INTERNATIONAL JOURNAL OF SCIENTIFIC PROGRESS AND RESEARCH (IJSPR) ISSN: 2349-4689
Issue 164, Volume 64, Number 01, October 2019

The Comparison of Rate of Heat Transfer For Helical Coil Heat Exchanger at Multiple Cross-Section using CFD

Ankit Kumar Maurya[1], Sachin Baraskar[2], Anil Verma[3], Prof. G.R. Selokar[4]

[1]*Research Scholar, Department of Mechanical Engineering, School of Engineering, SSSUTMS, Sehore*

[2]*Assistant Professor, Department of Mechanical Engineering, School of Engineering, SSSUTMS, Sehore*

[3]*Head of the Department of Mechanical Engineering, School of Engineering, SSSUTMS, Sehore*

[4]*Registrtar, SSSUTMS, Sehore*

Abstract - Enhancing the heat transfer by the use of helical coils has been studied and researched by many researchers, because the fluid dynamics inside the pipes of a helical coil heat exchanger offer certain advantages over the straight tubes, shell and tube type heat exchanger, in terms of better heat transfer and mass transfer coefficients. Various configurations of coil structure are possible, and the configuration in which there is a series of vertically stacked helically coiled tubes is the most common type. This configuration offers a high compact structure and a high overall heat transfer coefficient; hence helical coil heat exchangers are widely used in industrial applications such as power sector, nuclear power generation, food processing plants, heat recovery systems, refrigeration, food industry, industrial HVACs etc. Convective heat transfer between a surface and the surrounding fluid in a heat exchanger has been a major issue and a topic of study in the recent years. In this particular study, an attempt has been made to analyse the effect of Rate of heat transform from a three different cross-sections on the helical tube, where the hot fluid flowing in tube and outer surface of tube having less temperature then hot fluid. Different cross-sections of the pipes are taken into consideration while running the analysis. The contours of pressure, temperature, velocity magnitude and the mass transfer rate from the tubes were calculated and plotted using ANSYS FLUENT 14.5 where the governing equations of mass, momentum and energy transfer were solved simultaneously, using the k-$\mathcal{E}$ two equations turbulence model. The fluid flowing through the tube was taken as water.

1.0 INTRODUCTION

Heat exchange among flowing fluids is one of the mainly physical process of concern, and a selection of heat exchangers are used in unlike type of installations, as in process industries, compact heat exchangers nuclear power unit, Heating Ventilation Air Conditions, food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get a proficient way of heat transfer from one fluid to another, by direct contact or by indirect contact. The heat transfer occurs by three principles: conduction, convection and radiation. In a heat exchanger the heat transfer through radiation is not taken into consideration as it is small in comparison to conduction and convection. Conduction takes place when the heat flow from the high temperature fluid flows through the adjacent solid wall. The conductive heat transfer can be maximized by taking a least thickness of wall of a highly conductive material. But convection is acting the key role in the performance of a heat exchanger. Forced convection in a heat exchanger transfers the heat from one flowing stream to other stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it.

Heat exchangers are one of the mostly used equipment in the process industries. Heat exchangers are used to transfer heat between two process streams. One can realize their usage that any process which involve cooling, heating, condensation, boiling or evaporation will require a heat exchanger for these purpose. Process fluids, usually are heated or cooled before the process or undergo a phase change. Different heat exchangers are named according to their application. For example, heat exchangers being used to condense are known as condensers, similarly heat exchanger for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transfer using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements.

1.1 Heat exchangers are of two types:- (i) where both media between which heat is exchanged are in direct contact with each other is Direct contact heat exchanger.

(ii)Where both media are separated by a wall through which heat is transferred so that they never mix, indirect contact heat exchanger.

A typical heat exchanger, usually for higher pressure applications up to 552 bars, is the shell and tube heat exchanger. Shell and tube type heat exchanger, indirect contact type heat exchanger. It consists of a series of tubes, through which one of the fluids runs. The shell is the container for the shell fluid. Generally, it is cylindrical in shape with a circular cross section, although shells of different shape are used in specific applications. For this particular study shell is considered, which a one pass shell is generally. A shell is the most commonly used due to its low cost and simplicity, and has the highest log-mean temperature-difference (LMTD) correction factor. Although the tubes may have single or multiple passes, there is one pass on the shell side, while the other fluid flows within the shell over the tubes to be heated or cooled. The tube side and shell side fluids are separated by a tube sheet.

1.2 CLASSIFICATION OF HEAT EXCHANGER'S:

a) Tubular Heat Exchanger
b) Double Pipe Heat Exchanger
c) Shell and Tube Heat Exchanger
d) Helical Tube Heat Exchanger

2.0 COMPUTATIONAL FLUID DYNAMICS (CFD)

CFD is a sophisticated computationally-based design and analysis technique. CFD software gives you the power to simulate flows of gases and liquids, heat and mass transfer, moving bodies, multiphase physics, chemical reaction, fluid-structure interaction and acoustics through computer modeling. This software can also build a virtual prototype of the system or device before can be apply to real-world physics and chemistry to the model, and the software will provide with images and data, which predict the performance of that design. Computational fluid dynamics (CFD) is useful in a wide variety of applications and use in industry. CFD is one of the branches of fluid mechanics that uses numerical methods and algorithm can be used to solve and analyze problems that involve fluid flows and also simulate the flow over a piping, vehicle or machinery. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic and turbulent flows are on going research. Onwards the aerospace industry has integrated CFD techniques into the design, R & D and manufacture of aircraft and jet engines. More recently the methods have been applied to the design of internal combustion engine, combustion chambers of gas turbine and furnaces also fluid flows and heat transfer in heat exchanger (Figure 1). Furthermore, motor vehicle manufactures now routinely predict drag forces, underbonnet air flows and surrounding car environment with CFD. Increasingly CFD is becoming a vital component in the design of industrial products and processes.

2.1 APPLICATION OF CFD:

CFD not just spans on chemical industry, but a wide range of industrial and nonindustrial application areas which is in below:

a) Aerodynamics of aircraft and vehicle.
b) Combustion in IC engines and gas turbine in power plant.
c) Loads on offshore structure in marine engineering.
d) Blood flows through arteries and vein in biomedical engineering.
e) Weather prediction in meteorology.
f) Flow inside rotating passages and diffusers in turbo-machinery.
g) External and internal environment of buildings like wind loading and heating or
h) Ventilation system.
i) Mixing and separation or polymer moldings in chemical process engineering.
j) Distribution of pollutants and effluent in environmental engineering.

2.2 CFD COMPUTATIONAL TOOL:

This section describes about the CFD tools which are required for the CFD analysis of the problem. There are the three main elements for the processing of the CFD simulations: the pre-processor, solver, and post-processor are described.

a. **Pre-processor:** A pre-processor is defined to the geometry of the problem. It is fixed to the domain for the computational analysis and then generates the mesh of the geometry. Here also set the nomenclature like inlet, outlet, and wall etc.

Generally, the finer the mesh of the geometry in the CFD analysis gives more accurate solution. Fineness of the grid also determines the computer hardware and more time needed for the calculations.

b.Solver: - In the solver processor the calculations is done by using the numerical solution methods. There are the many numerical methods which are used for the calculations for example: - The finite element method (FEM), finite volume method (FVM), the finite difference method and the spectral method. Most of them in CFD codes use finite volume method. In this project the finite volume method is used. The solver performs the following steps:-

- Firstly the fluid flow equations are integrated over the control volume
- Then these integral equations are discredited (producing algebraic equations through converting of the integral fluid flow equations),
- And then finally an iterative method is used to solve the algebraic equations.
- Pressure based coupled solution method CFD code is used for solving the simulations in this project.

c. **Post-Processor**: The post-processor is provided to the visualisation of the results of the solutions. It includes the capability to display the geometry and mesh also. And in this processor we can create the vectors, contours, and 2D and 3D surface plots of the problem solutions. Here the model also can be manipulated. By this process we can also see the animation of the problem.

2.3 Problem-solving with CFD:

The decisions which should be formed before setting of CFD code

Some of the decisions to be made can include.

- Decide the problem is 2D or 3D.
- Generate the geometry.
- Do the meshing of the geometry; give the size of the elements.
- Decide the type of boundary conditions to use in the analysis.
- Confirm the type of solution method either pressure based or density based.
- Initialized the problem
- Give the iteration number
- Converged to the problem
- Calculate the results.

2.4 CFD GOVERNING EQUATION:

In this section summarisation of the governing equations are given that are used in to solve the fluid flow and heat transfer mathematically. This solution is based on the principle of conservation of mass, momentum, and energy. CFD Computational equations are given below:-

This equation is a mass conservation equation:-

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0.$$

This is the rate of change of momentum equation:-

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_j}{\partial x_i}\right) - \frac{\partial p}{\partial x_j}.$$

This is the Energy equation;-

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_i}\left(\frac{k}{C_p}\frac{\partial u_j}{\partial x_i}\right).$$

Where velocity vector **u** (with components of the velocities *u*, *v*, and *w* in the direction of *x*, *y*, and *z*), pressure P, density ρ, viscosity μ, temperature T, and heat conductivity k. The changes in these given fluid properties can occur over space because this project works on only steady state condition.

3.0 LITERATURE REVIEW:

[1] *Miyer Valdes et al. may 2019* The purpose of this research is to evaluate the effect of twist in the internal tube in a tube-in-tube helical heat exchanger keeping constant one type of ridges. To meet this goal, a Computational Fluid Dynamic (CFD) model was carried out. The effects of the fluid flow rate on the heat transfer were studied in the internal and annular flow. A commercial CFD package was used to predict the flow and thermal development in a tube-in-tube helical heat exchanger. The simulations were carried out in counter-flow mode operation with hot fluid in the internal tube side and cold fluids in the annular flow. The internal tube was modified with a double passive technique to provide high turbulence in the outer region. The numerical results agree with the reported data, the use of only one passive technique in the internal tube increases the heat transfer up to 28.8% compared to smooth tube.

[2] *Nidhi R. Singh et al. 2018*In the present study effect of steam temperature on heat transfer coefficient is studied using ANSYS Fluent (2015). In this study CFD analysis is performed to validate experimental data of condensation heat transfer coefficient. Steam temperature is varied from1030C -1150C and its effect on heat transfer coefficient is done. Three helical coils having different coil diameter is used. It is observed that as saturation temperature of steam increases heat transfer coefficient increases and as coil diameter increases heat transfer coefficient decreases and the percentage of error is within 9-15%

[1] *R.Berlin Samuel Raj-March 2017* The study deals with the CFD simulation of the shell and tube heat exchanger by using a straight tube and a helical tube and comparing both of its performance based on heat transfer. Straight tube heat exchanger (STHE) have large heat transfer surface area-to-volume ratios to provide high heat transfer efficiency which are mostly used in industries. But the Helical Coil Heat Exchanger (HCHE) is also widely used in industrial applications because it can accommodate greater heat transfer area in a less space, with higher heat transfer coefficients. So both of the STHE and HCHE performances are compared to prove that HCHE is better in usage and the analysis is done using ANSYS Fluent 14.5 software. The model is created using CATIA V5 software.

[1] *Syed Mohammed Arif et al.*The purpose of this research is to study the effect of coil diameter and inlet temperature of steam on the heat transfer coefficient in the helical coil. The study was based on fluid to fluid heat transfer. CFD analysis was carried out for two different mass flow rate of water. Heat transfer coefficient was calculated for various inlet temperatures ranging from 100-105°C. A CFD simulation was carried out for three different coils. The results show that there is an increase in heat transfer coefficient with the increase in inlet temperature of steam.

[2] *Daniel Flórez-Orrego,* Walter Arias, Diego Lopez and Hector Velasquez [2012] have worked on the single phase cone shaped helical coil heat exchanger. The study showed the flow and the heat transfer in the heat exchanger. An empirical correlation was proposed from the experimental data for the average Nusselt number and a deviation of 23% was found. For the cone shaped helical coils an appreciable inclination of the velocity vector components in the secondary flow was seen, even though the contours of velocity were similar. The study showed that some of the deviations and errors were due to the non-uniform flame radiation and condensed combustion products which modified the conditions of the constant wall heat flux assumptions. The correlations for the Nusselt number values were not totally reliable. There was no proper data available for the effect of the taper in the local Nusselt number and also the effect of curvature ratio, vertical position and the pitch of the heat exchanger.

[2] *J. S. Jayakumar [2008]* observed that the use of constant values for the transfer and thermal properties of the fluid resulted in inaccurate heat transfer coefficients. Based on the CFD analysis results a correlation was developed in order to evaluate the heat transfer coefficient of the coil. In this study, analysis was done for both the constant wall temperature and constant wall heat flux boundary conditions. The Nusselt numbers that were obtained were found to be highest on the outer coil and lowest in the inner side. Various numerical analyses were done so as to relate the coil parameters to heat transfer. The coil parameters like the diameters of the pipes, the Pitch Circle Diameters have significant effect on the heat transfer and the effect of the pitch is negligible.

[3] *Timothy J. Rennie [2004]* studied the heat transfer characteristics of a double pipe helical heat exchanger for both counter and parallel flow. Both the boundary conditions of constant heat flux and constant wall temperature were taken. The study showed that the results from the simulations were within the range of the pre-obtained results. For dean numbers ranging from 38 to 350 the overall heat transfer coefficients were determined. The results showed that the overall heat transfer coefficients varied directly with the inner dean number but the fluid flow conditions in the outer pipe had a major contribution on the overall heat transfer coefficient. The study showed that during the design of a double pipe helical heat exchanger the design of the outré pipe should get the highest priority in order to get a higher overall heat transfer coefficient.

[4] *J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, RohidasBhoi [2008]* studied the constant thermal and transport properties of the heat transfer medium and their effect on the prediction of heat transfer coefficients. Arbitrary boundary conditions were not applicable for the determination of heat transfer for a fluid-to-fluid heat exchanger. An experimental setup was made for studying the heat transfer and also CFD was used for the simulation of the heat transfer. The CFD simulation results were reasonably well within the range of the experimental results. Based on both the experimental and simulation results a correlation was established for the inner heat transfer coefficient.

[5] *Usman Ur Rehman [2011]* studied the heat transfer and flow distribution in a shell and tube heat exchanger and compared them with the experimental results. The model showed an average error of around 20% in the heat transfer and the pressure difference. The study showed that the symmetry of the plane assumption worked well for the length of the heat exchanger but not in the outlet and inlet regions. The model could be improved by using Reynolds Stress models instead of k-ε models. The heat transfer was found to be on the lower side as there was not much interaction between the fluids. The design could be improved by improving the cross flow regions instead of the parallel flow.

4.0 RESEARCH METHODS:

Three different cross-sections of helical coil heat exchanger have been analysed

4.1 Domain Properties:

The mesh is converted into the required format and is imported to Ansys CFX-14.5. The properties of domains and fluid are defined in ANSYS CFX Pre 14.5. The summary of Flow domain properties are given in Table 4.4.

Table - 4.1.1: Helical Coil Heat Exchanger Domain Properties

Location	Helical Coil Heat Exchanger
Domain Type	Fluid Domain
Fluid List	Water
Coordinate Frame	Coord 0
Reference Pressure	0 [atm]
Buoyancy Option	Not Buoyant
Domain Motion Option	Stationary
Mesh Deformation Option	None
Turbulence Model	k-Epsilon
Turbulence Wall Function	Scalable
Density of Water	998.2 kg/m^3
Combustion	None
Thermal Radiation	None

4.2 Boundary Conditions:

Boundary conditions are used according to need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a cross flow with one tube so there are respective inlet and outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for wall. The details about all boundary conditions can be seen in the table below:

Table 4.2.1: Boundary Conditions

-	Boundary condition type	Velocity Magnitude	Temperature
Inlet	Velocity Inlet	0.5 (m/s)	350
Outlet	Pressure Outlet	0.5 (Pascal)	300

4.3 Geometrical Dimensions:

Table 4.3.1: Geometrical Dimensioning

S.No.	Geometry	Dimension (mm)	Area (mm^2)
1	Circular	d= 12, t=1	113
2	Rectangular	l=11.3, b= 10	113
3	Triangular	H=15.03, B=15.03	113

4.4 Post Processing:

3D-Streamlines of velocity and pressure contours starting from inlet of heat exchanger to outlet of heat exchanger can be seen. Using the function calculator in tools average values of various parameters like the pressure, velocity, mass flow rate, area at the boundaries and the required domain can be found out for calculation of performance parameters.

5.0 RESULTS:

5.1 General:

Three different cross-sections of helical coil heat exchanger have been analysed and their mass flow rates variations has been evaluated at constant parameters. Only inlet velocity has been taken into consideration with value of 0.5 m/s.

Each cross-section has been taken a area of fluid type and the fluid which is taken, is water with reference pressure as 1 atmospheric. Heat transfer option has been set to none, fluid temperature 77°C and turbulence model κ-ϵ has been taken for the domain. We know that the Density of water is defined as 998.2 kg/$m^3$and kinematic viscosity as 0.8926 X $10^{-6} m^2/s$. The outlet of heat exchanger reference pressure is set equal to 0.5 Pascal (gauge). Consider the inlet and outlet area of the heat exchanger is $1.13x10^{-5}m^2$. Surface of heat exchanger is taken to be smooth and no slip condition is taken.

High resolution advection scheme with second order upwind numeric for 200 iterations were given. The timescale control was set to Auto Timescale. The RMS residual target was set to $1x10^{-5}$for termination of the calculations. After completion of the iterations results are obtained. The variation of the pressure and velocity using pressure contours and velocity streamlines respectively on the surface of the draft tube could be observed.

6.0 CONCLUSION

The computation and comparison of different mass flow rate of various geometric configurations using CFD will help to optimize shape of cross-sections of helical heat exchanger. In this dissertation Computational Fluid Dynamics (CFD) approach has been used to predict the optimum cross-section of different cross-section of helical coil heat exchanger

REFERENCES

[1] CFD Analysis of Heat Transfer in Helical Coil by Syed Mohammed Arif, Prof. Rashed Ali , Dr. Dhanraj P. Tambuskar3, Prof. DivyaPadmanabhan Mar -2017

[2] Heat Transfer Analysis Of A Straight Tube And Helical Coil Heat Exchanger Using Cfd by R.Berlin Samuel Raj-March 2017

[3] Computational Model to Evaluate the Effect of Passive Techniques in Tube-In-Tube Helical Heat Exchanger by Miyer Valdes, Juan G. Ardila Dario Colorado and Beatris A. Escobedo-Trujillo 3-18 May 2019

[4] CFD Analysis of Condensation Heat Transfer in Helical Coil Heat Exchanger by Nidhi R. Singh, Rashed Ali-2018

[5] Experimental and CFD study of a single phase cone-shaped helical coiled heat exchanger: an empirical correlation by Daniel Flórez-Orrego, ECOS June 26-29, 2012.

[6] Helically Coiled Heat Exchangers by J.S.Jayakumar 2008.

[7] Numerical and Experimental Studies of a Double pipe Helical Heat Exchanger by Timothy John Rennie, Dept. of Bio-resource Engg. McGill University, Montreal August 2004.

[8] Experimental and CFD estimation of heat transfer in helically coiled heat exchangers by J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, P.K. Vijayan, and RohidasBhoi, 2008, Chemical Engg Research and Design 221-232.

[9] Heat Transfer Optimization of Shell-and-Tube Heat Exchanger through CFD Studies by Usman Ur Rehman, 2011, Chalmers University of Technology.

[10] Structural and Thermal Analysis of Heat Exchanger with Tubes of Elliptical Shape by Nawras H. MostafaQusay R. Al-Hagag, IASJ, 2012,Vol-8 Issue-3.

[11] Design and Thermal Evaluation of Shell and Helical Coil Heat Exchanger by Amitkumar S. Puttewar, A.M. Andhare, International Journal of Research in Engineering and Technology Volume: 04 Issue: 01 ,Jan-2015.

[12] "Analysis of Tub- in-Tube Helical Coil & Straight Tube Heat Exchanger" by P. P. Gavade, S.S. Malgave, D.D. Patil, H.S. Bhore, V. V. Wadkar, Journal of Mechanical Engineering and Technology (JMET) Volume 3, Issue 2, July-Dec 2015, pp. 14-19.

[13] "Modeling of a Helical Coil Heat Exchanger for Sodium Alanate Based On-board Hydrogen Storage System" by MandhapatiRaju, Sudarshan Kumar, Proceedings of the COMSOL Conference 2010 Boston.

[14] "An Experimental Performance Investigation of Counter Flow Helical Coiled Heat Exchanger" by Satish. B. Ingle, Snehal S. Borkar, Volume 4 (8): 260-271, IJPRET, 2016.

[15] "Fabrication and Analysis of Counter Flow Helical Coil Heat Exchanger" by SwapnilAhire, PurushottamShelke, BhalchandraShinde, NileshTotala International Journal of Engineering Trends and Technology (IJETT) Volume 15 Number 5 – Sep 2014.