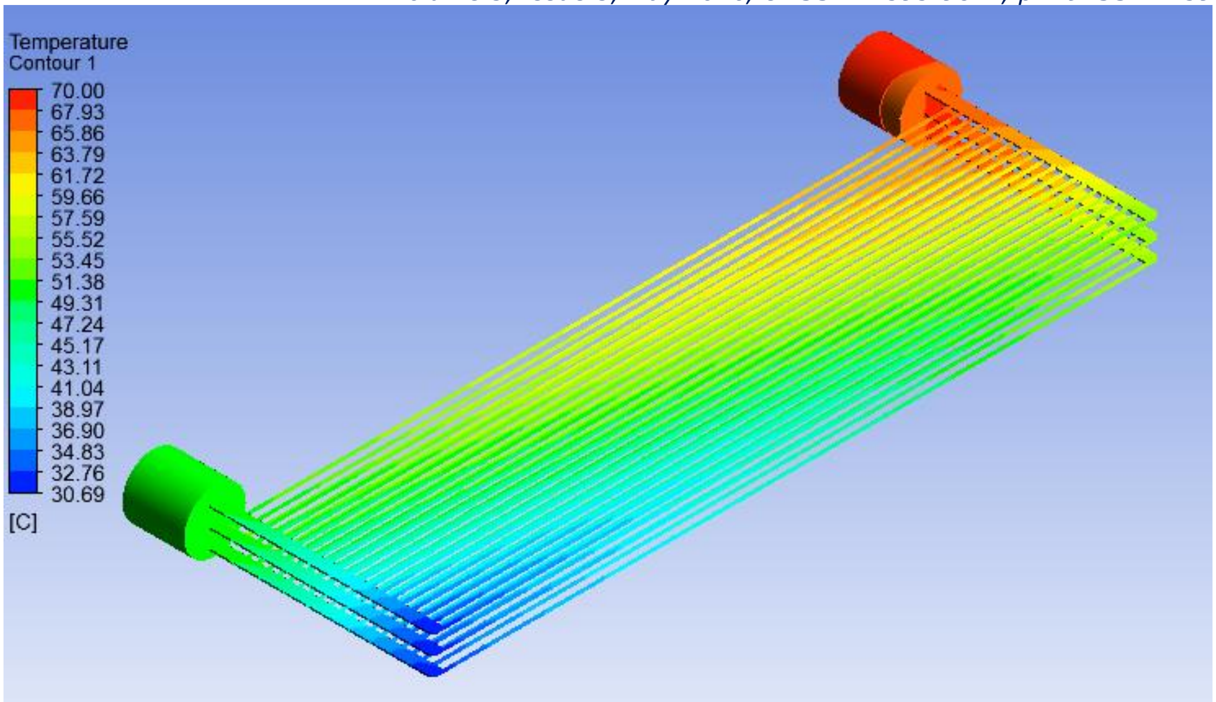
Sr. No.	Description	Value
1.	Inlet temperature of cold water (^{o}C)	25
2.	Outlet temperature of cold water (^{o}C)	47.85
3.	Temperature increase for cold water (^{o}C)	22.85
4.	Inlet temperature of hot water (^{o}C)	80
5.	Outlet temperature of Hot water (^{o}C)	56.85
6.	Temperature decreases for hot water (^{o}C)	23.15
7.	Cold water mass flowrate (kg/s)	0.2
8.	Hot water mass flowrate (kg/s)	0.2

Now we do counter flow mini channel heat exchanger analysis from case II to V. as shown below:

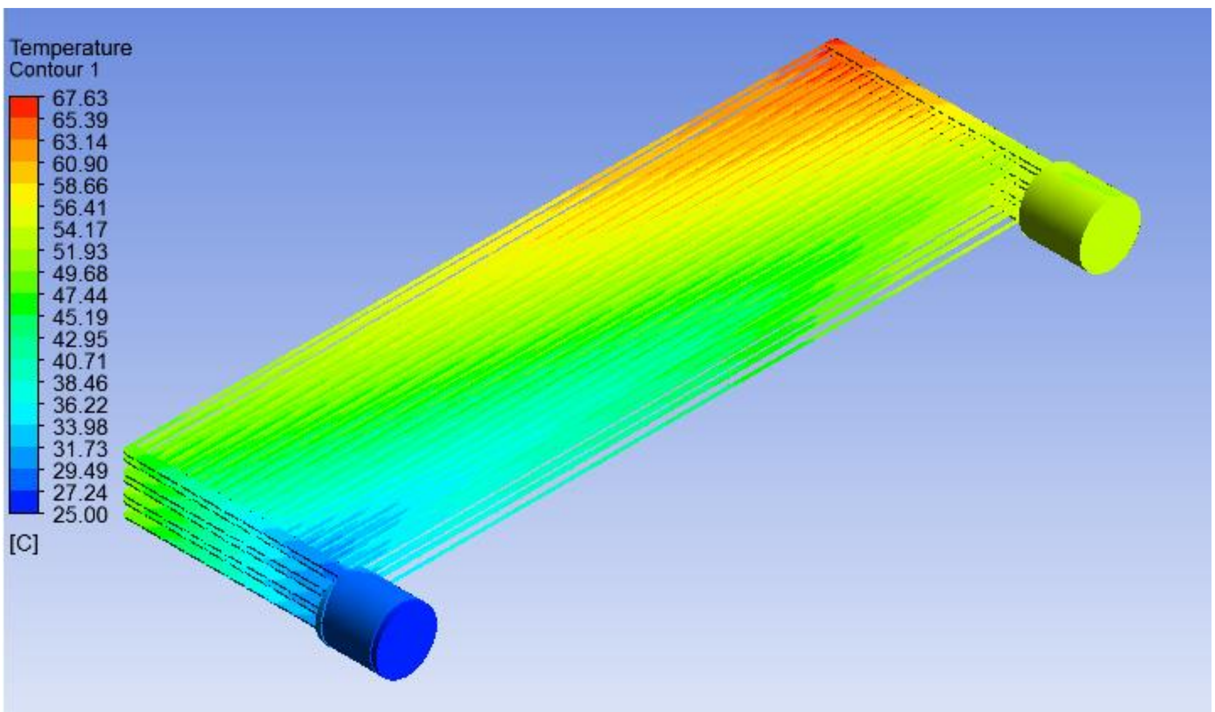
Case II



As shown in figure cover plate with inlet temperature 27.53 c and outlet temperature 69.32 c and hot fluid mass flow rate 0.2 kg/s and cold fluid mass flow rate is 0.1 kg/s.

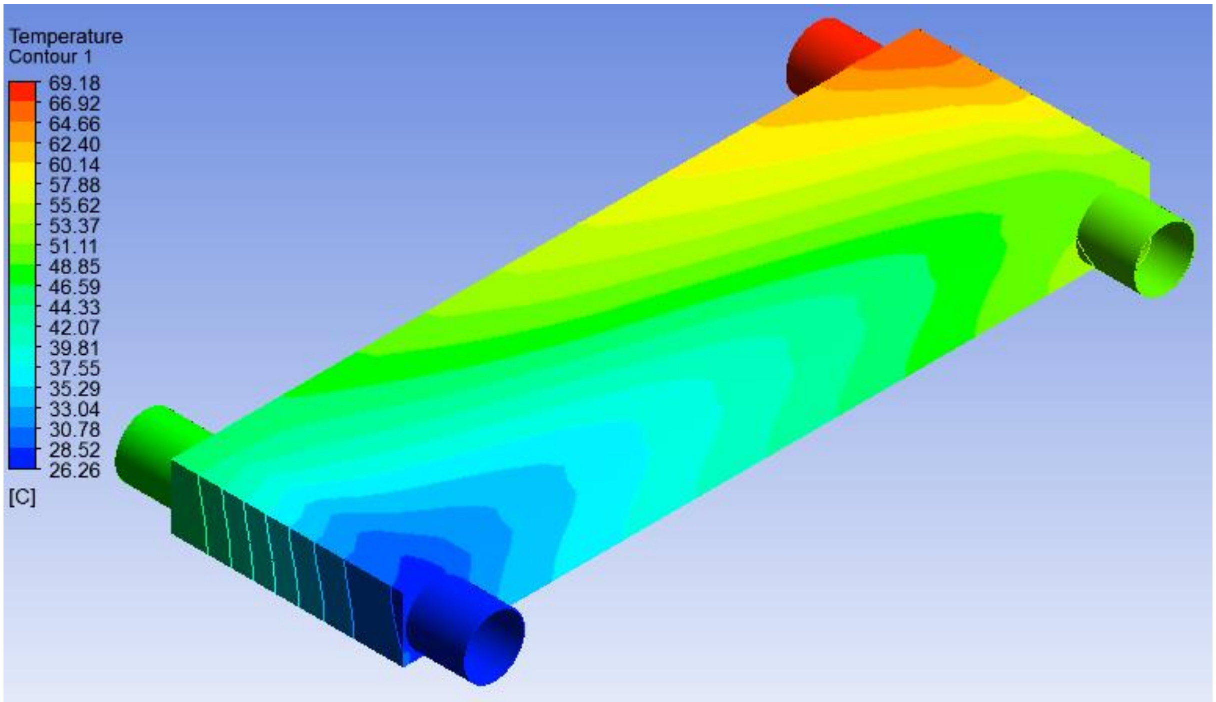


In above figure hot plate with inlet temperature is 70.00 c and its cold to 30.69 c.

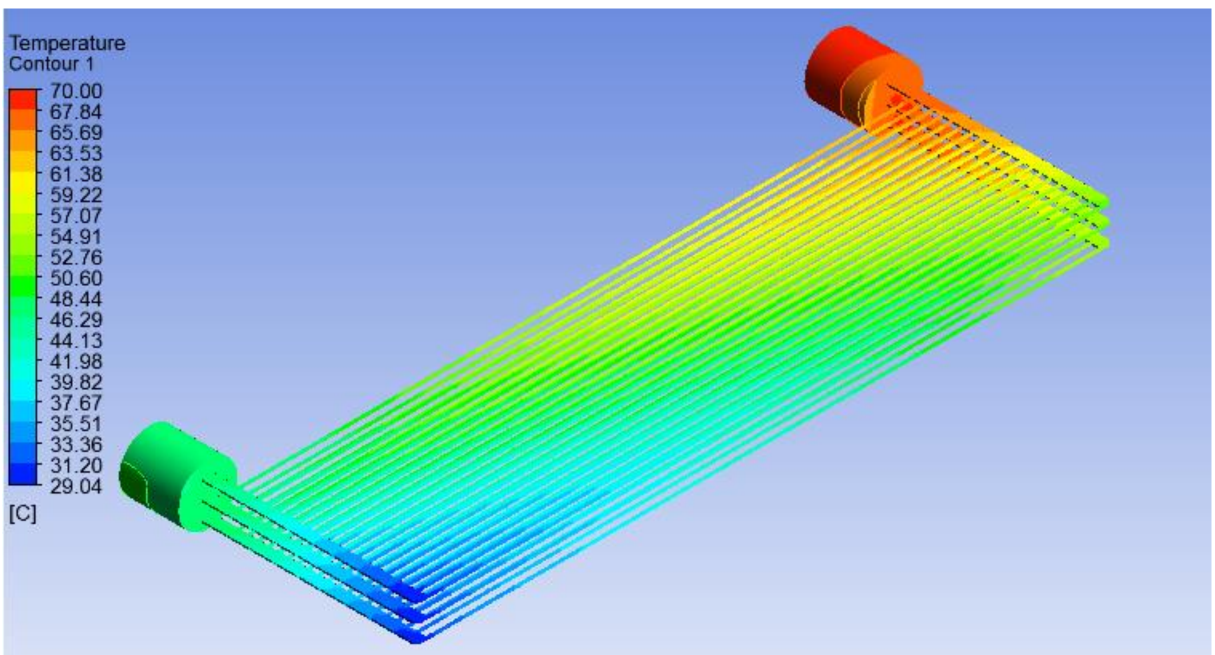


In above figure cold fluid inlet temperature is 25.00 and it hot to 67.63 c .

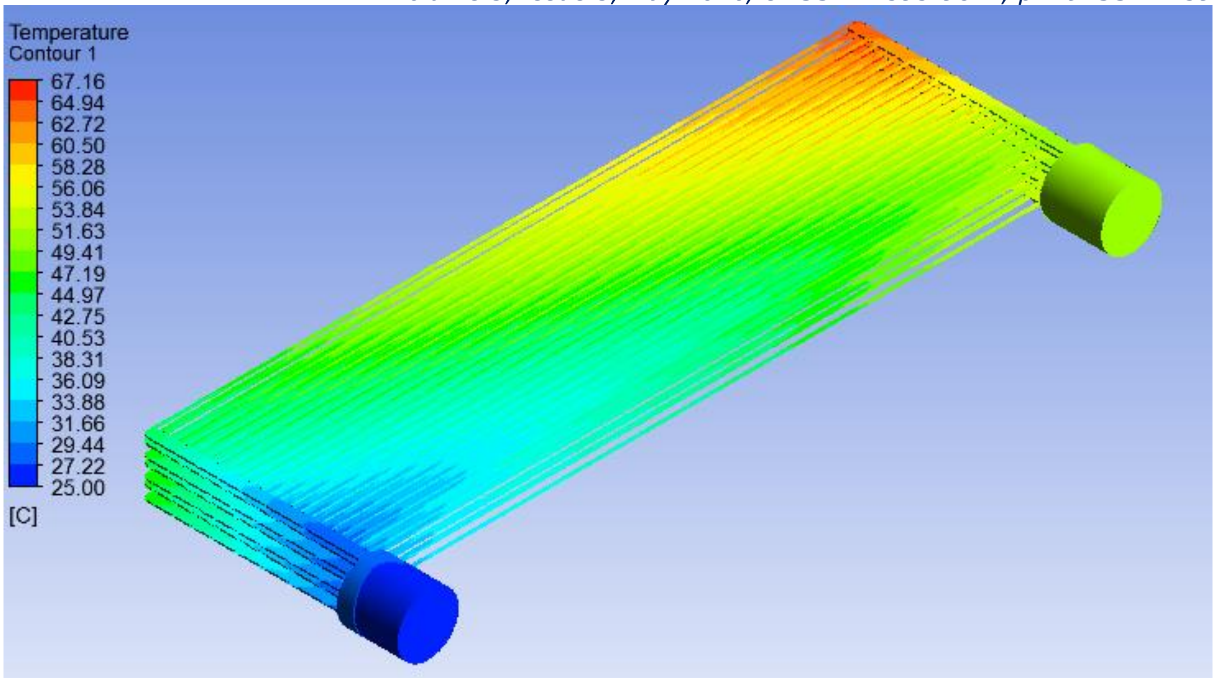
CASE III



As shown in figure cover plate with inlet temperature 27.53 c and outlet temperature 69.32 c and hot fluid mass flow rate 0.2 kg/s and cold fluid mass flow rate is 0.15 kg/s.

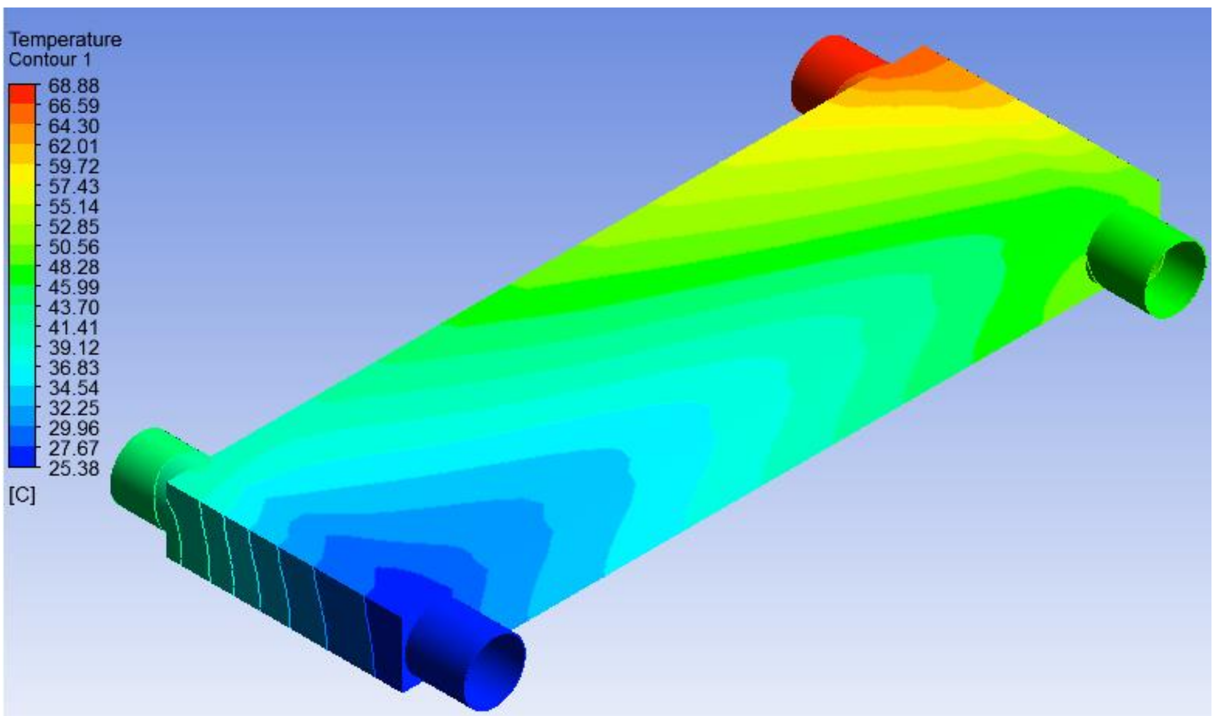


In above figure hot plate with inlet temperature is 70.00 c and its cold to 29.04 c.

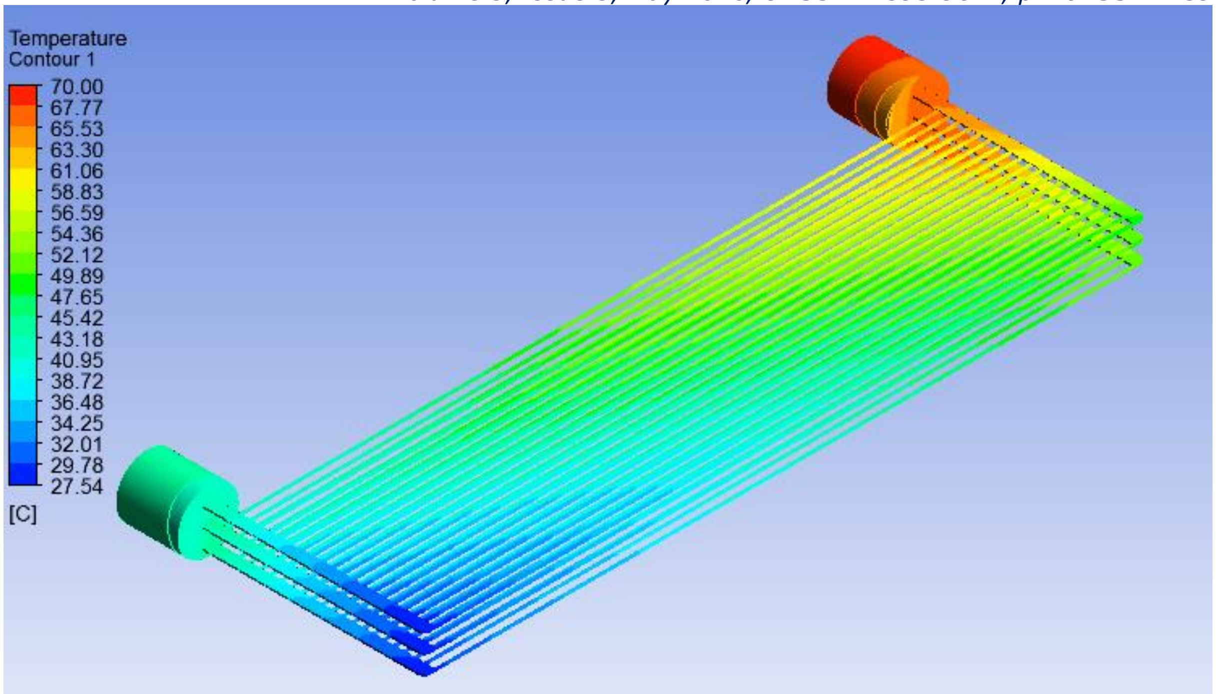


In above figure cold fluid inlet temperature is 25.00 and it hot to 67.16 c .

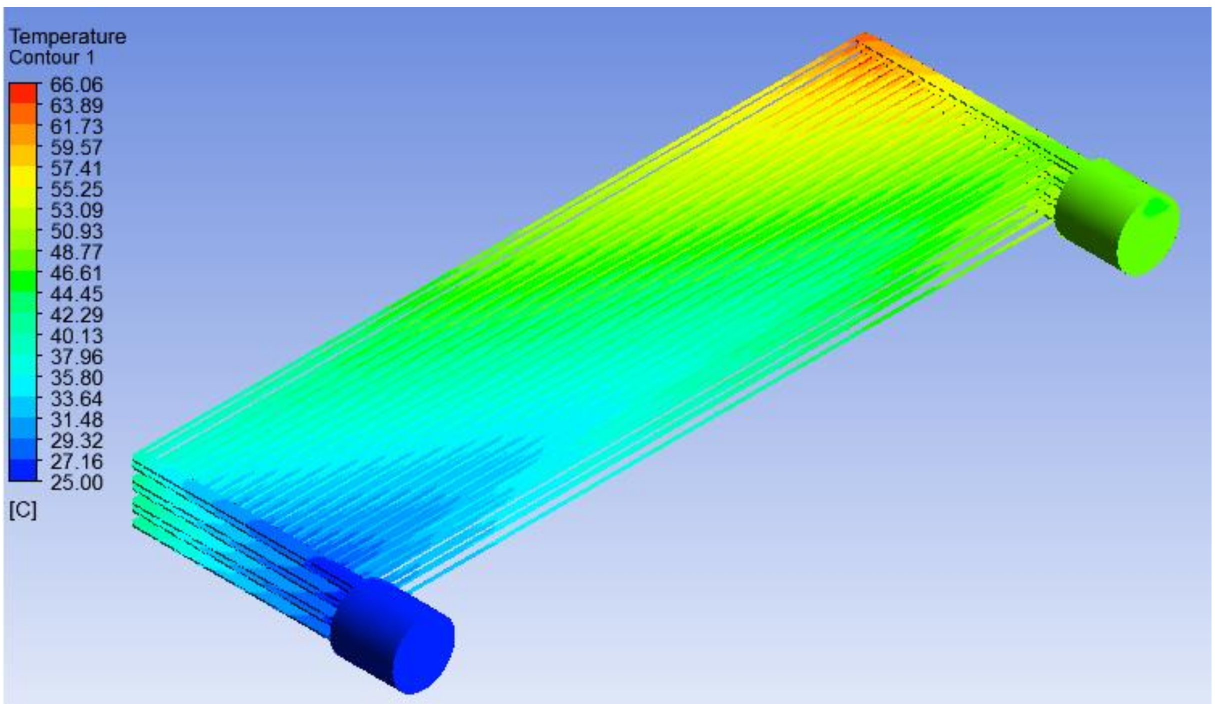
CASE IV



As shown in figure cover plate with inlet temperature 25.38 c and outlet temperature 68.88 c and hot fluid mass flow rate 0.2 kg/s and cold fluid mass flow rate is 0.25 kg/s.

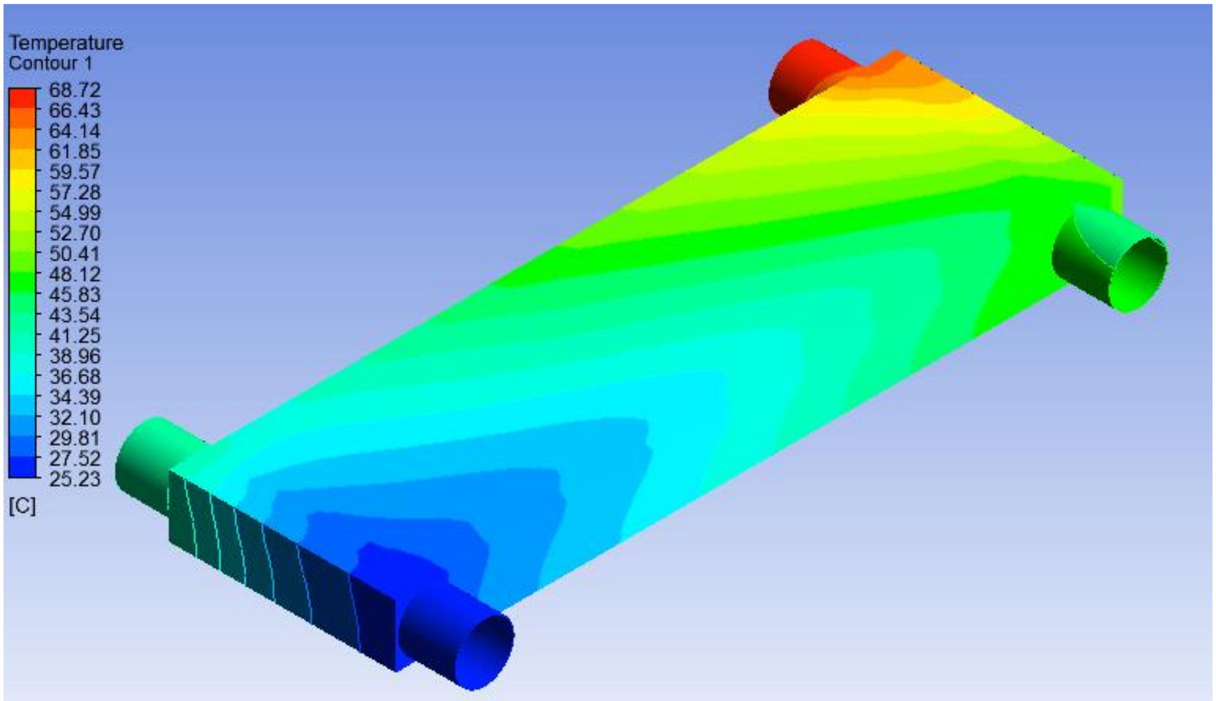


In above figure hot plate with inlet temperature is 70.00 c and its cold to 27.54 c

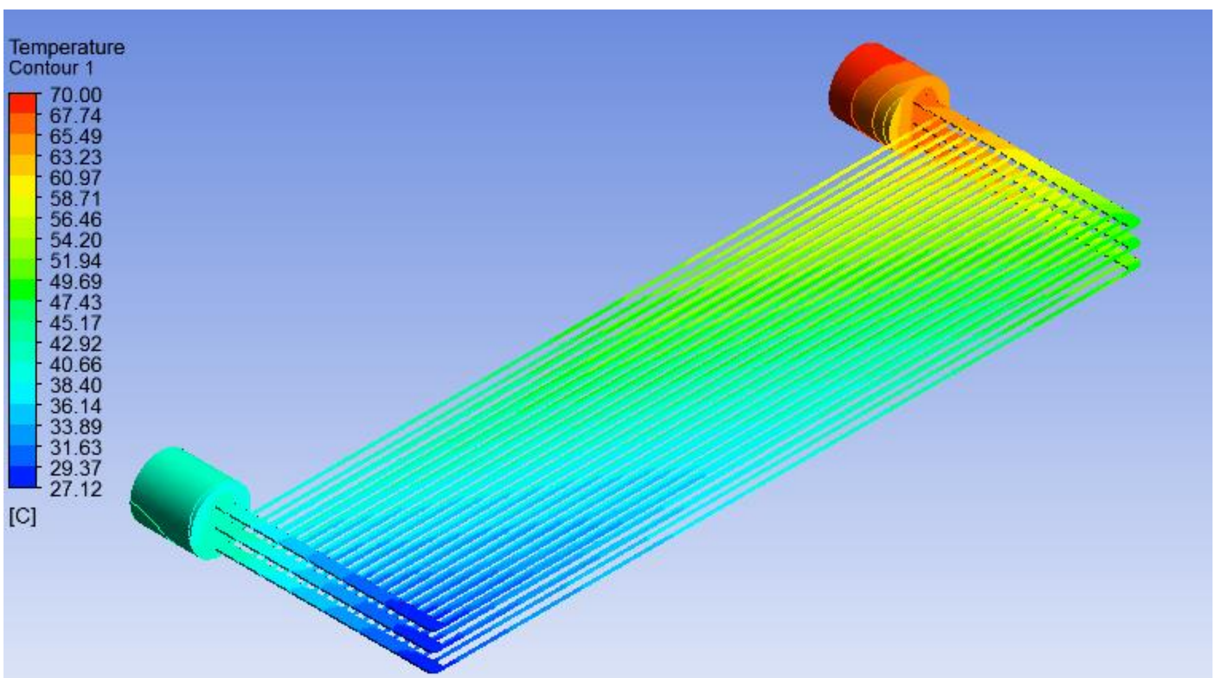


In above figure cold fluid inlet temperature is 25.00 and it hot to 66.06 c .

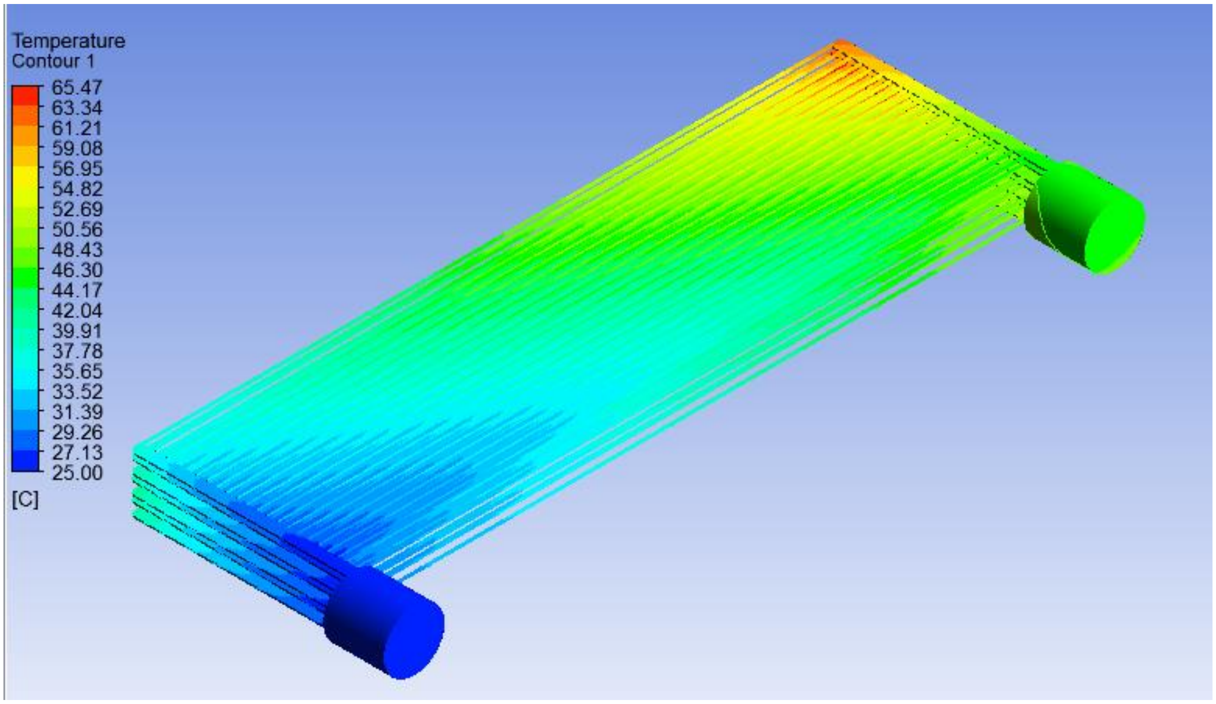
CASE V



As shown in figure cover plate with inlet temperature 25.23 c and outlet temperature 68.72 c and hot fluid mass flow rate 0.2 kg/s and cold fluid mass flow rate is 0.30 kg/s.



In above figure hot plate with inlet temperature is 70.00 c and its cold to 27.12 c



In above figure cold fluid inlet temperature is 25.00 and it hot to 65.47 c .

RESULT TABLE:

SR NO	Hot fluid mass flow rate (kg/s)	Cold fluid mass flow rate (kg/s)	Cold water inlet temperature (°C)	Cold water outlet temperature (°C)	Hot water inlet temperature (°C)	Hot water outlet temperature (°C)
1	0.2	0.10	25.00	52.31	70.00	50.98
2	0.2	0.15	25.00	50.59	70.00	48.43
3	0.2	0.20	25.00	49.07	70.00	46.48
4	0.2	0.25	25.00	47.19	70.00	44.62
5	0.2	0.30	25.00	45.73	70.00	43.21

V. "CONCLUSIONS"

- Heat exchanger was design considering one dimensional heat transfer through channels. From the result of performance test, it is clear that assumption of one dimensional flow is valid.
- We achieve hot water temperature drop is 23.15 ^{o}C and cold water temperature rise is 22.85 ^{o}C for parallel for heat exchanger.
- From the result table hot fluid mass flow rate remain constant and cold fluid mass flow rate increase we get outlet temperature decrease in hot fluid and outlet temperature rise in cold fluid in counter flow mini channel heat exchanger.
- Heat balance form the CFD analysis for hot water to cold water is achieve in the range of 1.2 % error of total heat of hot fluid i.e. heat is extracted by the cold fluid is 1.2 % less and this is due convection is take place from the outer surface of the aluminum plate.

REFERENCES

1) Kandlikar,S.G, " Micro channels and Minichannels- History,terminology,classification and current research needs", first international conference on micro channels and minichannels ,april 24-25,2003,Rochester ,new york,usa,page no:1 -3.

2) Kandlikar,S.G, and mark,E.Steinke, "Examples of micro channels mass transfer processes in biological systems", ", first international conference on micro channels and minichannels ,April 24-25,2003,Rochester ,new york,usa,page no:1-2

3) Paisarn Naphon and Osod Khonseur , "Study on the convective heat transfer and pressure drop in the micro-channel heat sink", international communication in heat and mass transfer,page 39-43.

4) Thanhtrunng Dang and Jyh-Tong Teng, "Comparisons of the heat transfer and pressure drop of the micro channel and minichannel heat exchangers", Heat and mass transfer (2011) 47:1311-1322.

5) R. Walchli, T. Brunschwiler, B. Michel, D. Poulikakos, "Combined local micro channel-scale CFD modelling and global chip scale network modelling for electronics cooling design" International Journal of Heat and Mass Transfer (2009).

6) S.M.Gaikwad, J.D.Patil, R.B.Gurav, Pritee Purohit, A.A.Ramgude, "CFD Simulation on Fluid flow in Micro channel Heat Exchanger"

7) Reiyu Chein , Janghwa Chen, "Numerical study of the inlet/outlet arrangement effect on microchannel heat sink performance", International Journal of Thermal Sciences(2009).

8) Heat transfer and fluid flow in minichannel and microchannel by Satish.G.Kandlikar, Srinivas Garimella,Dongqing le,Stephin Colin,Michel R King

9) Levi A Campbell and Satish Kandlikar , "Effect of entrance condition on frictional losses and transition to turbulence in minichannel flow", microchannel and minichannels -2004, June 17-19,2004 Rochester, new york,usa

10) Compact Heat Exchanger -selection design and operation by J.E. Hesselgreaves

11) Fundamental of heat exchanger design by Ramesh k shah & Dusan P Sekulic