A brief review on CFD work of helical coil and straight tube heat exchangers

S. S. Pawar, Siddhant Nandi, Tejas Salian , Ganesh Shenoy , Sanket Shetty
S. I. E. S Graduate School of Technology, Nerul, Navi Mumbai – 400 706.

ABSTRACT-

Heat exchangers are used to transfer heat from fluid at high temperature to fluid at lower temperature. Heat exchangers are used in industrial purposes in chemical industries, nuclear power plants, refineries, food processing, etc. Sizing of heat exchangers plays very significant role for cost optimization. Also, efficiency of heat exchangers is an important parameter while selection of industrial heat exchangers. Methods for improvement on heat transfer have been worked upon for many years in order to obtain high efficiency with optimum cost. In this paper, a brief review of CFD work carried out by various researchers related to helical coil and straight tube heat exchangers is presented. It includes the consideration of the effect of important parameters such as flow rate, coil curvature ratio, pitch of helical coil, length of the tube, correlations developed under various conditions, types of flow, etc. on heat transfer rate.

Keywords: CFD, Helical Coil, Straight Tube.

1. Introduction-

Heat exchangers have been an important part in process industries and also in heat recovery, nuclear power plants, refineries, etc. Heat exchangers basically perform heat transfer between two or more fluids of different temperatures. In industries, maximum efficiency is expected with minimal use of resources. Straight tube heat exchanger has been the oldest type of heat exchanger that has been in use. Research work has been performed by various investigators on enhancing the performance of straight tube heat exchanger by changing geometric such as baffle arrangement, types of tube arrangement, length of the pipe and pitch in case of helical coil heat exchangers, use of augmentation techniques in straight tube heat exchangers, etc. However, it was found that straight tube heat exchangers have restriction in terms of sizing and space which are significant parameters while designing industrial heat exchangers. Helical coil heat exchangers are reported to be the most useful heat exchanger due to its higher heat transfer coefficient as compared to straight tube heat exchanger under the same experimental conditions and hence, it requires less space. It is also found out that helical coil has better heat transfer characteristics as compared to straight tube heat exchangers. There are various experimental research works available in the literature which analyses the performance of heat exchangers. However, it is observed that the experimental analysis of heat exchangers is difficult and costly. Also, in experimental analysis, it is difficult to provide different conditions for testing or to prove the hypothesis as it is not feasible economically. In this sense, CFD technique play an important role to overcome all difficulties faced in experimental work. Computational Fluid Dynamics (CFD) is a computational method used in recent years to obtain results for various parameters by numerical computation using software. CFD modeling and analysis allows to impose different constraints and boundary conditions for testing purposes without any involvement of actual cost experimentation. Also, results can be analyzed for different conditions. This paper presents a brief review on CFD work available in the literature on heat transfer in straight tube and helical coil heat exchangers. The numerical work reported in the literature for straight tube heat exchangers and helical coil heat exchangers is presented in separate sections.

2. Review on CFD work in Straight tube heat exchangers

Shell-and-Tube heat exchanger is the basic type of straight tube heat exchanger. Mostly, tube used is of circular cross-section. Nasr and Shafegat[1] developed a setup of shell and tube heat exchanger with segmented helical baffles for their research. Fluid flow pattern were studied in this paper. Comparison of numerical analysis had been carried out between different types of baffles in the heat exchanger. Segmented and helical baffles with 40° angle were compared. Also helical baffles of different angles were compared in this paper. The comparison was done on FLUENT 6.0 on various heat exchanger parameters of the CAD model. The results of this numerical analysis were shown graphically and various parameters were studied by the observation of the inferences drawn from the graph. By the results, it was concluded that the helical baffles heat exchanger was found to be perfect replacement for the shell and tube heat exchanger. Relationship between area, heat exchanger coefficient and pressure drop were derived for shell side developed in the helical baffles. All the parameters were better for helical baffles. However, when a pressure drop is observed, thermal duty has to be filled with a bigger cross sectional area. Usually the optimum results were seen in small cross sectional area only. Results were seen to be optimum at 40° whereas results for 20° were very close to the results seen in 40°.Results for other angles was found to be varying.

Ozden and Tari[2] studied the effect of baffles on straight tube heat exchangers and their performance. The shell side design of a shell-and-tube heat exchanger; in particular the baffle spacing, baffle cut and shell diameter dependencies of the heat transfer coefficient and the pressure drop were investigated by numerically modeling a small heat exchanger. The flow and temperature fields inside the shell were resolved using a commercial CFD package. They used Spalart–Allmaras and $k-\varepsilon$ model for analyzing the turbulence effect. Particle velocity path line was observed for baffle cut 25% and 36% by changing the number of baffles i.e. 6,8,10, & 12. Recirculation was observed for lower number of baffles. Thus, it was reported that in order to improve effectiveness, number of baffles must be increased.

Sarma and Das[3] predicted the performance of a shell and tube finned heat exchanger for waste heat recovery. The performance parameters such as effectiveness, overall heat transfer coefficient, energy extraction rate etc., were considered. The CFD analysis for given parameters was performed in FLUENT 6.3.16. The model for analysis was developed in GAMBIT 2.4.14. $k-\varepsilon$ model was used for analysis of turbulence in fluid. Analysis was performed for different varying velocity range of fluid from 0.85 m/s to 1.45 m/s with temperature of $25^\circ C$, keeping the shell side fluid constant at the speed of 0.0709 m/s and has a temperature of $120^\circ C$ Pressure drop along the tube was found to have greater value. This implies that the pumping power of fluid should be high to nullify the pressure drop in order to attain desired pressure. It was observed that length of the pipe can be shortened to overcome the pressure drop. No back pressure flow was observed in the CFD results. Outlet temperature of fluid seems to decrease with increase in tube diameter. This indicates better heat transfer between the fluids. The CFD results were compared with the experimental results for effectiveness. The results were found to be in good agreement, having an error range of 6.4%.

Thundil and Ganne[4] studied various heat transfer parameters in the numerical analysis of the baffles of the shell and tube heat exchanger. The inclinations used were 0°, 10° and 20° and their results were compared. The numerical analysis was done on ANSYS CFX 12.1. It was observed that temperature gradually increases from 300° K to about 340° K. The pressure drop was found to be less for 20° baffles as compared to the other two. A smoother flow was observed in baffles other than 0°. The flow and temperature fields were resolved in the shell side of a small shell and tube heat exchanger. The mass flow rate must be below 2 kg/s, if it was more than 2 kg/s, the pressure drop was found to increase at a faster rate with little change in the outlet temperature. The baffles could be used effectively if the angle was beyond 20°, since the centre row of tubes are not supported. All the parameters were found to be better for the baffle with 20° inclination angle and hence it was the optimum angle for the given numerical problem. Patel and Mavani[5] designed shell and tube heat exchanger and conducted numerical analysis. HTRI software was used for the numerical analysis of the problem. Heat transfer area and pressure drop were varied in this paper and various heat transfer parameters were observed in order to design the best shell and tube heat exchanger for the given conditions. It was observed that pressure drop was directly proportional to the fluid flow rate. The pitch ratio, length and the layout of the tube and the baffle spacing were found to have a major effect with the change in pressure ratio.

You *et.al.*[6] performed analysis by developing a numerical model and meshed using GAMBIT and the numerical analysis was done using FLUENT. Concepts of porosity and permeability were studied for the shell-side flow and heat transfer of shell-and-tube heat exchangers (STHXs). The distributed resistance, heat source, the distributed turbulence kinetic energy and its dissipation rate were studied. The numerical analysis was done at shell-side Reynolds numbers over a wide range from 6813 to 22,326 for a shell tube heat exchanger with flower baffles. The contours of the velocity and temperature fields were studied for the heat exchangers with and without flower baffles. It was observed that due to flower baffles, the fluid velocity magnitude as well as convective heat transfer coefficient vary in a periodical way at the centre of the FB-STHX, and three regions with small, moderate or large convective heat transfer coefficients are generated after the flower baffles. The heat transfer rate is affected on the shell side considerably. Overall thermal hydraulic performance was better for FB-STHX than the SG-STHX.

Parmar and Chopra[7] presented mathematical modelling of cross-counter flow heat exchanger. In this paper an attempt was made to analyze the performance of shell and tube type cross counter flow heat exchanger by changing the various parameters like both hot and cold fluid flow rate, direction of fluid flow. After changing the various parameters, the maximum performance obtained. For that the mathematical model of counter flow heat exchanger was adopted and also the analysis of the heat exchanger was carried out. Ɛ-NTU Method A new numerical methodology for thermal performance calculation in cross-flow heat exchangers was developed. Effectiveness-number of transfer units (ƐNTU) data for several standard and complex flow arrangements was obtained using this methodology. The maximum performance for cross counter flow heat exchanger by varying the various parameters was determined. A method based on Ɛ-NTU approach was utilized for the analysis of heat exchanger performance. For that the data of shell & tube cross-counter flow heat exchanger was used. One experiment on cross counter flow shell & tube heat exchanger was performed in laboratory and data of four different temperatures of hot and cold fluid was obtained by changing the various parameters. Here, one mathematical model was also utilized for calculation of effectiveness of heat exchanger. It was observed that with increase in the cold fluid flow rate by 10% to 90%, the effectiveness and also NTU decreases slowly up to 40% and then very fast from 50% to 90%.Also, with decrease in the cold fluid flow rate by 10% to 80% ,it was observed that the effectiveness and also NTU increases continuously. By changing the direction of fluid flow i.e. using parallel flow instead of counter flow arrangement, and keeping the NTU constant, the effectiveness decreases by 4.43%. It was concluded that the maximum performance (effectiveness) could be obtained by decreasing the hot fluid flow and keep the cold fluid flow constant for this particular heat exchanger.

Jadhav and Koli[8] analyzed shell and tube heat exchanger for heat transfer properties and effect of baffles on pressure drop. Numerical modeling was performed on shell side of heat exchanger to observe change in heat transfer co-efficient and pressure drop. Analysis was performed in ANSYS FLUENT. A small heat exchanger model was developed with 6 baffles, as larger model would consume more computational time. For the given model with two types of baffles, it was found that the CFD analysis showed a lower pressure drop value at baffle cut of 30% than that of value observed for baffle cut at 25%. Thus it was noticed that, less pumping power was required and greater heat transfer was achieved in the model.

Yang *et.al.*[9] performed 3-D numerical simulations of a rod-baffle shell-and-tube heat exchanger with four different modeling approaches were developed and validated with experimental results. In two models there were subsection of the heat exchanger, in one heat exchanger was a porous medium and in one it was a whole medium. Meshing was done using GAMBIT. Five different grid systems were considered, 5×10^4, 1.9×10^5, 3.8×10^5, 5.3×10^5 and 7.5×10^5 cells were adopted for calculation. Grid independence test was performed and forth model was chosen. The standard wall function method was adopted for the near-wall region, and the non-slip boundary condition was adopted on all solid surfaces. The surfaces of the rod baffles were set as adiabatic because the impact caused by thermal conduction of the rod-baffle could be neglected, and taking the rod-baffle surface as adiabatic allows for a coarser grid. The inner wall of shell-side was also set as adiabatic because the heat exchanger is thermally isolated during the experiment. It was concluded that the periodic model, porous model and whole model have a high accuracy on predicting heat transfer performance, while the unit model had a relatively low accuracy; it was concluded that the porous model and whole model has high accuracy on predicting pressure drop, while unit model and periodic model are unable to directly predict hydraulic performance.

Wen *et.al.*[10] numerically compared the shell side flow patterns in original plain helical baffle and the improved ladder type helical baffle in straight tube heat exchangers. The investigators used periodic modelling with 2 cycles to study the shell side performance of straight tube heat exchangers with helical baffles. They used ICEM CFD software for meshing the numerical domain with unstructured tetrahedral grids. The investigators have conducted a series of grid independent test on both the models and came to the conclusion that optimum grid numbers were 6,482,355 and 6,299,216 for improved and original model respectively. $k-\varepsilon$ model was used to study the turbulent flow rates. ANSYS Fluent 15 was used to simulate fluid flow and heat transfer in both the models. Non slip boundary condition was adopted on the wall. Conductive oil was used as shell side working fluid. Mass flow was used as periodic boundary condition at inlet. Both the models were solved by finite volume method with SIMPLE pressure velocity algorithm. Convective terms were discretized by QUICK scheme and 3rd order precision. 1 x 10^{-4} for flow equations and 1 x 10^{-8} for energy equation was set as convergence criteria. The investigators observed the variation of overall heat transfer coefficient with flow rate and tube bundle pressure drop with flow rate in both original and improved model. They observed that velocity was lower near shell centre and higher near shell wall in plain baffle and velocity was higher and uniformly distributed in ladder type baffle. Temperature was also uniformly distributed in ladder type baffles whereas in plain baffle type, temperature was higher near shell wall and gradually reduces towards the shell centre. They came to the conclusion that uniform distribution of temperature improves the performance of the heat exchanger. Numerical analysis was validated with experimental results. They observed 5.55 to 9.62% deviation in overall heat transfer coefficient for experimental and numerical data and 6.72 to 12.68% deviation in tube bundle pressure drop for experimental and numerical data. Thus, they came to conclusion that their periodic model was quite accurate.

3. Review on Helical heat exchangers-

Li *et.al.*[11] analysed for effect of buoyancy over turbulent flow of fluid in helical coil. A dimensionless ratio parameter considered by them which was $\sqrt{Gr/Dn}\sqrt{1+\mathrm{Pr}}$. The Reynolds Number was taken as 5×10^4, gap ratio was taken as 0.05, the Grashoff number was considered in the range 5×10^8 to 10^{10}. SIMPLEC method was used to resolve the coupling between the velocity and pressure. The Grashoff Number was found to be increasing and secondary flow was developed showing three vortices. Earlier papers did not show the effect of buoyancy. CFD results showed good agreement with existing experimental data referred. It revealed about the development of flow and temperature fields. They derived that the local Nusselt Number is affected by buoyancy forces only when

$$\sqrt{Gr / Dn}\sqrt{1+\Pr} \geq 1.2. \quad (1)$$

Acharya *et.al.*[12] worked on steady heat transfer enhancement due to chaotic particle paths in steady, laminar flow through a tube. Computation of velocity vectors and temperature fields was done. Spatially varying local and constant bulk Nusselt number and bulk friction factors were determined for a range of governing parameters. They established quantitatively the viability of using the concept of chaotic mixing as a useful tool in designing as efficient coiled tube heat exchanger. They also reported that alternating axis coil geometry displays a heat transfer enhancement of 7-20% in terms of the fully developed Nusselt number with little change in the pressure drop. The value of Reynolds number was found in the given range of $50 \leq Re \leq 1200$.

Kumar *et.al.*[13] performed numerical analysis, hot fluid was considered in the tube side and cold fluid was considered in the annulus area. Outer tube consisted of semi-circular plates that supported inner tube which provides high turbulence in annulus region. FLUENT 6.0 was used for the numerical analysis. Lebnon and NH software were used for the CAD modelling. Nusselt number and friction factor were found for inner and outer tube. Tests were conducted for 3 different diameters of inner and outer tube which covered a wide range of Reynolds number. Friction factor is given as-

$$f_c = 0.02985 + \frac{75.89[0.5\tan^{-1}((N_{De} - 39.88)/77.56)/\pi)]}{(D/(d_{i,out} - d_{o,in}))} \quad (2)$$

where $35 < N_{De} < 20000$, $1.61 < d_{i,out}/d_{o,in} < 1.67$, $21 < D/(d_{i,out} - d_{o,in}) < 32$. Here, N_{De} is Dean Number. Deviation of 15% was found in friction factor of experiments and correlation. Standard k-ε model was used for flow and heat transfer results. Reynolds stress was related to mean velocity gradient on the basis of Boussinesq hypothesis. It was observed that when the overall heat transfer results increases the value of dean number for the inner coiled tube also increases. It was also observed that the deviation in the value of Nusselt number was within 4% for inner tube and 10% for outer coiled tube.

Kumar *et.al.*[14] studied the hydrodynamics and heat transfer characteristics of a coiled flow inverter (CFI) on a pilot plant scale. Heat transfer results were studied for Reynolds number ranging from 1000-16000. The main aim was to characterize the performance of CFI as heat exchanger for a water-water or water-air counter current flow on a pilot plant scale. It was found that when the Reynolds number increases, the influence on heat transfer decreases. Dean roll cells avoids mixing in hot walls and values of Dean Number = 3 and dispersion number = 0.0013. It was observed that flow rate was higher for tube side as compared to the shell side. It was observed that at low Reynolds number the heat transfer results was 25% higher and at higher Reynolds number it was 12% higher as compared to the coiled tube data reported in the literature. Jayakumar *et.al.*[15] worked on fluid to fluid heat transfer which weren't researched earlier. Conjugate heat transfer is considered. FLUENT 6.2 was used for the numerical analysis of the problem. Specification of constant temperature or constant heat flux boundary condition for an actual heat exchanger does not yield proper modelling. Inside and outside convective heat transfer and wall conduction was considered. Temperature dependent values of thermal and transport properties of heat transfer medium are also used which was not done earlier. Relations for μ, ρ, κ, Cp were obtained by regression analysis using MATLAB and were later programmed in FLUENT. An error Nu is about 24% when properties at ambient conditions were used. When the same was seen for mean temperature error was about 10%.Flow rate of the hot fluid through helical pipe was low.

Kumar *et.al.*[16] performed analysis of setup consists of a tube in tube heat exchanger. CVFDM was used for the numerical analysis of the problem. Renormalization group (RNG) k-ε model is used to model the turbulent flow and heat transfer in TTHC heat exchanger. Friction factor in TTHC was found to be different. For inner tube, as the Reynolds number increases, the friction factor also increases. For outer tube, following relation was developed-

$$\frac{fc}{fs} = 1 + 0.0927 N_{DE}^{0.551} \quad \text{For } 300 < N_{De} < 900. \tag{3}$$

The accuracy of the numerical analysis as compared to the experimental analysis was ±2%. The overall heat transfer coefficient increases with increases in inner coiled tube flow rate for a constant flow in the annulus region. As the operating pressure increases, the overall heat transfer coefficient for the inner tube also increases. Kharat *et.al.*[17] carried out CFD analysis for concentric helical coil heat exchanger in order to develop a correlation for heat transfer coefficient in the annular region between two coils. In the coil used the pitch was equal to the diameter of the tube. Parameters like gap between the coil, tube diameter & coil diameter was varied. Solid edge V20 was used for making the model which was imported by GAMBIT for meshing and the numerical analysis was carried using fluent 6.3.26. Heat transfer coefficients were plotted against coil gap using CFD analysis and it was concluded that heat transfer coefficient decreases with increase in gap. A correlation for Nusselt number was developed which is

$$Nu = 0.02652604 \mathrm{Re}^{0.834694285} \mathrm{Pr}^{0.3} (\text{Gap ratio})^{-0.096856199} \tag{4}$$

Jayakumar *et.al.*[18] provided detailed study of fluid flow characteristic and heat transfer for helical coil heat exchanger done by CFD analysis. They analyzed the variation in local Nusselt number along length and circumference. Analysis was done by varying various parameters like Pitch circle diameter, tube pitch & tube diameter. Meshing was done using Gambit 2.3 & then exported to Fluent 6.3 for numerical analysis. For turbulent flow $k-\varepsilon$ model was used. Movement of 10 fluid particles was plotted and fluid particles took various trajectories from inside to outside and vice versa. Nusselt number showed oscillatory behaviour; also Nusselt number at the outer side was higher than inner side. Similar results were observed for constant heat flux boundary condition. The Nusselt numbers increased with increase in pitch as well as tube diameter whereas Nu decreased with increase in PCD.

Elazm *et.al.*[19] conducted research on the effect of taper angle and helical pitch on the heat transfer characteristics of helical cone coils. 22 cases were taken under consideration for analysis by varying taper angle, coil pitch and pipe diameter. A comparison was done between CFD results of the developed taper model and the values obtained through equation suggested by Manlapaz and Churchill for ordinary helical coil. The values for equations were calculated through MATLAB. The equation is as follows-

$$Nu = [(3.675 + \frac{4.343}{x_1})^3 + 1158(\frac{De}{x_2})^{3/2}]^{1/3} \tag{5}$$

Where $x_1 = (1 + \frac{957}{De^2 \mathrm{Pr}})^2$ and $x_2 = (1 + \frac{0.477}{\mathrm{Pr}})$

On observing results, they founded out that there is no significant error until $\theta = 40^\circ$. They deduced that with introducing a taper angle, the outlet temperature of the fluid had significant changes. The value of outer temperature increases with increase in taper angle. This was the effect of taper angle, which provided greater heat transfer area. Accordingly, it was found that the Nusselt number also increased with increase in taper angle. Also increase in heat transfer rate was observed. With increasing pitch, heat transfer co-efficient and Nusselt number increases when increasing the pitch. Kumar and Karanth[20] presented a case of cooling hot water through heat exchanger by applying fixed wall temperature boundary conditions. Temperature drop, pressure drop, heat transfer co-efficient and Nusselt number are the parameters that are calculated experimentally and used for comparing it with the obtained CFD results. CFD computation was carried out for three different mass flow rate - 0.005 kg/s, 0.02 kg/s, 0.05 kg/s for helical heat exchangers. The particle behaviour in helical coil was observed to be in line at inlet side, but found to be in scattered state at outlet. Due to this, Nusselt values undergoes fluctuation and thus the heat transfer co-efficient is affected. For Nusselt number, average error found was up to 5% for helical coil. It is observed that in helical coil, temperature drop observed was high. When mass flow rate increases from 0.005 kg/s to 0.05 kg/s, heat transfer rate was observed to be increased by 11%. The CFD results and correlation had good agreement of results.

Tayde *et.al.*[21] studied the characteristics of heat transfer inside a helical coil for different boundary conditions. CFD methodology was applied for the analysis by using FLUENT. The main objective was to compare the CFD simulations results for heat transfer analysis obtained by using the above boundary conditions. Fluid properties tend to change with change in temperature. Thus, temperature dependent analysis was also done to check the thermal and transport properties of the heat transfer medium. Comparison of all these results showed that use of constant thermal properties and transport properties gives inaccurate prediction. Constant wall temperature as an arbitrary boundary condition is also a reason of inaccurate result prediction. The analysis done by considering fluid properties based on temperature was found to give near to accurate results and have less error. Chaves *et.al.*[22] compared helical coil heat exchanger with two and three coils using CFD.Analysis was carried for various inlet hot water temperature (25,30,35, & 40) and cold water temperature 20 C.ANSYS CFX V12 was used for numerical analysis. No slip boundary condition was considered. Hot fluid velocity was 0.01 m/s (Re=2562) , cold water velocity was 0.1 m/s (Re=425). The total number of nodes were 398596 and number of iteration was 200.CFD analysis for various hot water temperature was performed and results were obtained which concluded that the 3 coil is more effective than 2 coil heat exchanger only for higher temperature difference. Thundil *et.al.*[23] performed CFD analysis for helical coil heat exchanger was performed in order to determine the tube side heat transfer coefficient for constant wall temperature. $k-\varepsilon$ Model was used for turbulent region, wall temperature, hot water temperature, velocity inlet boundary condition at top and pressure outlet boundary condition at the bottom was specified. Conservation of mass, momentum and energy equation are fundamental equations. The values from CFD analysis was found to be in good agreement with experimental values. There were deviations noted which increased with increase in dean number. From the temperature contour it was concluded that larger pitch was better for heat transfer.

Pawar and Sunnapwar[24] conducted numerical computation in FLUENT for helical coils of three different dimensions. Heat exchange condition was modeled with temperature of 62.5 $\pm$ 0.2 $°C$ for natural convection in vessel, conduction through tube wall and forced convection heat transfer. Values of Nusselt numbers by CFD and experiment were compared. The average value of Nusselt number based on experimental data was found to be 6.38% higher as compared to the average value of the Nusselt number for temperature dependent fluid properties. Value of outlet temperature by thermometer measured was higher than CFD by 3.03%. For outer wall temperature, experiment and CFD results had deviation of 2%. Value of heat transfer rate differs for 11.10% for CFD results with experimental results for temperature dependent properties, while 9.84% for constant temperature properties, thus proving the authenticity of measured parameters.

Bandhopadhyay *et.al.*[25] performed detailed study of fluid flow characteristic, pressure drop and heat transfer for helical coil heat exchanger for Non Newtonian psuedoplastic liquid by experimental and CFD analysis. Fluent 6.3 was used for CFD analysis. Standard schemes – STANDARD for momentum and 1st order upwind for other variables was used and Pressure-velocity SIMPLE coupling is used. The contour plot of static pressure decreases with increasing coil turn. It is observed that the pressure is more at the outer side wall and less at the inner wall. This is due to the action of centrifugal force. Frictional pressure drop per unit length of coil increases with increasing coil diameter. CFD results are validated by experimental results and results from Mishra and Gupta (1979). Seyyedvalilu and Ranjbar[26] studied CFD analysis of helical double pipe heat exchanger, in which hot fluid flows through inner tube and cold fluid flows through annular region. Several parameters were taken into account such as Dean Number, curvature ratio, tube diameter, and coil pitch. Curvature ratio was founded to be important parameter in helical double tube heat exchangers and it is presented by $d_o/2R$. Dean Number includes curvature ratio and it has to be noted that both parameters of d_o and R can change curvature ratio. As curvature ratio increases, the heat transfer increases drastically. As the radius of the loops increases, fluid's torsion behaviour approaches to linear behaviour and also helical tube turns to straight tube. The increase in pitch decreases the heat transfer, however this decrease is negligible. Increasing of number of coils means longer heat transfer path and observed decrease in Nusselt Number due to number of coils and this decrease in Nusselt Number due to number of coils indicates that flow is thermally developing.

Imran *et.al.*[27] researched upon turbulent flow model with counter flow heat ratio to find the conversions ratio. ANSYS FLUENT was used for numerical analysis of the problem. It was found that as the Reynolds Number increases the Nusselt Number increases for inner tube. The mixing of the fluid enhances. The curvature ratio was found to be inversely proportional to the Nusselt Number. The value of the Nusselt Number was found to be maximum, when the curvature ratio is 25. The outer wall has no major effect on the Nusselt Number. As the friction factor decreases, the Reynolds Number increases due to the relative roughness of the surface and the velocity of the flowing fluid. Logarithmic mean temperature difference increases at a steady rate as Reynolds Number increases. For hot fluid any condition can be assumed for the outer wall because it does not affect heat transfer rate significantly. Palande and Mhaske[28] conducted CFD analysis for helical coil and founded that inner tube flow rate increases so that the Reynolds Number and Dean Number increases for constant cold water flow rate. As the inner tube flow rate increases the logarithmic mean temperature difference increases, the temperature difference ($\Delta t_1 - \Delta t_2$) for constant cold water flow rate. As the inner tube flow rate increases, heat gained by the cold water increases, overall heat transfer coefficient increases for constant cold water flow rate.

Mishra[29] performed study of the deviation of Nusselt Number for different curvature ratio and Reynolds Number. Heat flux for outer wall was kept constant and other conditions were varied. As the coil diameter was increased the length of the heat exchanger was also increased. Fluid particles were undergoing an oscillatory motion inside both the pipes. Values for velocity and pressure were higher for outer tube comparatively. Values for the shear stress were higher for outer tube comparatively. It was found that Nusselt Number depends on the curvature ratio. Nusselt Number and frictional factor has maximum value for curvature ratio 10 and minimum value for curvature ratio 30. Balachandran[30] performed detailed study of effectiveness, overall heat transfer is performed by experimental and CFD analysis for water as a fluid. Solidworks Flow simulation is used to perform the analysis. When there is an increase in mass flow rate, the De and Nu increases for a particular hot fluid inlet temperature. When the Dean number increases the overall heat transfer coefficient U also increases. It was predominantly seen when the flow transmits from laminar to turbulent. More amount of heat transfer occurred between hot and cold fluid as the mass flow rate of hot fluid increases. It was seen that effectiveness of helical coil heat exchanger increases as the mass flow rate increases. So performance of this heat exchanger was found to be comparatively higher than shell and tube heat exchanger.

4. Conclusion

From literature review, considerable amount of CFD and numerical analysis for straight tube and helical coil heat exchangers are carried out under various geometrical and experimental conditions. For the different parameters and their values in various conditions, helical coil heat exchanger is found to have better efficiency as compared to straight tube heat exchanger. However, it was found that there is no single evidence of CFD work which represents analysis of these both heat exchangers by considering same length under isothermal condition in laminar and turbulent flow regimes to know actual advantages of these two geometries. This would be helpful in better understanding of both types of heat exchangers and their performance parameters.

REFERENCES

1. M.R. Jafari Nasr, A. Shafeghat, 'Fluid flow analysis and extension of rapid design algorithm for helical baffle heat exchangers', 2008.

2. Ender Ozden, Ilker Tari, 'Shell side CFD analysis of a small shell-and-tube heat exchanger', 2010.

3. Swapnaneel Sarma, D. H. Das, 'CFD Analysis of Shell and Tube Heat Exchanger using triangular fins for waste heat recovery processes', 2012.

4. Rajagapal Thundil Karuppa Raj, Srikanth Ganne, 'Shell Side Numerical Analysis Of A Shell And Tube Heat Exchanger Considering The Effects Of Baffle Inclination Angle On Fluid Flow', 2012

5. Sandeep K. Patel, Prof. Alkesh M. Mavani, 'Shell & Tube heat exchanger thermal design with optimization of mass flow rate and baffle spacing', 2012.

6. Yonghua You, Aiwu Fan, Suyi Huang, Wei Liu, 'Numerical modeling and experimental validation of heat transfer and flow resistance on the shell side of a shell-and-tube heat exchanger with flower baffles', 2012.

7. Parmar Kalpesh D., Prof. Manoj Chopra, 'Performance Analysis Of Cross Counter Flow Shell And Tube Heat Exchanger By Experimental Investigation & Mathematical Modelling', 2013.

8. Avinash D. Jadhav, Tushar A. Koli, 'CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop', 2014.

9. Jie Yang, Lei Ma, Jessica Bock, Anthony M. Jacobi, Wei Liu, 'A comparison of four numerical modeling approaches for enhanced shell-and-tube heat exchangers with experimental validation', 2014.

10. Jian Wen, Huizhu Yang, Simin Wang, Yulan Xue, Xin Tong, 'Experimental investigation on performance comparison for shell-and-tube heat exchangers with different baffles', 2015.

11. L. J. Li, C. X. Lin, M. A. Ebadian, 'Turbulent mixed convective heat transfer in the entrance region of a curved pipe with uniform wall temperature', 1998.

12. Narasimha Acharya, Mihir Sen, Hseuh-Chia Chang, 'Analysis of heat transfer enhancement in coiled-tube heat exchangers', 2001.

13. Vimal Kumar, Supreet Saini, Manish Sharma, K.D.P. Nigam, 'Pressure drop and heat transfer study in tube-in-tube helical heat exchanger', 2006.

14. Vimal Kumar, Monisha Mridha, A.K. Gupta, K.D.P. Nigam, 'Coiled flow inverter as a heat exchanger', 2007.

15. J. S. Jayakumar, S. M. Mahajani, J. C. Mandal, P. K. Vijayan, Rohidas Bhoi, 'Experimental and CFD estimation of heat transfer in helically coiled heat exchangers', 2008.

16. Vimal Kumar, Burhanuddin Faizee, Monisha Mridha, K.D.P. Nigam, 'Numerical studies of a tube-in-tube helically coiled heat exchanger', 2008.

17. Rahul Kharat, Nitin Bhardwaj, R.S. Jha, 'Development of heat transfer coefficient correlation for concentric helical coil heat exchanger', 2009.

18. J.S. Jayakumar, S.M. Mahajani, J.C. Mandal, Kannan N. Iyer , P.K. Vijayan, 'CFD analysis of single-phase flows inside helically coiled tubes', 2010.

19. Mohamed Abo Elazm, Ahmed Ragheb, Ahmed Elsafty, Mohamed Teamah, 'Computational Analysis For The Effect Of The Taper Angle And Helical Pitch On The Heat Transfer Characteristics Of The Helical Cone Coils, 2012.

20. Shiva Kumar, K. Vasudev Karanth, 'Numerical Analysis of Helical Coiled Heat Exchanger using CFD', 2013.

21. Madhuri Tayde, Jeevan Wankhade, Shylesha Channapattana, 'Heat Transfer Analysis of a Helically Coiled Heat Exchanger', 2014.

22. C. A. Chaves, D. R. F. de Castro, W. Q. Lamas, J. R. Camargo, F. J. Grandinetti, 'Computational Fluid Dynamics (CFD) simulation to analyze the performance of tube-in-tube helically coiled of a heat exchanger', 2014.

23. R. Thundil Karuppa Raj, Manoj Kumar S., Aby Mathew C., T. Elango, 'Numerical Analysis Of Helically Coiled Heat Exchanger Using CFD Technique', 2014.

24. S. S. Pawar, Vivek Sunnapwar, 'Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger', 2014.

25. Tarun Kanti Bandyopadhyay, Sudhansu Sandhibigraha, Sudip Kumar Das, 'Experimental and CFD Analysis of Non-Newtonian Pseudoplastic Liquid Flow through Vertical Helical Coil', 2014.

26. Mir Hatef Seyyedvalilu, S. F. Ranjbar, 'The Effect Of Geometrical Parameters On Heat Transfer And Hydro Dynamical Characteristics Of Helical Exchanger', 2014.

27. Mohammed Imran, Gaurav Tiwari, Alwar Singh Yadav, 'CFD Analysis of Heat Transfer Rate in Tube in Tube Helical Coil Heat Exchanger', 2015.

28. Prof. D. D. Palande, Ganesh B. Mhaske, 'Experimental and CFD Analysis of Tube in Tube Helical Coil Heat Exchanger', 2015.

29. Triloki Nath Mishra, 'Modeling and CFD Analysis of Tube in Tube Helical Coil Heat Exchanger', 2015.

30. M. Balachandran, 'Experimental And CFD Study Of A Helical Coil Heat Exchanger Using Water As Fluid', 2015.