Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 5, Issue 04, April -2018

COMPARISON OF EXPERIMENTAL HEAT TRANSFER PARAMETERS FOR U-TUBE, SPIRAL & HELICAL COILS OF SAME LENGTH WITH CFD RESULT

Prof. S. S. Pawar [1], Harshal Kolte [2], Suchit Bhalerao [3], Vishal Kakad[4], Sachin Kharat[5]

[1] *Professor and Guide, Mechanical Engineering, A.C. Patil College of Engineering, Navi Mumbai.*
[2, 3, 4, 5] *Students of, Mechanical Engineering, A.C. Patil College of Engineering, Navi Mumbai.*

Abstract — *Heat exchanger is an important unit operation that contributes to efficiency and safety of many processes. The heat exchange is performed between hot and cold water. Heat exchanger is important appliance in field of thermal, heat mass, fluid flow, as in nuclear reactor, steam power plant (in super heater), fertilizers factory, etc. This report presents on comparative study of u-tube, spiral & helical coils for different heat transfer parameters by using SOLIDWORKS and ANSYS FLUENT. The experimental results are compared with CFD calculations results using the CFD package FLUENT 16.2 and experimental setup is fabricated by other group. Mild Steel was chosen as the as material for the construction of the u-tube, spiral & helical coils. The fluid flowing through the coil was taken as water. While the heat transfer characteristics of u-tube, spiral & helical coils heat exchanger are available in literature, there exist no publish comparison of Experimental and CFD analysis of u-tube, spiral & helical coils heat exchanger. Heat transfer characteristics of u-tube, spiral & helical coils heat exchanger for various boundary conditions are compared.*

Keywords*- U-tube coil, helical coil, spiral coil, Nusselt number, Overall heat transfer coefficient*

I. INTRODUCTION

A heat exchanger is a piece of equipment built for efficient heat transfer from one medium to another. The media may be separated by a solid wall to prevent mixing or they may be in direct contact. The classic example of a heat exchanger is found in an internal combustion engine in which a circulating fluid known as engine coolant flows through radiator coils and air flows past the coils, which cools the coolant and heats the incoming air. Heat exchange between flowing fluids is one of the most important physical process of concern, and a variety of heat exchangers are used in different type of installations, as in process industries, compact heat exchangers nuclear power plant, food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modelling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)

II. LITERATURE REVIEW

[1] **J.S.Jaykumar, S.M.Mahajani, J.C.Mandal, P.K. Vijayan & Rohidas Bhoi (2008)** compared heat transfer characteristics inside a helical coil for various boundary conditions and compared experimental results with the CFD results using CFD package FLUENT 6.2 and a correlation was developed to calculate the inner heat transfer coefficient of the helical coil. Experimental and CFD estimation of Heat transfer in helically coiled heat exchanger. This experiments were performed for coil with Pipe I.D. -10mm, O.D. - 12.7mm, Tube material- SS 316, PCD of the coil - 300mm, Tube pitch - 30mm. They also observed that the use of constant values for the thermal and transport properties of the heat transport medium results in prediction of inaccurate heat transfer coefficients.

[2] **S.S.Pawar and Vivek K. Sunnapwar (2013)** carried experimental studies on heat transfer to Newtonian and non-Newtonian fluids. In helical coil with laminar and turbulent flow. Experimental studies on isothermal steady state and non-isothermal unsteady state conditions were carried out in helical coils for Newtonian and non-Newtonian fluids. Water, Glycerol-water mixture as non-Newtonian fluids and dilute aqueous polymer solutions of sodium carboxymethyl cellulose (SCMC), Sodium Alginate (SA) as non-Newtonian fluids were used in this study. This experiments were performed for coil with curvature ratio δ = 0.0757, 0.064 and 0.055 in laminar and turbulent regimes. Further comparison of overall heat transfer coefficient U_o and Nusselt numbers for Newtonian and non-Newtonian under isothermal and non-isothermal is presented in this paper. For the first time an innovative approach of correlating Nusselt number to dimensionless number 'M' for Newtonian fluid based on experimental data. The results in this work showed that overall heat transfer coefficient is higher for smaller helix diameter as compared to larger helix diameter

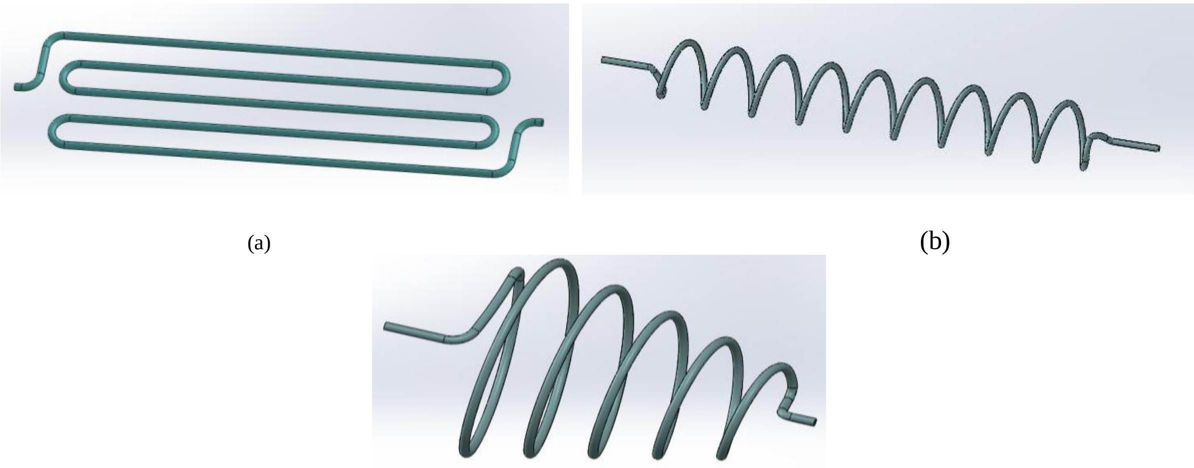
due to significant effect of centrifugal force on secondary flow in coil. They have also observed from results that heat transfer coefficient for pure water are higher than glycerol-water mixture and non-Newtonian fluids for same condition.
[3] **S.S.Pawar, Vivek K. Sunnapwar (2014)** carried out Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger. The author studies isothermal steady state and non-isothermal unsteady state conditions of experiment which was carried out in helical coils for Newtonian as well as for non-Newtonian fluids.

[4] **Pramod Deshmukh (Jan 2016)** used ANSYS FLUENT 15.0 for the numerical study of heat transfer characteristics of a helical coiled double pipe heat exchanger of the counter-flow. The authors compared CFD results with the experimental results from different studies and were well within the error limits. The simulation was carried out for water to water heat transfer characteristics and different inlet temperature.

[5] **Jamshid Khorshidi & Salman Hedari (April 2016)** carried design and construction of spiral heat exchanger. The author builds LAB-sized Spiral Plate Heat Exchanger by using Galvanized-Iron and concluded that Nusselt number is higher at the entrance because of the more temperature difference but it decreases as fluids go forward through the heat exchanger toward the exit points and as the temperature difference decreases and also they have concluded that the Nusselt number increases as the velocity of fluid increases.

III. CFD MODELLING

This study was carried out to estimate the influence of same boundary conditions on different coil geometry's of same length. The heat transfer characteristics of these geometry's was analyzed for various flow rates. The geometry of U-Tube, Spiral & Helical Coils are of same length i.e. 5.2 m and same pipe thickness i.e. 1mm used for modeling. The geometry of U-Tube, Helical & Spiral Coils were created on Solidworks shown in Figure 1. and mesh was created using ICEM CFD of the FLUENT package.
Dimensions of geometry are as follows:-
U-Tube – Number of turns = 4, Pipe diameter = 19.03 mm.
Spiral coil – No. of turns = 5, Pitch = 110 mm, Pipe diameter = 19.03 mm.
Helical coils – No. of turns = 9, Coil diameter = 163 mm, Pitch = 110 mm, Pipe diameter = 19.03 mm.



(a) (b)

(c)

Figure 1. 3-D Model in solidworks (a) 3D Model of u-tube (b) 3D Model of helical coil (c) 3D Model of spiral coil

In this model, unstructured non-uniform grids were used to mesh the pipe fluid volume and boundary layer mesh was generated with 3 prism layers called inflation and Growth rate-1.2 (unstructured non-uniform grids were also used by earlier investigators like **S.S.Pawar and Vivek K. Sunnapwar (2014)** Further increase in prism layers or decrease in element size was found to be out of computer capacity (Intel CORE i7 processor with 8 GB RAM and 4 GB graphic card) due to heavy geometry as compared to **Jayakumar et al. (2008)**. The method used for meshing is tetrahedrons as it gives maximum elements for analysis. Coil wall thickness of 0.001 m is considered in this analysis and shell conduction model of FLUENT is used. This will reduce computation time and excess load on computer. By using shell conduction the meshing of wall thickness is not necessary. Heat exchange from the hot fluid in the vessel to the cold fluid in the tube was modelled with convective heat transfer in the vessel maintained at $50 \pm 0.2^{o}C$ (natural convection), conduction through the tube wall and convective heat transfer to the coil fluid (forced convection). The inner and outer walls of the coil were defined as coupled for energy transfer from hot fluid outside the vessel to the cold fluid flowing through coil. For momentum equation, they are treated as no-slip ones. The coil material is mild steel considered for shell conduction model. The properties of water and mild steel are given in Table 1 and Table 2.

Table 2: Properties of water

Description	Value	Units
Density	1000	kg/m3
Specific heat	4187	J/kg-K
Thermal conductivity	0.6	W/m-K

Table 2: Properties of mild steel

Description	Value	Units
Density	7850	kg/m3
Specific heat	620	J/kg-K
Thermal conductivity	52	W/m-K

The temperature of cold water at inlet of coil is taken as 31^{o}C at inlet boundary condition. The inlet cold water and outlet hot water conditions are defined as velocity inlet and pressure outlet with zero back pressure respectively. Cold fluid at the inlet of coil is taken for various mass flow rate of 50,100,150 and 200 LPH flow. The temperature of wall is consider as 50^{o}C. The SIMPLE algorithm is used to treat the coupling equations of continuity and momentum for solving velocity and pressure distribution with skewness correction factor 1. Second order upwind scheme is used to treat the convection terms included in the equations. In laminar flow condition: for pressure, second order discretization scheme was used and for momentum and energy, second order upwind scheme was used. In turbulent flow condition: for momentum, turbulent kinetic energy and turbulent dissipation rate of second order upwind scheme was used. For laminar flow, viscous-laminar model and for turbulent flow, k–ε model were used to solve the governing equations. The relative convergence criterion of 1.0e−6 was used for continuity, x, y, z velocities and energy equation. While that for the k and ε was 1.0e−6 in turbulent flow analysis.

Grid generation:-

A grid dependency of the solution for two different mesh element's size was studied for three different geometries before finalizing the optimum coarse mesh as shown in Figure 2 (d). Grid independency of the solution was established. Fine meshing for smaller element size or more than three number of prism boundary layers is not possible for this heavy geometry which is out of computer capacity. Further increase in elements and nodes, the computation cost will increase because the convergence and the stability of iterations will become more difficult, as well as increasing memory and CPU run time. Grids shown in Figure 2 (d). (tetra/mixed) unstructured non-uniform grids is chosen for this present analysis which gives closer results to the experimental results with less than ±5% deviation in outlet water temperature and Nusselt number. This is resulted into the cost and time effective results which is affordable to the industrial applications.

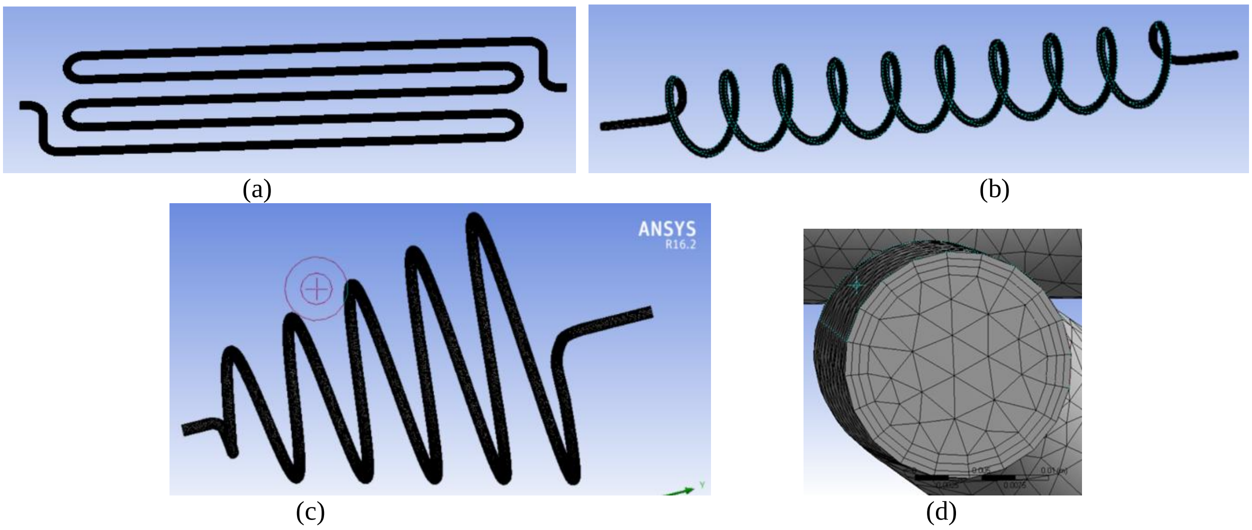


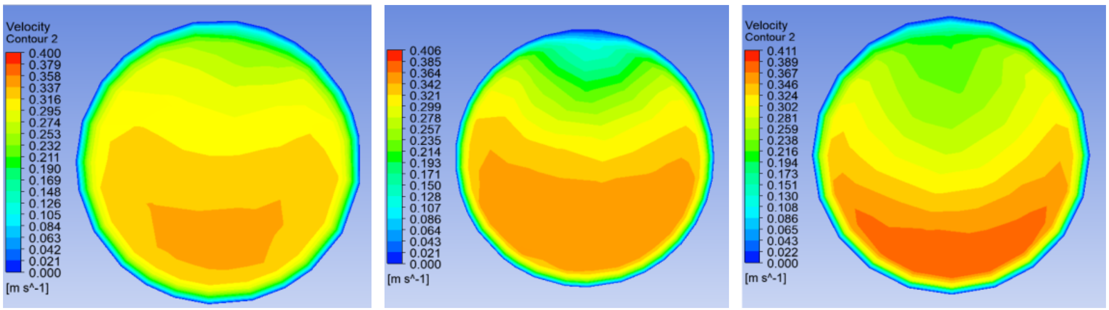
Figure 2. – Grids used in the analysis (a) Meshing of u-tube (b) Meshing of helical coil (c) Meshing of spiral coil (d) Enlarged cross section view of grid

The optimum mesh chosen for u-tube coil has 1979569 elements & 631830 nodes, helical coil has 1157224 elements & 399609 nodes and spiral has 741089 elements & 254657 nodes. The maximum time taken by FLUENT for some of the runs were observed about 6 hrs in an Intel CORE i7 processor with 8 GB RAM.

IV. CFD RESULTS

In this section, the results obtained by CFD for three different coil geometry are presented. ANSYS Fluent has been used to obtain the numerical results. Numerical investigation is carried out for varying flow rate with constant vessel temperature. In all these cases outlet temperature of water, velocity fields, overall heat transfer coefficient and Nusselt number results are evaluated and compared with experimental results.

Velocity Contour:-

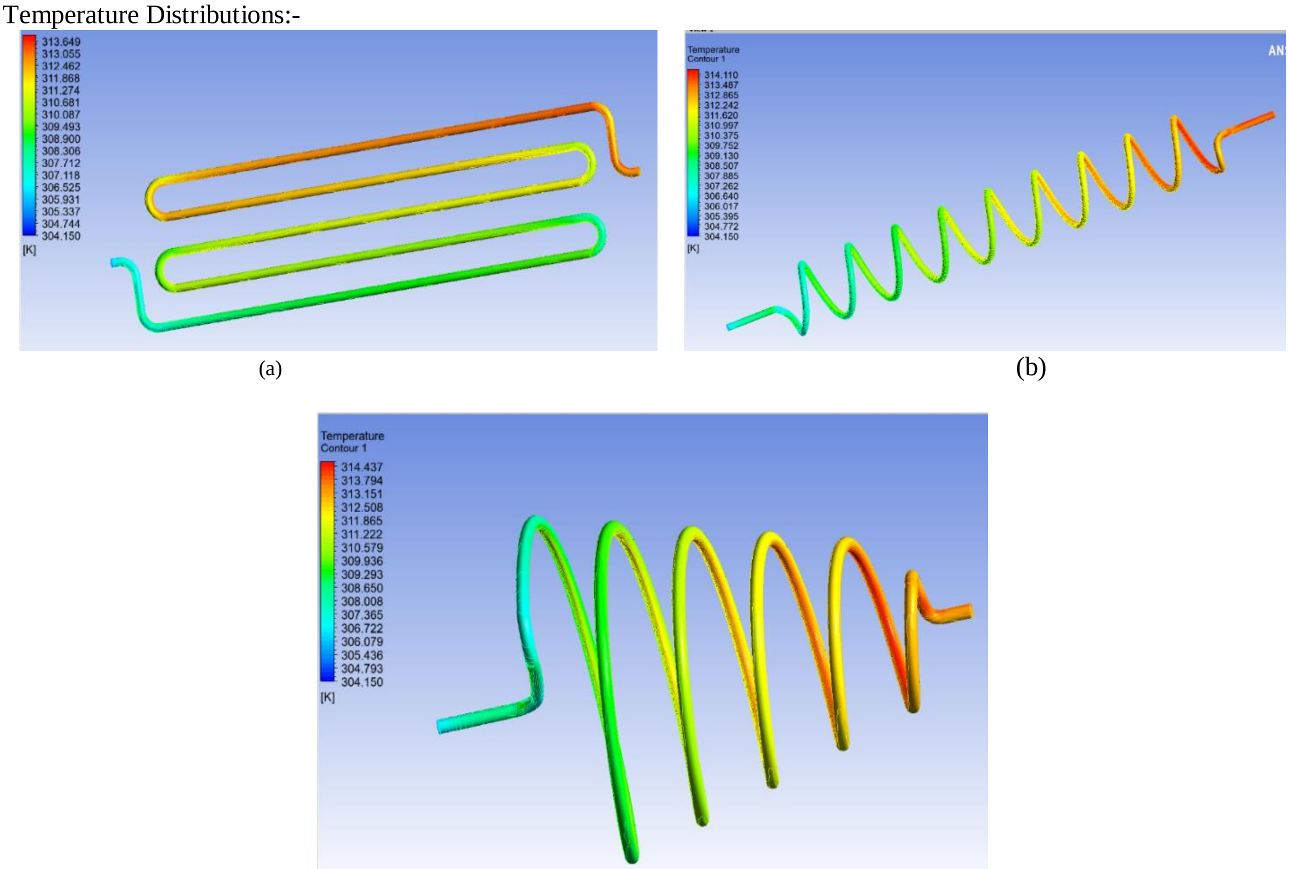


(a) (b) (c)

Figure 3. – Velocity contour (a) Velocity contour of u-tube (b) Velocity contour of helical coil (c) Velocity contour of spiral coil

Figure 3. shows 2D velocity contours which are plotted at outlet of coiled tube heat exchanger. Among all three coils spiral coil shows maximum velocity of 0.411 m/s at outlet which is more than other two which is because of its geometrical shape and curvatures. The curvature of the spiral and helical coil induces the centrifugal force, which further results in the development of secondary flow. Due to mixing of primary and secondary flow pattern simultaneously in coil, it becomes difficult to characterize the hydrodynamics of flow. Due to coil curvature effect of the coiled tube, water flows faster on the outer side of the coiled tube than that on the inner side as seen from Figure 3. for fully developed flow. This velocity difference at inner and outer side of the coil due to centrifugal force acting on moving fluid induces diametrical pressure drop. The intensity of secondary flows is dependent on the diametrical pressure drop. There is Uniform velocity vectors at inlet cross section which is not affected by centrifugal force as flow is not in curved tube at the entry.Velocity is kept on increasing till some extend of length of pipe in each coil, this length is known as flow settlement length, but in case of our geometries due to curvature no proper flow settlement has occur. Also Velocity is maximum or more than average at curvature of coil as turbulence occurs there.

Temperature Distributions:-



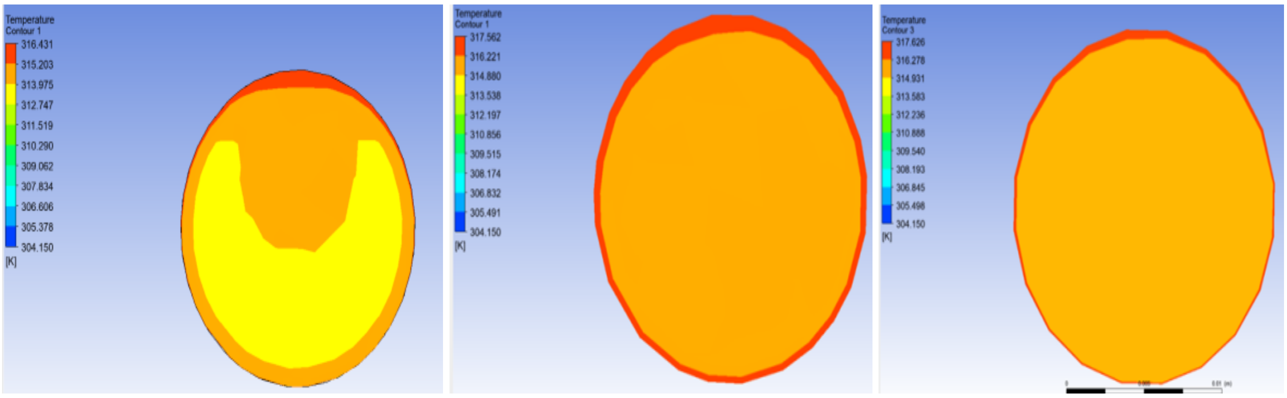
(a) (b)

(c)

Figure 4. – Temperature distributions (a) U-tube temperature distributions (b) Helical coil temperature distributions (c) Spiral coil temperature distributions.

Temperature distribution at various planes along the length of coil is as shown in Figure 4. In case of temperature variation, all three coil shows linear increase in water temperature along the length of coil as the coil has been placed in a vessel with constant temperature of freestream maintained at 50^{0}C. At the inlet, it is observed from temperature profile that temperature is uniform. As water in the coil moves in heated zone in the vertical direction, wall of the coil get heated due to surrounding temperature. Spiral coils shows maximum temperature of 314.437K at outlet which is maximum than other two coils, this is due to the more turbulence in spiral coil which causes heat transfer coefficient to be increased than normal.

Temperature Contour:-



(a) (b) (c)

Figure 5. – Temperature contour for flow rate of 50 LPH at outlet (a) Outlet temperature contour of u-tube (b) Outlet temperature contour of helical coil (c) Outlet temperature contour of spiral coil

Temperature contour at outlet of coil for flow rate of 50lph is as shown in Figure 5. From above temperature contours it can be seen than U-tube shows water temperature at outlet with maximum variation this is because heat absorbed from hot water is not evenly distributed in flowing water. But in case of other two geometries heat absorbed from hot water is evenly distributed into the water flowing through coils thus the contours of temperature at outlet of coils shows even distribution of temperature. This happens because of more turbulence in two coils whether as U- tube has less turbulence. Total temperature at outlet is calculated by FLUENT using volume integral method is presented in Table 3 for various flow rate.

Table 3 – Outlet temperature of water for different flow rates

Flow-Rate, Q (LPH)	Temperature (k)		
	U-Tube	Helical	Spiral
50	316.43	317.25	317.63
100	314.46	315.00	316.28
150	313.93	314.35	316.05
200	313.47	313.73	314.67

Above Table shows outlet temperature of water from coil at different flow rates. It shows that outlet temperature of water is maximum in case of spiral coil than other two for constant flow rate and it keeps decreasing as water flowrate is increase.

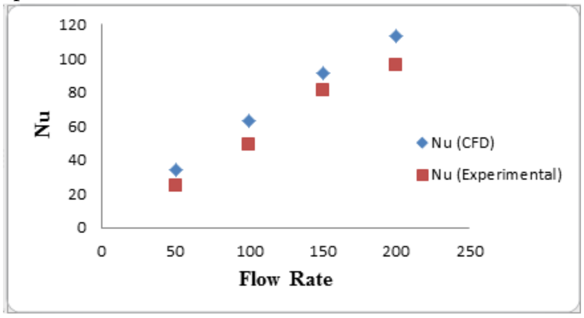
Comparison of CFD results with experimental results:-

Table 4 – Comparison for outlet temperature of CFD with experimental results

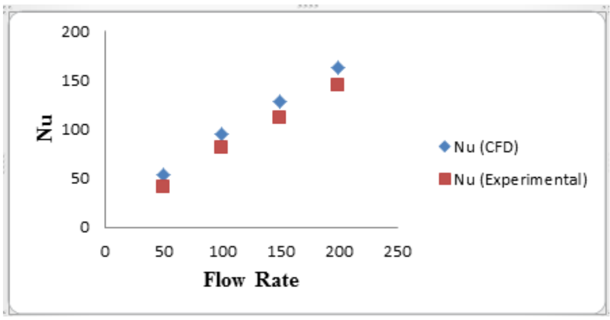
Flow rate (LPH)		Temperature (^{o}C)					
		U-Tube		Helical		Spiral	
		CFD	Experimental	CFD	Experimental	CFD	Experimental
Laminar	50	43.28	43.5	44.1	44.5	44.48	46.1
	100	41.31	41.5	41.85	42.5	43.137	43.5
Turbulent	150	40.78	41	41.2	42	42.90	43
	200	40.32	40.5	40.58	41.5	41.52	42

Above table shows variation of outlet water temperature with respect to flow rate, which depicts linear decrease in outlet water temperature with increase in water flow rate. From Table 4 it can be seen that there is only $\pm$ 5% deviation in temperature.

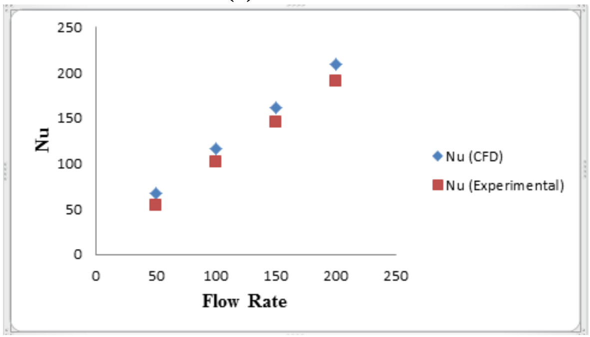
Comparison of Nusselt number:-



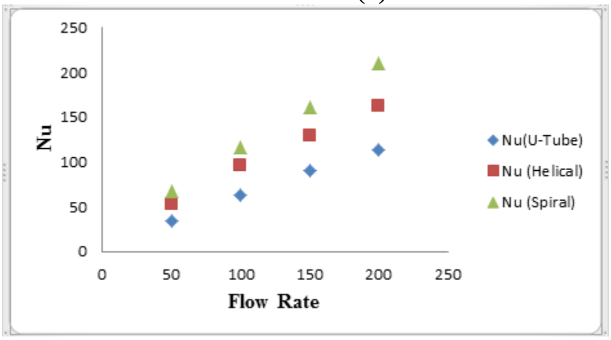
(a)



(b)



(c)



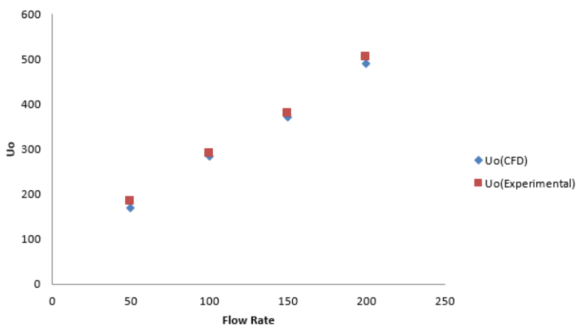
(d)

Figure 6. – Nusselt vs Flow Rate (a) Nusselt vs Flow Rate for U-Tube Coil (b) Nusselt vs Flow Rate for Helical Coils (c) Nusselt vs Flow Rate for Spiral Coil (d)Comparison of Nusselt for U-tube, Helical and Spiral coil (CFD results)

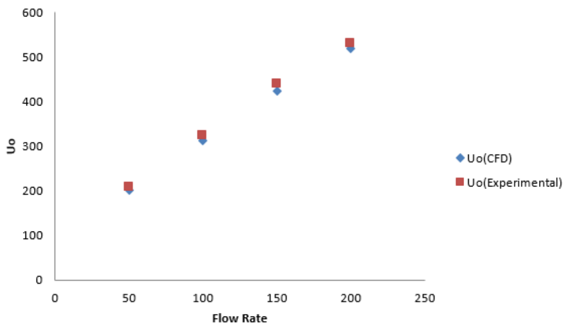
Figure 6 (a), (b), (c) shows comparison between experimental and CFD results for Nu v/s Q (flow rate) which shows approximately $\pm$ 5 % variation in results. Figure 6 (d) shows comparison of nusselt number obtained from CFD results. It clearly shows that nusselt number is higher for Spiral coil as compared to U-tube and Helical and it also shows that, as flow rate increases the nusselt number also increases.

Comparison of Overall heat transfer coefficient for U-tube, Helical, Spiral coil:-
The overall heat transfer coefficient of CFD result is calculated using Eq.1 from its components: inner and outer heat transfer coefficient. In this equation, wall conductance is neglected due to the smaller thickness (1 mm) of coil wall. This method of calculating overall heat transfer coefficient for smaller wall thickness was also found in the literature. In this analysis, overall heat transfer coefficients were calculated for temperature dependent fluid properties, constant temperature fluid properties and based on experimental data for comparison purpose.

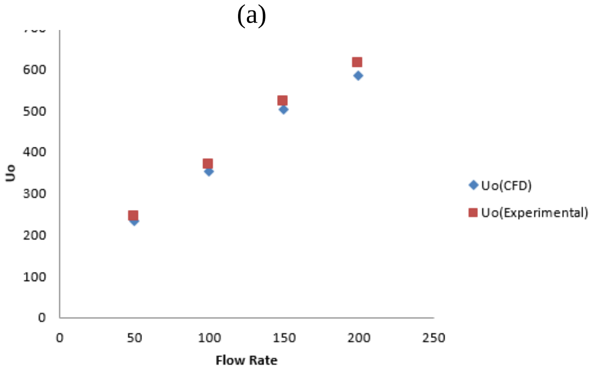
$$\frac{1}{ho} + \frac{1}{hi} = \frac{1}{Uo} \quad (1)$$



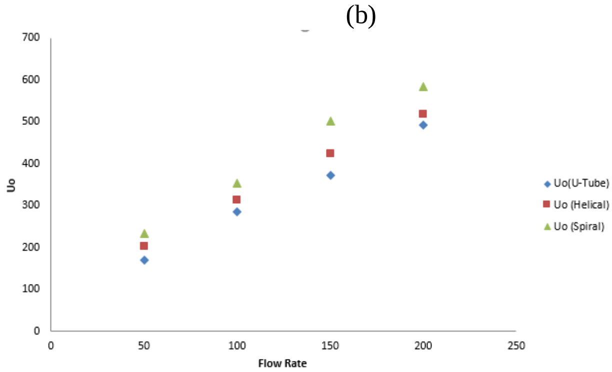
(a)



(b)



(c)



(d)

Figure 7. – Overall heat transfer coefficient Vs Flow Rate (a) Overall heat transfer coefficient Vs Flow Rate (U-tube) (b) Overall heat transfer coefficient Vs Flow Rate (helical coil) (c) Overall heat transfer coefficient Vs Flow Rate (spiral coil) (d) Comparison of Overall heat transfer coefficient for U-tube, Helical and Spiral coil (CFD results)

Figure 7 (d) shows variation of U_o v/s Q (flow rate) of three coils for results of CFD. It shows linear increase in U_o with increase in water flow rate. Overall heat transfer coefficient for Spiral coil is more than other two coils thus it has more heated water at outlet.

V. CONCLUSION

We observed that the outlet temperature of spiral coil is more than the other two coils i.e. U-tube and helical coils, for all the flow rate this shows that it is more effective than other two. It was also observed that, as the flow rate increases the Nusselt number also increases of all three coils. The overall heat transfer coefficient of spiral coil is more than the other two coils as we increases the flow rate the overall heat transfer coefficient increases. After comparison of CFD results with Experimental results it is seen that the CFD predictions match reasonably (+ 5% deviation) well with the experimental results within experimental error limits. Average percentage increase in heat transfer for Spiral coil with respect to U-Tube coil is 28.75%. Average percentage increase in heat transfer for Spiral coil with respect to helical coil is 15.25%. It can be say that one should go with the spiral coil for compact geometry which is more effective than the other two geometries for same length.

VI. REFERENCES

[1] S.S.Pawar and Vivek K. Sunnapwar,"Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger", CHERD-1475.
[2] S. S. Pawar and Vivek K. Sunnapwar, "Experimental studies on heat transfer to Newtonian and non-Newtonian fluids in helical coils with laminar and turbulent flow", Experimental Thermal and Fluid Science 44 (2013) 792–804.
[3] Jayakumar, J.S., Mahajani, S.M., Mandal, J.C., Kannan, N.I., Vijayan, P.K.,"CFD analysis of single-phase flows inside helically coiled tubes". Comp. Chem. Eng. 34, 430–446
[4] Jayakumar, J.S., Mahajani, S.M., Mandal, J.C., Vijayan, P.K., Bhoi, R., "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers", Chem. Eng. Res. Des. 86,221–232
[5] Mohammed Imran, Gaurav Tiwari and Alwar singh Yadav, "CFD Analysis of Heat Transfer Rate in Tube in Tube Helical Coil Heat Exchanger" IJISET – International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 8, July 2015.

[6] Pramod Deshmukh, Vikram D Patil and Baviskar Devakant, "CFD Analysis of Heat Transfer in Helical Coil Tube in Tube Heat Exchanger" International Journal of Innovations in Engineering Research and Technology [Ijiert] Issn: 2394-3696 Volume 3, Issue 1, Jan.-2016.

[7] Umang K Patel and Krunal Patel, "CFD Analysis Helical Coil Heat exchanger", IJARIIE-ISSN (O)-2395-4396 Vol-3 Issue-2 2017

[8] Syed Mohammed Arif, Rashed Ali, Dhanraj P. Tambuskar and Divya Padmanabhan, "CFD Analysis of Heat Transfer in Helical Coil", International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 - 0056 Volume: 04 Issue: 03 | Mar -2017.

[9] Shiva Kumara and K.Vasudev Karantha,"Numerical Analysis of a Helical Coiled Heat Exchanger using CFD", International Journal of Thermal Technologies ISSN 2277 – 4114.

