# Numerical Study of Heat Transfer Techniques in Curved Tubes: A Review

Anshuman Tiwari[1], Dr. Parag Mishra[2], Dr. Ajay Singh[3]
*M.E. Scholar, Mechanical Engineering Department, Radharaman Institute of Technology and Science Bhopal, Madhya Pradesh, India.[1]*
*Associate Professor, Mechanical Engineering Department, Radharaman Institute of Technology and Science Bhopal, Madhya Pradesh, India.[2]*
*Professor and Head, Mechanical Engineering Department, Radharaman Institute of Technology and Science Bhopal, Madhya Pradesh, India.[3]*
*Email: tiwarianshuman4@gmail.com[1], parag.mishra05@gmail.com[2], ajaysinghrits@gmail.com[3]*

**Abstract-** Heat exchangers have remarkable usage in chemical reactors, refrigeration & air conditioning systems, food processing, heat recovery processes, food industry, power industry and other energy concentrated areas. This article presents a literature review on numerical study of heat transfer properties of different type of flow in curved tubes in which the main categories of curved tubes i.e. helically curved tubes and spiral tubes are discussed. In all the literatures discussed below, numerical solution methods have played vital role to investigate these sophisticated model by saving time and money. The objective of paper is to study different methods employed in curved tube heat exchanger to improve heat transfer characteristics such as coefficient of heat transfer, Nusselt number etc. It is found that introduction of turbulence in fluid stream improves the characteristics of heat exchanger. Turbulence is imparted by many different ways such as by using multi tubes, by providing corrugation to the tubes, by the application of rib roughened surfaces, by using sinusoidal configuration on the tube surfaces etc. the future perspective of this literature study is to develop a tube of rectangular cross section and then twist the heat transfer surface of tube, this will reduce the effective-length of heat exchanger and will make it compact and turbulence of fluid flowing in the tube and heat transfer characteristics will be studied.

**Index Terms-** Helical Coiled tube, Heat Exchanger, CFD analysis, Heat transfer enhancement, Numerical techniques of heat transfer.

## 1. INTRODUCTION

Devices which exchanges heat between two fluids which are at different temperatures are commonly known as heat exchanger. Heat transfer occurs in three ways: conduction, convection and radiation, But Heat transfer mechanism in a heat exchanger usually involves convection in each fluid and conduction through the wall separating two fluids, numerous applications of heat exchangers in day to day life are found. Heat exchanger which consists of two concentric pipes of different diameters are called the double-pipe heat exchanger, these are the simplest type of heat exchanger. Parallel flow and counter flow are two types of flow arrangement which can be provided in a double-pipe heat exchanger, in parallel flow heat exchangers both the hot and cold fluids flows in the same direction also enter the heat exchanger at the same end, Whereas in counter flow heat exchanger, the hot and cold fluids flow in opposite directions and enter the heat exchanger at opposite ends. Among all the most common type of heat exchanger in industrial applications is the shell-and-tube heat exchanger, Shell-and-tube heat exchangers contain a large number of tubes (sometimes several hundred) packed in a shell with their axes parallel to that of the shell, another variation in heat exchanger design is Curved tube heat exchanger. Recent developments in design of heat exchangers, to full fill the demand of industries has led to the evolution of curved tube heat exchanger as these tubes has many advantages over a straight tube, such as (a)Heat transfer rate in helical coil are higher as compared to a straight tube heat exchanger. (b)Compact structure. It required small amount of floor area compared to other heat exchangers. (c)Larger heat transfer surface area. Basically there are three categories of curved tubes; helically coiled tubes, spirally coiled tubes, and other coiled tubes which are discussed in this paper. While numerical analysis is a broad term that describes various kinds of system modelling and analysis approaches in engineering, Computational Fluid Dynamics (CFD) has more specific uses. For studying the patterns of fluid flow in various systems such as pumps, the flow over airplanes and their parts, and the flow inside devices such as internal combustion engines CFD is proven a valuable kind of numerical analysis tool used. Analysis of complex flow geometries with the same ease is become possible due to cfd, as that faced while solving idealized simple problems using conventional methods. In CFD analysis geometry is generated according to problem then meshing is performed on the geometry, meshing plays very important role in CFD simulation because both computation time and accuracy of solution depend on the meshed structure. That is finer the grid is more accurate the result is. But the grid size should not made unnecessary finer so that the computation takes extra time for computing. Then solver solves governing equations and solution is generated. Furthermore, in real life tests are constricted to measure limited amount of quantities at a time, CFD simulations can provide predictions without being limited to a particular range of system parameters.. CFD software can be considered as a area of study combining fluid dynamics and numerical analysis.

## 2. LITRATURE REVIEW

H.F. Elattar et al. investigated on "Thermal and hydraulic numerical study for a novel multi tubes in tube helically coiled heat exchangers: Effects of operating/geometric parameters", in this paper the thermal and hydraulic analysis of turbulent flow conditions is carried out and for this analysis author used multiple tubes in one tube type heat exchanger with helical coiled configuration. Different operating parameter and geometrical variables of heat exchanger coil are analyzed and different heat transfer characteristics such as Nusselt numbers, thermal hydraulic index evaluated also both cold and hot water are taken into consideration for evaluation.. By this paper we come to know at Inner tubes number (N) =3 and coil inclination angles (ß) = 0° & 90° heat transfer coefficient is maximum, the pumping power increases as inner tubes numbers are increased from 1 to 5 and this pumping power variation is free from coil inclination angles. Moreover, the effectiveness of the coil is maximum when three inner tubes are used with coil inclination angle zero degree and ninety degree, [1]

J.I. Córcoles-Tendero et al. studied on "Numerical simulation of the heat transfer process in a corrugated tube", in this work heat transferred and friction factor are studied in a simple tube with spiral corrugation. In this study Prandtl numbers and Reynolds numbers are calculated in smooth tube and spirally corrugated tubes for turbulent flow conditions, analysis is carried out for smooth tube and spirally corrugated tube and comparison is made, by this we come to know that when compared with the experimental data an average Nusselt number comes within 17% of maximum relative error for the corrugated tube, and the Fanning factor differences are lower than 9%.[2]

Erfan Khodabandeh et al. analysed on "Application of nanofluid to improve the thermal performance of horizontal spiral coil utilized in solar ponds: Geometric study" In this paper horizontal spiral coil is studied in laminar flow for different cross section of tube numerically also for each cross section the effect of nanofluid concentrations are investigated numerically hybrid nanofluid of Water-graphene nanoplatelet/platinum have used as working fluid and three different volume concentration of nanofluid for different cross section rectangle, elliptic, trapezoid and circle are considered, by this paper we come to know that for elliptical section Nusselt number is maximum and average Nusselt number does not depends on shape of cross section of flow at lower mass flow rates, and also we come to know that when compare with curved corner surfaces then variation in velocity are found more for the angular cornered surfaces because of this behavior pressure loss in the pipe increases which also results in more pumping power.[3]

Zhouhang Li et al. studied on "Orientation effect in helical coils with smooth and rib-roughened wall: Toward improved gas heaters for supercritical carbon dioxide Rankine cycles", In this work, Horizontal and vertical arrangements of coil are made and investigation of these arrangement in smooth coil and in rib roughened coil is performed, by this we come to know that when the effect of buoyancy became sound then only effect of coil arrangement becomes evident. The performance of vertical coil was a bit better than the horizontal one also the downward orientation was slightly better than the upward orientation. Only a slight improvement was found in horizontal coil when hydraulic diameter of tube increases. Finally, rib roughened internal surface, a passive method of heat transfer enhancement was very effective for mixed convection. [4]

J. Hærvig et al. investigated on "On the fully-developed heat transfer enhancing flow field in sinusoidally, spirally corrugated tubes using computational fluid dynamics" in this paper 28 geometrically different sinusoidal, spirally corrugated tubes have investigated for heat transfer numerically. In order to vary corrugation the height and the length between two successive corrugated sections are varied, by this we come to know that as height of corrugation increases tangential velocity also increases and exceeds the axial velocity because of this at the center of corrugation pressure increases .[5]

Zhang Cancan et al. investigated on "Numerical investigation of heat transfer and pressure drop in helically coiled tube with spherical corrugation", in this paper spherically corrugated helical coiled tube were studied numerically for evaluation of heat transfer coefficient and pressure drop, by this we come to know that corrugation resulted in eddies which further increases the heat transfer rate, this structure destroyed the flow boundary layer which results in, increase in the turbulence of flow and improved the heat transfer mechanism. For the same condition heat transfer characteristics were found better in spherically corrugated tube then the simple helical tube. [6]

Hyder H. Balla studied on "Enhancement of heat transfer in six-start spirally corrugated tubes", in this work spiral shaped corrugation are provided in the tubes of heat exchanger, numerical and experimental both the studies are taken into consideration and analysis is performed for lower Reynold number also the results were compared, by this study we come to know that significant enhancement in the efficiency of heat transfer for spirally corrugated profile with geometric expression is occurred with significant increased friction factors.[7]

Yang Mei analyzed on, "Numerical study of the characteristic influence of the helically coiled tube on the heat transfer of carbon dioxide" in this paper heat transfer performance of supercritical carbon dioxide in horizontal helical tube heat exchanger is investigated numerically, Based on the findings of study, along the direction of flow, fluctuations are found in the heat transfer coefficient. Heat transfer coefficient increases as geometrical parameters such as the coil pitch(p), tube diameter (d), curvature radius (R) increases. The result also shows that on the heat transfer effect of the buoyant force are smaller than the effect of centrifugal one under the given condition [8]

Kishor Kumar Sahu et al investigated on, "Computational Fluid Dynamic Analysis for Optimization of Helical Coil Heat Exchanger" In this investigation heat transfer characteristics of heat exchanger with helical coil is determined by CFD software package. Experimental data is also collected and both experimental and CFD result were compared. Study is performed for the parameters like mass flow rate of fluids and pitch length of helical coil heat exchanger, by this we come to know that the CFD analysis results fairly matches with the Experimental Results. The comparison has proved that increase in helical coil pitch increases rate of heat transfer also increase in rate of heat transfer is seen when helical coil fluid velocity increases.[9]

R. Maradona et al has investigated on "CFD analysis of Heat transfer characteristics of Helical Coil Heat Exchangers" In this investigation, heat transfer rate is enhanced by the application of passive method . heat transfer rate is increased by using tube in tubes with helical configuration. Heat transfer characteristics are studied by varying Reynolds number (hot fluid) through cfd analysis Nusselt number and coefficient of heat transfer are estimated by result of velocity and temperature distribution in the heat exchanger. This helps to arrive at an optimum value of Reynolds number and Nusselt number for the corresponding tube-to-coil diameter ratios, by this we come to know that The simulation results indicate that the heat transfer characteristics such as Nusselt number and coefficient of heat transfer increases for the helical tube in tube than the straight tube in tube.[10]

Wei-Cheng Lin et al. investigated on "Numerical computations on flow and heat transfer characteristics of a helically coiled heat exchanger using different turbulence models", this paper presents thermal characteristics and hydraulic parameters heat exchangers with helically coiled tubes for high temperature gas reactors, water having low-temperature flows through the coiled tube and the high-temperature helium flows in shell side of heat exchanger, Three turbulence models are incorporated in the numerical simulations: realizable k–e, low-Reynolds k–e and Reynolds stress, by this we come to know that because of the centrifugal force coiled tube velocity and its gradient are more in the outside region of tube, the study further revealed that On the shell side, velocity is higher for fewer turns (e.g. seven turns) while in different turbulence models, shapes and sizes are different for different wake regions. On the surfaces located on periphery of the tube cross sections, the local heat flux varies for coiled heat exchangers with different magnitudes in different turbulence models. in comparison with the other two models the heat exchanger characteristics were at best when k–e turbulence model is used with lower Reynold numbers.[11]

Marco Colombo et al. studied on "CFD study of an air-water flow inside helically coiled pipes", in this paper air water mixture is flowed inside helically coiled pipes and studied with CFD, in which the adiabatic flow conditions are evaluated in ANSYS FLUENT, by this we come to know that water circulates in outer region of tube wall by the application of centrifugal force because water is the heavier phase, but air remains in inner space of the tube. Hence mixture velocity is maximum near air have density lower than water hence air flows faster than water also separation and stratification occurs between phases in helical tube, reason for this is existence of gravitational force as well as of centrifugal force. The one disadvantage was found int the model that Dispersion of secondary phase due to gravitational and centrifugal force limits the application of model discussed in the study.. Therefore, improvement in the model present in the study is required which can take care separation of phase with more accurate and advanced formulation.[12]

Zhongyuan Shi et al. analysed on "Numerical investigation of developing convective heat transfer in a rotating helical pipe", in this paper helical pipe is rotated to improve centrifugal effect inside fluid and characteristics of heat transfer are investigated numerically Enhancement of local heat transfer rate in two condition of rotation I.e. clockwise and counterclockwise rotation are analyzed, by this paper we come to know that thickness of boundary layer in laminar region develops faster For clockwise rotational cases, than in stationary cases, Rate of heat transfer increase for counter rotation but decreases for co rotation although thermal boundary layer becomes unstable in both rotational cases.[13] András Zachár investigated on "Investigation of a new tube-in-tube helical flow distributor design to improve temperature stratification inside hot water storage tanks operated with coiled-tube heat exchangers", in this investigation cylinder of hot water is taken into consideration with a tube-in-tube helical heat exchanger, Steady process and transient process of heat transfer are investigated also storage tank temperature variation and developed flow inside tank are studied and Comparison It is also found that local Nusselt number oscillates slightly along the average Nusselt number for outer side of tube but tendency of this local Nusselt number is found constant along tube of heat exchanger.[14]

K. Narrein et al. studied on "Influence of nanofluids and rotation on helically coiled tube heat exchanger Performance", in this paper numerical investigation of helically tube heat exchanger with a number of nanofluids are considered. The characteristics of heat exchanger are evaluated for different concentration of nanoparticles and for different diameter of particle also base fluid types (water, ethylene glycol, engine oil) are studied comprehensively for evaluation of the heat transfer properties and for characteristics of fluid flow, by this we come to know that thermal characteristics of heat exchanger with helical coils improves by the use of nanofluids although pressure drop also increases slightly. It is also found that for Copper oxide –water nanofluid the Nusselt number is maximum.[15]

## 3. METHODOLOGY

In above discussion numerical methods are constrained to CFD analysis of different curved tube heat exchanger. At first flow parameters of a problem are introduced into the CFD software and this is provided by a user friendly then then Grid generation takes place i.e. the grids are small division of the cell into a non-overlapping sub-domains also known as mesh or elements. Basic concept of CFD methods is to find values of different flow quantities at thousands of different nodes in the system, which are also knowns as grid points. Now physical problem that need to be Simulated is selected and properties of fluid used in problem, are provided with of suitable boundary conditions specification. Now different flow variables such as pressure drop, temperature variations, velocity variations etc. are calculated. The reliability and precision of a CFD solution depends on cell numbers in the mesh grids. Generally larger cell numbers leads to the better accuracy of the solution. Although non uniform meshes gives best and optimum results because finer meshing is provided for complex geometry, in regions of large variations and meshing is coarser in regions with relatively simple geometry. Now this meshed structure is transferred in to the solver. Solver sets up the equations which are selected according to the boundary conditions and solves them to compute the flow field. Basically The governing differential equation for the fluid flow is given by conservation of mass equation, conservation of momentum equation and conservation of energy equation. These equations are imparted in solver, which are defined as the equations shown below:

$$\nabla\rho.\mathbf{u} = 0$$

$$\nabla.(\rho\mathbf{u} \times \mathbf{u}) = -\nabla P + \nabla.[\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)]$$

$$\rho C_p\mathbf{u}.\nabla T = k\nabla^2 T$$

where ρ is the fluid density, u is the velocity, P is the pressure, μ is the dynamic viscosity,  is the specific heat, k represents thermal conductivity and variable T represents the temperature.

When solver solves all the required equation then it generates results. The results are then shown by graphical formats which gives us idea of variation in different flow and heat transfer parameters such as friction coefficient, coefficient of heat transfer and Nusselt number etc. along the geometry of the problem. The latest CFD software are now coming with many adaptable tools which can show the results in various pictorial form with different coloring such as colorful display of geometry, Vector plots, temperature contour plots, Particle tracking, streamlines path lines etc. Now a days these facilities may also include animation for dynamic result display.

## 4. Conclusion

The curved tubes can be classified broadly in three categories: helical tubes, spiral tubes and other curved tubes with two phase flow. It is seen in different analysis that curved tube provides greater characteristics of heat transfer in comparison to straight tubes, the reason behind this is centrifugal forced produced due to curved tubes this effect also contributes in better mixing inside fluid. For other curved tubes, Characteristics of heat transfer of 2-phase flow and of one phase flow are discussed numerically, But in this paper only one work is presented for 2-phase flow. This present study is emphasized on numerical method of heat transfer because CFD simulations possess great capabilities which helps us in saving the time of analysis for the heat transfer process. Analysis through CFD software are faster and economic than traditional methods of testing and data acquiring. Furthermore, in real life tests are constricted to measure limited amount of quantities at a time but CFD simulations can provide predictions without being limited to a particular range of system parameters, hence large volume of the results from the parametric studies can be generated.

**REFERENCES**

[1] H.F. Elattara, A. Foudab, S.A. Nadaa, H.A. Refaeyc, A. Al-Zahranid “Thermal and hydraulic numerical study for a novel multi tubes in tube helically coiled heat exchangers: Effects of operating/geometric parameters” International Journal of Thermal Sciences, vol-28, **(2018),** pp.70–83.

[2] J.I. Córcoles-Tenderoa, J.F. Belmontea, A.E. Molinaa, J.A. Almendros-Ibáñeza, “Numerical simulation of the heat transfer process in a corrugated tube” International Journal of Thermal Sciences ,vol-126, **(2018),** pp.125–136 .

[3] Erfan Khodabandeh, Mohammad Reza Safaei, Soheil Akbari, Omid Ali Akbari, Abdullah A.A.A. Alrashed, “Application of nanofluid to improve the thermal performance of horizontal spiral coil utilized in solar ponds: Geometric study” Renewable Energy, vol-122, **(2018),** pp.1-16.

[4] Zhouhang Li, Dapeng Bi, Yuling Zhai, Kongzhai Li, Hua Wang, Junfu Lu, “Orientation effect in helical coils with smooth and rib-roughened wall: Toward improved gas heaters for supercritical carbon dioxide Rankine cycles” Energy, vol-140, **(2017),** pp.530-545.

[5] J. Hærvig, K. Sørensen, T.J. Condra, “On the fully-developed heat transfer enhancing flow field in sinusoidally, spirally corrugated tubes using computational fluid dynamics” International Journal of Heat and Mass Transfer, vol-106, **(2017),** pp.1051–1062.

[6] Zhang Cancan, Xiang Sa, Wang Dingbiao, Peng Xu, Han Yong, “Numerical investigation of heat transfer and pressure drop in helically coiled tube with spherical corrugation”, International Journal of Heat and Mass Transfer, vol-113, **(2017),** pp.332–341.

[7] Hyder H. Balla, “Enhancement of heat transfer in six-start spirally corrugated tubes”, Case Studies in Thermal Engineering, vol-9, **(2017),** pp.79–89.

[8] Yang Mei, “Numerical study of the characteristic influence of the helically coiled tube on the heat transfer of carbon dioxide”, Applied Thermal Engineering, vol-102, **(2016),** pp.882–896.

[9] Kishore Kumar sahu, N.K.saikhedkar, “Computational Fluid Dynamic Analysis for Optimization of Helical Coil Heat Exchanger”, International Journal of Engineering Research & Technology, vol-5, **(2016).**

[10] R. Maradona and S. Rajkumar, “CFD analysis of heat transfer characteristics of helical coil heat exchangers”, Applied Mechanics and Materials, vol-787,**(2015),** pp.172-176

[11] Wei-Cheng Lina, Yuh-Ming Fernga, Ching-Chang Chienga, “Numerical computations on flow and heat transfer characteristics of a helically coiled heat exchanger using different turbulence models”, Nuclear Engineering and Design, vol-263, **(2013),** pp.77– 86.

[12] Marco Colombo, Antonio Cammi, Gael Raymond Guedon, Fabio Inzoli, Marco Enrico Ricotti, “CFD study of an air water flow inside helically coiled pipes”, Progress in Nuclear Energy, vol-85, **(2015),** pp.462-472.

[13] Zhongyuan Shi, Dong Tao, “Numerical investigation of developing convective heat transfer in a rotating helical pipe”, International Communications in Heat and Mass Transfer, vol-57, **(2014),** pp.170–182.

[14] András Zachár, “Investigation of a new tube-in-tube helical flow distributor design to improve temperature stratification inside hot water storage tanks operated with coiled-tube heat exchangers”, International Journal of Heat and Mass Transfer, vol- 63, **(2013),** pp.150–161.

[15] K. Narreina, H.A. Mohammed, “Influence of nanofluids and rotation on helically coiled tube heat exchanger performance”, Thermochimica Acta, vol-564, **(2013),** pp.13– 23.