CFD Analysis of Helical Coil Heat Exchanger

Palash Goya[1], Nitin Kushwah[2], Rahul Dabi[3]

[1,2,3]Assistant Professor
Mechanical Engineering Department
SVVV, Indore, India

***Abstract* - A helical coil heat exchangers are widely used in industrial applications such as chemical and food industries, power production, electronics, environmental engineering, manufacturing industry, air conditioning, waste heat recovery etc. over the straight and shell type heat exchangers due to due to its compact structure, larger heat transfer area and higher heat transfer capability etc.**

In the present study a tube in tube helically coiled heat exchanger has been modeled for fluid flow and an attempt has been made to analyze heat transfer characteristics for different fluid flow rates in the inner as well as outer tube. Copper was chosen as the as metal for the construction of the helical tube. The fluid flowing through the tube was taken as water. The total heat transfer rate from the wall of the tube were calculated and plotted using ANSYS FLUENT 16.0

***Index terms:* - heat exchanger, Heat Transfer, Helical Coil, and CFD.**

I. INTRODUCTION

A heat exchanger is a device that is used to transfer thermal energy between two or more fluids, and a variety of heat exchangers are used in different type of in engineering application, the purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another. The tube in tube type heat exchanger consists of one pipe placed concentrically inside another pipe having a greater diameter. It can be arranged in a lot of series and parallel configurations to meet the different heat transfer requirements. It is found that the heat transfer in helical circular tubes is higher as compared to Straight tube due to their shape. Helical coils offer advantageous due to their compactness and increased heat transfer coefficient. The increased heat transfer coefficients are a consequence of the curvature of the coil, which induces centrifugal forces to act on the moving fluid, resulting in the development of secondary flow. Due to the curvature effect, the fluid streams in the outer side of the pipe moves faster than the fluid streams in the inner side of the pipe.

Objectives of this project work.

- Numerical analysis was carried out to determine the heat transfer characteristics for a double-pipe helical heat exchanger for the different parameters and also to determine the fluid flow pattern in helical coiled heat exchanger. The objective of the project is to obtain a better understanding into the heat transfer process when a fluid flows in a helically coiled tube.

II. LITERATURE REVIEW

Kharat et al. (2009) had done the experiments to study the heat transfer rate on a concentric helical coil heat exchanger and develop the correlation for heat transfer coefficient by using CFD simulation and the experimental study. They found that the heat transfer coefficient decreases with the increase in coil gap. With increase in tube diameter the heat transfer coefficient increases. [1] Timothy J. Rennie studied the heat transfer characteristics of a double pipe helical heat exchanger for both counter and parallel flow. They had found results 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. [2] J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, Rohidas Bhoi had done experiments to study the constant thermal and transport properties of the heat transfer medium and their effect on the prediction of heat transfer coefficients. An experimental setup was made for studying the heat transfer and also CFD was used for the simulation of the heat transfer. Based on both the experimental and simulation results a correlation was established for the inner heat transfer coefficient [3] Ferng (2012) carried out a numerical investigation and found that, effects of different Dean (De) number and pitch size on the thermal hydraulic characteristics in a helically coil-tube heat exchanger. The CFD methodology in this paper investigates the flow and heat transfer phenomena in a helically coil-tube heat exchanger. Effects of inlet De number and pitch size on these characteristics have been also studied. [4].

III. METHODOLOGY:

CFD simulation of the system starts with the construction of desired 3D geometry and mesh. Meshing is the discretization of the domain into small volumes where the partial differential equations (PDEs) are then applied to each cell. Therefore, each cell now becomes a domain.

The simulation procedure has following steps:

- Geometry
- Mesh
- Solution

GEOMETRY

Heat exchanger is built in the ANSYS workbench design module. First, the fluid flow (fluent) module from the workbench is selected. The design modeler opens as a new window as the geometry is double clicked. Out of 3 planes, i.e., XY-plane, YZ-plane and ZX-plane, the XY-plane is selected for the first sketch. A line for the height of the helical structure is made. A new plane is created in reference with the XY-plane. Cross section of the coil is drawn. Sketch 2 swept along the line made in sketch made in sketch 1 using the" add frozen" operation to construct the 3D model with different parts. The helical sweep is of 2 turns because the twist specification is defined in number of turns

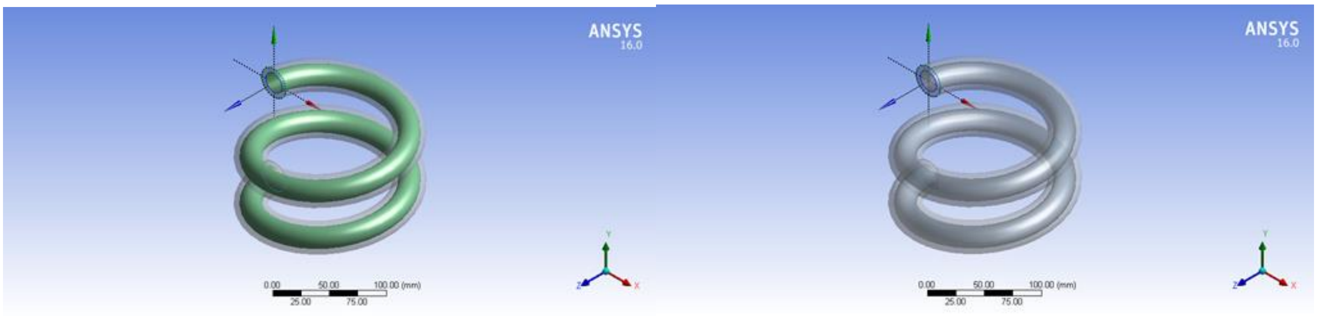

Figure 3.1: 3D Geometry Model of Helical heat exchanger

MESH

Initially a relatively coarser mesh is generated. This mesh contains mixed cells (Tetra and Hexahedral cells) having both triangular and quadrilateral faces at the boundaries. Care is taken to use structured hexahedral cells as much as possible. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, a fine mesh is generated.

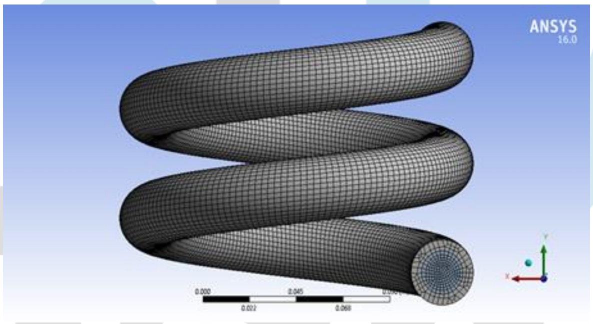

Figure 3.2 : Close View of the Meshed Parts

SOLUTION

It is achieved in following steps: The mesh is checked and quality is obtained. The analysis type is changed to Pressure Based type. The velocity formulation is changed to absolute and time to steady state. Gravity is defined as $y = -9.81\ m/s^2$. Model: Energy is set to ON position. Viscous model is selected as "k-ε model (2 equations). Materials: Water-liquid and copper is selected from the fluent database. Cell zone conditions: Fluid. Boundary conditions: Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. No slip condition is considered for each wall.

The solution methods are specified as follows:

Scheme = Simple, Gradient = Least Square Cell Based, Pressure = Standard, Momentum = Second Order Upwind, Turbulent Kinetic Energy = Second Order Upwind, Turbulent Dissipation Rate = Second Order Upwind. Solution initialization: initialization methods- Standard Initialization. Run Calculation: number of iteration- 1000, reporting interval- 1, profile update interval- 1

IV. ANALYSIS

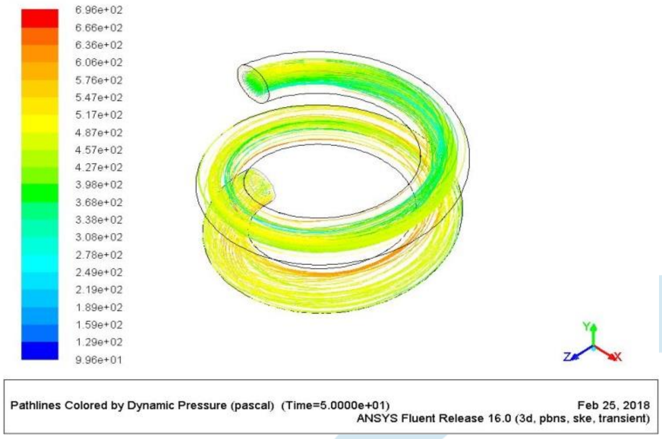

Figure 4.1: Dynamic Pressure

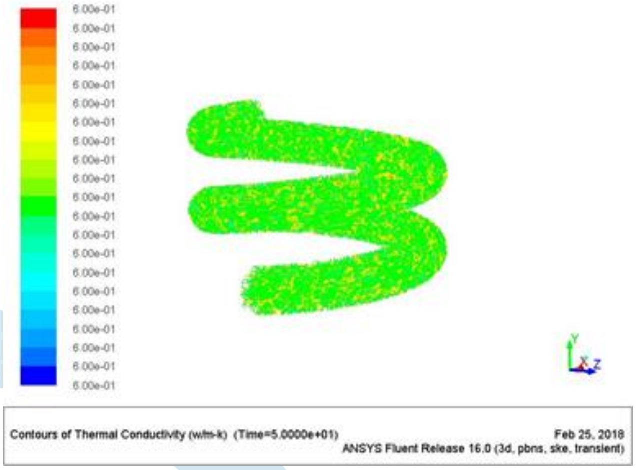

Figure 4.2: Thermal Conductivity

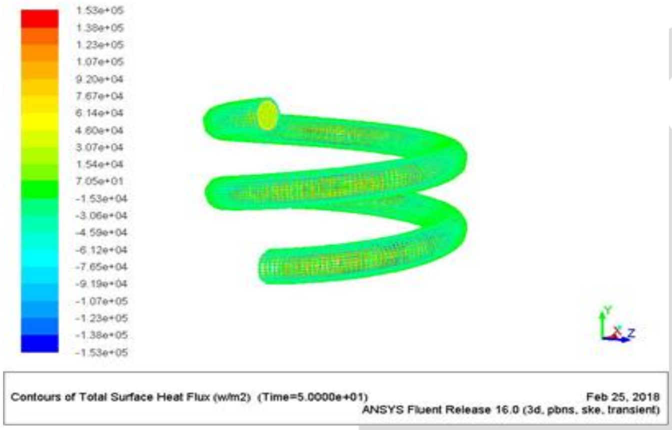

Figure 4.3: Total Surface Heat Flux

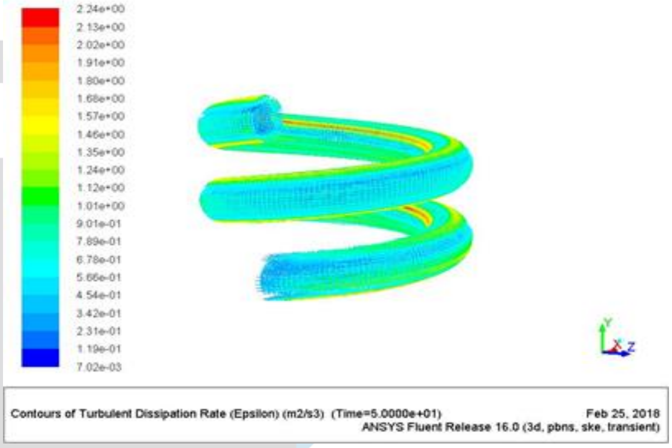

Figure 4.4: Turbulent Dissipation rate

Figure 4.5: Total Pressure

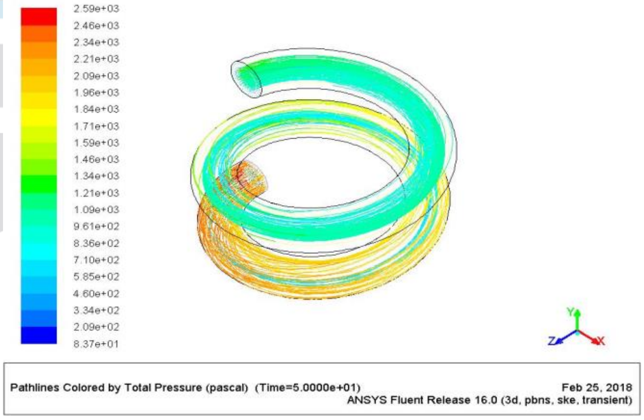

Figure 4.6: Pathline colored by Total Pressure

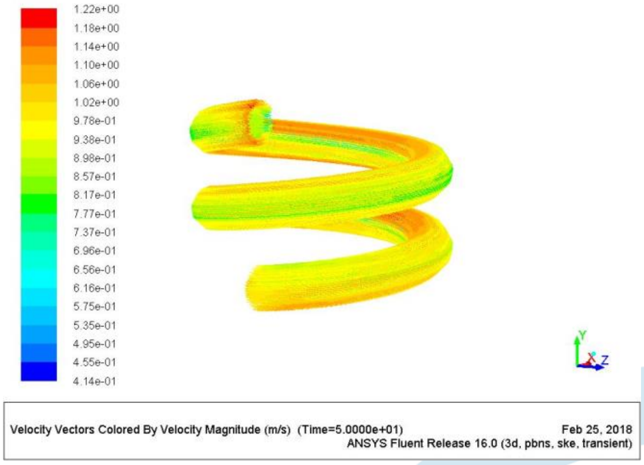

Figure 4.7: Velocity Magnitude

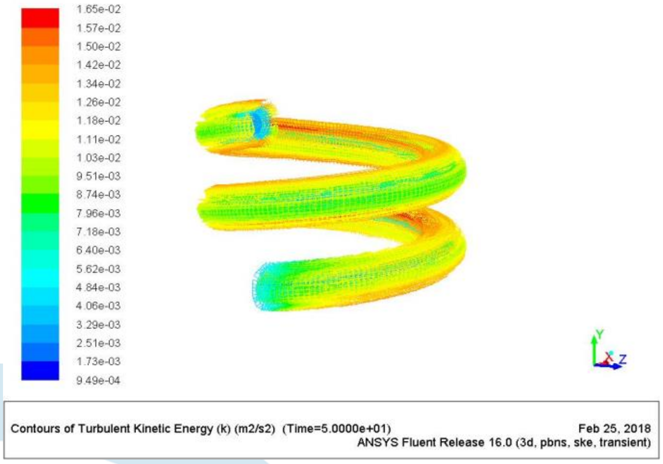

Figure 4.8: Turbulent Kinetic Energy

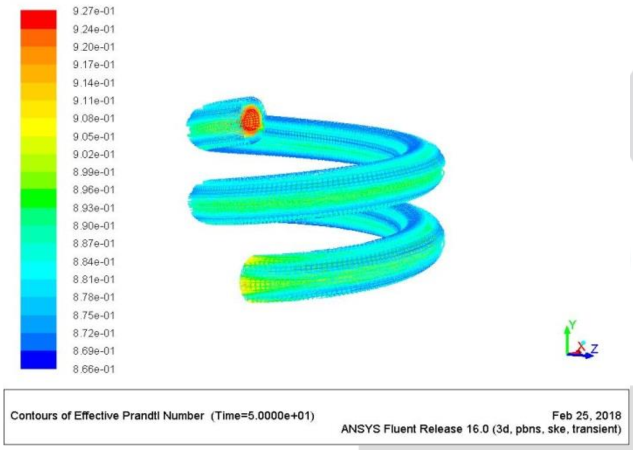

Figure 4.9: Effective Prandtl number

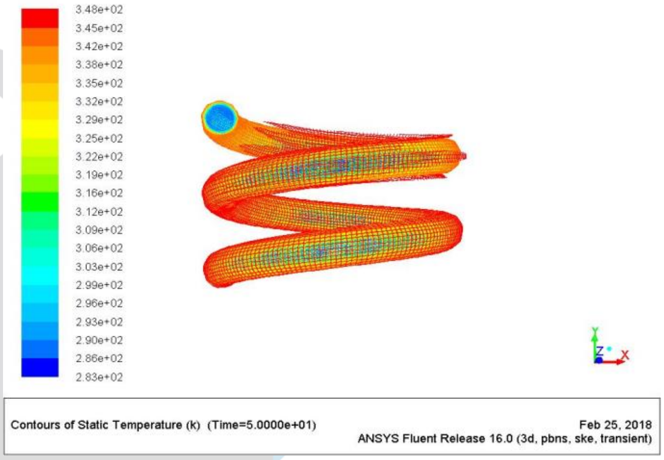

Figure 4.10: Static Temperature

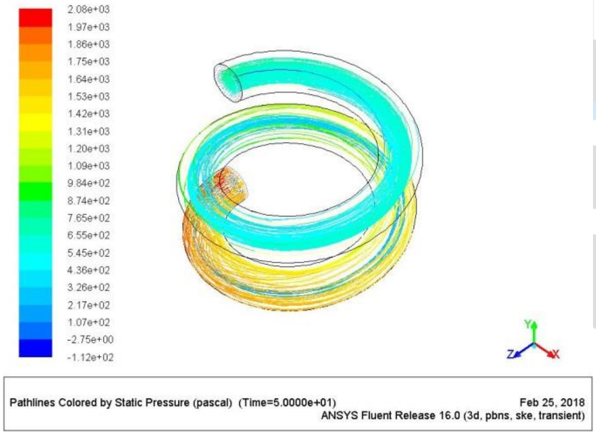

Figure 4.11: Pathline Colored by Static Pressure

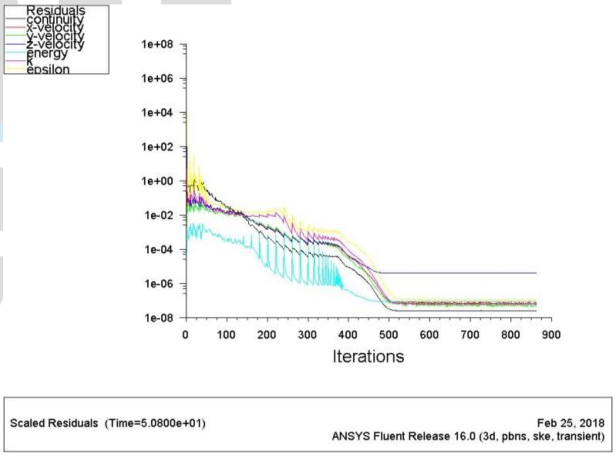

Figure 4.12: Scaled Residuals

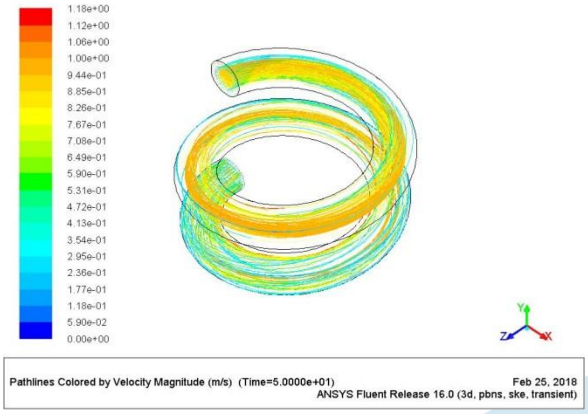

Figure 4.13: Pathline Colored by Velocity Magnitude

V. RESULTS

Table 1 Mass Flow Rate

Mass Flow Rate	(kg/s)
Inlet cool	0.27525607
Inlet hot	0.38911143
Out cool	-0.27525607
Out hot	-0.38911146
Net	-2.9802322e-08

Table 2 Heat Transfer Rate

Heat Transfer Rate	(w)
Inlet cool	-17439.715
Inlet hot	81119.289
Out cool	10022.132
Out hot	-73701.664
Net	0.041992188

Table 3 Static Temperature

Static Temperature	(k)
Inlet cool	287.18579
Inlet hot	342.38395
Out cool	291.6167
Out hot	339.30517

Table 4 Velocity Magnitude

Velocity Magnitude	(m/s)
Inlet cool	0.73764749
Inlet hot	0.41424999
Out cool	0.7654017
Out hot	0.42492407
Net	0.61919068

V CONCLUSION

From the pressure and temperature contours it was found that along the outer side of the pipes the velocity and pressure values were higher in comparison to the inner values. Condensation heat transfer coefficient increases with decrease in coil curvature ratio and heat transfer co-efficient increase with increasing inlet temperatures of water flow rate.

REFERENCES

[1] Kharat Rahul., Bhardwaj Nitin., Jha R.S., Development of heat transfer coefficient correlation for concentric helical coil heat exchanger, International Journal of Thermal Sciences, vol.-48 (2009) 2300–2308.

[2] 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.

[3] 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 Rohidas Bhoi, 2008, Chemical Engg Research and Design 221-232.

[4] Ferng, Numerically investigated effects of different Dean Number and pitch size on flow and heat transfer characteristics in a helically coil-tube heat exchanger