Analysis of heat transfer and fluid flow characteristics of concentric tube heat exchanger with dimpled tube

J.Sathees Babu
Associate Professor, Department of Mechanical Engineering
Vinayaka Mission's Kirupananda Variyar Engineering college, Vinayaka Mission's Research Foundation
(Deemed to be University) Salem - 636308, Tamil Nadu, India

P. Sellamuthu
Associate Professor, Department of Mechanical Engineering
Vinayaka Mission's Kirupananda Variyar Engineering college, Vinayaka Mission's Research Foundation
(Deemed to be University) Salem - 636308, Tamil Nadu, India

Abstract- Heat exchanger is a equipment which transfers the heat from a hot fluid to a cold fluid. Waste heat is heat, which is generated in a process by way of fuel combustion or chemical reaction, and it could still be reused for some useful and economic purpose. The main objective of the research is to study the heat transfer and fluid flow characteristics of concentric tube heat exchanger with dimpled tube. The Studies on heat transfer, friction factor and thermal performance of concentric tube heat exchanger using dimple tube. A CFD package (ANSYS FLUENT 17.2) was used for the numerical study of heat transfer characteristics of a double pipe heat exchanger for parallel flow and the results were then compared with that of the with and without fins. The with fin double pipe heat exchanger gives the less temperature compare to without fin double pipe heat exchanger.

Keywords – Dimpled tube, Double pipe , Heat Exchanger, Heat transfer, Friction factor

I. INTRODUCTION

The study of improved heat transfer performance is termed as heat transfer augmentation, enhancement, or intensification. Lot of research has focused on various augmentation techniques on rough surfaces, transverse or spiral ribs, transverse grooves, knurling, corrugated and spirally corrugated tubes, straight and spiral and annular fins. In this research , augmented surface was achieved with dimples located in a structured way along the tube of a double-pipe heat exchanger with the increased area on the tube side. Improved heat transfer surfaces are often utilized in advanced designs since they can increase heat transfer through a combination of: increased fluid turbulence; secondary flow generation; boundary layer disruption; and additional heat transfer surface area. These factors lead to an increase in performance and product life. Energy recovery from many heat exchangers can be improved since they were initially developed to use plain (smooth) heat transfer surfaces. Heat exchangers using enhanced heat transfer surface scan improve efficiency since enhanced surfaces provide an augmented geometry that produces a combination of a larger overall heat transfer coefficient and additional area for heat transfer
Augmented surfaces can create one or more combinations of the following conditions that are favorable for increasing the heat transfer coefficient with a consequent increase in the friction factor. Heat exchangers are used in variety of applications. Performance increase in heat exchanger leads to more economical design of heat exchanger to make energy, material & cost savings related to a heat exchange process. Thermal performance of heat exchangers are increased by heat transfer augmentation techniques. Classification of these techniques are passive and active. Active techniques involves external power input for the enhancement of heat transfer. Passive techniques generally used in surface to the flow channel by incorporating inserts or additional devices. One of the fluid flows through the tube and the other through the annulus. many methods are applied to increase thermal performance of heat transfer devices such as treated surfaces, rough surfaces, swirling flow devices, coiled tubes, and surface tension devices. Out of these dimple tube method is used to increase the thermal performance.

II. LITERATURE REVIEW

[1] J.E.Kim.et.al, (2012) This paper reveals that, Four different protrusion heights were considered and protrusion height to channel height (h/H) of 0.05, 0.10, 0.15, and 0.20. This experiment under performed by turbulent flow. Water as test fluid. This experiment results 40% negligible pressure drop, 24% increase heat transfer, increase friction factor up to 5–6% and volume goodness factor slightly increases by 4%.

[2] J.Kukulka.et.al, (2011) This paper reveals that, Enhancement tube and smooth tubes are compared and material is enhanced 304 L stainless steel tube and steel. This experiment under performed by Turbulent flow in the range of Reynolds Numbers near 2900, Water as working fluid increases in heat transfer for Enhancement tubes are in excess of 120% over smooth tubes.. Fouling rate for the smooth stainless steel tubes were compared to the average values of the four dimpled tubes. Dimpled tubes minimizes the fouling rate and also provides heat transfer performance in excess of 100%.

[3] A. García.et.al, (2011) This paper reveals that, Corrugated tubes, dimpled tubes and wire coils are compared. Water as working fluid and the range of Reynolds numbers between 200 and 2000 in Laminar, transition and turbulent flows are used. The heat transfer co-efficient on maximum Nusselt number augmentations of 250% can be expected at low Prandtl numbers. Incase of Reynolds numbers higher than 2000, the use of corrugated and dimpled tubes is favored.

[4] S.Suresh.et.al, (2010) This paper reveals that, experimental studies on heat transfer and friction factor characteristics of CuO/water nano fluid under turbulent flow in a helically dimpled tube and plain tubes are compared. This experiment was performed in Turbulent flow with Reynolds number range between 2500 and 6000. Nano fluids are prepared by sol-gel method. Observed that with 0.3% volume concentration of copper nanoparticles dispersed in ethylene glycol, its thermal conductivity increased by 40%. The heat transfer results showed that Nusseltnumber with dimpled tube and nano fluids under turbulent flow is 19%, 27% and 39% (0.1%, 0.2% and 0.3% volume concentrations of nanoparticles in a fluid).

[5] Wang.et.al,(2010) This paper reveals that, Heat transfer and hydrodynamics analysis of a novel dimpled tube, Ellipsoidal shape dimpled tube and spherical shape dimpled tubes are compare than smooth tube. Laminar and transition flow are used. Working fluid as water and range of Reynolds number e is less than 1000. Friction factor of dimpled tube increased by 26.9–75%. Friction factor of the ellipsoidal and spherical dimpled tubes increased by 32.9–92%. Nusselt number increased by 38.6–175.1% for the ellipsoidal dimpled tube and 34.1–158% for the spherical dimpled tube.

[6] Chinaruk Thianpong.et.al,(2009) This paper reveals that, Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator, Dimpled tube with twisted tape and plain tubes are compared. The range of Reynolds number are from 12,000 to 44,000 with hot and coldwater as working fluid. Aluminum made twisted tapes with thickness (δ) of 0.5 mm, width (w) of 22 mm, and twist ratios y/w=3, 5 and 7 are used in this experiment. The experimental results shows that both heat transfer coefficient and friction factor in the dimpled tube fitted with the twisted tape, are higher than those in the dimple tube acting alone as plain tube. Also the heat transfer coefficient and friction factor in the combined devices increase as the pitch Heat transfer rate and friction factor dimpled tube with twisted tape, are respectively 1.66 to 3.03 and 5 to 6.31 times in the plain tube.

[7] Chang.et.al, (2008) This paper reveals that, Heat transfer and pressure drop in dimpled fin channels, three types of dimpled fin channels follow the order of (a) convex -concave (b) convex–convex (c) concave–concave at each tested. Reynolds number (Re) ranging from 1500 to 11,000 and Water as working fluid are used as parameters. Results of above three types of dimpled fin channels follow the order of (c) convex–convex > (b) concave–concave > (a) concave– convex shapes increased pressure drops and decreasing L/d from 6.2 to 3.5 and increases f factors at Y/L = 1. Variations of Nusselt number against Reynolds number for each type of dimpled fin channel with L/d = (a) 8.9, (b) 6.2 and (c) 3.5 are compared.

[8] Pedro G.Vicente.et.al, (2002) This paper reveals that Experimental study of mixed convection and pressure drop in helically dimpled tubes for laminar and transition flow are compared. Water and ethylene glycol as test fluids, this experiment was performed in laminar and transition flows. Heat transfer increases and is up to 5 times higher than the smooth tube value corresponding to laminar flow. The results of isothermal pressure drop for laminar flow showed dimpled tube friction factors between 10% and 30% higher than the smooth tube. Heat transfer of dimpled tubes can increases up to 30%.

III.METHODOLOGY

The step by step procedure of the methodology is listed below,

i. Study the existing
ii. Identify the parameters of the analysis
iii. Mathematical calculation for the benchmarking
iv. 3D Geometry
v. De-Crystallization using pre-processor
vi. Analysis and post processing using ANSYS
vii. Identify the parameters for improving the system.
viii. Revised analysis for new variant
ix. Identify the best variant
x. Validation for benchmarking system

xi. Through physical experimentation

IV. MODELING OF DOUBLE PIPE HEAT EXCHANGER

CAD modeling of the complete Double Pipe Heat Exchanger structure is performed by using Solid works 2018 software. This software is having special tools in generating surface design to construct typical surfaces, which are later converted into solid models.

All part models are then assembled to make a complete structure. The process of assembly is very much common to general process of fabricating structures in real production. The CAD model of single and Double Pipe Heat Exchanger used for FE Analysis during assembly is shown in figure. The assembled CAD model has been prepared from various part modelling drawings. After geometry creation these models were imported in ANSYS for thermal analysis.

Software Tools

1) *Solid Works*
   i. The part modelling environment in which the extrude command is used for the modelling of the Double Pipe Heat Exchanger and the stiffeners are made using the glide command.
   ii. The parameters required for the modelling of Double Pipe Heat Exchanger are contour dimensions height, length, fillet radius, hole diameter.
   iii. Holes were made using surface trim command. The 3D SOLID is prepared by thickness in the third dimension provided after selecting the 2-D shell element.

2) *ANSYS 17*
   iv. ANSYS 17 used to Analysis the Double Pipe Heat Exchanger. In design simulation Quadra paver element is used for meshing of Double Pipe Heat Exchanger.
   v. Equivalence of the nodes is executed for this element. ANSYS has a quadratic displacement behavior and is well suited to modeling irregular meshes
   vi. For the meshing of the Double Pipe Heat Exchanger with and without stiffeners the above described nodal Quadra element is used with Double Pipe Heat Exchanger meshing.
   vii. The material properties like modulus of elasticity and poisons ratio is assigned to the Double Pipe Heat Exchanger in materials list in ANSYS.
   viii. The analysis was carried out for the load calculated. At first the Double Pipe Heat Exchanger is analyzed by considering the load calculated.
   ix. The deflection and maximum and minimum temperature are obtained which will give comparative results for modifications in the Double Pipe Heat Exchanger.

Sl. No.	Parameter		Forms
1	Hot pipe	Inner diameter	16.5mm
		Outer diameter	21.5mm
2	Cold pipe	Inner diameter	42mm
		Outer diameter	48.5mm
3	cold pipe inlet	Inner diameter	11mm
		Outer diameter	12mm
4	Cold pipe outlet	Inner diameter	11mm
		Outer diameter	12mm
5	Hot pipe length		750mm
6	Cold pipe length		450mm

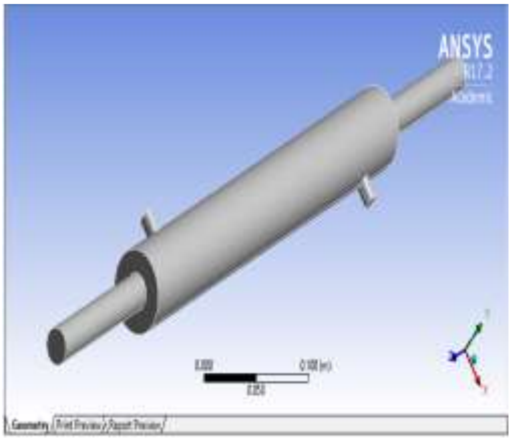


Figure 1. Heat exchanger Solid Model Figure 2. Double Pipe with Fin Model

V. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

Computational fluid dynamics (CFD) analysis includes modeling of the geometry, meshing and analysis of the part geometry against boundary conditions.

In the ANSYS workbench design module parallel -flow heat exchanger heat exchanger is modeled. Initially the fluid flow is selected and geometry is created.

Table - 2. Naming of various parts of the body with state type

Part number	Part Of The Model	State Type
1	Inner Fluid	Fluid
2	Inner_Pipe	Solid
3	Outer_Fluid	Fluid
4	Outer_Pipe	Solid

Meshing of model :

Initially a relatively coarser mesh is generated which contains mixed Tetra and Hexahedral cells having both triangular and quadrilateral faces. Finally a fine mesh is generated with the edges and regions of high temperature and pressure gradients are meshed.

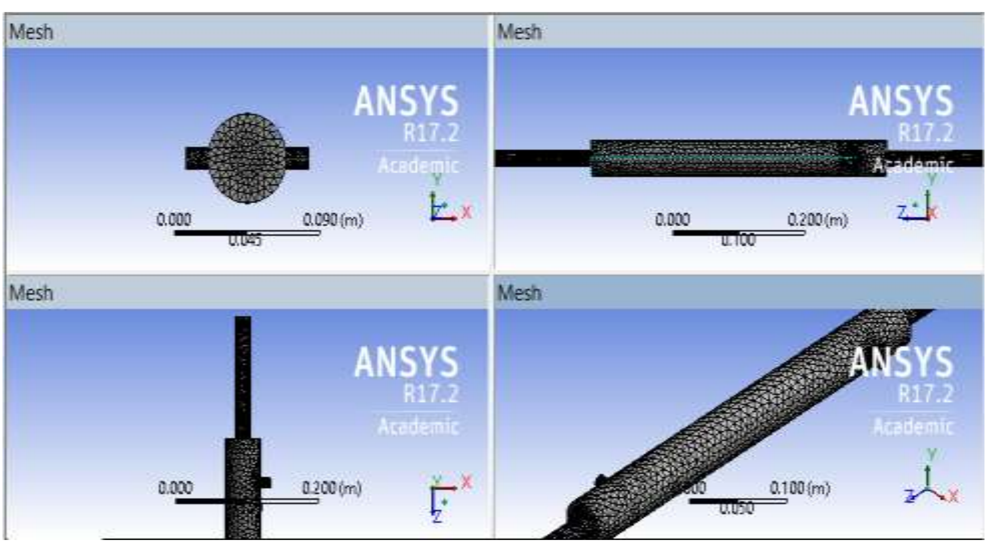


Figure 3. Various view face of the Mesh module

Named Selection

The surfaces of the solid are named as inlets and outlets for inner and outer fluids. The outer wall is named as insulation surface.

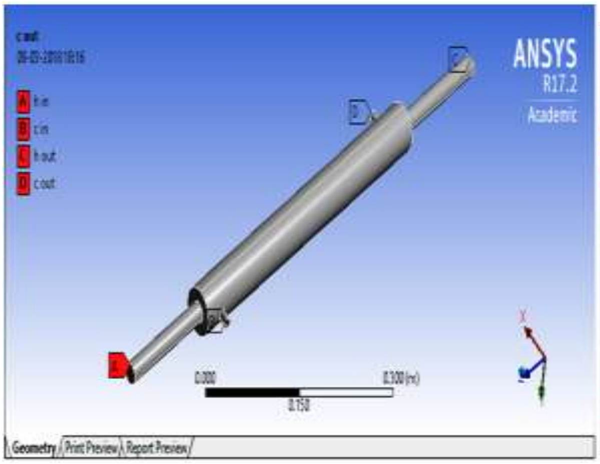


Figure 4. Named Sections of heat exchanger 1

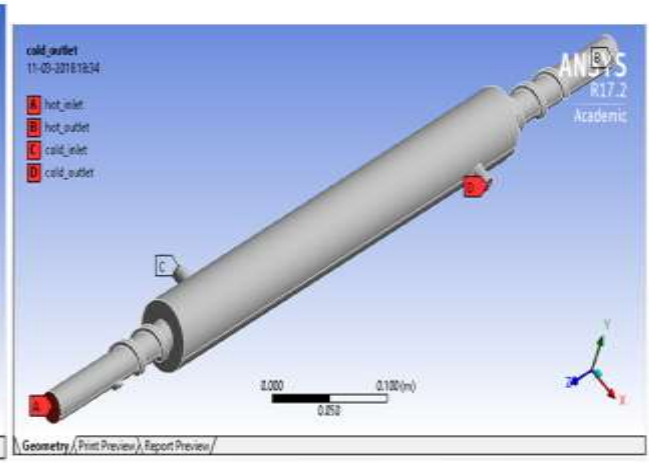


Figure 5. Named Selections heat exchanger 2

Save project and close the window. Refresh and update project on the workbench

Solution:

1) *Problem Setup*

The mesh is checked and quality is obtained. Change the analysis type to Pressure Based type. Change the velocity formulation to absolute and time to steady state. Gravity is defined as y = -9.81 m/s^2

2) *Models*

Energy is set to ON position. k-ε model (2 equations) is selected as Viscous moddel. Radiation model is changed to Discrete Ordinates.

3) *Materials*

Add water-liquid and copper, steel to the list of fluid and solid respectively from the fluent database.

4) *Cell zone conditions*

The parts are assigned as water and copper, steel as per fluid/solid parts.

5) *Boundary Conditions*

Boundary conditions are used according to the need of the model. Mass flow rate inlet and pressure outlet are defined as inlet and outlet conditions. As this is parallel-flow with two tubes so there are two inlets and two outlets. The walls are y specified with respective boundary conditions. No slip condition is considered for each wall. Heat flux condition is set to zero for each wall except the tube walls. The details of all boundary conditions are listed in the Table 3 as given below.

Table -3 Boundary Conditions

Quantities	**Boundary condition**
Working fluid	**Water**
Inner pipe (hot fluid)	**Hot inlet Mass flowrate = 0.125 kg/s Temperature = 82.22 c**
Outer pipe (cold fluid)	**Cold inlet Mass flowrate =0.215kg/s Temperature = 32.22 c**

6) Reference Values

i. Area = 1 m^2
ii. Density = 998.2 kg/m^3
iii. Length = 39.37008 inch
iv. Temperature = 348 K
v. Velocity = 0.9942 m/s
vi. Viscosity = 0.001003 kg/m-s
vii. Ratio of specific heats = 1.4

7) Solution Methods

The solution methods are specified as follows:
i. Scheme = Simple
ii. Gradient = Least Square Cell Based
iii. Pressure = Standard
iv. Momentum = Second Order Upwind
v. Turbulent Kinetic Energy = Second Order Upwind
vi. Turbulent Dissipation Rate = Second Order Upwind

8) Solution Control and Initialization

i. Under relaxation factors the parameters are
ii. Pressure = 0.3 Pascal
iii. Density = 1 kg/m3
iv. Body forces = 1 kg/m2s2
v. Momentum = 0.7 kg-m/s
vi. Turbulent kinetic energy = 0.8 m2/s2
vii. Then the solution initialization method is set to Standard Initialization whereas the reference frame is set to Relative cell zone.
viii. Measure of Convergence

To get the nice convergence throughout the simulation residuals are given as per the Table 4 that follows.

Variable	Residual
x-velocity	**10-6**
y-velocity	**10-6**
z-velocity	**10-6**
Continuity	**10-6**
Specific dissipation energy/ dissipation energy	**10-5**
Turbulent kinetic energy	**10-5**
Energy	**10-9**

9) Run Calculation

The number of iteration is set to 1000 and the solution is calculated and various contours, vectors and plots are obtained.

VI. RESULTS

Mass flow rate and total heat trasfer rate.

The mass flow rate and total Temperature are given in the tables below.

Table: Show the heat exchanger result

Double pipe Heat exchanger		Mass Flow Rate Mass Flow Rate (kg/s)	Temperature (ºC)
Without fin	**Hot pipe outlet**	**0.125**	**326.0**
	Cold pipe outlet	**0.251**	**332.9**
With fin	**Hot pipe outlet**	**0.125**	**320.3**
	Cold pipe outlet	**0.251**	**325.4**

Contours

The temperature, pressure and velocity distribution along the heat exchanger can be seen through the contours.

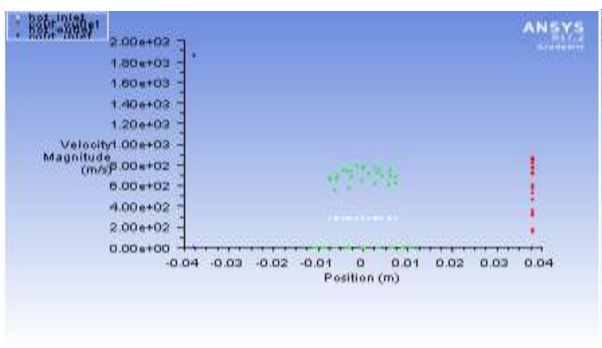


Figure 6. Velocity magnitude in different position

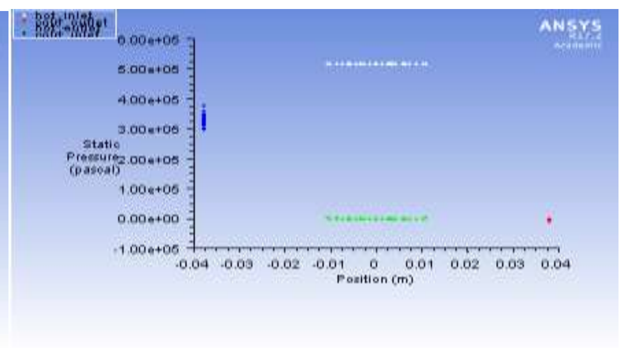


Figure 8. Static pressure magnitude in different position

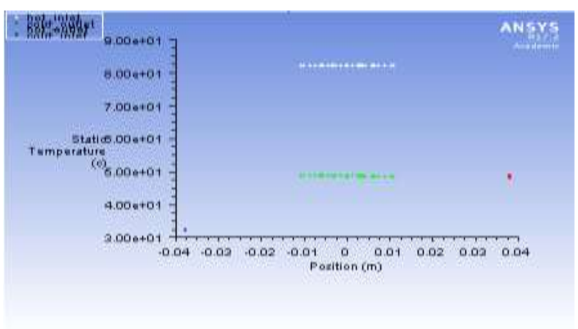


Figure 9. Temperature magnitude in different position.

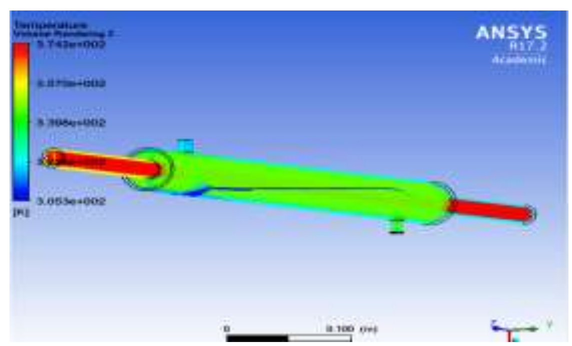


Figure 10. Double pipe heat exchanger Temperature Disribtuion (With fin)

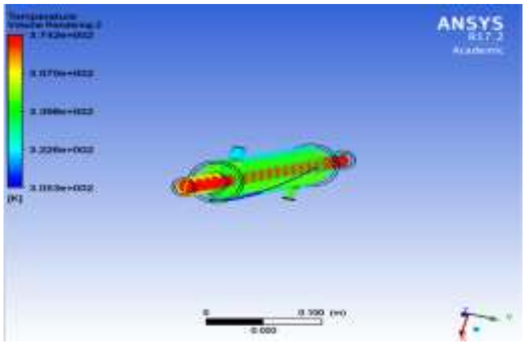


Figure 11. Double pipe heat exchanger Temperature (Without fin)

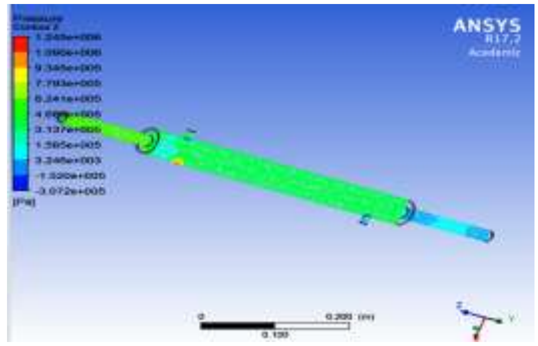


Figure12. Heat exchanger Fluid Pressure Distribution

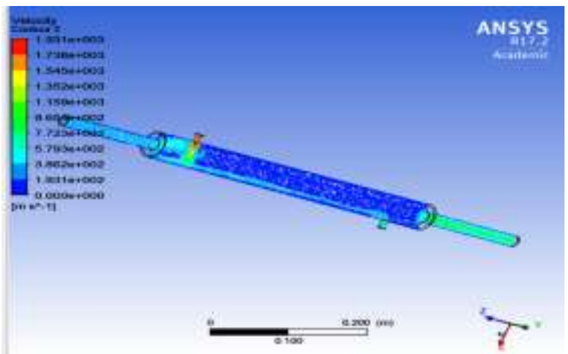


Figure 13. Heat exchanger Fluid Velocity Distribution

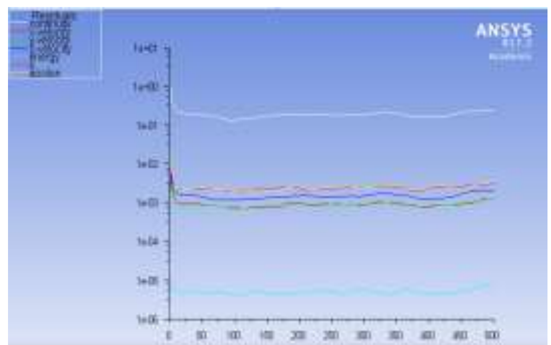
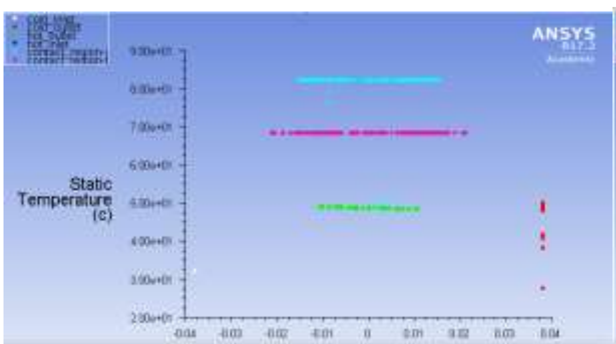


Figure14. : Fluid Velocity magnitude in different position.



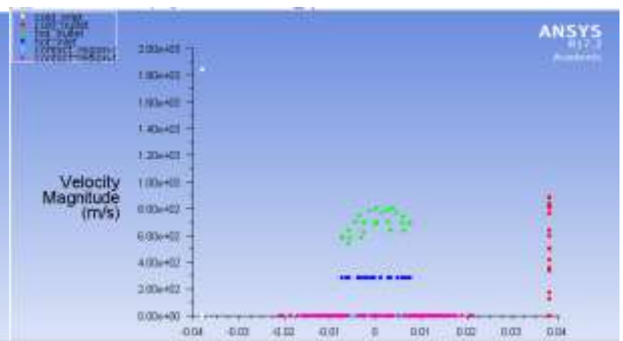


Figure 15. Static Pressure magnitude in different position Figure16. Velocity magnitude in different position

With fin		Temperature (k)	Pressure (Pa)	Velocity (m/s)
Hot fluid	inlet	355.4	535100	380.3
	outlet	320.3	36120	760.6
Cold fluid	inlet	305.3	535100	1711.0
	outlet	325.4	30100	950.7

Without fin		Temperature (k)	Pressure (Pa)	Velocity (m/s)
Hot fluid	inlet	360.4	624100	386.2
	outlet	326.0	3246	772.3
Cold fluid	inlet	305.3	313700	1738.0
	outlet	332.9	3246	579.3

VII. CONCLUSION

A CFD package (ANSYS FLUENT 17.2) was used for the numerical study of heat transfer characteristics of a double pipe heat exchanger for parallel flow and the results were then compared with that of the with and without fins. The results of this research described that there is not much difference in the heat transfer performances of the parallel-flow configuration. The simulation was carried out for water to water heat transfer characteristics and different inner pipe were studied. We are absorbed temperature and pressure, velocity in double pipe heat exchanger with and without fins conditions in inner pipe . From the velocity vector plot it was found that the fluid particles were undergoing an oscillatory motion inside the pipes. From the pressure and temperature contours it was found that along the outer side of the pipes the velocity and pressure values were higher in comparison to the inner pipes. The with fin double pipe heat exchanger gives the less temperature compare to without fin double pipe heat exchanger.

REFERENCES

[1] J.E.Kim, J.H.Doo, Man Yeong Ha, H.S.Yoon, "Numerical study on characteristics of flow and heat transfer in a cooling passage with protrusion-in-dimple surface", International Journal of Heat and Mass Transfer 55(s 23–24):pp. 7257–7267 , 2012

[2] David J.Kukulka, Rick Smith, Kevin.G.Fuller, "Development and evaluation of enhanced heat transfer tubes", Journal od Applied thermal Engineering, Vol 31, pp Issue 13, pp. 2141 – 2145, 2011

[3] Alberto Garcia , Juan Pedro Solano, Pedro.G.Vicente, A.Viedma, "The influence of artificial roughness shape on heat transfer enhancement", Journal of Applied thermal Engineering , 35(1) · January 2011

[4] S.Suresh, M.Chandrasekar, S.Chandra shekar, "Experimental studies on heat transfer and friction factor characteristics of Cuo/water nano fluid under turbulent flow in a helically dimpled tube", Journal of Experimental and fluid science, Vol 35, IIsue 3, pp. 542-549, 2011

[5] Wang, Y, He, Y. L, Lei, Y. G., Zhang, J., "Heat transfer and hydrodynamics analysis of a novel dimpled tube", Journal of Experimental and fluid science, pp 1273-1281, 2010

[6] ChinarukThianpong, PetpicesEiamsa-ard, Khwanchit Wong charee SmithEiamsa-ard, "Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator", Jounal of International communications in in Heat and Mass Transfer, Vol 36, Issue 7, pp. 698-704, 2009.

[7] S.W.Chang,K.F.Chiang,T.L.Yang,C.C.Huang," Heat transfer and pressure drop in dimpled fin channels", Journal of Experimental and fluid science, Vol 33, Issue 1, pp . 23- 40, 2008.

[8] Pedro.G.Vicente, Alberto Garcia , A.Viedma, "Experimental study of mixed convection and pressure drop in helically dimpled tubes for laminar and transition flow ", International Journal of Heat and Mass Transfer, 45(26):pp 5091-5105, 2002.