Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

A Review of Comparative Study of Heat Transfer by Experimentation and CFD for Helical Coiled Heat Exchangers

Gourav Vivek Kulkarni
B.E. (Mechanical Engineering)
KLS Gogte Institute of Technology
Belagavi, Karnataka, India

***Abstract*— Heat Exchangers are of utmost industrial importance in the field of Power generation, Refrigeration, Air Conditioning, Nuclear energy and numerous other applications. However it is required to utilize the heat energy of the hot fluid to the optimum extent in order to come up with much more efficient energy systems. The classic design of a heat exchangers mainly consists of a shell and tube. The tube is enclosed within the shell and generally caries the hot fluid. The shell carries the cold fluid. There are various flow arrangements that can facilitate enhanced heat transfer. However by continuous research, it was found that straight tube heat exchangers were not providing the required cooling effect to the desired extent. This has led to the development of Helical coil Heat exchangers in which, instead of a straight tube, a helical tube is used. The heat transfer analysis in helical coil heat exchangers is the next step towards performance evaluation and optimization of the system. This can be accomplished in two ways. One being experimentation and the other being numerical estimation by means of Computational Fluid Dynamics (CFD). This work elaborates the methodology undertaken in experimentation as well as the analysis done by means of CFD considering three cases. The experimental procedure comprises of making the fluid to flow at various mass flow rates and obtaining the heat transfer characteristics. However the experimental results are prone to a certain external factors that can affect the results. This limitation can be overcome by using CFD where everything is pre-programmed and solutions totally depend on the mathematical models and equations. Further the same model is developed using suitable software and the fluid flow simulation is carried out. Various boundary conditions like temperature, heat flux and other temperature dependent thermal and transport properties are well defined during the pre processing stage. Solutions are carried out to obtain the results of simulation. The chief objective is to establish a correlation between the experimental results and the CFD results. On obtaining the results, critical comparison is done with regards to the corresponding results obtained from experimentation and CFD. Thus we can look at CFD as an excellent tool to analyze fluid flow systems with lesser investment as compared to experimentation. Since the fluid flow through the Helical coil Heat exchangers has got complex flow characteristics, there may be certain areas where CFD can provide results that have got much more realistic values. At the end a conclusion is drawn as to which method of analysis can prove to be handy as well as reliable since the areas in which Helical Coil heat exchangers are used now a days are of extreme technical importance.**

Keywords*— *Computational Fluid Dynamics(CFD), Heat Exchanger, experimentation, simulation

I. INTRODUCTION

A Heat exchanger is a device used to carry out heat transfer between two fluids so as to utilize the thermal energy of the hot fluid to accomplish the desired task. Extensive thermal analysis is required in order to come up with a heat exchanger that can efficiently transfer heat between the fluids. A conventional heat exchanger comprises of a tube enclosed within a shell. Various flow arrangements can be done in order to obtain the heat transfer. These type of heat exchangers are categorized as straight tube heat exchangers. There are various arrangements including parallel flow, counter flow, cross tube, mixed flow and so on.

Over the years, experimentation has shown that straight tube heat exchangers provide lesser heat transfer rates for the same boundary conditions as compared to the helical coil heat exchangers. This has led to extensive research and development of helical coil heat exchangers. Helical coil heat exchangers are devised in the form of a helical tube that contains the flowing fluid. These heat exchangers have shown better performance as compared to the conventional double pipe heat exchangers.

The helical coil heat exchanger can be better understood by its nomenclature. It comprises of a tube with a fixed diameter coiled to form a helix with a fixed coil diameter. The following figure represents the nomenclature of a helical coil heat exchanger.

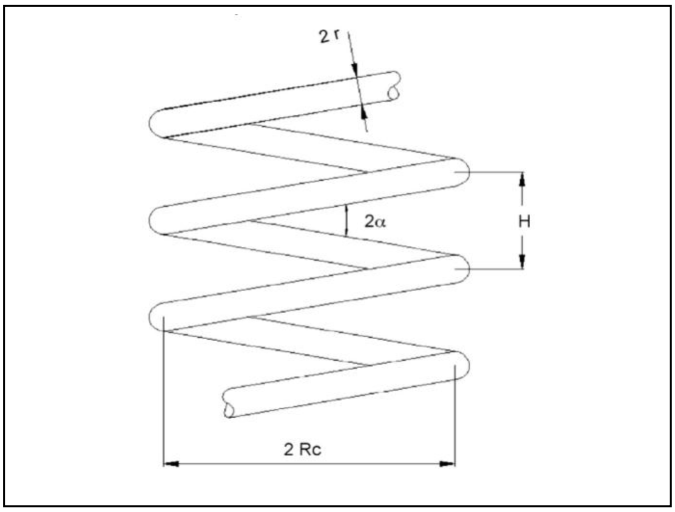

Fig. 1. Nomenclature of Helical Coil Heat Exchanger[1]

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

Fig. 1. shows the general convention followed in the nomenclature of a helical coil heat exchanger. A tube of diameter '2r' is coiled with a coil radius of 'R_C'. From the figure, 'H' represents the pitch of the helical coil. It is the distance between two adjacent turns. The coil diameter is also known as the Pitch Circle diameter (PCD). The helix angle 'α' is defined as the angle which is made by the projection of one turn of the coil with the plane perpendicular to its axis.

Another term called Curvature ratio (δ) is defined as the ratio of radius of the tube to the coil radius. It mainly signifies the size factor of the coil. It can be inferred that as we vary there parameters, there can be excessive change observed in the performance of the heat exchanger. Reduction in the pitch or helix angle may lead to decrease in heat transfer beyond a certain extent due to increased convection resistance. This can be overcome by a compact design comprising of intersecting tubes of the hot and cold fluid. Such numerous variations can be undertaken to improve the performance of the heat exchanger.

A typical schematic diagram of a helical coil heat exchanger is as follows. It shows the various features that are of importance with regards to the further analysis.

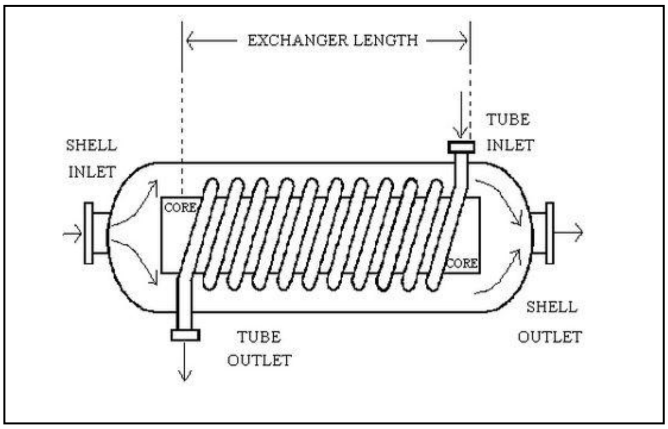

Fig. 2. Schematic Diagram of Helical Coil Heat Exchanger[1]

The above diagram can be easily referred to understand the basic working principle of a helical coil heat exchanger. It comprises of a cylindrical shell with inlet and outlet ports. A centrally located core, houses the helical tube of the heat exchanger. The tube inlet is provided at one end while the tube outlet is provided at the other end. The axial distance between the tube inlet and tube outlet is called as the exchanger length. Larger the exchanger length, larger is the heat transfer up to a certain value beyond which, the thermal resistance increases to resist the heat transfer thereby decreasing the heat transfer rate.

The helical coil heat exchanger is preferred to straight tube heat exchangers for a numerous reasons. The most important advantage of a helical coil heat exchanger is the effective space utilization and lesser space requirements. This makes a helical coil heat exchanger to be installed at applications where space is a major issue and its effective utilization is of utmost importance. Straight tube heat exchangers on the other hand, although easy to design, cannot be installed in a limited space for the desired performance. At lower flow rates, the double pipe heat exchangers reveal lesser heat transfer rates thus making them uneconomical for practical applications where, along with the effectiveness, cost also has to be considered. Maintenance of helical coil heat exchanger tubes is easier and economical as compared to the straight tube heat exchanger. This is because multiple phases of the fluid can be much more easily accommodated in a helical coil heat exchanger tube.

On understanding the basic structure and working of a helical coil heat exchanger, the next step is to carry out the heat transfer analysis for its optimum performance. This can be done by having an experimental setup in which the fluids can be made to flow and the results can be obtained thereby. There is another route in which we can use Computational Fluid Dynamics (CFD) in order to simulate the flow through the flow field. Both these techniques have their own requirements and procedures. This review emphasizes on a comparison between the procedure of experimentation and CFD in order to carry out the heat transfer analysis.

II. LITERATURE SURVEY

A number of journal papers were referred in order to come up with the contents of this work. However a few of them that have evidently influenced this review work have been elaborated in this section.

The paper entitled, "Experimental and CFD estimation of heat transfer in Helically coiled heat exchanger" by J S Jayakumar et al.[1], mainly considers fluid to fluid heat transfer at specified boundary conditions of constant heat flux and constant temperature. The analysis was carried out by heat dependent properties of heat transport media. The methodology undertaken was experimentation, simulation and the comparison of results. Suitable correlations were developed thereafter. There was a specially devised experimental setup with shell and helical tube within it. The experiment was carried out at steady state. The mass flow rate of fluid within the tube was varied at various values of temperature. Five different mass flow rates and three different temperatures were considered during the experiment. Shell side flow was kept constant and it was observed that the heat transfer coefficient remains constant on the shell side. Further numerical simulation includes the use of Computational Fluid Dynamics(CFD) in order to analyze the heat transfer. The effect of using temperature dependent thermal and transport properties was studied and comparison was made with effect as observed during experimentation. It was inferred that the Nusselt number value as obtained when ambient properties were used was associated with an error or 24%. But when mean temperature properties were used, the error in Nusselt number reduced to 10%. This signifies that specified boundary conditions can improve the performance. Further, contours of temperature variation and velocity variation along the pipe were obtained and analysed. On carrying out the final comparison, it was found that specification of constant temperature and constant heat flux does to give proper modelling conditions to which further boundary conditions can be applied. This had led to the use of a model with conjugate heat transfer. It is concluded towards the end that CFD matches reasonably with the experimental results within the error limits.

The paper entitled, "Experimental and CFD study of a helically coiled heat exchanger using water as a fluid" by M Balachandran [2] reveals that compared to straight tubes, curved tubes provide more advantages in the context of heat transfer. In all general cases, the helix is wound inside the case but in this work, it is wound outside the case. This avoids insulation to be provided on the outer side of the casing. The

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

flow simulation is carried out using Solid Works software with water as the working fluid. The flow conditions for both the fluids were considered to be laminar with the mass flow rate of cold fluid being constant with variable mass flow rate of the hot fluid. Here also the experiment is carried out at a steady state and the results comprise of the variation in effectiveness, overall heat transfer, heat transfer coefficient with change in mass flow rate of the hot fluid. The contours of velocity and temperature were also analyzed. The outcome of the work was that with the increase in mass flow rate of the hot fluid, various parameters like effectiveness, overall heat transfer, heat transfer coefficient, Nusselt number and heat transfer rate of the cold fluid also increase due to better flow distribution in a helical coil. Thus it was concluded that a helical coil heat exchanger can be much better as far as the performance is concerned as compared to the conventional straight tube heat exchanger.

In the paper entitled, "A comparative analysis of Thermal characteristics between experimental values and FEM values in helically coiled heat exchanger" by Revendra Verma et al.[3] begins with the description of enhancement in the heat transfer by using a helically coiled heat exchanger as compared to a straight tube heat exchanger. In this work, the constrains are velocity and mass flow rate. These variables are considered to be the key factors that influence the heat transfer. With these constrains, the values of heat transfer rates and heat transfer coefficients are easily obtained. The experimental procedure is similar to that followed by J S Jayakumar et al.[1] The FEM procedure comprises of modeling the cavity using a helically coiled heat exchanger with unstructured mesh and using hydrogen gas as the fluid. The coolant used is liquid nitrogen. It concludes to say that the CFD results are in good agreement with the experimental values and that CFD can be a very powerful tool to replace the complex and expensive experimental procedure which may require use of liquid nitrogen which may prove to be an expensive affair.

The paper entitled, "Heat Transfer Analysis of helically coiled heat exchanger" by Madhuri Tayde et al.[4] considers the effect of using the actual fluid properties instead of constant values. The effect is studied and the importance of using the actual fluid properties is established. The characteristics of heat transfer inside the helical coil are considered and are examined. The turbulence model used is Shear Stress Transport(SST) and k-epsilon which have a blending function that can provide standard values in main stream flow and near the boundary of the shell where the gradient is steeper. Further it was found that specification of constant temperature and heat flux does not yield proper modeling conditions. Therefore conjugate heat transfer and temperature dependent properties of heat transport media are considered for the analysis of the heat exchanger.

The paper entitled, "CFD analysis of Heat transfer rate in Tube in Tube Heat Exchanger" by Mohammad Imran et al.[5] was studied in order to understand the procedure of CFD analysis in a heat exchanger. The flow in this case was turbulent with constant heat transfer rate specified as the boundary conditions. A new parameter reciprocal to the Curvature ratio, termed as the D/d ratio where 'D' stands for diameter of the coil and 'd' stands for diameter of the tube, was considered as the key variable. Optimisation of this ratio for the flow characteristics was the objective of this numerical analysis. Turbulent flow model with counter flow heat exchanger was considered for which it was found that as the Reynold's number increases, the Nusselt number also increases showing enhanced mixing at increased turbulence levels. Further, as the D/d ratio increases, the Nusselt number id found to decrease which establishes certain size limitations on the heat exchanger design. The optimum value of D/d ratio at which the Nusselt number is maximum was found to be 25. The outer wall boundary condition was found to have no significant effect on the Nusselt number. It was also reported that as the Reynold's number increases, the log mean temperature difference in the heat exchanger also increases depicting better heat transfer as the flow becomes turbulent.

For the review of comparative study of heat transfer by experimentation and CFD, papers authored by J. S. Jayakumar et al.[1], M Balachandran[2] and Revendra Verma et al.[3] will be studied in detail in the following sections to understand the experimental and simulation procedure followed in each work and the comparative study undertaken thereby.

III. METHOD OF EXPERIMENTATION IN HEAT TRANSFER ANALYSIS OF A HELICAL COIL HEAT EXCHANGER

This section comprises of three parts in which the experimental procedure as followed by three different papers is studied. The experimental setup and the analysis thereby will be discussed.

A. Experimental Procedure followed by J. S. Jayakumar et al.[1]

The following figure depicts the experimental setup established for the research.

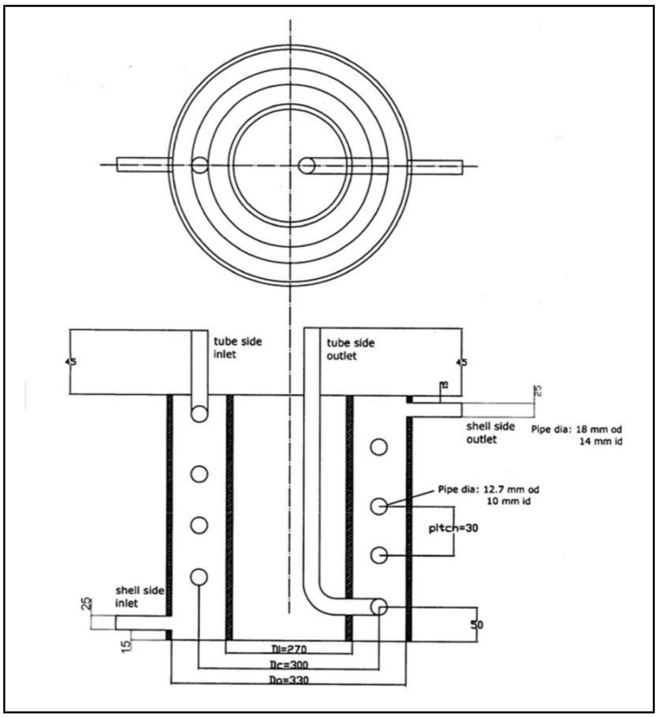

Fig. 3. Front Sectional view of the experimental setup[1]

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

Fig. 3. depicts the various dimensions of the experimental setup employed in the work. The setup has been devised based on previous results and study conducted by the authors. A block diagram of the same is shown as follows.

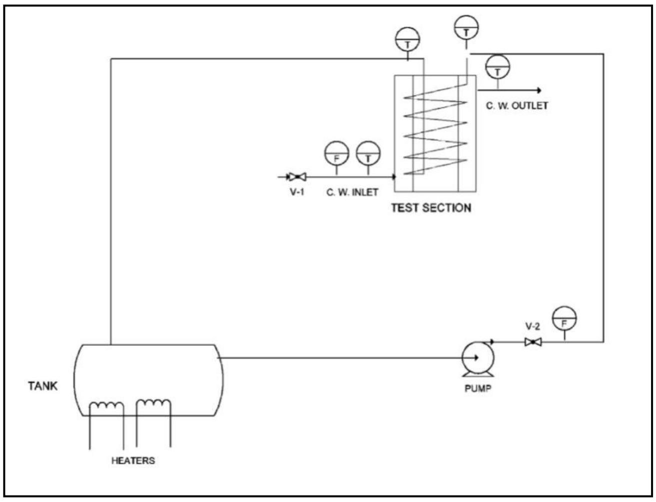

Fig. 4. Block diagram of Experimental Setup[1]

The tube of the heat exchanger has an internal diameter of 10mm and an external diameter of 12.7 mm. The tube material is Stainless Steel, SS 316. The Pitch Circle Diameter (PCD) of the coil is 300 mm while the pitch is 30 mm. The helical coil is enclosed in a vessel to simulate the shell side of heat exchanger. The cold fluid enters the heat exchanger from the bottom and flows upwards. A tank with electrical heaters is provided to heat the water to be circulated through the helical coil. There are three heaters, with a total power of 5000 WA controller maintains the water temperature at the inlet. A centrifugal pump is used to pump the hot fluid to the heat exchanger. Flow rate of hot fluid is measured using a rotameter. Resistance Temperature Detectors i.e. RTDs are used in order to carry out the temperature measurements. The flow rate , inlet temperature and outlet temperature of the cold fluid are measured by this equipment. The arrangement is made such that there is a constant rise in temperature of the cold fluid.

The experimental procedure employed is as follows. The setup is at first allowed to reach the steady state. Thereafter, the mass flow rate is varied along with temperature. There are five values of mass flow rate and three values of temperature that are considered for the experiment. For each set, the shell side flow is kept constant which leads to constant heat transfer coefficient on the shell side. The experiment is carried out by changing the flow rate through the tube. The results include the values of temperatures of hot and cold fluids at the inlet and outlet as well as the power consumption of the pump and the power input to the heater.

Various results are obtained that include the variation of Inner Nusselt number with respect to the Dean number. Dean number is similar to the Reynold's number and is used in this study. It is defined as follows

$$De = Re*(r/R_c)^{0.5} \quad (1)$$

where De is the Dean's number

Re is the Reynold's number

r is the tube radius in mm

R_c is the coil radius in mm

The graphs of variation of inner Nusselt number vs. the Dean number are as follows.

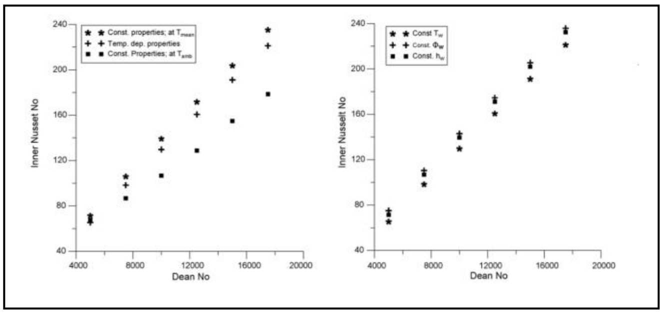

Fig. 5. Variation of Inner Nusselt number vs. Dean number at various properties at the wall and mean values of Temperature heat flux and heat transfer coefficient[1]

It is evident from the graphs that, at constant properties at ambient temperature as well as at mean temperature, the value of inner Nusselt number varies from 60 to 240. Whereas the same for constant wall conditions depicts a steady rise from 60 to 240. Thus for constant wall conditions, the heat transfer coefficient increases rapidly as compared to its corresponding cases for mean and ambient conditions.

B. Experimental Procedure followed by M. Balachandran[2]

Following is the experimental setup of this research work.

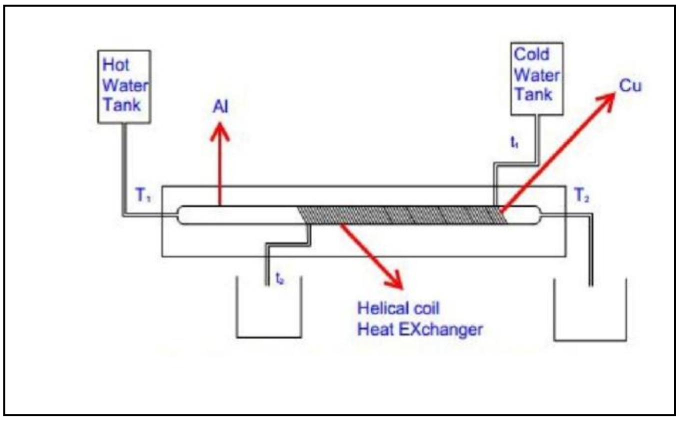

Fig. 6. Experimental setup[2]

The experimental Setup comprises of a Aluminium tube which is connected to the hot water tank and a copper coil around it which is connected to the cold water tank. The helix is wound outside the the aluminium tube to avoid the extra insultion required. The copper coil has an inner diameter of 4.5mm and an outer diameter of 6.5 mm while the aluminium tube has an outer diameter of 57.5 mm. The straight coil length is 1675 mm and the tube length is 800 mm. The number of turns is 82 with the piitch being 8mm. The fluid used is water.[2]

The experimental procedure involves the variation of mass flow rate of fluid at different temperatures and obtaining corresponding plots of Dean number vs. mass flow rate at various temperatures. Similar plots are obtained for variation of Overall heat transfer coefficient vs. mass flow rate and another plot of variation of Nusselt number against mass flow rate.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

Since the individual plots are not available in literature, the comparative plots are shown in the section dealing with the comparative study.

C. Experimental Procedure followed by Revendra Verma et al.[3]

The Experimental setup is same as that established by J S Jayakumar et al[1]. The difference is in the experimental procedure. Here, velocity of flow and the mass flow rate are considered to be the key constrains. Cooling water is circulated through the shell and the flow rate, inlet and outlet temperature of the same is measured.

The experiment in this case too, is conducted at steady state. There are six different values of mass flow rate at which the experiment is conducted maintaining constant temperature. During each iteration of a new mass flow rate, the mass flow rate on the shell side of the heat exchanger is constant which ensures that the heat transfer coefficient is constant.

On obtaining the various results at various values of mass flow rate and velocities, a tabulation was obtained as follows.

Velocity (m/s)	Toutlet (K)	Mass flow rate (kg/s)	Q (KW)	hi ($KW/m^2.K$)
0.2	52	1.35E-5	-0.0003524	0.33831
0.21	52.4	1.42 E-5	-0.0003593	0.31675
0.22	53.11	1.49 E-5	-0.0003557	0.29241
0.23	53.64	1.55 E-5	-0.0003535	0.26227
0.24	54.05	1.62 E-5	-0.0003562	0.24925
0.25	54.65	1.69 E-5	-0.0003512	0.22704

Fig. 7. Experimental Results as formulated by Revendra Verma et al.[3]

Thus it can be inferred that as the velocity of increases, the mass flow rate increases as also the rate of heat transfer increases. While it is observed that the value of heat transfer coefficient decreases. These results will be compared with those obtained by CFD in the further parts in the next section.

IV. CFD APPROACH TO HEAT TRANSFER ANALYSIS OF A HELICAL COIL HEAT EXCHANGER

The previous section dealt with the experimental analysis of Heat transfer in a helical coil heat exchanger. In this section, cases will be considered as to how the same research was carried out by means of CFD. It is intended to give a clear picture of the various techniques of CFD followed in case of heat transfer analysis.

A. CFD simulation Procedure followed by J. S. Jayakumar et al.[1]

For the modelling purpose, regression analysis was carried out using MATLAB and the following set of equations were devised to enter the temperature dependent properties. These polynomial functions were programmed in FLUENT.

$$\mu(T) = 2.1897e-11T^4 - 3.055e-8T^3 + 1.6028e-5T^2$$
$$-0.0037524T + 0.33158$$

$$\rho(T) = -1.5629e-5T^3 + 0.011778T^2 - 3.0726T + 1227.8$$

$$k(T) = 1.5362e-8T^3 - 2.261e-05T^2 + 0.010879T - 1.0294$$

$$Cp(T) = 1.1105e-5T^3 - 0.0031078T^2 - 1.478T + 4631.9$$

Fig. 8. Polynomial Equations formulated to model temperature dependent properties[1]

In the above figure, T is temperature in kelvin.
μ represents dynamic viscosity in Pa-s
ρ represents density in m^2/s
k represents thermal conductivity in W/m-K
C_p represents the specific heat in J/kg-K

Further, a grid independence study was carried out to obtain the following grids for helical pipe fluid volumes.

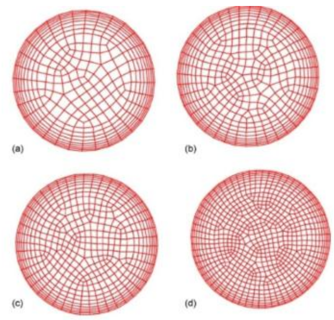

Fig. 9. Grids used for helical pipe fluid volume[1]

From the Fig. 9, the sub sections a, b, c and d represent the successive grids studied during the grid independence study. The last grid was finalized as beyond that, the accuracy of results would not be affected even though the number of nodes would be increased.

On carrying out the simulation the following contours were obtained. Prominent among them were the velocity and temperature profiles considering constant properties and then considering temperature dependent properties.

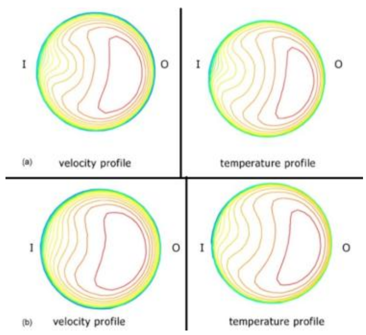

Fig. 10. Velocity and Temperature profiles at the exit of the tube for (a), constant properties and (b), temperature dependent properties[1]

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 10, October - 2017

From Fig. 10, I represents Inner side and O represents Outer side of the tube. It can be observed from all the figures that the velocity at the right end i.e. the outer side is higher than that at inner side. The variation in shapes of the contours can be attributed to the consideration of different boundary conditions.

On further processing, results were obtained at constant wall properties and for conjugate heat transfer. Graphs of Nusselt number vs. Dean number were plot using the data obtained from the simulations. The variation in Nusselt number is similar to that observed in the experimental results.

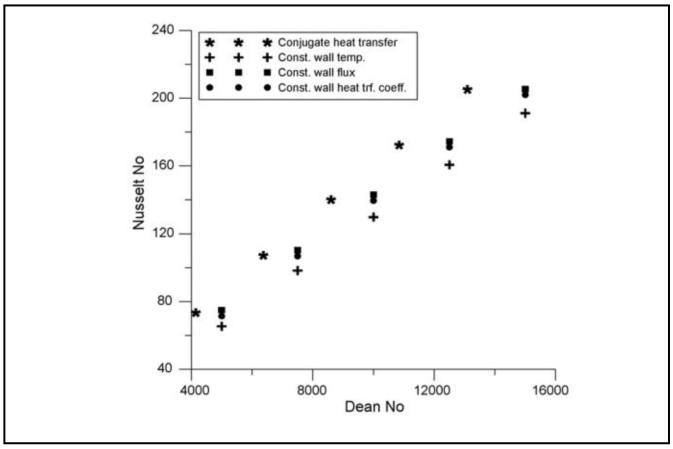

Fig. 11. Variation of Nusselt number vs. Dean number at various boundary conditions[1]

It can be observed that for simulations the upper limit of Nusselt number is slightly lower than the experimental values depicting better approximation at lesser values of Dean number.

B. CFD simulation Procedure followed by M. Balachandran[2]

In this work, the CFD simulation was done using the Solid Works Flow simulation(Cosmos Express) software and various contours were obtained. Suitable CAD tools were used to carry out the modelling and to apply the boundary conditions. Since the literature contains all graphs and results discussed in a comparative form for experimentation and CFD, the same has been incorporated in the next section.

C. CFD simulation Procedure followed by Revendra Verma et al.[3]

This research work has considered a very systematic approach towards the CFD simulation. At first a 3D Cavity model is developed in Solid Works. Further a grid was generated within the flow domain. The type of mesh used was unstructured mesh with 116128 nodes[3] and 3901305 elements[3]. While defining the fluid properties, the hot fluid was selected as hydrogen gas and the coolant was selected as liquid nitrogen. The heat transfer model used was set to the mode of Total Energy with the material defined as Steel. Further, the velocity of flow was varied and the Temperature contours were obtained for different velocities. A table extracted from the original work is attached below which shows the variation of Outlet temperature of coolant i.e. liquid nitrogen with the increase in velocity at constant inlet temperature of the coolant.

S.No.	Outer temp. of drum(K)	Inlet Coolant Temp. (K)	Outlet Temp. of Coolant(K)	Velocity (m/s)
1	40	65	51.67	0.2
2			52.15	0.21
3			52.98	0.22
4			53.21	0.23
5			53.68	0.24
6			54.12	0.25

Fig. 12. Outlet Temperature table at constant coolant inlet temperature for varying velocity[3]

It can be clearly observed that for a constant value of inlet coolant temperature, as the velocity of flow increases, the outlet temperature of the coolant increases which shows better reception of heat and enhanced heat transfer at higher velocities of flow.

V. COMPARISON OF RESULTS OBTAINED FROM EXPERIMENTATION AND CFD

Having seen the individual procedures to carry out the heat transfer analysis, the next step is to compare the results of experimentation and those of simulation and come up with a conclusion as to what is the correlation between the two methods and which one can be preferred as per the requirements. This section deals with the results and comparative discussion of the papers elaborated till now.

A. Comparative Study according to J. S. Jayakumar et al.[1]

On critical examination of the results as obtained from the experimentation and simulations, it has been observed that the results obtained from both the methods are in good accordance with each other. As stated earlier, correlations are devised for the variation of Nusselt number and Dean number as follows.

$$Nu = C*De^{m}*Pr^{n} \tag{2}$$

where C, m and n are constants which are to be determined. Using multiple regression analysis of MATLAB. Pr stands for Prandtl number.

Towards the end it has been mentioned that the methodology of heat transfer analysis was successfully validated with the experiments. It is further proposed to extending the CFD simulation to various Pitch circle diameters, tube pitch values and pipe diameters.

The paper concludes that CFD predictions match reasonably with the experimental values within the error limits and can be further extended for enhanced research.

B. Comparative Study according to M. Balachandran[2]

The most important part of the comparative study conducted in this paper is the variation of Overall Heat Transfer coefficient against mass flow rate at various temperatures and the variation of Dean number against mass flow rate at various temperatures.

Following graphs clearly distinguish between the results as obtained from the experimentation and those obtained from the CFD simulations. It can be observed that the overall value of heat transfer coefficient is lesser as obtained from CFD in most of the cases as there is certain convergence that has to be considered at which the simulation tends to bring out balance in the governing equations.

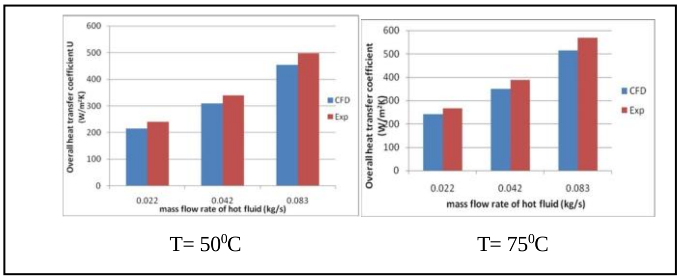

Fig. 13. Variation of Overall heat transfer coefficient against mass flow rate at constant temperature[2]

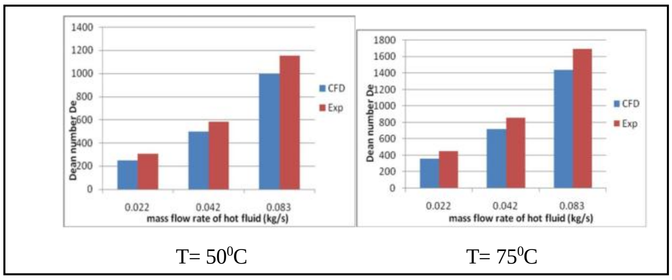

Fig. 14. Variation of Dean number against mass flow rate at constant temperature[2]

From Fig. 13. and Fig. 14. it can be clearly observed that the results as obtained from experimentation and from CFD are in good agreement with each other. The trend in variation of the values is similar. However certain correction factors are required to be used along with the simulation results in order to obtain more realistic values.

C. Comparative Study according to Revendra Verma et al.[3]

Two cases were considered in this work for the comparison of heat transfer. The first case deals with the variation of Heat Transfer Rate against velocity. The second case deals with the variation of Heat transfer coefficient against velocity. The graphs extracted from the original work are as follows.

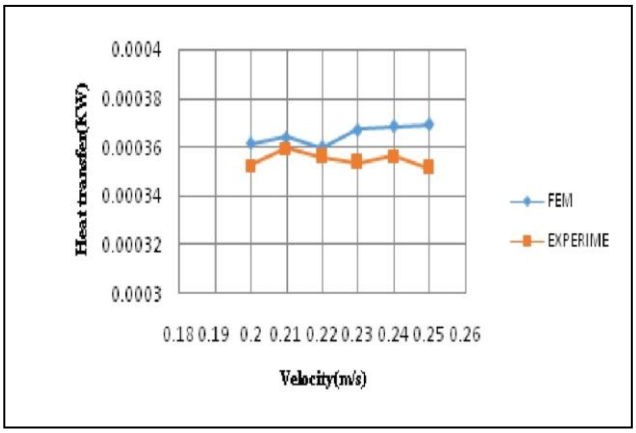

Fig. 15. Variation of Heat transfer vs. Velocity for experimentation and FEM results[3]

From Fig. 15. it can be observed that the as the velocity of flow increases, the heat transfer rate initially increases and decreases thereafter. Also that the trend of variation of values obtained from FEM approach is similar to that of experimental results. However at higher speeds there is a deviation observed in the values. This can be attributed to the convergence criteria in which the simulation does not further yield accurate results once the equation imbalances are resolved.

Similarly, variation of Heat transfer coefficient can be obtained as follows.

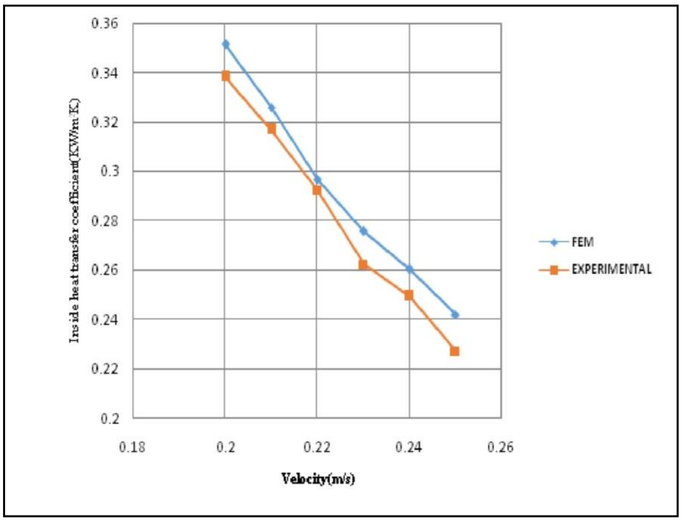

Fig. 16. Variation of Heat transfer coefficient vs. Velocity for experimentation and FEM results[3]

Fig. 16. depicts the comparison in an apt manner from which it can be inferred that the FEM values are found to be slightly higher than the Experimental values. However the trend in both the results is much more accordance with each other as compared to Fig. 15. The heat transfer coefficient decreases as the velocity increases. But the point of consideration is the correlation between the experimental and FEM values, which are found to be in good agreement.

VI. CONCLUSION

By carrying out the study of Heat transfer in a Helical Coil Heat Exchanger by experimentation and by simulations, the following can be concluded

1. For all the cases as considered in the study, it is found that the Experimental Values and the values of parameters obtained from CFD simulations are in good agreement with each other. There are certain boundary conditions at which the results are identical and there is no requirement of any correction factor.

2. The correlations established can prove to be an important tool for further research in the field of Helical coil heat exchangers. Since the experimental and simulation values are in good agreement with each other, a linear mapping can be established between the methods.

3. CFD simulations can be preferred over complex experimentation since the results obtained by CFD are within the desired range and more number of iterations can be undertaken easily with the help of CFD rather than

complex experimental procedures. Thus CFD can be used for fluid flow simulation through the helical coil heat exchanger to obtain the flow characteristics. CFD can prove to be a much more cost effective approach as compared to complex and expensive experimentation.

ACKNOWLEDGMENT

Author thanks Prof. Vivek V. Kulkarni, Assistant Professor, Department of Mechanical Engineering, KLS Gogte Institute of Technology, Belagavi, for his guidance and support. Author also thanks Visvesvaraya Technological University, Management and Principal of KLS Gogte Institute of Technology, Belagavi for their encouragement and constant support.

REFERENCES

[1] J S Jayakumar, S M Mahajani, J C Mandal, P K Vijayan, Rohidas Bhoi, " Experimental and CFD estimation of heat transfer in Helically coiled heat exchanger", Elsevier, Chemical Engineering Research and Design 86 (2008) pp 221-232

[2] M Balachandran, " Experimental and CFD study of a helically coiled heat exchanger using water as a fluid ", International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Vol 3, Issue-5, May-2015. pp 87-91

[3] Revendra Verma, Hitesh Kumar, Satyashree Ghodke, " A comparative analysis of Thermal characteristics between experimental values and FEM values in helically coiled heat exchanger " , International Journal of Engineering Research and Technology, ISSN: 2278-0181, Vol 2, Issue-11, November-2013. pp 3646-3651

[4] Madhuri Tayde, Jeevan Wankhade, Shylesha Channapattana, "Heat Transfer Analysis of a Helically coiled Heat Exchanger", International Journal of Innovative Research and Development, ISSN: 2278-0211, Vol 3, Issue-13, December 2014. pp 265- 268

[5] Mohammed Imran, Gaurav Tiwari, Alwar singh Yadav, "CFD Analysis of Heat Transfer Rate in Tube in Tube Helical Coil Heat Exchanger", International Journal of Innovative Science, Engineering & Technology, ISSN: 2348-7968, Vol 2, Issue-8, July 2015. pp 53- 57

[6] R. Beigzadeh, A Parvareh. M Rahimi, "Experimental and CFD study of the tube configuration effect on the shell side Thermal Performance in a shell and Helically Coiled Heat Exchanger", Iranian Journal of Chemical Engineering, Vol 12, No-2, Spring-2015. pp 13-25.