Experimental and CFD estimation of heat transfer in helically coiled heat exchangers

J.S. Jayakumar\textsuperscript{a,b,*}, S.M. Mahajani\textsuperscript{a}, J.C. Mandal\textsuperscript{a}, P.K. Vijayan\textsuperscript{b}, Rohidas Bhoi\textsuperscript{a}

\textsuperscript{a}Indian Institute of Technology, Mumbai, India
\textsuperscript{b}Bhabha Atomic Research Centre, Mumbai, India

**ABSTRACT**

Enhancement in heat transfer due to helical coils has been reported by many researchers. While the heat transfer characteristics of double pipe helical heat exchangers are available in the literature, there exists no published experimental or theoretical analysis of a helically coiled heat exchanger considering fluid-to-fluid heat transfer, which is the subject of this work. After validating the methodology of CFD analysis of a heat exchanger, the effect of considering the actual fluid properties instead of a constant value is established. Heat transfer characteristics inside a helical coil for various boundary conditions are compared. It is found that the specification of a constant temperature or constant heat flux boundary condition for an actual heat exchanger does not yield proper modelling. Hence, the heat exchanger is analysed considering conjugate heat transfer and temperature dependent properties of heat transport media. An experimental setup is fabricated for the estimation of the heat transfer characteristics. The experimental results are compared with the CFD calculation results using the CFD package FLUENT 6.2. Based on the experimental results a correlation is developed to calculate the inner heat transfer coefficient of the helical coil.

© 2008 The Institution of Chemical Engineers. Published by Elsevier B.V. All rights reserved.

1. Introduction

It has been widely reported in literature that heat transfer rates in helical coils are higher as compared to a straight tube. Due to the compact structure and high heat transfer coefficient, helical coil heat exchangers are widely used in industrial applications such as power generation, nuclear industry, process plants, heat recovery systems, refrigeration, food industry, etc. (Xin et al., 1996; Abdulla, 1994; Bai et al., 1999; Jensen and Bergles, 1981; Futagami and Aoyama, 1988; Patankar et al., 1974).

Heat exchanger with helical coils is used for residual heat removal systems in islanded or barge mounted nuclear reactor system, wherein nuclear energy is utilised for desalination of seawater (Manna et al., 1996). The performance of the residual heat removal system, which uses a helically coiled heat exchanger, for various process parameters was investigated by Jayakumar and Grover (1997). The work had been extended to find out the stability of operation of such a system when the barge on which it is mounted is moving (Jayakumar et al., 2002). In all these studies, empirical correlations were used to estimate the amount of heat transfer and pressure drop in the helical coils. The appropriateness of the correlation used in the above work is uncertain for the specific application and in the present work, it is proposed to generate the desired correlations using numerical and experimental work.

Heat transfer and flow through a curved tube is comprehensively reviewed first by Berger et al. (1983) and subsequently by Shah and Joshi (1987). The latest review of flow and heat transfer characteristics is provided by Naphon and Wongwises (2006). The characteristics of flow, pressure drop and heat transfer have been reported by many investigators. The heat transfer enhancement in helical coil systems is reported by Prabhanjan et al. (2004), Berger et al. (1983), Janssen and Hoogendoorn (1978) and Ruthven (1971). Condensing heat...
transfer and pressure drop of refrigerant R 134A in helicoidal (helical double pipe heat exchanger) is experimentally investigated by Kang et al. (2000). The effect of torsion on the flow in a helical tube of circular cross-section is experimentally investigated by Yamamoto et al. (1995) for a range of Reynolds numbers from about 500 to 20,000.

Most of the investigations on heat transfer coefficients are for simplified boundary conditions such as constant wall temperature or constant heat flux (Prabhanjan et al., 2004; Shah and Joshi, 1987; Nandakumar and Masliyah, 1982). The situation of constant wall temperature is idealised in heat exchangers with phase change such as condensers. The boundary condition of constant heat flux finds application in electrically heated tubes and nuclear fuel elements. However, the case of fluid–fluid heat exchange has not been studied well. Experimental studies of a double pipe helical heat exchanger was conducted by Rennie and Raghavan (2005). The double pipe helical coil heat exchanger was further numerically investigated by Rennie and Raghavan (2006a, b). Pressure drop and heat transfer in tube-in-tube helical heat exchanger was studied by Kumar et al. (2006). For a double pipe heat exchanger, the co-current or counter-current flow situation can be applied. However, in the helically coiled heat exchanger, which is taken up in the present study, practically cross flow exists in the shell side and hence the analysis is entirely different from those reported in earlier studies.

In this work, it is proposed to generate correlations for inner heat transfer coefficient considering fluid–fluid heat exchange in a helically coiled heat exchanger. An experimental setup is built for carrying out the heat transfer studies representing the equipment under study. For further details of the equipment and its applications, refer to Jayakumar and Grover (1997). In addition, the heat transfer phenomena in the exchanger is analysed numerically using a commercial CFD code FLUENT Version 6.2 (2004). In contrast to the earlier similar analyses, instead of specifying an arbitrary boundary condition, heat transfer from hot fluid to cold fluid is modelled by considering both inside and outside convective heat transfer and wall conduction. In these analyses, we have used temperature dependent values of thermal and transport properties of the heat transfer medium, which is also not reported earlier. The numerical predictions are verified against the experimental results.

The article is organised as follows: we begin with the introduction of helically coiled system followed by the description of the experimental setup used for heat transfer studies and the methodology of experimentation. In the next section, the numerical results for heat transfer characteristics of a helical pipe are presented. Subsequently the actual heat exchanger is analysed for the case of fluid–fluid heat transfer and the computational results are compared with the experimental ones. In the last section, the validated results are converted into a correlation for the estimation of heat transfer inside a helical coil as appropriate to the experimental conditions.

### 2. Characteristics of helical coil

Fig. 1 gives the schematic of a helical coil. The pipe has an inner diameter 2r. The coil has a diameter of 2Rc (measured between the centres of the pipes), while the distance between two adjacent turns, called pitch is H. The coil diameter is also called pitch circle diameter (PCD). The ratio of pipe diameter to coil diameter (r/Rc) is called curvature ratio, δ. The ratio of pitch to developed length of one turn (H/2πRc) is termed non-dimensional pitch, λ. Consider the projection of the coil on a plane passing through the axis of the coil. The angle, which projection of one turn of the coil makes with a plane perpendicular to the axis, is called the helix angle, α.

Similar to Reynolds number for flow in pipes, Dean number is used to characterise the flow in a helical pipe. The Dean

![Fig. 1 – Basic geometry of a helical pipe.](image)
number, \( De \) is defined as,

\[
De = Re \sqrt{\frac{r}{R_c}}. \tag{1}
\]

where, \( Re \) is the Reynolds number = \( \frac{2u_0 \rho}{\mu} \).

Many researchers have identified that a complex flow pattern exists inside a helical pipe due to which the enhancement in heat transfer is obtained. The curvature of the coil governs the centrifugal force while the pitch (or helix angle) influences the torsion to which the fluid is subjected to. The centrifugal force results in the development of secondary flow (Darvid et al., 1971). 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. The difference in velocity sets-in secondary flows, whose pattern changes with the Dean number of the flow.

The critical Reynolds number for the transition from laminar to turbulent flow in helical coils is a function of the coil parameters. The critical Reynolds number may be determined using the correlation by Schmidt (1967).

\[
Re_{cr} = 2300 \left[ 1 + 8.6 \left( \frac{r}{R_c} \right)^{0.65} \right] \tag{2}
\]

For curvature ratios less than 1/860, the critical Reynolds number becomes equal to that of a straight tube.

3. **Experimental setup**

3.1. **Helical coil test section**

The pipe used to construct the helical section has 10 mm i.d. and 12.7 mm o.d. The tube material is SS 316. The Pitch Circle Diameter (PCD) of the coil is 300 mm and tube pitch is 30 mm. The remaining parts of the setup are made of SS 304.

The helical coil is enclosed in a vessel to simulate the shell side of heat exchanger. The details are given in Fig. 2. The cold fluid enters the shell through the bottom connection and flows up. It leaves the shell through the nozzle at the top. The coil and the baffles are welded to a top flange in such a way that they can be replaced with another coil assembly.

The helical coil test section is connected to a loop, which provides the necessary flow through the tube and shell side of the test section and the required instrumentation. Fig. 3 illustrates the loop used for the estimation of heat transfer coefficient.

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 W. A controller is provided to maintain the temperature of water at the inlet of the test section at the set value. The hot fluid from the tank is pumped through the test section using a centrifugal pump of 1/2 hp power rating. Flow rate of hot fluid is measured using a rotameter. Both inlet and outlet temperatures of the hot fluid are measured by using RTDs and the values are available on digital displays.

Cooling water from a constant temperature tank is circulated through the shell side. Its flow rate, inlet and outlet temperatures are measured. Flow rate of cooling water is kept constant. Its flow is adjusted such that the rise in temperature of the cooling water is approximately 3 to 5 °C.

3.2. **Experimental procedure**

Measurements are taken only after the temperatures attain steady values. Experiments are conducted for five different flow rates through the coil and for three different values of temperature at the inlet of the helical pipe. During the course of each set of experiments, the flow rate through the shell side is kept constant, which ensures a constant heat transfer coefficient on the shell side. The experiment is carried out by changing the flow rate through the tube. Once a steady state is attained, values of flow rates of the hot and cold fluids, temperatures at the inlet and exit of the hot and cold fluid, and the power input to the heater and the pump are noted.

4. **Numerical experiment**

CFD has been used to investigate the performance of heat exchangers. Such an early study was done by Rustum and
Soliman (1990). A vertical mantle type of exchanger was analysed by Shah (2000) and reliable results were found to be obtained. Grijseerdt et al. (2003) have analysed plate type heat exchangers using CFD tool for design optimisation. Van der Vyver et al. (2003) have validated heat exchange process in a tube-in-tube heat exchanger using the commercial CFD code STAR-CD against standard correlations and then applied the code for performance evaluation of Fractal heat exchangers. Heat transfer coefficients for parallel and series flow arrangement in a plate type heat exchanger were determined experimentally and numerically by Flavio et al. (2006). They have used FLUENT 6.1.22 to carry out the numerical studies.

Numerical studies of helically coiled double pipe heat exchangers have also been carried out. Such a heat exchanger was numerically modelled for laminar fluid flow and heat transfer characteristics were studied by Rennie and Raghavan (2005). They have modelled the heat transfer from hot fluid to cold fluid using the CFD package PHOENICS 3.3 and found out the overall heat transfer coefficients for counter-current and parallel flows. Kumar et al. (2006) have investigated hydrodynamics and heat transfer characteristics of double pipe helical heat exchanger using the CFD package FLUENT 6.0. In their analyses, they have concentrated on the turbulent flow regime.

The contents of the present section can be divided into three parts. The effect of using temperature dependent thermal and transport properties of the heat transfer medium for flow inside a helical pipe is investigated first. Then the comparison of the heat transfer characteristics of the helical coil obtained for various boundary conditions such as constant wall temperature, wall heat flux and wall heat transfer coefficients are presented. The analysis has then been extended to a shell and tube heat exchanger with helically coiled tube bundles, in which conjugate heat transfer from hot fluid in the tube to cold fluid in the shell is modelled. In these investigations, commercial CFD package FLUENT 6.2 (double precision, segregated, 3D version) was used.

Before carrying out CFD analysis of flow through helical pipe, the methodology of modelling of conjugate heat transfer has been validated. For this, a straight pipe in a shell and tube heat exchanger was analysed using FLUENT for fluid-to-fluid heat transfer through the metal wall. It has been found that the heat transfer coefficient predicted by the Dittus–Boelter equation (Dittus and Boelter, 1930) are comparable with those calculated by FLUENT, with a maximum error of 5%. Hence, the use of the CFD modelling for the prediction of heat transfer coefficient can be employed with confidence.

4.1. Effect of property values on CFD modelling of heat transfer in a helically coiled tube

Heat transfer coefficients for the cases of constant wall temperature have been experimentally determined by various investigators such as Rogers and Mayhew (1964), Kubair and Kuloor (1996), Mori and Nakayama (1967a), Akiyama and Cheng (1971) and Janssen and Hoogendoorn (1978). Characteristics of heat transfer from a helical coil, in which the fluid inside the tube is subjected to constant heat flux is studied by many investigators such as Mori and Nakayama (1967b), Kalb and Seader (1972) and Futagami and Aoyama (1988). In various numerical studies reported (Rennie and Raghavan, 2006a,b; Kumar et al., 2006) on heat transfer in a double pipe heat exchanger, constant properties of the heating/cooling medium were considered. In a later work by Rennie and Raghavan (2006b), the effect of change of Prandtl number on heat transfer was studied. Two studies were performed by them; in the first one, they have considered three Prandtl numbers obtained by changing the thermal conductivity of the fluid. The other properties such as specific heat, viscosity and density were taken to be constants. In the second study, they have considered thermally dependent thermal conductivities by assuming a relationship $k = A + BT$. All other properties were kept constant.

In the present analysis, we consider the transport and thermal properties of the heat exchange medium (both hot and cold fluids) to be dependent on the temperature; rather than studying the influence of Pr alone as reported by Rennie and Raghavan (2006b). The plots of density, viscosity, thermal conductivity and specific heat of water are given in Fig. 4. It can be seen that the viscosity of water changes from $14e-04 \text{ Pa s}$ to $3 \text{ Pa s}$ as the temperature is increased from 280 to 370 K. Also significant changes are observed in density and thermal conductivity.

For modelling temperature dependent properties, the following polynomial functions (Eqs. (3)–(6)) were programmed in FLUENT. In CFD code, the governing equations are solved with the fluid properties evaluated at the cell temperatures.

$$\mu(T) = \mu_0 \left(1 + \frac{T}{5000}\right)^3$$

$$\rho(T) = \rho_0 \left(1 + \frac{T}{5000}\right)^2$$

$$k(T) = k_0 \left(1 + \frac{T}{5000}\right)^2$$

$$C_p(T) = C_p_0 \left(1 + \frac{T}{5000}\right)^2$$

These relationships were obtained by regression analysis using MATLAB. In the above relationships, temperature is specified in K. It may be noted that since the pressure of the fluids does not change appreciably and, also since the pressure dependency of the properties of an incompressible fluid is negligibly small, only the temperature dependency was taken into account in the studies.

In order to study the effect of fluid properties on the modelling of heat transfer, the case of heat transfer to fluid flowing inside a helical tube, which is heated to a constant wall temperature is analysed. The grid used in the analysis is shown in Fig. 5. A constant wall temperature of 300 K was specified as the boundary condition. Hot water at a temperature of 360 K is entering the helical coil at the top (velocity inlet boundary condition) and leaving at the bottom (pressure outlet boundary condition). Analysis has been carried out by varying the inlet flow rate. In the first series of analyses, the properties of water were kept constant corresponding to the fluid inlet temperature and pressure (360 K temperature, 1 atm pressure). Second set of analyses were done using the temperature dependent properties of water, as given by Eqs. (3)–(6). These equations were programmed in FLUENT as polynomial functions to compute the properties. In both the sets of analysis, shell conduction through the pipe wall of 1.35 mm thickness was taken into account.

Before carrying out the actual analysis, a grid dependency of the solution was studied. The grids used are given in Fig. 6(a–d). The grid given in (c) was chosen because further refinement does not result in reduction of mass and energy errors.
The velocity and temperature profiles at the exit of the helical pipe for the constant and temperature dependent properties of the cooling medium are given in Fig. 7. In all the figures, the right side is the outer side of the coil. It can be readily seen that unlike for flow through a straight tube high velocity is on the outer side of the coil. It is also noted that the shapes of contours are different for the constant and temperature dependent cases. For the case of constant properties, the profiles are much skewed on the inner side of the coil. Since the fluid properties in each cell volume are same to satisfy the continuity, smaller area is available for the high velocity regions. This is especially prominent in the inner side of the coil, as the velocity gradients at this location are higher.

The results of computation considering temperature dependency of the fluid properties are given in Fig. 8. The Dean number is calculated based on the fluid properties at the inlet. These figures show the results of three analyses. In Fig. 8(a), the first one (symbol □) is the results where the properties at ambient conditions (as reported by Rennie and Raghavan, 2006a) are used for computations. The next one (symbol *) is the results obtained when properties evaluated at mean temperature of the fluid is used in the entire computational domain. The last one, represented with symbol +, is the results of temperature dependent properties as given by Eqs. (3) to (6) are used.

From the analysis, it can be seen that an error $Nu$ is about 24% when the properties at ambient conditions are used. The maximum error in $Nu$ for the case when the properties at mean temperature are used is about 10%. When the flow rate of the hot fluid through the helical pipe is low, its change in
temperature along the pipe length is small. This leads to the 
$Nu$ of the three cases to be closer at low values of $De$.
The amount of heat removed from the hot fluid in these 
cases is given in Fig. 8(b). It is seen that the amount of 
heat transferred predicted when constant values of properties 
are used differ appreciably from the actual situation, where 
temperature dependant properties are considered. Since the 
implications in considering properties temperature depend­
ant are high, subsequent analyses were carried out with 
incorporation of variable properties.

Fig. 6 – Grids used for helical pipe fluid volumes.

Fig. 7 – Velocity and temperature profiles at the exit of the tube for constant and temperature dependant properties for the 
case of inlet velocity 1.5 m/s: (a) constant properties and (b) temperature dependent properties.
4.2. CFD modelling of heat transfer in a helically coiled tube for various boundary conditions

A study was carried out to estimate the influence of boundary condition on the nature of heat transfer. The heat transfer characteristics for flow inside a helical pipe were analysed for three boundary conditions. The boundary conditions selected were (a) constant wall temperature (300 K), (b) constant heat flux (50 kW m\(^{-2}\)) and (c) constant convective heat transfer coefficient (2000 W m\(^{-2}\) K\(^{-1}\)). The sensitivity of the results on the value of the imposed boundary condition was studied. On changing wall flux from 50 to 250 kW, changes in Nusselt number were marginal. This change is attributable to differences in the fluid temperature distribution along the pipe. Similar results were obtained for other BCs, viz., constant wall temperature and constant wall heat transfer coefficient.

4.3. CFD modelling of a helically coiled tube in a shell and tube heat exchanger

The methodology established in the previous section is extended to simulate one of the helically coiled tubes inside a shell and tube type heat exchanger. The geometry used for numerical modelling is same as the experimental setup. However, in order to reduce the number of cells used in the computation, only two turns of the helical pipe are considered (the experimental setup has 3.5 turns). In this case also, the investigations were carried out using the CFD package FLUENT 6.2 (double precision, segregated, 3D version). Heat exchange from the hot fluid inside the helical pipe to the cold fluid in the shell was modelled with convective heat transfer in the tube, conduction through the tube wall and convective heat transfer to the shell fluid.

4.3.1. Grid and boundary conditions

The geometry and the mesh were created using GAMBIT 2.2 of the FLUENT package starting from its primitives. In this model, the pipe wall thickness is considered. Structured grids were used to mesh the pipe fluid volume and the pipe solid volume. The grid for the pipe is the same as the one given in Figs. 5 and 6(c). Boundary layer mesh was generated for the pipe fluid volume. Due to the highly irregular nature of the shell side fluid volume, unstructured grid was generated. The inner and outer walls of the pipe were defined as coupled for energy transfer from the hot fluid (inside the pipe) to the cold fluid (in the shell). For momentum equation, they
were treated as no-slip ones. The inner and outer wall of the shell were taken as no-slip adiabatic ones. The grid for the analysis domain, which includes the pipe fluid volume, pipe solid volume and the shell fluid volume, is shown in Fig. 10.

Hot fluid inlet is at the top with a velocity inlet boundary condition. Hot fluid outlet was specified as a pressure outlet with zero back pressure. For conjugate heat transfer, outflow boundary condition cannot be specified. Cold fluid enters the tank through the top nozzle (velocity inlet BC) and leaves through the bottom nozzle (pressure outlet BC).

4.3.2. CFD modelling
Heat transfer from the hot fluid flowing inside the helical pipe to the cold fluid flowing through the tank is modelled using FLUENT. The mass flow rate of the cold water was kept at 0.2124 kg s\(^{-1}\), which is same as the one used in the experiment. Flow velocity through the pipe considered were 1.0 to 3.0 m s\(^{-1}\) in steps of 0.25 m s\(^{-1}\). This gives a range of De from 3500 to 11,000. Hot water inlet was at 360 K and the cold water inlet was at 300 K. The Realisable \(k-\varepsilon\) turbulence with standard wall functions was used in this analysis.

Pressure–velocity coupling was resolved using the SIMPLEC algorithm with skewness correction factor 1. For pressure, linear discretisation was used. For momentum, turbulent kinetic energy and turbulent dissipation rate the Power law scheme was used. For the energy equation, second order upwinding was employed. A convergence criterion of 1.0e\(^{-05}\) was used for continuity and x, y and z velocities. The convergence criteria for energy equation was 1.0e\(^{-08}\), while that for the \(k\) and \(\varepsilon\) was 1.0e\(^{-04}\).

Grid independency of the solution was established. Grid for the simulation is the same as the optimum one selected earlier, as given in Figs. 5 and 6(c). The mesh density of the shell side fluid was changed from 3.5563e8 cells m\(^{-3}\) to 12.365e8 cells m\(^{-3}\). It is found that refinement after a mesh density of 9.963e8 cells m\(^{-3}\) does not decrease the energy and mass errors in any appreciable way. This grid size is approximately 30 times smaller that those used by Kumar et al. (2006). The optimum mesh was chosen for the further analysis and it has 873,760 nodes, 4,516,224 faces and 1,915,833 volumes. Each of the runs took about 30 h of time in a Xeon 2.8 processor machine with 2 GB RAM.

Temperature profile of the hot fluid for of the cases (inlet velocity = 1.5 m s\(^{-1}\)) is given in Fig. 11. The effect of cross flow in the shell region is visible from the temperature contours. Velocity profile along a section on the YZ plane is shown in Fig. 12. This is the plane passing through the shell inlet/outlet. Irregularity of the flow in shell as compared to a double pipe heat exchanger is very dominant in this case. The velocity contours in the XZ plane are given in Fig. 13. The coil section at top left is hot fluid inlet, where a circular velocity contours are observed. As the fluid flows down, the flow gets developed due to the centrifugal and torsional effects induced by the helical nature of the pipe. This figure also indicates the irregular velocity profiles existing in the shell side.

The results of the analysis of the CFD simulation are used to estimate the overall heat transfer coefficient. Fig. 14 gives a comparison of the overall heat transfer coefficients obtained from the experiment and those calculated using the CFD code. It is found that the values are well within 5%.

During the experiments, care was taken to obtain non-oscillatory flow rates. For this, stabilised power supply was used for the experimental setup, which maintains speed of the pumps at a constant rpm. Proper thermal insulation was provided for the heating tank and the pipes. No insulation need to be provided to the test section as the cold water at ambient temperature was flowing on the shell side. Thus, enough care was taken to minimise uncertainties in the experiments. An error of 0.2 °C is expected in temperature
measurements due to the accuracy of the temperature sensors and indicators. Error due to accuracy of the rotameters was 3%.

The figure also shows overall heat transfer coefficient predicted by the CFD code when constant values are used for fluid properties. Marked differences in overall heat transfer coefficients are observed between (i) temperature dependent properties and (ii) constant properties used in CFD calculations. It is also observed that the CFD predictions when temperature dependent properties are used are closer to the experimentally observed values. The figure also shows the overall heat transfer coefficient calculated from its components, viz., inside heat transfer coefficient, wall conductance and outside heat transfer coefficient. This value is found to be very close to the overall heat transfer coefficients calculated from overall energy balance.

The inside Nusselt numbers reported by FLUENT for the case of helical pipe heat exchanger assembly are compared with those obtained for a helical pipe, where constant wall boundary conditions are applied. The results are shown in Fig. 15. It can be seen that the heat transfer characteristics of the actual heat exchanger cannot be predicted by an assumed boundary conditions such as constant heat flux, wall temperature, etc.

In our application, the flow is in turbulent regime and the ranges of interest of the Dean number and Prandtl number are 2000 to 12,000 and 1.0 to 3.5, respectively. Based on the nature of the correlations available in the literature, it is proposed that
the Nusselt number for inside heat transfer can be represented in the form,

$$Nu = CDe^mPr^n$$  \hspace{1cm} (7)

where $C$ and $m$ are the constants; which are to be determined. Index of the Prandtl number, $n$, is selected to be equal to 0.4. Using multiple regression analysis of MATLAB software, the following correlation was generated for estimating the inner heat transfer coefficient.

$$Nu = 0.025De^{0.9112}Pr^{0.4}$$  \hspace{1cm} (8)

which is applicable to $2000 < De < 12,000$.

The methodology for estimation of heat transfer for a helically coiled heat exchanger has been successfully validated against experiments. The CFD simulations can now be extended to coils of various (i) pitch circle diameters, (ii) tube pitches and (iii) pipe diameters. The next step would be to...
develop a unified correlation generated by CFD analysis applicable to all helical configurations.

As mentioned earlier, CFD analysis was carried out using Realizable $k$-$\varepsilon$ model with standard wall functions. Applicability of this model with strong centrifugal and torsional effects predominant in the case of flow through helical coils needs to be further investigated. In addition, usage of a fixed set of model constants without considering the temperature effects on them, can result in uncertainties in the CFD predictions. In future analysis, these factors need to be taken into account.

5. Conclusion

It is observed that the use of constant values for the thermal and transport properties of the heat transport medium results in prediction of inaccurate heat transfer coefficients. Also for prediction of heat transfer in a situation of fluid-to-fluid heat transfer, as it occurs in the case in a heat exchanger, arbitrary boundary conditions such as constant wall temperature, constant heat flux, etc., are not applicable. In this situation, it is essential to model the equipment considering conjugate heat transfer. An experimental setup is fabricated to study fluid–fluid heat transfer in a helically coiled heat exchanger. Heat transfer characteristics of the heat exchanger with helical coil are also studied using the CFD code FLUENT. The CFD predictions match reasonably well with the experimental results within experimental error limits. Based on the results a correlation was developed to calculate the inner heat transfer coefficient of the helical coil. Based on the confidence gained in the CFD predictions, the results generated under different conditions may used further to obtain a generalised correlation, applicable to various coil configurations.

REFERENCES


