JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

CFD Investigation of Convective Heat Transfer in Spiral Coiled tubes

[1]Mangesh Shashikant Bidkar, [2]Dr. Rashed Ali
[1]ME Scholar, [2]Associate Professor
[1]Department of Mechanical Engineering,
[1]Pillai college of Engineering, New Panvel, India.

Abstract: Heat transfer efficiency will improve when fluid passes via pathway curvature pipes. It is generally known that heat transmission in a spiral coil is greater than in a straight pipe. The detailed characteristic of Heat transfer & flow of fluid is not shown in this report. This report brings out cfd analysis of how Improves transfer of heat & drop in pressure. With the help of this analysis, we will be able to predict Transferring heat & pressure drop inside a Spiral tube. CFD simulation is carried out for Spiral coils by varying coil parameters such as (i) diameter of tube (ii) Pitch of coil (iii) Number of Turns and their effects on heat transmission have been investigated. Since there is no Published Numerical, Analytical and Experimental details about spiral coils systems. In this report the laminar of flow pressure drop & Characteristics of transfer heat from spiral coil systems will be investigated with commercially available ANSYS FLUENT CFD package. The influences of parameters such as Newtonian fluid, Reynolds number, tube area or cross section, length to diameter ratio, coil pitch, number of turns on the Fluid flow as well as transfer of heat in spiral coils are investigated and presented. The CFD results are compared to previous researchers of experimental results, and they indicate good agreement.

IndexTerms - *Spiral Coiled tube; Curvature ratio; CFD simulation; Curved Tubes.*

I. INTRODUCTION

Spiral coils are commonly used curved tubes used in a wide range of technical applications such as heating, cooling, HVAC field, steam generators and condensers in power plants and nuclear reactor. In a coiled tube flow of secondary originates in the pipe cross section because of the variations between the inertial and centrifugal powers. The output of the secondary flow produced has a big impact on how heat and flow transfer in these tubes develops. Active and passive approaches can be used to improve the efficiency of heat exchangers or any other thermal system that uses coils for heat transfer. The coils under examination are exposed to external forces in the active approach, whereas heat transfer enhancement is done in the passive technique by using alternative surface geometries or adding different additives to the fluid. It is mentioned in Kubair and Kuloor[8] analyzed spiral coil without taking account the coil's indicative variables. For 'Graetz problem,' Kubair and Kuloor[1] examined the motion of fluids passing through the 2 spiral coils. Their analysis shows the effect of the length to mean spiral coil diameter ratio on transfer of heat. The curvature ratio is the most essential indicative geometric factor of the spiral coil. Kubair and Kuloor[2] studied the two spiral coils for the 'Graetz problem', taking into account the curvature ratio of both spiral coils. This is the only relation available for 'Graetz problem' on spiral coils which implies curvature ratio term. Kubair & Kuloor study with such a combination of helical coils, straight tube, and spiral coil for thermal transfer [3]. The efficiency of spiral coils has also been proven to be superior to that of straight pipe or helical coils. This same author [4] created correlations of friction considerations throughout spiral coil tubes through laminar, transition and turbulent flowing for various liquids. Paisarn Naphon et al. [6] studied to influence of a curvature ratio on both the transfer of heat & flow formation throughout the spirally coil horizontally tubing. A effects for the convective heat transmission and flowing patterns that they expected was in good accordance with experimental experiment. Their studies show that the centrifugal force has a major impact on improving heat transfer and dropping pressure. Since of the centrifugal force that Nusselt numbers as well as decrease in pressure derived from spiral coil tube is 1.49 to 1.50 times greater than with the regular straight tubes. Naphon et al. performs a study of both the liquid flow & transfer of heat for curved coils [7]. A recent development in the field of Anthony Bowman and Hyunjae Park [5], study the transfer of heat and pressure drop properties in toroidal & spiral coil. CFD Examination for toroidal tube system has been performed to anticipate the laminar stream and heat transfer characteristics and compared with the available Experimental test and numerical results. The analysis was conducted for non-slip wall condition and it is imposed at the inner wall surface of the coiled tube. From their results it is imposed at the inner wall surface of the coiled tube. From their results it is concluded that effectiveness ratio for spiral tube heat exchangers is 20-30% higher than toroidal tube heat exchangers. Due to this preferable selection was spiral coil heat exchanger over toroidal. A new type of spiral heat exchanger was designed by Dr. Madhukar S. Tandale and Dr. Sandeep M. Joshi.[8] for a process industry in which waste heat was recovered from a producer gas and it was used to generate the steam. The design was programmed to conduct simulation tests on a computer in order to access the performance of the prototype. In their design they had used a new type of flow arrangement for Fluid Warm and cold in which Guiding of the hot fluid was in the axial path while the cold fluid flow was in spiral path. 12% maximum deviations were observed when the calculated values of overall heat transfer & the theoretical system was compared to experimental system. Comparison of the pressure fall value with the known correlations and the actual values observed which Was Discovered within acceptable range. Effect of Coil Diameter on Pressure Drop in Archimedean Spiral Coils was investigated by Rakesh Baghel et.al [9] according to their investigation presence of the secondary flow inside the coil and centrifugal flow, due to curvature ratio increases the pressure drop inside the Archimedean spiral coils. For this, appropriate experimental results should be produced to test above said hypothesis for liquid-to fluid mix convection type transfer of heat in spiral coils tube. In designing spiral coil heat exchangers, the calculation of heat transfer coefficients for isothermal equilibrium conditions is an essential step. The main goal of this work in this field was also to verify an experimental observation with the results of the CFD analysis for Newtonian liquids using constant fluid property, and to observe differences in spiral coil flow patterns for different Nusselt number values. The general objective of conducting CFD analysis in Ansys was to validate the experimental data such as: inner Nusselt number, total heat transfer coefficient, exit temperatures, calculated by thermocouples with CFD calculation results using FLUENT CFD package solver in a turbulent, laminar flow region.

II. EXPERIMENTAL DATA ANALYSIS

The experimental results that is oil outlet temperature, inside heat transfer coefficient, and Nusselt Number as reported by Rahul patil, Nadar, Rashed Ali [13]. The arrangement of experimental setup used for validating the CFD results. The experimental procedure is as shown in Fig. 1. The range of variables covered in various coils and tests fluids used in the present investigation are also reported by Rahul patil, Nadar and Rashed Ali (2017). The physical dimensions of spiral coils Are they given in Table: 01 and Table 2 provides the different test fluids used in this work. Within the spiral coil test segment, the thermo-physical effects for study fluids are considered to be stable along the length of the coil and tested at average bulk temperature. Dynamic viscosity, kinematic viscosity, SN150 oil thermal conductivity and density from Andrew (1993).

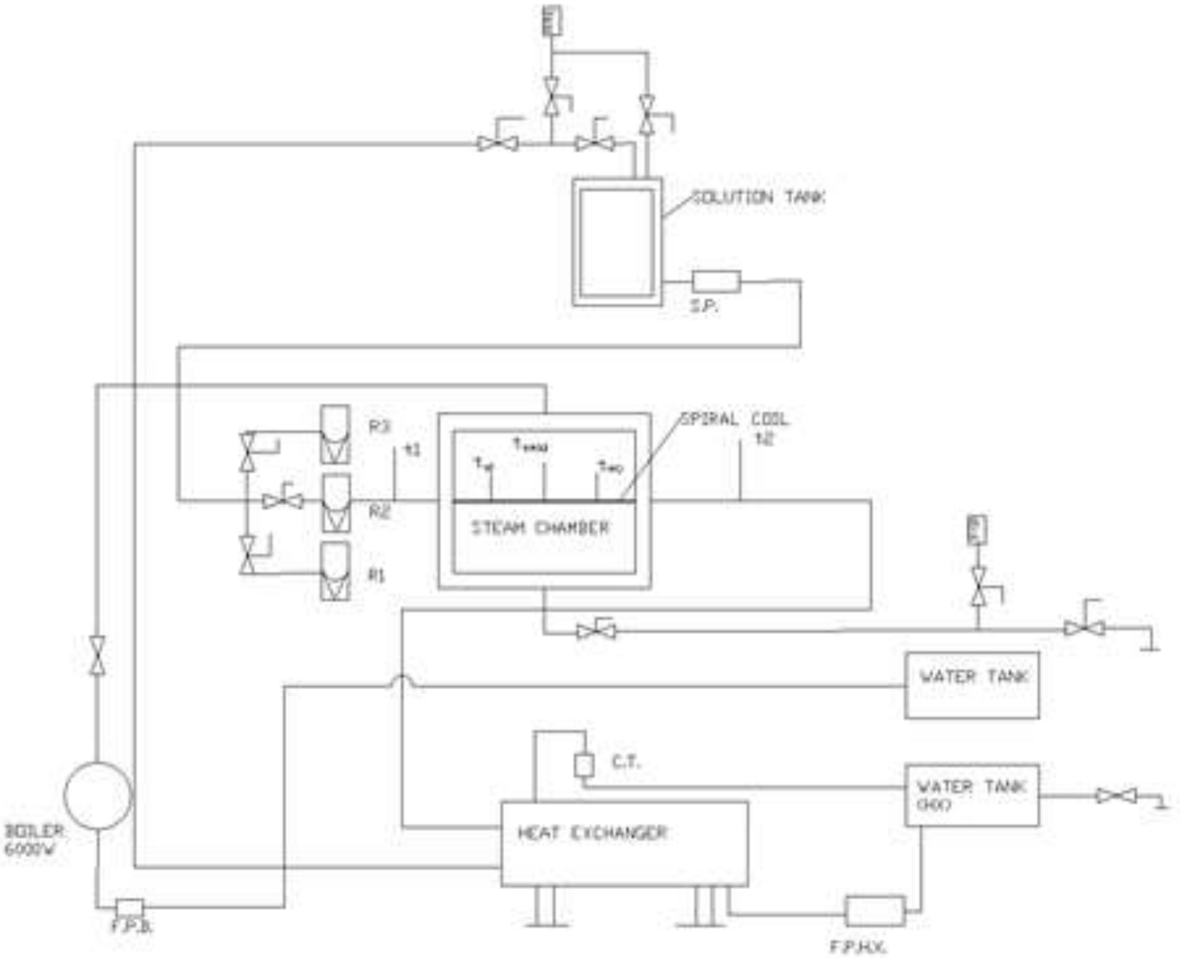


Fig. 1: Schematic layout of experimental setup.

Table 1: Physical Dimensions of Spiral coil used									
Spiral Coils	**Maximum diameter of coil (D_{co})**	**Minimum diameter of coil (D_{ci})**	**Avg. diameter of coil (D)**	**Pitch of coil (P)**	**Number of Turns (n)**	**Outer diameter of tube (D_o)**	**Inner Diameter of tube (D_i)**	**Ratio (D/D_i)**	**Ratio (P/D_i)**
Coil -I	700	100	400	25	12	9.85	8	50	3.125
Coil -II	542.4	100	321.25	18.44	12	7.91	5.9	54.45	3.125
Coil -III	400	100	250	12.5	12	5.91	4	62.50	3.125
Coil -IV	500	100	300	50	4	9.85	8	37.50	6.25

Table 2: Types of coil, Test fluid and Different flow rates used in this Experimental work.		
Spiral coils	**Test Fluid**	**Range of Different Mass Flow Rate (lps)**
Coil -I	SN150	0.0019 – 0.075
Coil -II	SN150	0.01898 – 0.06511
Coil -III	SN150	0.014973 - 0.049267
Coil -IV	SN150	0.00508 - 0.057049

III. NUMERICAL COMPUTATION

Numerical Simulations was run utilizing commercial CFD package software program for various Reynolds number in laminar as well as turbulent flow. The double precision, segregated, 3D version method is used to overcome the equations governing of mass, momentum and energy.

Equations and thermophysical properties regulating laminar flow:
SN150 oil is used here as a test fluid and is regarded as an incompressible, homogeneous, steady and Newtonian fluid of negligible impact of viscous heating. The Flow was Base on Equations Navier-Stokes using fluent package. The single-phase homogeneous flow governing equations in the Cartesian co-ordinates (x, y, z) are as follows:

Continuity equation:

$$\rho\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0$$

Navier–Stokes equations (momentum equations):

$$\rho\left(u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)-\frac{\partial p}{\partial x}$$

$$\rho\left(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)-\frac{\partial p}{\partial y}$$

$$\rho\left(u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)-\frac{\partial p}{\partial z}$$

Energy equation:

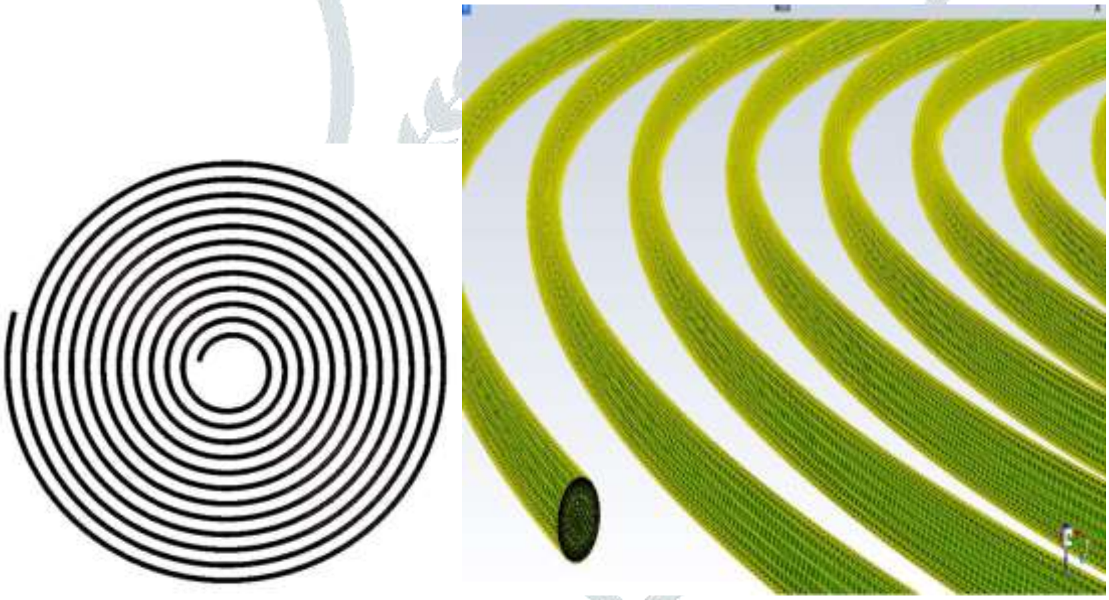
$$\rho\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}\right)=\frac{k}{C_P}\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial z^2}\right)$$

where p, T, u, v and w represents the pressure, temperature and Velocities in directions x, y & z, etc.

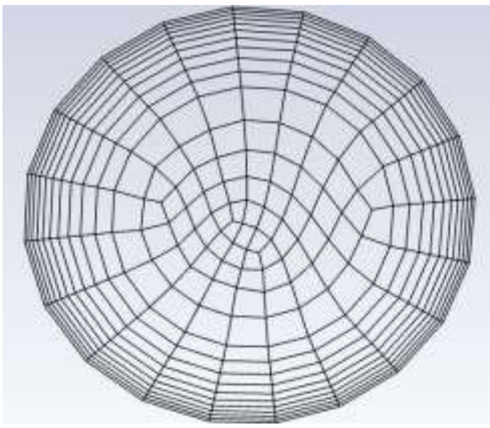
3.1 CFD MODELLING

The specific geometry used in the numerical modelling are the same as the As in the Figure 1 describes the experimental setup. & Figure 2 Shows a typical coiled spiral tube and the computational domain for this analysis, coil-1. The Coil-1 geometry was developed in solid works. Figure 2(a) & The mesh was then created, which is shown in Fig. 2(b) and (c) use of commercial CFD package. In this model, Volume of fluid in the pipe was meshed using uniform hexahedral grids as well as the mesh was formed at the boundary layer. Further increase or decrease in element size was found to be out of computer capacity. Heat flow from the hot steam in the steam chamber to the cold fluid in the spiral tube was modelled with convective heat transfer in the pipe (natural convection), conduction through the tube wall and convective heat transmission to the coil's fluid (forced convection). They are considered as no-slip ones for momentum equation. The material on the coil is pure copper with what heat conductivity specific heat 381 (J/kg K) Is considered a standard for shell conduction. The Coupled algorithm is used to treat thecoupling equations of continuity and momentum for solving velocity and pressure distribution. For laminar flow, viscous-laminar model Laminar model were used to solve the governing equations. In addition to such, a convergence criterion was provided as a surface monitor for outlet temperature based on the Facet average process. Table 3 describes the standard boundary requirements of the simulations used in this CFD study.

(a) CAD model of Spiral coil-1



(b) Complete Mesh Coil-1



(c) Enlarge Cross Sectional View of grids

Fig. 2 – Grids used in the analysis

3.2 Grid Generation

A grid dependence of the solution was examined for the size of three different mesh elements before finalizing the optimum coarse mesh, as shown in Fig. 2(c). For this heavy structure, that is out of range of the computer, fine meshing with smaller element size (Approx. 3289370 cells) Cannot possible. Further, with a greater number of cells of computational, cost of computing will increase as convergence and iteration consistency Are going to get more complicated, as well as memory and Processor time increase. Hence, grids shown in Fig. 2(c) uniform hexahedral grids is chosen for this present analysis which gives closer results to the experimental results with less than ±20% deviation in Nusselt number, inside heat transfer coefficient(hi) and outlet oil temperature. It results in cost-effective and time-efficient outcomes that are sustainable for the industrial applications. For an Intel, CORE i5 processor computer with 8 GB RAM the maximum time taken by FLUENT for some of the runs was observed at about 18 hrs.

Table 3: Typical Boundary conditions used for simulation.

Spiral coils	Test Fluid	Inlet Temp. of oil (°C)	Mass flow rate (lps)
Coil -I	SN150	36.4	0.019
	SN150	37	0.028
	SN150	37.7	0.034
	SN150	40.6	0.043
	SN150	41.8	0.053
	SN150	40.2	0.052
	SN150	41.8	0.075

IV. RESULTS AND DISCUSSION

In this section, description of temperature fields, and comparison of CFD heat transfer result with experimental data presented. This section is divided into following sections as: results and discussion for CFD analysis.

4.1 Comparison of CFD results with experimental results for different conditions.

Analysis of CFD tests on various conditions with experimental results in commercial CFD package, as shown in Table 3, a total of seven runs were performed for comparison of experimental results with CFD measurement results. All the parameters needed to calculate the outlet temp of oil, hi, & Nu as CFD results were taken from the results of seven runs of simulation performance.

4.1.1 Comparison of inside heat transfer coefficient (hi)

The inner heat coefficient values based on experimental data for coil-I, CFD output results constant temperature fluid properties are shown in figure 3. The average values of the coefficient of internal heat transfer for seven mass flow rates under different conditions are found as: 461.950026 W/m^{2}K for experimental results, 500.8916571 W/m^{2}K for fluid properties evaluated by using FLUENT. In above case, the minimum and maximum average deviation in values of the coefficient of internal heat transfer was found to be 6.37-11.81% which is reasonably well. CFD measurement results for SN150 fluid properties are estimated to be lower than experimental data.

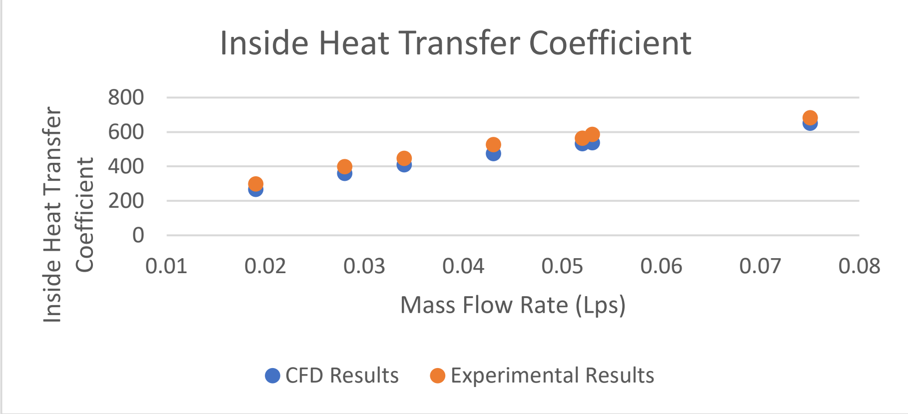


Fig. -3 Comparison of experimental inner heat transfer coefficient and CFD inner heat transfer coefficient

4.1.2 Comparison of inner Nusselt number

Nusselt number is the significant heat transfer parameter considered in That Heat Exchanger model. For both the CFD calculation Its value of internal heat transfer coefficient (hi) is drawn from the FLUENT output and the thermal conductivity value (k) is also taken from the FLUENT solver output. Displays Comparison values for the inner Nusselt number determined on the basis of experimental data and standard fluid properties for seven Various mass-flow rates in a spiral coil under isothermal heat transfer condition.

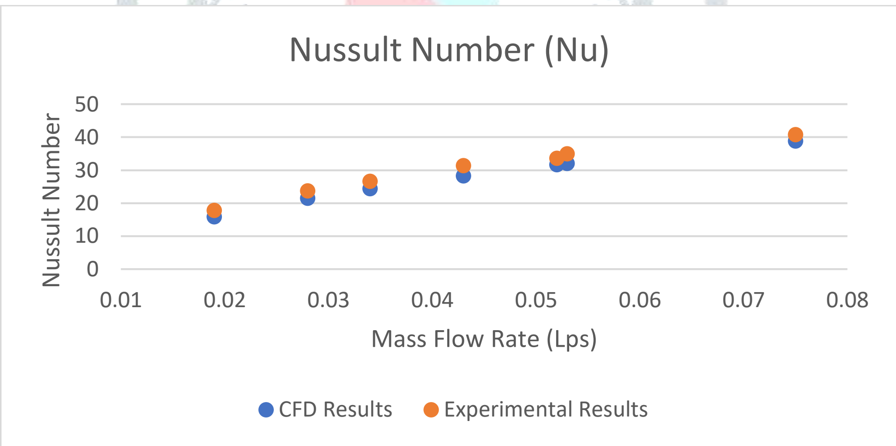


Fig.-4 Comparison of Experimental Nussult Number and CFD Nussult Number

The average values of Nusselt's inner numbers For seven mass flow rates, the following are: 29.86998714; based on experimental evidence, 27.55853995; for CFD results, for validation purposes. At any point in the fluid domain, however, we can not experimentally measure the Nusselt number, that is only achievable by CFD code In design terms, we conclude that there would be no major difference in heat exchanger efficiency. CFD calculation predicts accurate results and gives details of fluid physics which is significant.

4.1.3 Comparison of outlet temperature

Validation of experimental data determined outlet temperature values of oil using thermometer with CFD analysis data For temperatures obtained from outlets the study file of solver FLUENT by 2 temperature limits. The outlet temperature value determined by the thermometer is found to be Higher (1.12 percent) than the CFD estimation results. Experimental outlet temperature analysis predicts strong agreement with the outlet temperature calculation results for the CFD.

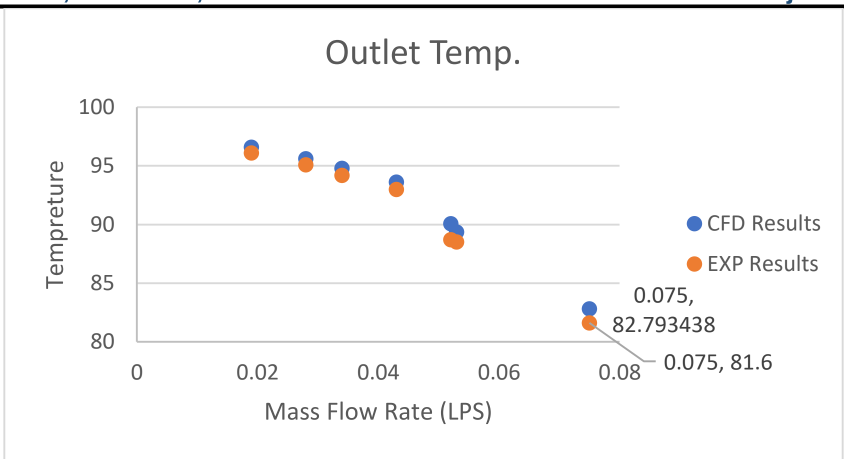


Fig.-5 Comparison of Experimental outlet Temp. and CFD outlet Temp.

The comparison of experimentally determined Temperatures at the exterior wall surface using thermocouples with CFD measurement results taken from FLUENT solver report file.

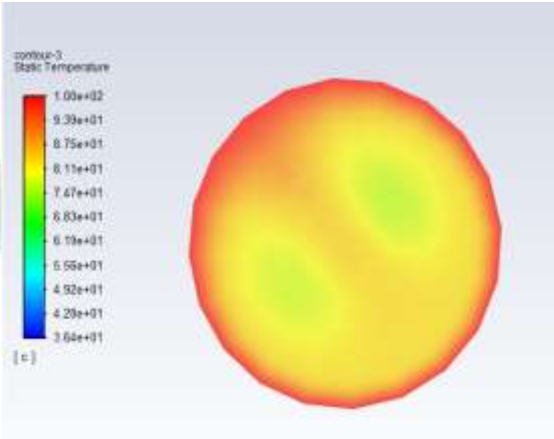


Fig.-6 Cross sectional view of Outlet Temp. contours

The experimentation assessment matched fairly well with CFD estimates of More than 1.12 percent deviation. The comparison of these experimental measurements of temperatures with the CFD calculation results shows of the measured parameters which was one of the prime objectives of this CFD analysis. For calculation of the Coefficient of internal heat transfer, based on experimental data, outer walls surface temperatures calculated by thermocouples instead of were taken the internal Wall temperature considering negligible surface temperature differences in outer and inner wall temperatures as the wall thickness is very low. It was verified by heat balance method and difference in Temperatures at Outside and Inside Walls was found to be less. However, using CFD code it is taken from report file of FLUENT solver and their values are found to be closer. These values of wall temperatures are used in calculating the inner heat transfer coefficient using CFD code. That's the excellent feature of CFD code, which we can take temperature and velocity values under analysis at any point in the fluid domain.

V. CONCLUSION

Commercial CFD package is used to validate experimental data of Outlet Temp. of oil, Inside heat transfer coefficient and Nusselt Number. The effects of steam temperature on Inside heat transfer coefficient is studied and It has been noted that as Inlet steam temperature increases coefficient of heat transfer increases and also observed that heat transfer coefficient is maximum for smaller diameter of coil and lower for bigger coil diameter. It has been discovered that as tube diameter increases heat transfer coefficient increases. From study of temperature variation at various locations It has been noted that temperature is higher at outer side of tube as compared to inner side which is the result of secondary flow generation. And the experimental results (outlet temp., Inside heat transfer coefficient and Nusselt Number) are compared with the CFD calculation results and are found to be reasonably well with the CFD predictions.

REFERENCES

1. V. Kubair, N.R. Kuloor, Heat transfer to Newtonian fluids in spiral coils at constant tube wall temperature in laminar flow, Indian J. Technol. 3 (1965) 144-146
2. V. Kubair, N.R. Kuloor, Heat transfer to Newtonian fluids in coiled pipes in laminar flow, Int. J. Heat Mass Transfer. 9 (1966) 63-75.
3. V. Kubair, N.R. Kuloor, Comparison of performance of helical and spiral coil heat exchangers, Indian J. Technol. 4 (1965) 1-4.
4. V. Kubair, N.R. Kuloor, Flow of Newtonian fluids in Archimedean spiral tube coils: Correlation of the laminar, transition and turbulent flows, Indian J. Technol. 4 (1966) 3-8.
5. Anthony Bowman, Hyunjae Park, 2004, CFD study on laminar flow pressure drop and heat transfer characteristics in Toroidal and spiral coil systems, ASME International Mechanical Engineering Congress and Exposition California USA, 1-9.
6. P. Naphon, J. Suwagrai, Effect of curvature ratios on the heat transfer and flow developments in the horizontal spirally coiled tubes. International Journal of Heat and Mass Transfer 08 (2006) 002.
7. P. Naphon, S. Wongwises, A review of flow and heat transfer characteristics in curved tubes, Renewable and Sustainable Energy Reviews 10 (2004) 463-490.
8. Dr. Madhukar S.Tandale, Sandeep M. Joshi 2008.Design of Heat Exchanger for Waste Heat Recovery from Producer Gas, 5th WSEAS Int. Conf. on Heat and Mass Transfer (HMT'08).pp 83-88.
9. Rakesh Baghel and Sushant Upadhyaya, 2013 "Effect of Coil Diameter on pressure Drop in Archimedean Spiral Coils". , IJAER 8. 2151 – 2156.
10. Jia-dong JI, Pei-qi GE, Wen-bo BI, 2014, Numerical Investigation of flow and heat transfer performances of horizontal spiral coil pipes, Journal oh Hydrodynamics (Elsevier), 576-584.
11. Deshpande, Pramod M., Dr. Dawande Shrikant, Study of Hydrodynamic of Horizontal Spiral coil tube, international journal of Advanced Engineering Research and studies, E-ISSN2249-8974.
12. S.S. Pawar, Vivek K. Sunnapawar, 2014, Experimental and CFD investigation of convective heat transfer in helically coiled tube heat exchanger, Chemical engineering research and design (Elsevier), 1475(1-19).
13. Rahul Patil, 2017, Experimental Studies on Heat Transfer to Newtonian Fluids through Spiral Coils, Experimental Thermal & Fluid Science (Springer), 1-18.
14. Rahul Patil, Nadar, Rashed Ali, 2016, The influence of Dean Number on heat transfer to Newtonian fluid through spiral coils with constant wall temperature in laminar flow, Springer-Verlag Berlin Heidelberg.
15. J. Eustice, 1910, Flow of water in curved pipes, Proc. R. Soc. A-84, 107-118.
16. Hui Zhu, Hanqing Wang, Guangxiao Kou, 2014, Experimental study on the heat transfer enhancement by Dean Vortices in spiral tubes, International Journal of Energy And Environment (IJEE), Volume 5,317-326.
17. Jay J. Bhavsar, V.K. Matawala, S. Dixit, July-2013, Design and Experimental analysis of spiral tube heat exchanger, International Journal of Mechanical and Production Engineering, 37-42.
18. Manoj Kumar, Vishal Gupta, Samarjeet Bagri, June-2017, A review on design and development of spiral coil heat exchangers, IRJET, Volume 04, 2251-225.

AUTHORS PROFILE

Mangesh Shashikant Bidkar
M.E. Scholar, B.E. (Mechanical Engg.)
Pillai College of Engineering, New Panvel.

Dr. Rashed Ali

Recognized Post Graduate Teacher in Mechanical Engineering, University of Mumbai
Pillai college of Engineering, New Panvel.