CFD Analysis of Heat Transfer in Helical Coil Heat Exchanger

Ashish Dewangan[1] , Ashok Kumar Yadav [2,*]

[1]Department of Mechanical Engineering, IIT (ISM), Dhanbad-826004, India

[2]Department of Mechanical Engineering, RKGIT, Ghaziabad, 201003, India

ABSTRACT

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 analyze the effect of two different flows (parallel and counter-flow) on the total heat transfer from a helical tube, where the cold fluid flows in the outer pipe and the hot fluid flowing in the inner pipes of the tube in tube helical coiled heat exchanger. Different dimensions of the pipes, helixes are taken into consideration while running the analysis. The surface Nusselt number, the contours of temperature and energy, velocity vectors and the total heat transfer rate from the wall of the tube were calculated and plotted using ANSYS FLUENT 15.0 where the governing equations of mass, momentum and heat transfer were solved simultaneously, using the k-e two equations turbulence model. Copper was chosen as the as metal for the construction of the helical tube. The fluid flowing through the tube was taken as water

I. INTRODUCTION

Heat exchange between flowing fluids is one of the most important physical process of concern, and a variety of heat exchangers are used in different type of installations, as in process industries, compact heat exchangers nuclear power plant, HVACs, food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. In a heat exchanger the heat transfer through radiation is not taken into account as it is negligible in comparison to conduction and convection. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. But convection plays the major role in the performance of a heat exchanger.

Forced convection in a heat exchanger, transfers the heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it.

Tubular heat exchangers are built of mainly of circular tubes although some other geometry has also been used in different applications. This type of construction offers a large amount of flexibility in design as the designing parameters like the diameter, length and the arrangement can be easily modified. This type is used for liquid-to-liquid (phase changing like condensing or evaporation) heat transfer.

The double pipe or the tube in tube type heat exchanger consists of one pipe placed concentrically inside another pipe having a greater diameter. The flow in this configuration can be of two types: parallel flow and counter-flow. It can be arranged in a various configurations to meet the different heat transfer requirements of this the helically arranged stands out as it has found its place in different industrial applications. As this configuration is widely used, knowledge about the heat transfer coefficient, pressure drop, and different flow patterns has been of much importance. The curvature in the tubes creates a secondary flow, which is normal to the primary axial direction of flow. This secondary flow increases the heat transfer between the wall and the flowing fluid. And they offer a greater heat transfer area within a small space, with greater heat transfer coefficients. Study has been done on the types of flows in the curved pipes, and the effect of Prandtl and Reynolds number on the flow patterns and on Nusselt numbers. The two basic boundary conditions that are faced in the applications are constant temperature and the constant heat flux of the wall.

Due to existence of the secondary flow, the heat transfer rates (& the fluid pressure drop) are greater in the case of a curved tube than in a corresponding straight tube at the same flow rate and the same temperature and same boundary conditions.

In the current study, heat transfer is analysed in Helical coil tube type heat exchanger because of certain advantages- a) Helical coils give better heat transfer characteristics since they have lower wall resistance & higher process side coefficient. b) surface area of the curved pipe is exposed to the moving fluid, which eliminates the dead-zones that are a common drawback in the shell and tube type heat exchanger. c) it offers a larger surface area in a relatively smaller reactor volume and a lesser floor area. d) spring-like coil of the helical coil heat exchanger eliminates thermal expansion and thermal shock problems, which helps in high pressure operations. e) fouling is comparatively less in helical coil type than shell and tube type because of greater turbulence created inside the curved pipes. But it has some drawbacks, such as- For highly reactive fluids or highly corrosive fluid coils cannot be used, instead jackets are used and Cleaning of vessels with coils is more difficult than the cleaning of shells and jackets. Coils play a major role in selection of agitation system. Sometimes the densely packed coils can create unmixed regions by interfering with fluid flow. The design of the helical tube in tube type heat exchanger is also a bit complex and challenging

II. COMPUTATIONAL FLUID DYNAMICS (CFD)

The calculations required simulating the interaction of fluids with surfaces defined by boundary conditions, and initial conditions are done by the ANSYS Fluent v13.0. The Navier stokes equations form the basis of all CFD problems. Two equation models are used for the simulations, and different models are discussed below. The continuity equation, energy equation and the Navier-Stokes momentum equation govern the flow of the fluid in curved tube. Continuity Equation gives the conservation of mass and is given by: $\partial \rho / \partial t + (\rho U_i) / \partial x_i = 0$

And for constant density, $\partial \rho / \partial t = 0$

The momentum balance, (Navier-Stokes equations) follows Newton's 2nd law. The two forces acting on the finite element are the body and the surface forces. In CFD programmes, the momentum equation is given as: $\rho (U_i \partial U_j / \partial x_i) = \rho g_j - \partial p / \partial x_j - \mu \partial^2 U_j / \partial x_i^2$

The governing energy equation is given by: $\rho Cp (U_i \partial T/\partial x_i) = k \partial^2 T/\partial x_i^2$

Turbulence is created because of the unstable nature of the fluid flow. The flow becomes turbulent for higher Reynolds number. In this model the k-ε (turbulent kinetics energy "k" and the turbulent dissipation "ε") model is used. The time constant for turbulence is determined from the turbulent kinetic energy and dissipation rate of turbulent kinetic energy, given as: $\tau = k/ \varepsilon$

Daniel Flórez-Orrego , Walter Arias, Diego Lopez and Hector Velasquez have worked on the single phase cone shaped helical coil heat exchanger and reported the inclination of velocity vector components in the secondary flow, even though the contours of velocity were similar, these deviation was due to the non-uniform flame radiation and condensed combustion products. J. S. Jaya Kumar observed the use of constant values for the transfer and thermal properties of the fluid resulted in inaccurate heat transfer coefficients and based on the CFD analysis, a correlation was developed in order to evaluate the heat transfer coefficient of the coil. Timothy J. Rennie studied the heat transfer characteristics of a double pipe helical heat exchanger and showed that the overall heat transfer coefficient 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 and concluded that the design of the outré pipe should get the highest priority in order to get a higher overall heat transfer coefficient. J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, Rohidas Bhoi studied the constant thermal and transport properties of the heat transfer medium and their effect on the prediction of heat transfer coefficients and a correlation was established for the inner heat transfer coefficient. Usman Ur Rehman studied the heat transfer and flow distribution in a shell and tube heat exchanger and 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. Nawras H. Mostafa, Qusay R. Al-Hagag studied on the mechanical and thermal performance of elliptical tubes used for polymer heat exchangers. A set of design curves were generated from which a number of geometries of the tube and different materials can be easily selected in order to meet the deformation constraints.

The area of design of a helical coil tube in tube heat exchanger is challenging because of the lack of experimental data available regarding the behavior of the fluid in helical coils and the heat transfer characteristics for a double-pipe helical heat exchanger by varying the different parameters like different temperatures and diameters of pipe and coil and also to determine the fluid flow pattern in helical coiled heat exchanger. The objective of the project is to obtain a better and more quantitative insight into the heat transfer process that occurs when a fluid flows in a helically coiled tube. The study also covered the different types of fluid flow range extending from laminar flow through transition to turbulent flow. The materials for the study were decided and fluid taken was water and the material for the pipe was taken to be copper for its better conducting properties.

III. METHODOLOGY

Computational fluid dynamics (CFD) study of the system starts with the construction of desired geometry and mesh for modeling the dominion. Heat exchanger is built in the ANSYS workbench design module. It is a counter-flow heat exchanger. Constructed geometry is shown in Fig. 1.

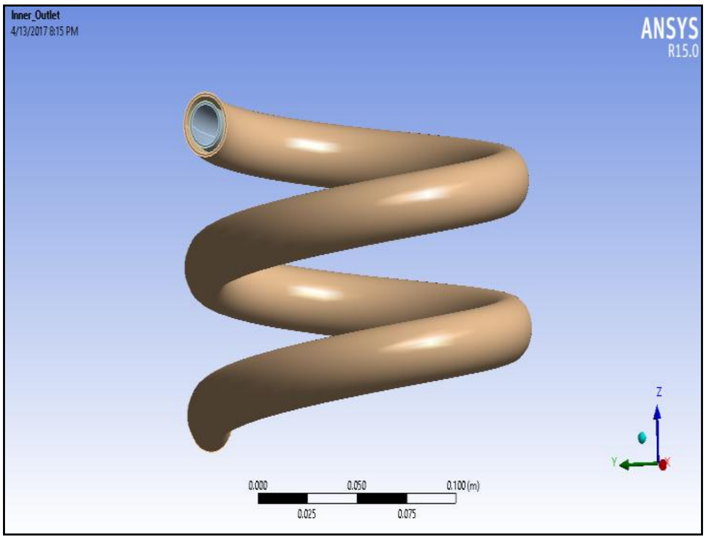


Figure 1. Geometry of Helical Coil Tube

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. For this fine mesh, the edges and regions of high temperature and pressure gradients are finely meshed. (Fig. 2) Total Mesh Nodes are 60852 and elements are 55660. The different surfaces of the solid are named as per required inlets and outlets for inner and outer fluids. The outer wall is named as insulation surface. In ANSYS Fluent Launcher dimension is taken as 3D, option as Double Precision and processing as Serial type.

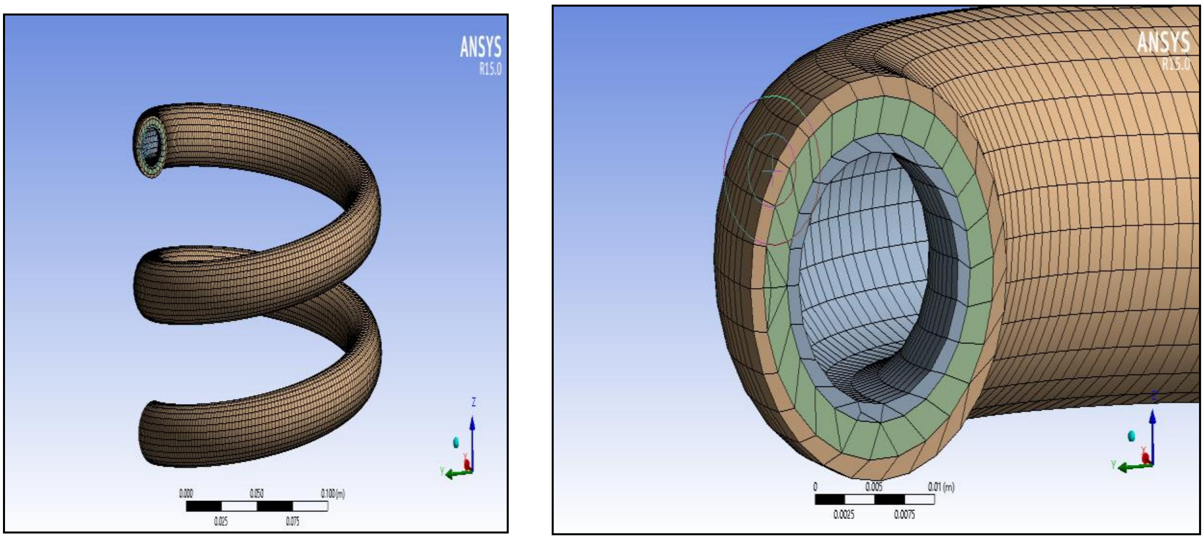


Figure 2. Meshing Structure of Helical Coil Tube

IV. SOLUTIONS

In Problem Setup, the analysis type is changed to Pressure Based type. The velocity formulation is absolute and time is steady state. Gravity is defined as y = -9.81 m/s^2. In Models, Energy is set to on position. Viscous model is selected as "k-ε model (2 equation). Radiation model is changed to Discrete Ordinates. In Materials, water-liquid and copper is selected for fluid and solid respectively from the fluent database. In Cell Zone Conditions the parts are assigned as water and copper as per fluid/solid parts. Boundary conditions are used according to the need of the model. The inlet and outlet conditions are defined as velocity inlet and pressure outlet. As this is a counter-flow with two tubes so there are two inlets and two outlets. The walls are separately specified with respective boundary conditions. No slip condition is considered for each wall. Except the tube walls each wall is set to zero heat flux condition. The details about all boundary conditions are given in the Table 1.

Table 1. Boundary Conditions

-	Boundary Condition Type	Velocity Magnitude	Turbulent Kinetic Energy	Turbulent Dissipation Rate	Temperature
Inner Inlet	Velocity Inlet	0.9942 m/s	0.01 m^2/^{s2}	0.1 m^2/s^3	348 K
Inner Outlet	Pressure Outlet	-	-	-	-
Outer Inlet	Velocity Inlet	0.8842 m/s	0.01 m^2/s^2	0.1 m^2/s^3	283 K
Outer Outlet	Pressure Outlet	-	-	-	-

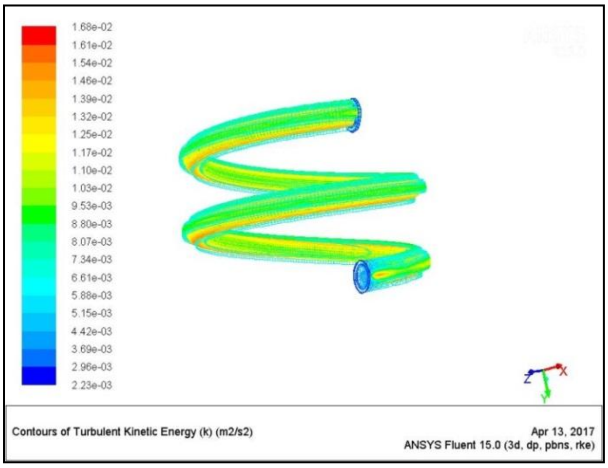
The Reference Values are - Area 1 m^2, Density 998.2 kg/m^3, Length 39.37008 inch, Temperature 348 K, Velocity 0.9942 m/s, Viscosity 0.001003 kg/m-s and Ratio of specific heats as1.4.

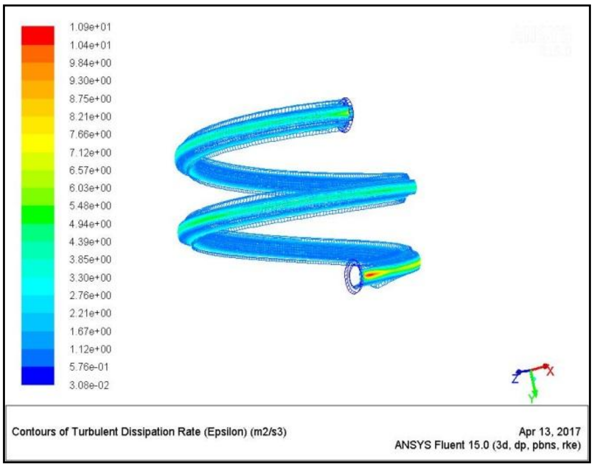
In Solution Method- scheme is simple, Gradient is Least Square Cell Based and Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate are taken as Second Order Upwind.

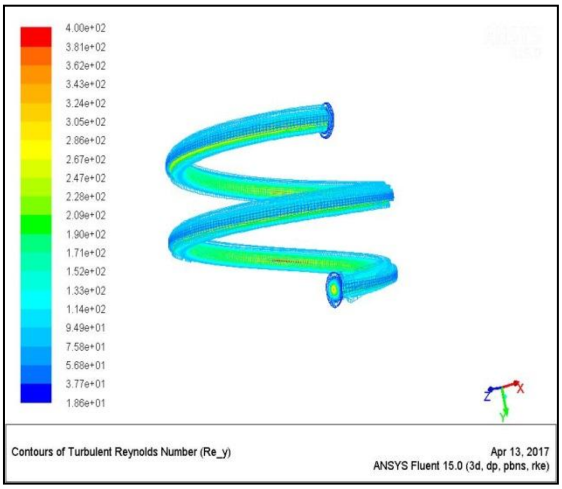
In Solution Control and Initialization, relaxation factors are as- pressure 0.3 Pa, Body forces 1 kg/m^2s^2, Momentum 0.7 kg-m/s and Turbulent kinetic energy 0.8 m^2/s^2.

V. RESULTS AND DISCUSSION

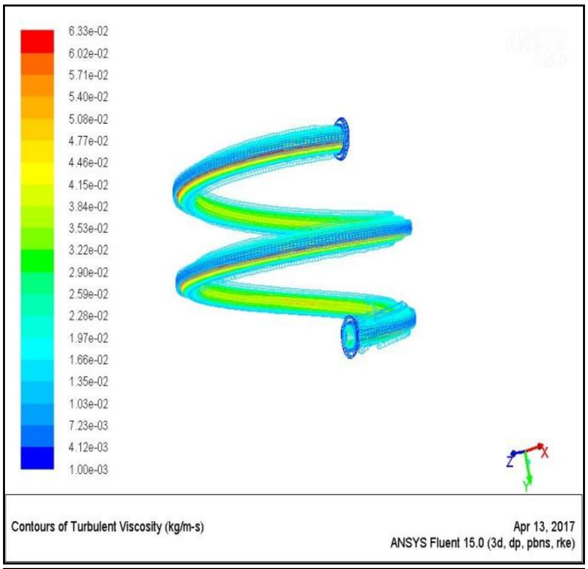
The temperature, pressure and velocity distribution along the heat exchanger can be seen through the contour. The plots give an idea of flow separation at several parts of the heat exchanger.

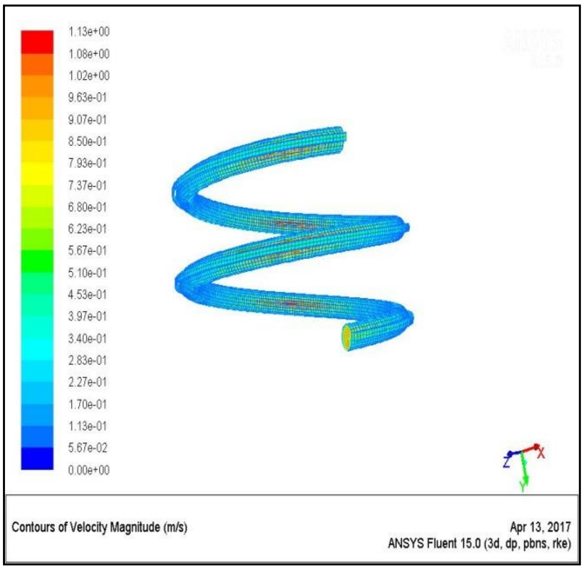


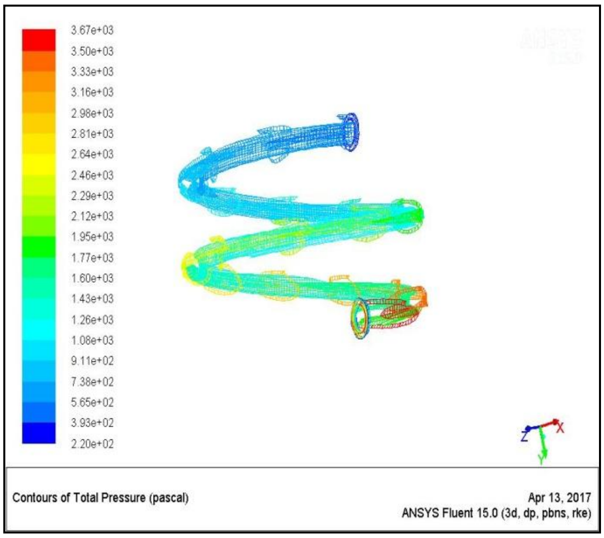


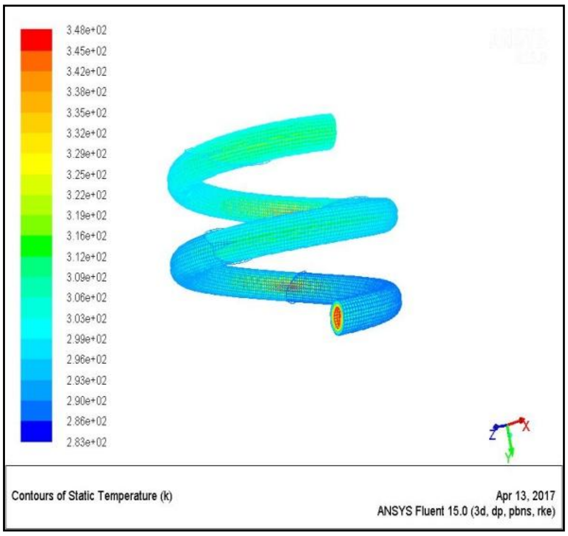


ISBN: 978-93-86171-85-6









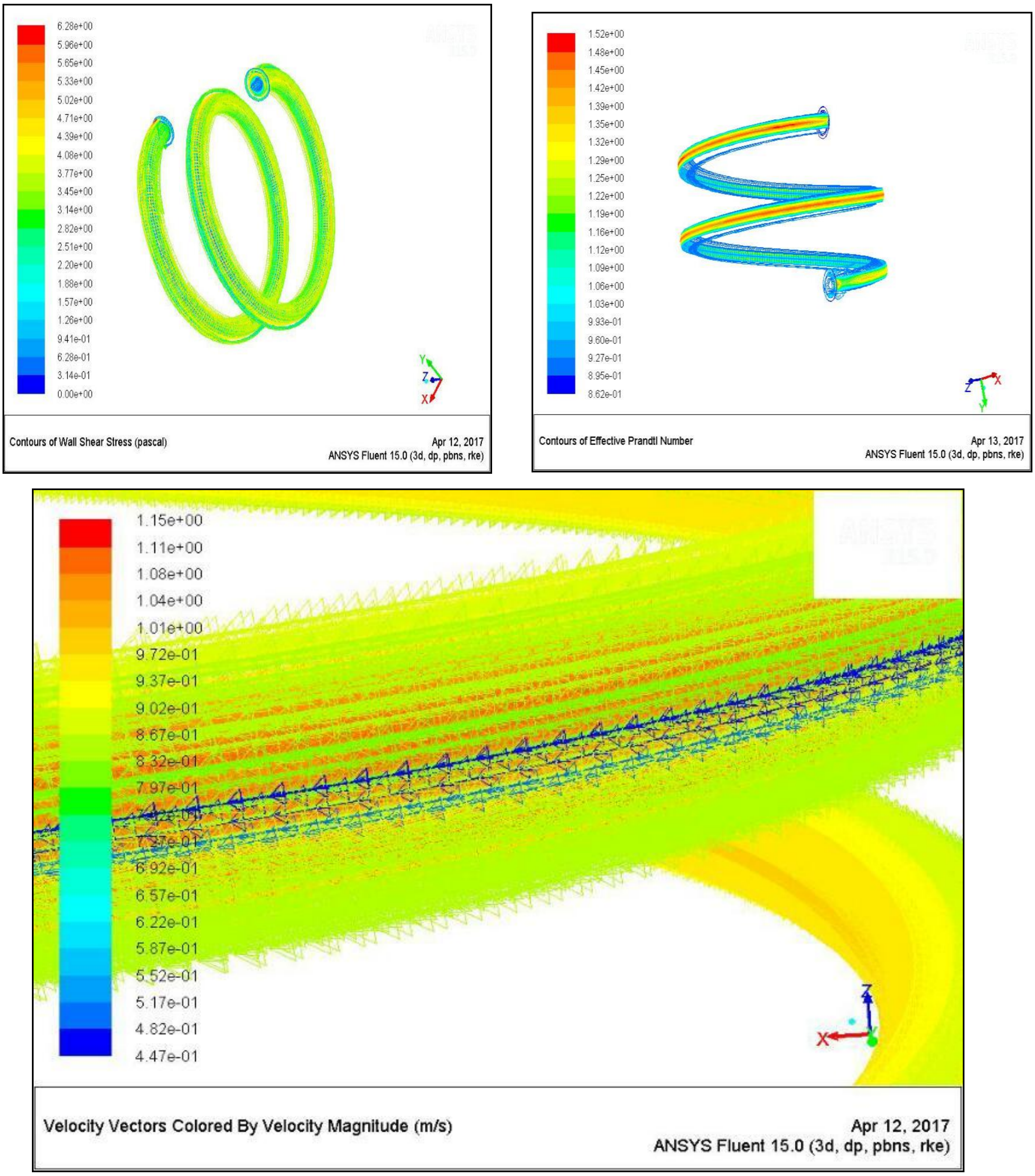


Figure 3. Contors and Vectors from CFD Analysis

VI. PLOTS

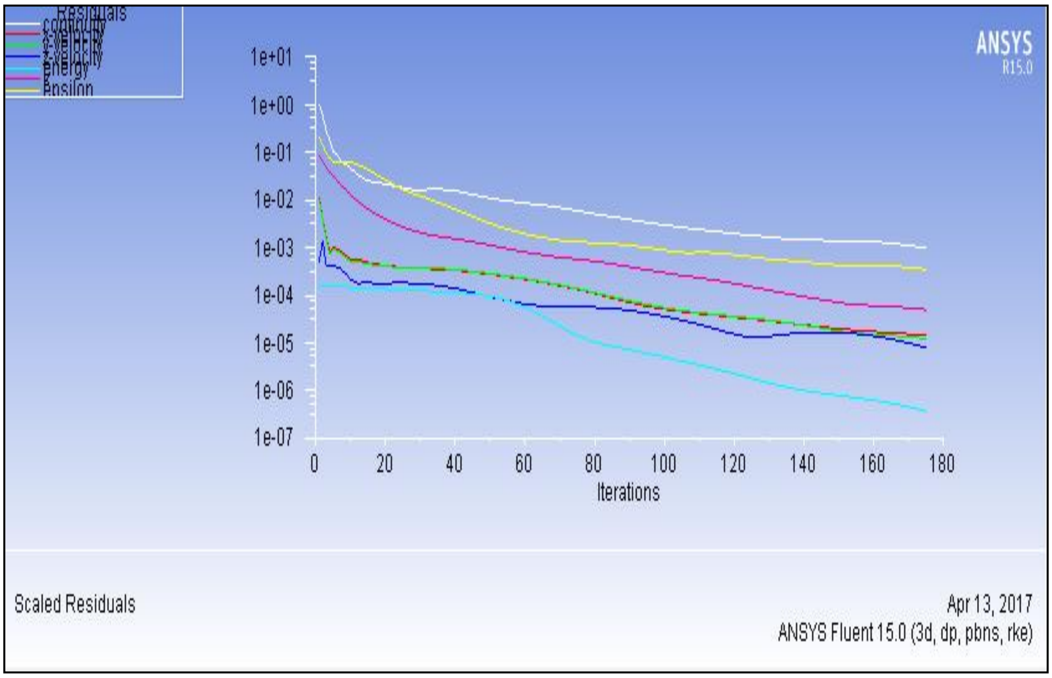


Figure 4: Scaled Residuals

VII. CONCLUSIONS

A CFD package (ANSYS FLUENT 15.0) was used for the numerical study of heat transfer characteristics of a helical coiled double pipe heat exchanger for parallel flow and the results were then compared with that of the counter-flow. The CFD results when compared with the experimental results from different studies and were well within the error limits. The study showed that there is not much difference in the heat transfer performances of the parallel-flow configuration and the counter-flow configuration. Nusselt number at different points along the pipe length was determined from the numerical data. The simulation was carried out for water to water heat transfer characteristics and different inlet temperatures were studied. Nusselt number for the pipes was found to be varying from 340-360. Characteristics of the fluid flow were also studied for the constant temperature and constant wall heat flux conditions.

- From the velocity vector plot (Fig. 3) it was found that the fluid particles were undergoing an oscillatory motion inside both the pipes.
- From the pressure and temperature contours (Fig. 3) it was found that along the outer side of the pipes the velocity and pressure values were higher in comparison to the inner values.

REFERENCES

[1] 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.

[2] Numerical And Experimental Studies of a Double pipe Helical Heat Exchanger by Timothy John Rennie, Dept. of Bio-resource Engineering 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 Engineering Research and Design 221-232.

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

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

[6] Numerical analysis of forced convection heat transfer through helical channels Dr. K. E. Reby Roy, IJEST, July-2012 vol-4.

[7] Minton P.E., Designing Spiral Tube Heat Exchangers, Chemical Engineering, May 1970, p. 145.

[8] Noble, M.A., Kamlani, J.S., and McKetta, J.J., Heat Transfer in Spiral Coils, Petroleum Engineer, April 1952, p. 723

[9] Heat Transfer Analysis of Helical Coil Heat Exchanger with Circular and Square Coiled Pattern by Ashok B. Korane, P.S. Purandare, K.V. Mali, IJESR, June 2012, vol-2, issue-6