Thermal Performance Investigation of Horizontal Spiral Coiled Tube for Design & Process Parameters Using CFD & DOE Techniques

Lokesh Singh[1]*, Dr. A. K. Raghav[2]

Abstract— In present research work the effect of process and design parameters of the horizontal spirally coiled flow tube, are investigated using CFD simulation and DOE techniques. In this research work total, six parameters are selected which are mass flow rate of water flow in the tube, water inlet temperature, heating surface temperature which is made of copper tube, tube inner diameter having 3 mm thickness, curvature ratio of tube and pitch of tube. The experiment table is made by using a mixed Taguchi method which generates a total of 36 cases for this research study. Experimental validation is also performed with previously published research work [01] to show a good agreement among CFD simulation and experiment results. The error among the simulation and experiment results are in a range of 3% to 12% max, which is acceptable for further research work. The results are then analyzed by using the signal to noise ratio which shows the rank among all parameters and as well as the best optimal solution for a selective response. In the present study, the research work is focused on Nusselt Number analysis.

Index Terms— CFD, Spiral flow, Taguchi Method, S/N ratio, validation, Curvature Ratio, Experiment Validation

1 Introduction

Spiral coil tubes are used in various engineering areas of heat transfer and fluid flow some areas are refrigeration devices, chemical Reactors and boilers, single-phase heat exchangers and custom heat transfer devices. In spiral tubes the heat transfer and flow conditions are more complicated than straight tube and that is the reason the spiral tube is more thermally efficient than a straight tube one another reason to use spiral tube is the low space used during installation in any device rather than straight tube which uses more space than you spiral tube. turbulent flow conditions in the spiral tube are more complicated than the straight tube so higher-pressure drop can occur in a spiral tube when compared with straight tube show proper designing is required in spiral tube flow study. In curved tubes helical and spiral tubes are most widely used but helical tubes are much wider tubes used in heat transfer and fluid flow areas.

2 The objective of the study

The objective of this study is to investigate the thermal performance of the spiral coiled tube in horizontal fluid flow conditions using CFD simulation and DOE techniques. Taguchi method is used to find the relation among input and output parameters using signal to noise ratio formula development.

3 CFD Theory

In CFD the governing equations for Newtonian fluid can be expressed in its most popular forms.

- *Lokesh Singh, a research scholar in career point university, Kota, India, PH-9996099963. E-mail: lokeshsingh2707@gmail.com*
- *Dr. A.K. Raghav, Research guide of scholar in career point university, Kota, India, PH-9911447296. E-mail: akraghava@gmail.com*

Mass conservation Equation

$$\frac{\partial \rho}{\partial t} + div(\rho u) = 0$$

Momentum conservation equation

$$\rho \frac{Du}{Dt} = \frac{\partial(-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx}$$

$$\rho \frac{Dv}{Dt} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial(-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My}$$

$$\rho \frac{Dw}{Dt} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial(-p + \tau_{zz})}{\partial z} + S_{Mz}$$

Energy Conservation equation

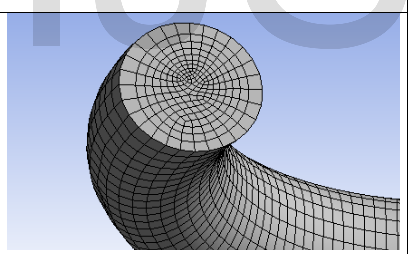
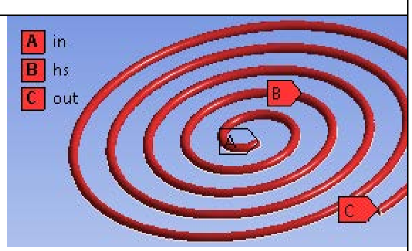
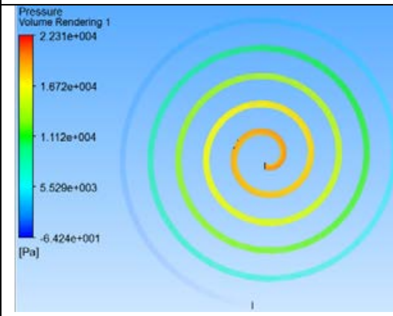
$$\rho \frac{DE}{Dt} = -div(pu) + \left[\frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{xy})}{\partial y} + \frac{\partial(u\tau_{xz})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z}\right] + div(K\ grad\ T) + S_E$$

The equations are in their general form although in this research work the steady-state form of equations is used for getting simulation results so proper assumptions are created in Ansys Fluent (Version 14.5) software. The turbulence model equation is selected for a two-equation format which is widely used for a single-phase flow CFD simulation study. In the present study K-ω (SST) turbulence model is used.

4.CFD Modeling and Grid Independence Test

Steady-state pressure-based CFD simulation is adopted for this research work and all cases are solved by using Ansys Fluent (Version 14.5). The important modeling steps are present in table 1.

Table 1
CFD Modeling Steps

Step	Description	Image/Detailing
I	**CFD Domain Formulation**	
II	**Discretization of CAD File**	
III	**Boundary Named Selection**	
IV	**Setup of Boundary Conditions**	Steady State, Pressure Based, k-ω (SST model), Energy Equation
V	**Setup for Solution Methods**	First and Second-Order Upwind Schemes for field variables
VI	**Solution**	Steady State
VII	**Results**	

All required important information regarding CFD simulation is present in table 1, now the grid independence test required to final the mesh size and its elements. The grid-independent test present in table 2 for spiral flow CFD simulation cases which have selective boundary condition.

Table 2
Grid Independence Test

Element Size	Elements	Nu-Ex	Nu-S	Constant Parameter
1.5	836172	83.13	82.07	Tw, in = 20C
2.0	740350	83.13	82.94	Twall = 35 C
2.5	657841	83.13	84.67	CR-0.02
3.0	512840	83.13	87.64	MFR 0.10 kg/s

Some important assumptions are carried out in this research work which is the following:

I.CFD domain is only the fluid part of the spiral coil tube, all other structural parts are not considered in this CFD simulation

II> Steady State pressure-based solver is selected because of the experiment is also set on steady-state [01] boundary condition.

III. Fluid properties are also set on room temperature and treat as constant fluid properties.

IV. Two equation turbulence model is adopted for this research

ISSN 2229-5518

work based on literature review.

V. The tube inner part is assumed fully polished in nature, so roughness of the inner tube is set to be zero for the CFD simulation.

5.Taguchi Method

The aim to develop the Taguchi method to design a robust experiment table for any kind of engineering research. This method can reduce the involvement of time and money during the engineering research work. The analysis part of the Taguchi method is based on Signal to Noise formulation which is present for "smaller is better" and "larger is Better" options.

Lower the better (for making the system response as small as possible):

$$S/N = -10\, log_{10}\left(\sum \frac{y^2}{n}\right)$$

6.Factor and Levels

In present research work, six factors are selected and each factor has some levels as per the literature requirement and the final factors and levels are present in table 3 for this research study.

Table 3

Factor and Levels for spiral flow coil

Factors	Description	Level-I	Level-II	Level-III	Unit
Inner Diameter	ID	8	9	NA	mm
Mass Flow Rate	MFR	0.1	0.12	0.14	Kg/sec
Heat Flux (Twall)	HF	30	33	36	°C
Fluid Temperature	FT	15	17	19	°C
Pitch	Pitch	25	30	35	mm
Curvature Ratio	CR	0.018	0.020	0.024	NA

7.Orthogonal Array

As seen in Table 3, the first factor which is the inner diameter of the tube have two levels and all other factors have three levels, so the mixed Taguchi method is applied for this research work and the final experiment table for this study is present in table 4.

Table 4 Mixed Orthogonal Array for Spiral Flow

Run	ID	MFR	HS	FT	PITCH	CR	Nu
1	8	0.1	25	15	25	0.018	98.04
2	8	0.12	28	17	30	0.020	112.09
3	8	0.14	31	19	35	0.024	110.20
4	8	0.1	25	15	25	0.020	102.24
5	8	0.12	28	17	30	0.024	105.36
6	8	0.14	31	19	35	0.018	118.93
7	8	0.1	25	17	35	0.018	93.14
8	8	0.12	28	19	25	0.020	119.80
9	8	0.14	31	15	30	0.024	116.21
10	8	0.1	25	19	30	0.018	93.81
11	8	0.12	28	15	35	0.020	101.46
12	8	0.14	31	17	25	0.024	126.83
13	8	0.1	28	19	25	0.024	101.79
14	8	0.12	31	15	30	0.018	112.49
15	8	0.14	25	17	35	0.020	118.13
16	8	0.1	28	19	30	0.018	93.78
17	8	0.12	31	15	35	0.020	105.51
18	8	0.14	25	17	25	0.024	126.87
19	9	0.1	28	15	35	0.024	84.86
20	9	0.12	31	17	25	0.018	147.16
21	9	0.14	25	19	30	0.020	125.54
22	9	0.1	28	17	35	0.024	84.67
23	9	0.12	31	19	25	0.018	109.65
24	9	0.14	25	15	30	0.020	116.30
25	9	0.1	31	17	25	0.020	91.10
26	9	0.12	25	19	30	0.024	106.83
27	9	0.14	28	15	35	0.018	117.96
28	9	0.1	31	17	30	0.020	94.32
29	9	0.12	25	19	35	0.024	99.58
30	9	0.14	28	15	25	0.018	127.75
31	9	0.1	31	19	35	0.020	85.56
32	9	0.12	25	15	25	0.024	112.46
33	9	0.14	28	17	30	0.018	117.42

Run	ID	MFR	HS	FT	PITCH	CR	Nu
34	9	0.1	31	15	30	0.024	89.47
35	9	0.12	25	17	35	0.018	103.48
36	9	0.14	28	19	25	0.020	126.11

Nusselt number is selected as a response parameter for this study and the formulas used for calculation of Nu is present here:

Step I Heat Transfer Calculation

Q = m*Cp*(Twater out - Tair in)

m = Mass flow rate

Cp = Specific heat at a constant pressure of air

Tair out = Temperature of air at the outlet section

Tair in = Temperature of air at the inlet section

Q = 0.14*4190*(299.8-288)

Q = 6921.88 J/s

Step II Convective Heat Transfer

$$h = \frac{Q}{A*(T_{Plate} - T_{Bulk})}$$

$$h = \frac{6921.88}{0.078*(304-293.818)}$$

$$h = 8715.582\ W/m^2K$$

Step III Nu Number Calculation

$$Nu = \frac{h*D_h}{k}$$

$$Nu = \frac{8715.582*8*0.001}{0.6}$$

$$Nu = 116.21$$

Where k=Thermal Conductivity of water

8.Experiment Validation

Experiment validation is an important part of the CFD simulation-based study because it verifies the CFD simulation for research work. In the present study, **Naphon and Suwagrai [01, 02]** based experimental research is used for the CFD validation study. All required boundary conditions are adopted form these research work and the same design is reconstructed for CFD validation and then the compare among experiment and CFD simulation is performed and present in table 5 and figure 1 for selective CR and mass flow rate boundary conditions. Water inlet temperature and wall temperature of the heating surface is set constant for this experiment validation study.

Table 5
Experiment Validation with CFD simulation [01]

MFR	Experiment (Tin=20 C and THS=35 C)				Simulation (Tin=20 C and THS=35 C)			
	Straight	CR-0.02	CR-0.04	CR-0.05	Straight-S	CR-0.02-S	CR-0.04-S	CR-0.05-S
0.050	35.04	49.23	46.90	43.09	36.87	53.24	52.43	45.07
0.066	41.62	61.52	57.28	53.68	44.15	66.92	62.66	56.24
0.085	47.96	71.90	67.24	60.67	51.84	79.18	69.84	66.84
0.100	50.93	83.13	75.93	69.15	55.41	92.92	79.57	73.79
0.117	54.95	90.33	81.22	75.72	60.70	98.65	84.40	79.43

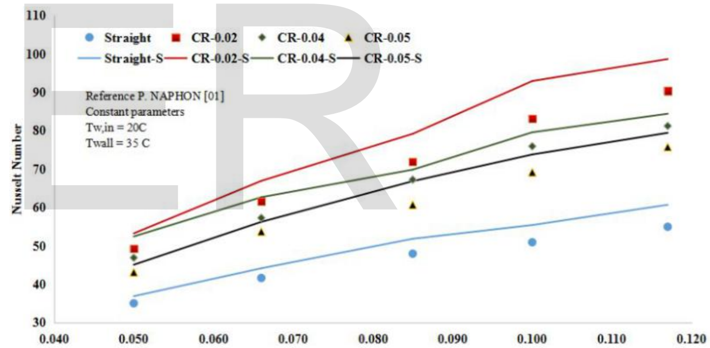


Fig. 1 Experiment Validation with CFD simulation results [01]

9.Result and Discussion

In the present research study, six factors are selected for CFD simulation work. A total of 36 cases are generated using the Taguchi method and the experiment table is present in table 4 with response Nusselt number. Taguchi analysis is performed using S/N ratio analysis for Nusselt Number and the results are present in table 6.

Table 6
S/N ratio analysis for Nusselt Number

Level	ID	MFR	HS	FT	PITCH	CR
1	40.68	39.33	40.63	40.54	41.2	40.84
2	40.54	40.89	40.57	40.72	40.54	40.62
3		41.62	40.64	40.58	40.1	40.38

Delta	0.14	2.3	0.07	0.19	1.1	0.45
Rank	**5**	**1**	**6**	**4**	**2**	**3**

As seen in Table 6, the best rank for Nusselt number using S/N ratio under the option of "larger is better" is mass flow rate of fluid, the second rank is the pitch of spiral tube, third rank is curvature ratio, the fourth rank is fluid temperature and the last rank is heating surface. The same result is present in figure 2 for the S/N ratio of the Nusselt number of spiral coil flow conditions.

As seen in figure 2, the optimal solution for Nusselt number is found by selecting the maximum S/N number value for the Nusselt number and the optimal solution is present in table 7 for Nusselt number.

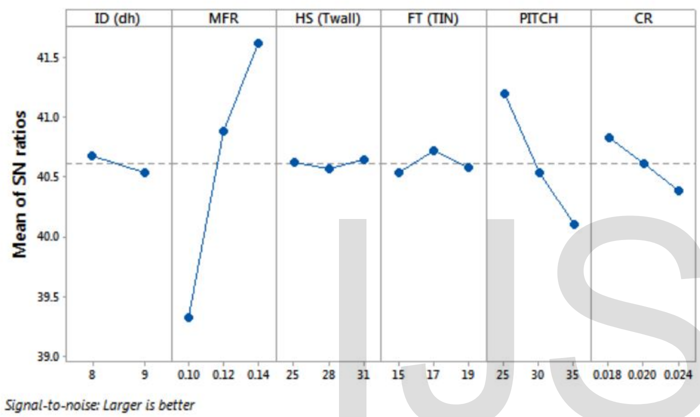


Fig. 2 S/N ratio analysis for Nusselt Number for spiral Flow

Table 7
Optimal Solution for Nusselt Number

Response	**ID**	**MFR**	**HS**	**FT**	**PITCH**	**CR**	**Nu**
Nu	8	0.14	31	17	25	0.018	153.26

The interaction plot for all six factors is shown in figure 3, which indicates relation among tow factors and one response which is Nusselt number.

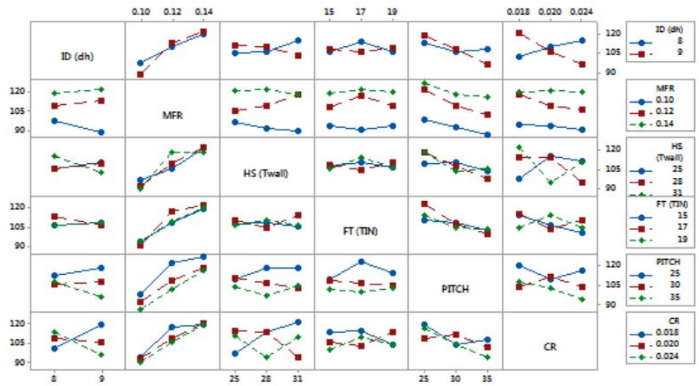
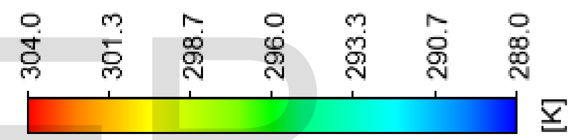
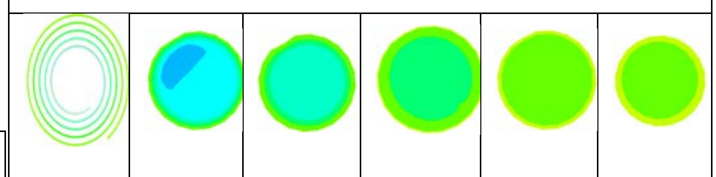
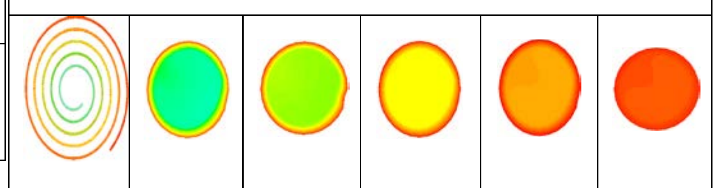
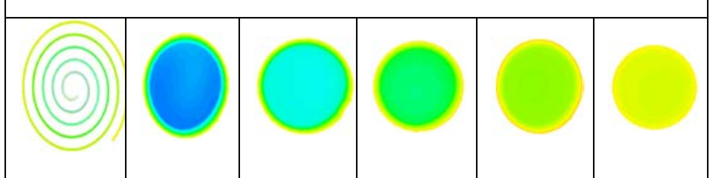


Fig. 3 Interaction Plot for Nusselt Number

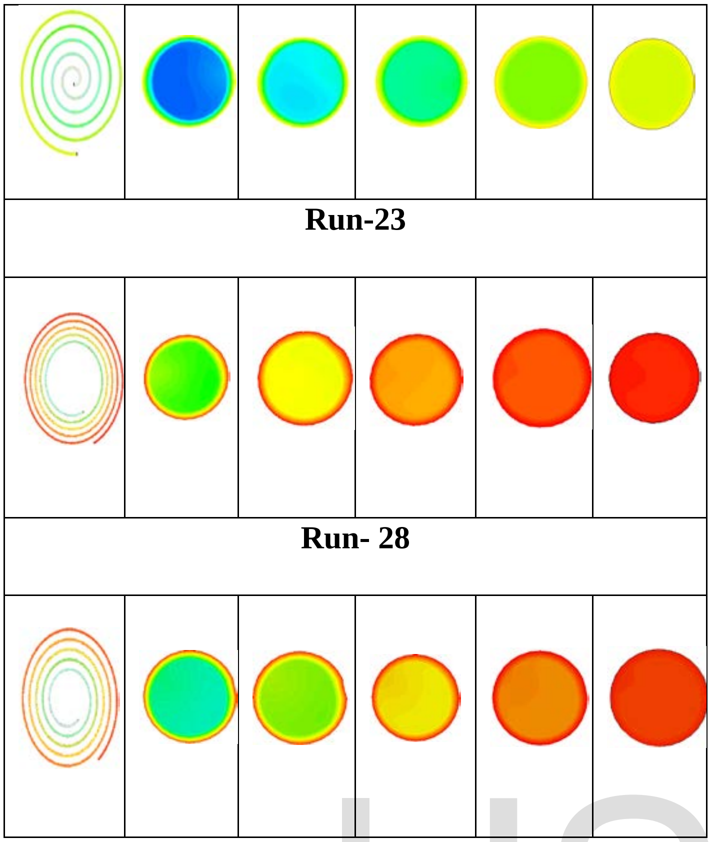
10. CFD Contours

The best part of CFD analysis is its visual part of a solved problem. In this section, the selective cases are present to show the capability of CFD simulation. As seen in table 8 the effect of temperature in different boundary conditions are clearly shown for experiment number 1, 6, 11, 19, 23 and 28.

Table 8
Temperature Contour


Run-1

Run-6

Run-11

Run-19

ISSN 2229-5518



10.Conclusion

This study finds the effect of various process and design papers on the thermal performance of spirally coiled flow conditions using CFD and DOE techniques. The main conclusion of this research work is present here:

I)Taguchi method is used for developing the experiment table for six parameters and each has three levels but the first-factor inner diameter has two levels only. A total of 36 experiment cases are designed in this method and then solved by CFD simulation software.

II)Signal To noise ratio analysis is performed for this experiment table and it is found that the Nusselt number can be increased by improving the mass flow rate of fluid, then the second rank for improving the Nu is the pitch of spiral tube and least effective parameter which can help to improve the Nu is heating surface temperature.

The optimal solution for the Nusselt number is to present the following table for the selective experiment.

Response	**ID**	**MFR**	**HS**	**FT**	**PITCH**	**CR**	**Nu**
Nu	8	0.14	31	17	25	0.018	153.26

References

[01] NAPHON P., SUWAGRAI J. Effect of curvature ratios on the heat transfer and flow developments in the horizontal spirally coiled tubes[J]. International Journal of Heat and Mass Transfer, 2007, 50(3-4): 444-451.

[02] NAPHON P. Study on the heat transfer and flow characteristics in a spiral-coil tube[J]. International Communications in Heat and Mass Transfer, 2011, 38(1): 67-74.

[03] YOO G. J., CHOI H. K. and DONG W. R. Fluid flow and heat transfer characteristics of spiral coiled tube: Effects of Reynolds number and curvature ratio[J]. Journal of Central South University of Technology, 2012, 19(2): 471-476.

[04] JAMAL-ABAD M. T., ZAMZAMIAN A., and DEHG- HAN M. Experimental studies on the heat transfer and pressure drop characteristics of Cu-water and Al-water nanofluids in a spiral coil[J]. Experimental Thermal and Fluid Science, 2013, 47(5): 206-212.

[05] ALTAÇ Z., ALTUN Ö. Hydrodynamically and thermally developing laminar flow in spiral coil tubes[J]. International Journal of Thermal Sciences, 2014, 77: 96- 107.

[06] LU X., DU X. P. and ZENG M. et al. Shell-side thermal- hydraulic performances of multilayer spiral-wound heat exchangers under different wall thermal boundary Conditions[J]. Applied Thermal Engineering, 2014, 70(2): 1216-1227.

[07] KURNIA J. C., SASMITO A. P. and MUJUMDAR A. S. Laminar heat transfer performance of power-law fluids in the coiled square tube with various configurations[J]. International Communications in Heat and Mass Transfer, 2014, 57: 100-108.

[08] GHOBADI M., MUZYCHKA Y. S. Fully developed heat transfer in mini scale coiled tubing for constant wall temperature[J]. International Journal of Heat and Mass Transfer, 2014, 72(5): 87-97.

[09] LI Cheng-Guang, XUE Wan-Yun, and HUAI Wen-Xin. Effect of vegetation on flow structure and dispersion in strongly curved channels[J]. Journal of Hydrodynamics, 2015, 27(2): 286-291.

[10] SALIMPOUR M. R. Heat transfer coefficients of shell and coiled tube heat exchangers[J]. Experimental Thermal and Fluid Science, 2009, 33(2): 203-207.