International Engineering Research Journal

Numerical Analysis of Heat Exchanger Tubes

Anita D. Patil[1], Dr. Rajendra K. Patil[2]

[1]Department of Mechanical Engineering,TSSM's Padmabhooshan Vasantdada Patil Institute of Technology, Bavdhan, Pune, India
[2] Department of Mechanical Engineering, TSSM's Padmabhooshan Vasantdada Patil Institute of Technology, Bavdhan, Pune, India

Abstract

Heat exchangers play important role in automotive, thermal management system and refrigeration and air conditioning applications. In such applications, heat exchangers tubes are used on water or coolant side. Energy saving, cost reduction and heat exchanger size reduction are need of the day.Conventionally, heat exchangers are made up of tubes and corrugated fins. The liquid flows through tube and gas or air flows over fins. To enhance the water side thermal performance, many designs of tubes such as dimple tube and microchannel tube are used now a days in heat exchangers instead of straight tubes.The present work focuses on thermal performance of straight tube, microchannel tube and dimple tube. Numerical analysis of heat exchanger tubes has been carried out using commercial CFD tool ANSYS FLUENT 17.0.Microchannel tubes and dimple tubes are compared with the straight tube at varying parameter like flow rate of coldwater. It has been found that dimpled tubes and microchannel tubesprovide better performance in terms of heat transfer rate, as compared to straight tubes.

Keywords:*Heat Exchanger, Thermal Performance, Tubes, CFD.*

1. Introduction

Automobile such as trucks, buses and cars use compact heat exchanger for their thermal management, heating and cooling systems. Tubes are widely used in heat exchanger for automotive applications such as radiator, intercooler, condenser and heater core. In automobile industry main challenge is to give high heat transfer rate at water side with minimum size and cost of heat exchanger. Heat transfer rate at water side can be varied by varying tube design. The present study reports the numerical analysis of thermal performance of straight tube, dimple tube and microchannel tube. Natural convection is a process of heat transfer, in which the flow of fluid is caused by density differences in the fluid occurring due to difference in temperature.
Here, the fluid which surrounds a heat source receives heat, becomes less dense and rises. The working fluid that is surrounding the high temperature fluid is cooler and then moves in to replace it. After that cooler fluid gets heated and the process continues, resulting convection current. In forced convection, the low temperature fluid removes heat from the comparatively high temperature fluid, as it flows along or across it. If it moves along the hot stream then it's called parallel flow and if they are across then its counter flow. In internal combustion engine in which anengine coolant flows through radiator tubes and air past the tubes through fins, which cools the coolant and heats the incoming air.
In a heat exchanger the heat transfer through radiation is negligible and hence can be neglected as in compare to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows through the surrounding solid wall. The conductive heat transfer is increased by selecting a minimum thickness of wall of a highly conductive material. But convection plays the major role in the performance of a heat exchanger. In Forced convectionthrough the wall of the pipe heat exchanger transfers the heat from one moving stream to another stream. The cooler fluid removes heat from the hotter fluid as it flows along or across it.
The methods used to improve heat transfer enhancement is by geometric modification prompting earlier transition to turbulence, creating vortices that increase mixing or restarting the thermal boundary layer to decrease its thickness. Heat transfer enhancement is required not only to improve the heat transfer, but also to minimize the flow resistance.Among various heat transfer enhancers, a dimpled tube and microchannel tube shows a high heat transfer capacity to other types of heat transfer enhancers that are available.
To enhance the water side thermal performance, many designs of tubes such as dimple tube and microchanneltubes are used now a day in heat exchangers instead of straight tubes. CFD simulation has been a good tool in identifying the possibilities of increasing the heat transfer. In the present study we have investigated different configuration and compared heat transfer rates.

2. Literature Review

Juin Chen et al. [2001] studied heat transfer enhancement in dimpled tubes. Heat transfer

enhancement was investigated in a coaxial-pipe heat exchanger using dimples as the heat transfer modification on the inner tube. Tube-side Reynolds numbers (Re) were in the range of 7.5×10^3–5.2×10^4 for water flow. A constant annular mass flow rate was chosen to obtain the highest possible Reynolds number of 1.1×10^4. Typically, the heating water inlet temperature was 68.1±0.1°C.All six variants with inward-facing, raised dimples on the inner tube increased the values of heat transfer coefficient significantly above those for the smooth tube. Heat transfer enhancement ranged from 25% to 137% at constant and from 15% to 84% at Re constant pumping power. At a constant Rethe relative *J* factor (ratio of heat transfer coefficient to friction factor, relative to smooth tube values), had values from 0.93 to 1.16, with four dimpled tube configurations having values larger than unity. Despite the extremely simple design, this outperforms almost all heat transfer enhancements recommended in the literature. A correlation is evaluated from results. A correlationused for predicting heat transfer coefficients and friction factors for the design of dimpled-tube heat exchangers.

John H. Jacoby et al. [1993] studieddimpled heat transfer surface and method of making same. The heat transfer surface includes at least one plate for transferring heat energy from the hot fluid on one side of the plate to the cool fluid on the other side of the plate, and a plurality of intact spaced depression on one side of the plate thereby creating a plurality of intact projections on the other side of the plate. The depressions and projections are arranged to increase both the heat transferfilm coefficient of the plate and the heat energy being transferred by the plate.The results provide an inexpensive, easy-to-manufacture dimpledheat transfer surface having an increased film coefficient and heat transfer coefficient.

Mr. B. Vijayaragavan et al.[2017] performed numerical analysis and investigated experimentally the performance of double pipe heat exchanger using dimples.Double pipe heat exchangers are widely used in various heat transfer applications starting from oil refineries to automobile radiators because of simplicity in design. The rate of heat transfer in a double pipe heat exchanger can be increased by using various heat transfer augmentation techniques out of which dimples is identified as a passive method with least value of pressure drop in comparison with other techniques. In their work the performance of double pipe heat exchanger with dimples of various shapes and configurations are investigated using the CFD package ANSYSFLUENT 16.0 and the arrangement providing efficient heat transfer was identified through CFD results and experimentally validated along with the plain tube model. The inline arrangement with counter flow was chosen for the study with dimple dimension of depth to diameter ratio 0.26. Out of the various pitches ranging from 300mm to 100mm and dimple shapes considered such as hemispherical, square, triangular and elliptical, the hemispherical dimpled tube with a pitch of 150mm arranged in two rows wasidentified to be the most efficient. Their experimental results were in agreement with the CFD results and the study show that performance of double pipe heat exchangers could be enhanced with the selected dimpled configuration which improves the heat transfer rate by creating turbulence in the flow at a minimum pressure drop.

Dr. Syed Azam Pasha Quadri et al. [2016] investigated numericallyconcentric tube heat exchanger with and without dimples using CFD.In this work, first concentric tube exchanger was designed and then dimple tube concentric heat exchanger was designed in solid works 2016 design software. CFD analysis was carried out in solid works flow simulation by using three different materials such as austenitic stainless steel, hastelloy and titanium. Efficient heat transfer rate for the given materials was studied in both types of heat exchangers. The results indicated that dimple tubes heat exchanger is more efficient than the heat exchanger without dimple tube.

Eugene Duane Daddis et al. [2006] studied foul resistant condenser using microchannel tubing. In their study a condenser coil for a refrigerated beverage and food service merchandiser included a plurality of parallel fins between adjacent tubes. In one embodiment, the tubes comprised microchannel tubes, with no fins there between, and the spacing between the microchannel tubes was maintained in the range of 0.75 inches to optimize the heat transfer performance. Plural rows of microchannel tubes were provided with separate inlet headings and with the rows being staggered in transverse relationship to enhance the heat transfer characteristic while minimizing the fouling.

3. Problem Statement

Heat exchanger play important role in automotive thermal management system and refrigeration and air conditioning applications. In such heat exchanger tubes are used on water or coolant and coolant side heat transfer coefficient is low. This paper focuses on thermal performance of straight tube, microchannel tube and dimple tube.3D models are created in CREO 3.0 for three types tubes.Numerical analysis of these tubes using ANSYS FLUENT 17.0 Solver.

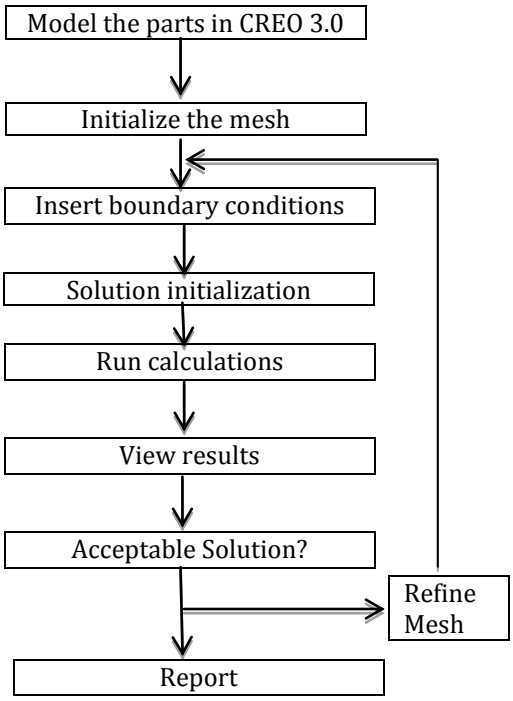
4. Objective

This study focuses on numericalinvestigation of heat transfer through straight tube, microchannel and dimple tube heat exchanger by varying parameters like flow rate of cold water. Performance of heat exchanger is compared for different configurations.

5. Methodology

The analysis of straight tube, microchannel and dimple tubevarying parameter like flow rate of cold water is done numerically. In numerical method, CFD analysis is done using ANSYS FLUENT 17.0 software. Three dimensional models of tubes are first prepared using CREO 3.0 software. The appropriate meshing is done

on these tube models using ANSYS software. Then, solver FLUENT is used to set boundary condition .The analysis is done with coupled algorithm and the *k-*εturbulent model. The temperature, velocity and pressure contours are obtained from the CFD results. The heat transfer rate is calculated using CFD output. Numerical analysis involves a number of basic steps that are shown in the following flowchart.



6. Simulation set up and data input

The governing equations are discretized by using the finite Volume method. The pressure-velocity coupling is achieved through the coupled algorithm. The grid-independent study is done for all cases. All simulations are run in FLUENTk-ε reliable model. Boundary conditions used are mass flow rate and temperature. The geometry of tubes, its internal, created in CREO 3.0. Drawings of tubes are used for geometry creation. Meshing is done in ANSYS. Inlet surface meshed and volume meshing done with hexahedral. Numbers of cells are varying from 7 to 10 million for the geometry dimensions and requirement for grid independent solution. Simulation is done on various configurations of tubes.

The following configurations are considered:

1. Straight tube
2. Dimple tube
3. Microchannel tube

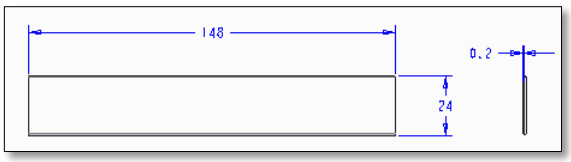


Fig.1 2D of Straight Tube

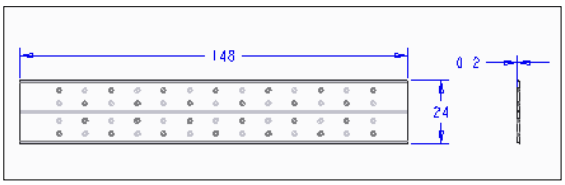


Fig.22D of Dimple Tube

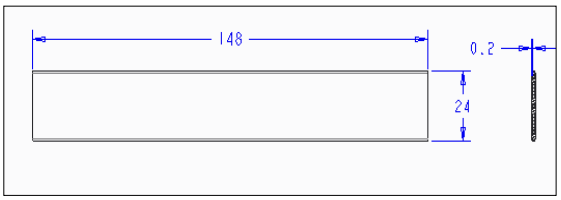


Fig.32D of Microchannel Tube

Design Data used:

Table 1 Water Parameter

Mass Flow Rate	4/6/10/15 LPM
Specific heat	4200 J/kg.K
Thermal Conductivity	0.672845W/m.K
Density	968.6223kg/m3
Inlet Temperature	85°C
Viscosity	0.000333Pa.s

Table 2 Tube Material Property

Conductivity	202.4W/m-K
Density	2719kg/m3

Table 3 Geometryof Tubes

Length	148 mm
Width	24 mm
Thickness	0.2 mm

Fluent set-up

Table 4 Boundary Conditions

Pressure Outlet	0 Gauge
Wall	No slip
Default interior	Fluid (Water)

Table 5 Solver Setting

Solver	Coupled
Formulation	Implicit
Time	Steady
Velocity formulation	Absolute
Gradient option	Cell based

Table 6 Solver control

Equations	Flow
Pressure	0.5
Density	1
Body force	0.8
Momentum	0.5

Table 7 Solver control

Pressure	Standard
Momentum	Second order
Turbulent kinetic energy	Second order
Turbulent dissipation rate	Second order

7. Numerical Simulation

Fluid carrier is phenomenon of great interest, since it is frequently found in many scientific fields and industrial processes. The numerical simulation of fluid transport used in FLUENT by fluid carrier requires the modeling of the continuous phase (fluid), the discrete phase and the interaction between them. The continuous phase-whether Liquid or gas has been modeled using an Eulerian formulation. The discrete phase may be approached as an Eulerian or form a Lagrangian point of view. This has given place to two distinctive strategies, the so called as Eulerian-Eulerian and the Eulerian-Lagrangian methods. In the Eulerian-Eulerian approach, fluid velocity and concentration fields are calculated for each point of the numerical domain. The Eulerian-Eulerian method can be employed by using a one fluid formulation and two fluid formulations.

7.1. k-εTurbulence model

One of the most prominent turbulence models, the *k*-ε (*k*-epsilon) model, has been implemented in most general-purpose CFD codes. It has proven to be stable and numerically robust and has a well-established regime of predictive capability. For general-purpose simulations, the *k*-ε model offers a good compromise in terms of accuracy and robustness.Turbulent kinetic energy and turbulent dissipation rate values are 1 m^2/s^2, $1m^2/s^3$ respectively.

7.2. Geometry Modeling

Geometry of the tubes of heat exchanger is modeled in CREO 3.0. This software gives an advantage of parametric modeling. Shading View of the tubes are shown in figure 4,5 and 6.Robustness and accuracy when the near wall Functions allow solution on arbitrarily fine near wall grids, which is significant improvement over standard wall functions.

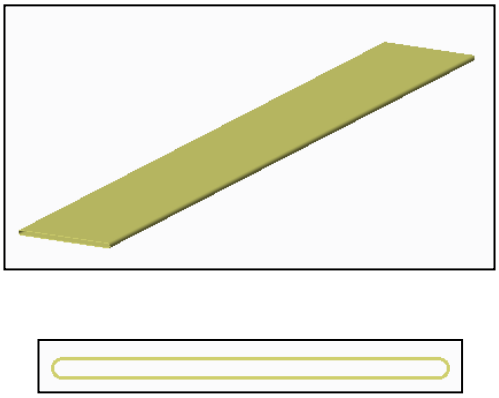


Fig.4 Schematic diagram of Straight Tube

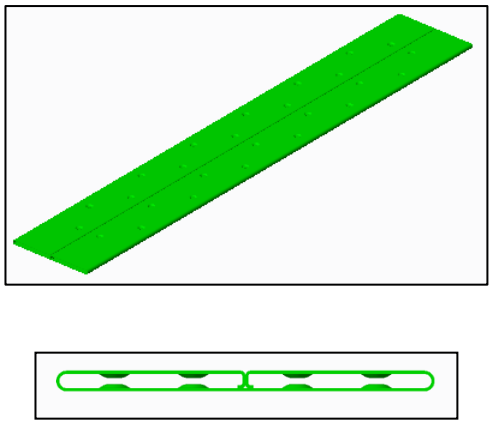


Fig.5 Schematic diagram of Dimple Tube

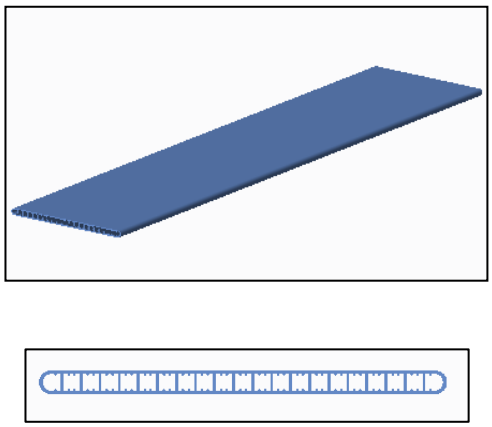


Fig.6 Schematic diagram ofMicrochannel Tube

3D models of straight tube,Dimple tube and microchannel tube as shown in figure 4,5 and6.The tube length, tube width and tube thickness are same for straight tube ,dimple tube and microchannel tube and these values are 148mm,24mm ,0.2mm respectively

7.3. Assumptions

Fluid flow and heat transfer are in steady state and three dimensional.

7.4. Meshing

The most important part in CFD simulation is discretization of geometry. Generally hexahedral and tetrahedral meshes are used for CFD codes. Hexahedral mesh gives better results, but meshing is very difficult. Hexahedral mesh is generated in CFD FLUENT. The interior water space is subdivided into a computational mesh consisting of rectangular elements of sufficient enough to capture significant gradient in velocity. Considering the symmetry of the planes, only representative part simplifies geometry.Element size range from 7 to 10 million for the geometry dimensions. Geometry dimensions are 148mm length, 24mmwidth, 0.2mm thickness respectively.

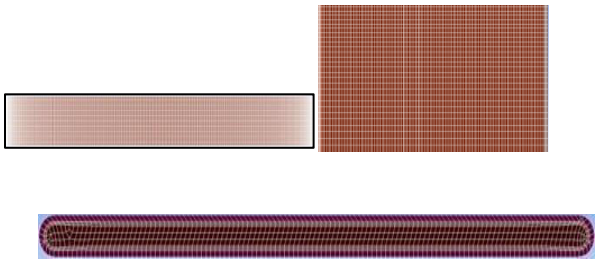


Fig 7 Meshing of Straight Tube

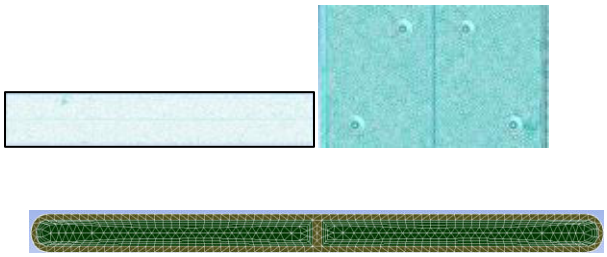


Fig 8 Meshing of Dimple Tube

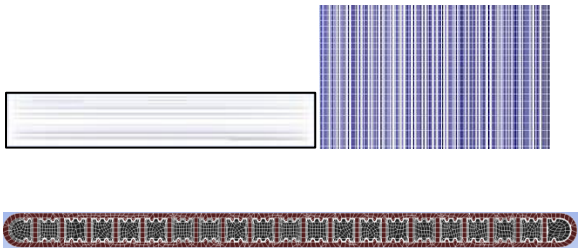


Fig 9 Meshing of Microchannel Tube

7.5. Results and Discussion

CFD Analysis was done in ANSYS FLUENT 17.0 solver using k-ε Realizable model.
After modeling, meshing and analysis using solver FLUENT we came up with the output readings. Some of the contours of pressure, velocity and temperature with flow rate, i.e. 4/6/10/15LPM is as shown as follows.

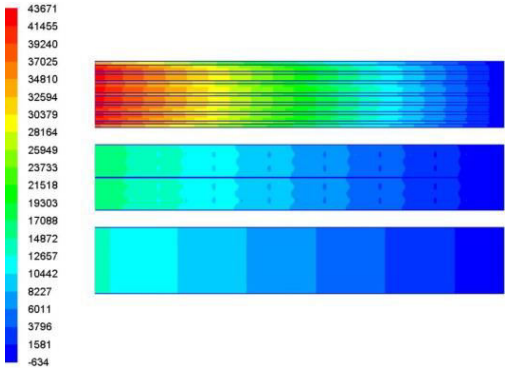


Fig 10 Pressure Contour at 4 LPM

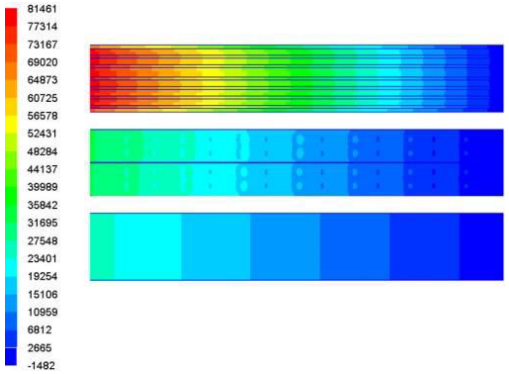


Fig 11 Pressure Contour at 6 LPM

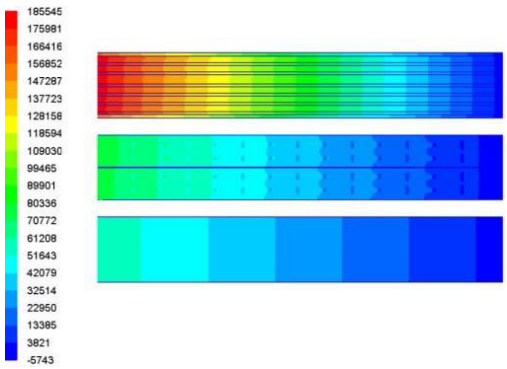


Fig 12 Pressure Contour at 10LPM

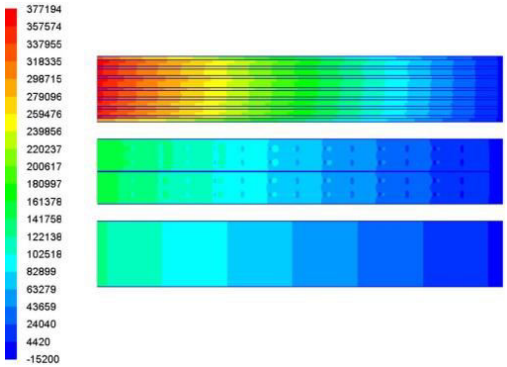


Fig 13 Pressure Contour at 15LPM

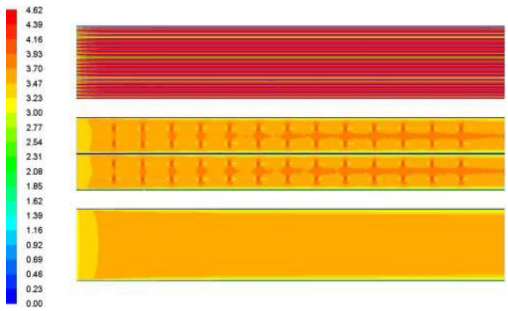


Fig 14 Velocity Contour at 4 LPM

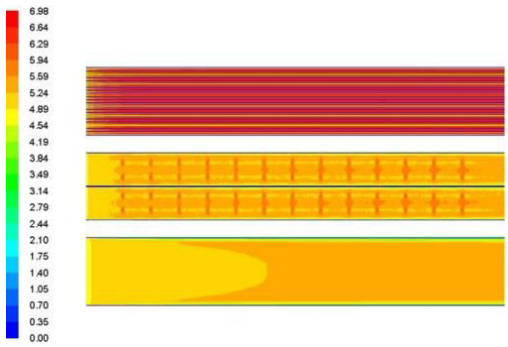


Fig 15 Velocity Contour at 6 LPM

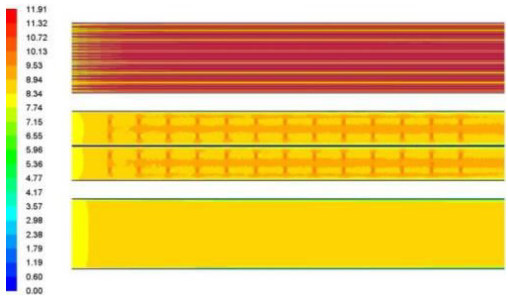


Fig 16 Velocity Contour at 10 LPM

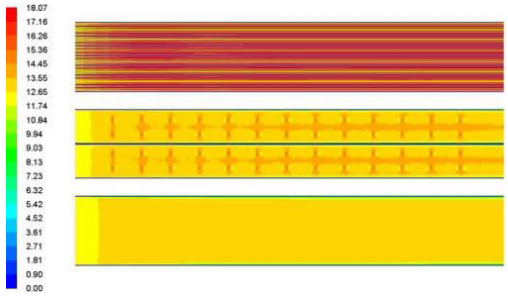


Fig 17 Velocity Contour at 15 LPM

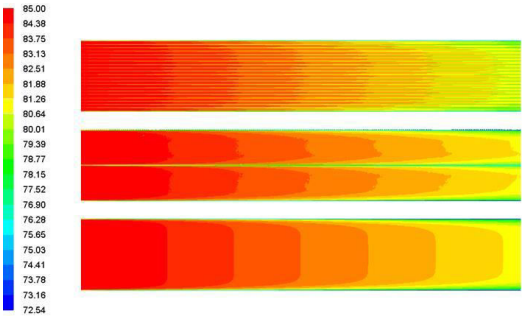


Fig 18 Temperature Contour at 4 LPM

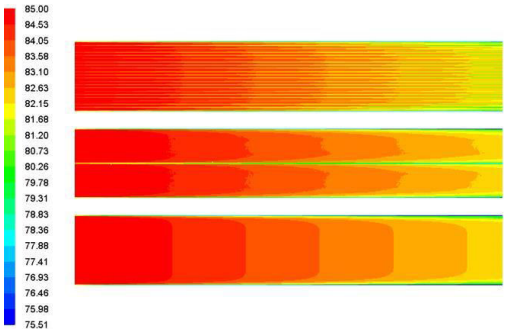


Fig 19 Temperature Contour at 6 LPM

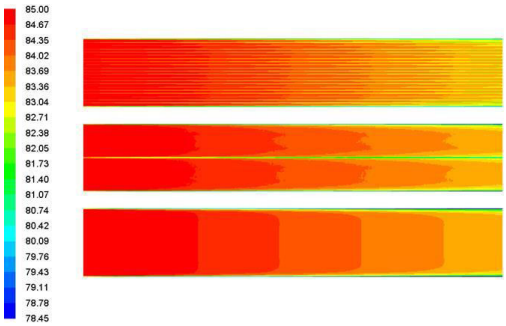


Fig 20 Temperature Contour at 10 LPM

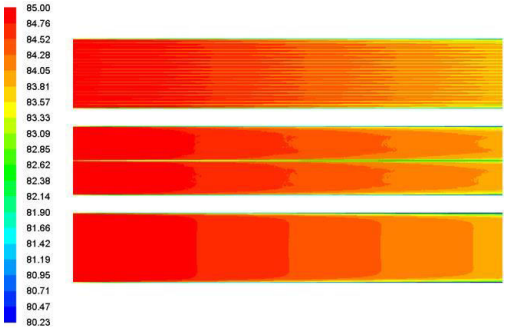


Fig 21 Temperature Contour at 15 LPM

Numericalanalysis has been carried out for straight tube, dimple tube and microchannel tube and results of pressure contour, velocity contour and temperature contour are presented in figure 10 to 21.It is very clear

that pressure and velocity is increased with mass flow rate .Pressure and velocity is highest for microchannel tube and dimple tube than straight tube. Heat transfer enhancement in dimple and microchannel tube due to interruption of the development of the boundary layer, increase of the degree of turbulence, increase of the heat transfer surface area and generation of the secondary flow.

Table 8 Mass Flow Rate vs. Pressure Drop

Pressure Drop (Pa)	Mass Flow Rate (LPM)			
	4	6	10	15
Straight Tube	13410	25493	59832	125810
Dimple Tube	16250	31882	74665	151700
Microchannel Tube	41880	78644	179751	358911

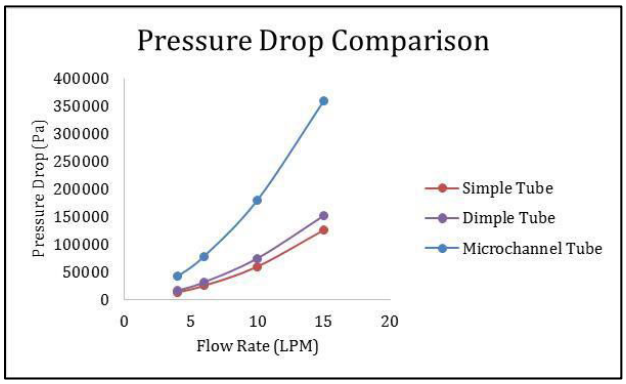


Fig 22 Flow Rate vs. Pressure Drop for various configurations

The Pressure drop across straight tube, Dimple tube and Microchannel tube is given in table no 8. Fig 22shows that pressure drop increases as flow rate increases.

Table 9 Mass Flow Rate vs. Pressure Drop

Heat Rejection (kW)	Mass Flow Rate (LPM)			
	4	6	10	15
Straight Tube	1192	1199	1206	1210
Dimple Tube	1195	1202	1208	1212
Microchannel Tube	1203	1208	1212	1215

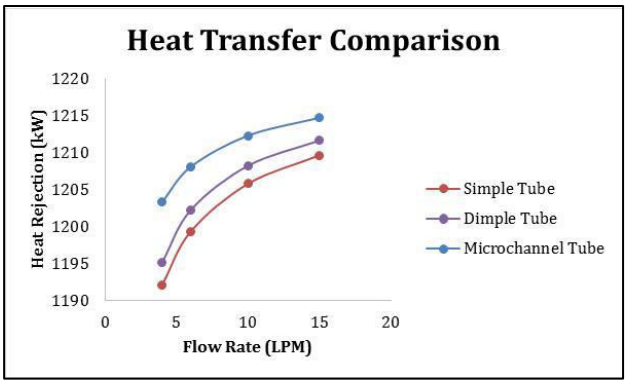


Fig 23 Flow Rate vs. Heat Rejection for various configurations

The Heat rejection across straight tube, Dimple tube and Microchannel tube is given in table no 9.Fig 23 shows that heat rejection increases as flow rate increases. Fig 23 depicts the variation of heat rejection for straight tube,dimple tube and microchannel tube for various mass flow rates which shows a remarkable increase of heat rejection for the dimple tube and microchannel tube when compared to the straight tube ,indicating that a dimple tube and microchannel tube is efficient than a straight .

8. Conclusion

The results show that when flow rate is increased in straight tube, dimple tube and microchannel tube there is increase in heat rejection. The microchannel tube and dimple tube shows heat transfer enhancement than simple tube. But at the same time pressure drop increases for microchannel tube and dimple tube than straight tube. Temperature distribution along tubesfor various configurations is investigated. Resultsare presented in graphical and tabular form.
Tolerating small difference, CFD results are close to the experimental results.

Acknowledgement

Sincere thanks to Prof. Dr. R. K. Patil and PVPIT Bavdhan for their tremendous efforts and guidance for completion of research work.

References

[1].Juin Chen, Hans Muller Steinhagen, Geoffrey G. Duffy (2001) "Heat transfer enhancement in dimpled tubes" *Applied Thermal Engineering*, Vol 21, Issue5, PP 535-547.

[2].John H.Jacoby, Jackson Pond (1993) "Dimpled heat transfer surface and method of making same" United States, No 5224538.

[3].Mr. B. Vijayaragavan., Mr. S. Rajasundar, Mr. C. Raju(2017)"CFD Analysis and Experimental Investigation on the Performance of Double Pipe Heat Exchanger using Dimples"*International Journal of*

Advanced Research Methodology in Engineering and Technology, Vol 1, Issue 3.

[4].Dr. Syed Azam Pasha Quadri, Shakib Javed Shakil Sheikh (2016)"Evaluating the Performance of Concentric Tube Heat Exchanger With And Without Dimples By Using Cfd Analysis"*IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*Vol 13, Issue 5 , PP 46-52.

[5].Eugene Duane Daddis JR. Manlius, NY (US); Robert H.L.Chang, Shanghai (CN)(2006)"Foul-Resistant condenser using microchanneltubing" United States,No 7000415B2,