Heat transfer analysis of a circular tubes using extended surfaces

Sanjay Kumar Sharma, Ashish Nayyar, Chandan Kumar
Department of Mechanical Engineering, Swami Keshvanad Institute of Technology, Management & Gramothan, Jaipur-302017 (INDIA)
Email- sanjaybagra84@gmail.com
Received 23.12.2018 received in revised form 28.01.2019, accepted 02.02.2019

Abstract: This paper analyze the use of extended surfaces in circular tube to enhance heat transfer. Heat transfer parameters in simple circular (SC) tube without insert, with rectangular insert (RI), with twisted insert (TI) and with wave insert (WI) have been calculated and compared. The heat transfer in air for Reynolds number having range from 5000-15000 was studied experimentally and numerically in a simple circular tube and in three extended surfaces of different design with same surface area (One insert area:0.000680m^2). In present study, design of experiment (DOE) is used to develop the experiment layout. Total five factors are selected i.e. Extended surface, velocity, heat flux, material of tube and air pressure in domain. The experimental results are validated by CFD simulation. It is concluded that extended surfaces improve heat transfer rate in circular tubes. The rectangular insert of extended surface is optimum among all tested combinations.

Keywords: Computational fluid dynamics, Forced convection, Heat transfer, Extended Surface, Nusselt number, S/N Ratio.

1. INTRODUCTION

Circular tube is an element which is used to transfer thermal energy among fluids & fluid and solid surfaces at different temperature when in thermally contact. It does not required any external heat and work interaction. The purpose of constructing a circular tube is to get an efficient method of heat transfer from one fluid to another, by direct contact or by indirect contact. It is used in various practical applications such as heating and cooling of fluids, space heating, refrigeration, air conditioning, power stations, chemical plant, petroleum refinery, sewage treatment etc. [1]. Some common applications of circular tube in engineering are shell & tube type heat exchangers, condensers, evaporator, automobile radiator, cooling tower and air preheater [2]. A circular tube play an important role in heat transfer applications therefore chosen for the works.

The heat transfer occurs by three manners: conduction, convection and radiation. The radiation heat transfer in a circular tube is negligible in comparison to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows to the surrounding through solid wall. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. Convection plays the major role in the heat transfer mechanism of a circular tube.

A. Tohidi et al. [1] on the characteristics of heat transfer and friction factor of laminar flow (480<Re<1750) in a circular tube fitted with central slant rods under constant heat flux condition. The computation result shows that Nusselt number and friction factor increase with a decrease in rod pitch. C. X. Lin et al. [2] done the numerical analysis on investigate three-dimension turbulent developing convective heat transfer in helical pipes with finite pitch. The studied involved k-ϵ turbulent model and FVM method. Circumferential average Nusselt number is discovered to be oscillatory and pitch curvature or Reynolds number increase than the oscillation phenomenon enhanced. A. M. Husseina et al. [3] numerical investigated the thermal and flow flied in transverly corrugated circular tube for turbulent flow (5000<Re<60,000) under constant heat flux. Friction factor is in-depended on Reynolds number for corrugated tube. F.Akbaridoust et al. [4] studied numerically and experimentally the pressure drop and convective heat transfer behavior of nanofluid (CuO) for steady state and laminar flow (200<Re<1000) in helically coiled tube at a constant wall temperature. The coil with different curvature and torsion ratio for is used. It is observed that heat transfer is enhanced with greater curvature ratio. Jalaluddin et al. [5] have discussed on thermal performance and pressure drop of the spiral tube ground source heat pump. It is reported that spiral pipe provides a better heat exchange rate than straight pipe. Pressure drop of water flow is also increased due to increasing the length of pipes per meter borehole depth therefore pumping power is also increased. P. Naphon et al. [6] have performed study on effect of curvature ratio on the heat transfer and flow development in the horizontal spirally coiled tube under constant wall temperature. The k-ϵ standard model and finite volume method have been for simulation. Heat transfer rate is directly proportional to the mass flow rate. Nusselt number is higher for lower curvature ratio. V. Singh et al. [7] have described heat transfer and flow friction characteristics in a tubular heat exchanger inserted with single, double, triple and quadruple twisted tape with solid ring. It is observed that number of twisted tape is increase than friction factor, Nusselt number and thermal performance factor is also increase.

Heat Transfer in circular tubes [8]

$$Q_1 = \dot{m} C_p (T_{out} - T_{In})$$
(1)

Where Here $\dot{m}$ is mass flow rate, T_{out} is out temperature and T_{in} is inlet temperature of circular tube.

Heat flux is given to this circular tube by electrical energy and then convective heat transfer is

$$Q_1 = hA(T_{sur} - T_{bulk})$$
(2)

Where T_{sur} is tube surface temperature and T_{bulk} is bulk temperature.

This is heat flow is equal to Q_1 and then

$$h = \frac{Q_1}{A(T_{sur} - T_{Bulk})}$$
(3)

This is used for calculation of Nu number [8]

$$Nu = \frac{hD_h}{k}$$
(4)

This Nu is then validated by Dittus-Boelter correlation which is presented here [8].

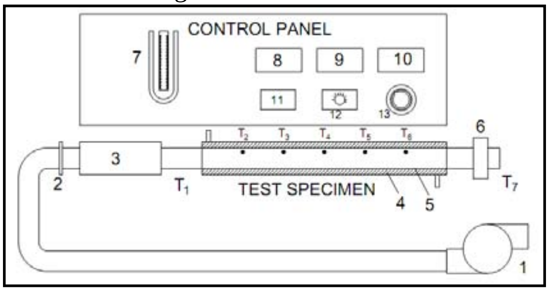
$$Nu = 0.02 \mathrm{Re}^{0.8} \mathrm{Pr}^n$$
(5)

Here, n = 0.4 for heating and 0.3 for cooling.

To develop a customized forced convection heat transfer analysis in circular tube which work on one fluid flow conditions. To the study heat transfer, Nusselt number and dimensionless factor experimentally in a circular tube which has an inner corrugated tube filled with various categories of extended surface (Rectangular insert, Rectangular twisted insert, Wave Insert) from conventional to modified insert types which include perforated and the Reynolds number is changed from 5000 to 15,000 of turbulent regime. To apply Signal to Noise ratio through Taguchi method for numerical analysis and to optimize the response parameters.

2. EXPERIMENTAL SETUP

Performance analysis of heat transfer through forced convection in circular tube is investigated in current research study for air as fluid medium. Selection of setup design is take care from literature review. The schematic line diagram of the experimental set up is shown in figure 1, Schematic diagram of the rectangular insert, twisted insert and wave insert in figure 2.



1. Blower, 2. Orifice, 3. Heater, 4. Insulation Wall, 5. Copper Tube, 6. Pressure Valve, 7. U-Tube Manometers, 8. Voltmeter, 9. Temperature, 10. Ammeter, 11. Dimmer for Blower, 12. Temperature Sensing Points 13. Voltage Controller
T1: Air inlet temperature to test specimen
T2-T6: Temperature test specimen surface along the length
T7: Air outlet temperature from test specimen

Figure 1 : Line diagram of setup

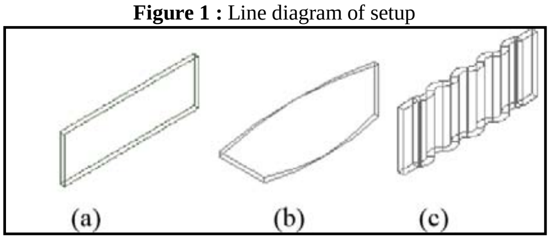


Figure 2 : Schematic of the inserts
(a) Rectangular insert, (b) Twisted insert, (c) Wave insert

4. CFD- ANALYSIS

Computation fluid dynamics (CFD) is a research tool of fluid dynamics which provides cost effective means, simulate real flows using the numerical solution of governing equations. Main governing equation used is Navies-Stokes equation. Computational technique used algebraic equations instead of partial differential equations and solve by digital computer [8]. It also provide testing conditions which are very difficult to measure experimentally and cannot solve analytically. Steps involves in CFD analysis are process:

- Mathematical modeling
- Numerical Analysis
- Software tool (solver, pre and post processing)

Governing equation of mass conservation [9, 10]:

$$\frac{\partial p}{\partial t} + div(\rho u) = 0 \quad (6)$$

Momentum conservation

X- Direction

$$\rho \frac{Du}{Dt} = \frac{\partial(-p+\tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \quad (7)$$

Y direction

$$\rho \frac{Dv}{Dt} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial(-p+\tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \quad (8)$$

Z direction

$$\rho \frac{Dw}{Dt} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial(-p+\tau_{zz})}{\partial z} + S_{Mz} \quad (9)$$

Energy conservation:

$$\rho \frac{DE}{Dt} = -div(\rho u) + \left[\frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{xy})}{\partial y} + \frac{\partial(u\tau_{xz})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} \right.$$
$$\left. \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z} \right] + div(KgardT) + S_E \quad (10)$$

Here

$$E = i + 1/2\,(u^2 + v^2 + w^2) \quad (11)$$

Experimental and simulation both studies are investigated for circular tube. The numerical part of this study is discussed. ANSYS WB and ANSYS FLUENT software's are used for CAD modelling and CFD simulation work.

All important steps are present in following section.

3.1 Formulation of flow Problem

CFD domain is required for simulation and validation so formulation of CFD is required. The two type of CFD domain used, first design is simple circular tube and second design is circular tube with extended surface.

3.2 Making of Flow Domain

CAD file is generated by ANSYS DM software. The dimensions of test specimen are measured by experimental setup.

3.3. Formulation of Boundary Conditions

Boundary conditions are developed in ANSYS ICAM CFD software.

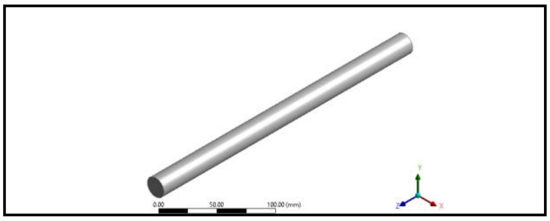


Figure 5 : CFD domain CAD diagram

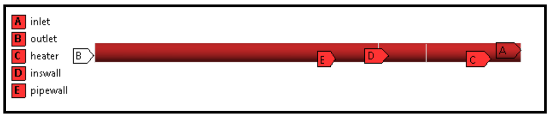


Figure 6: Boundary of circular tube

3.4 Grid Generation

Mesh generation is completed in ANSYS ICAM CFD software. Finite structured grids are generated. Grid Independence test is also performed for optimum grid size selection as shown in fig. 7 and table 1.

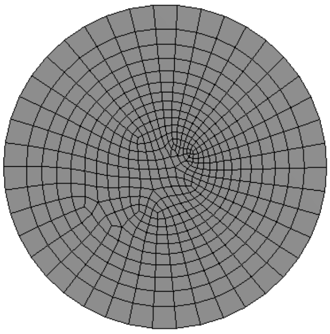


Figure 7 : Mesh of CFD domain

Table 1 Grid quality index for discretized domain of circular tube

Mesh Quality	**Avg.**	**Max.**	**Min.**	**St. Dv.**
Element Quality	0.61	0.99	9.05	0.23
Aspect Ratio	2.79	13.45	1.13	1.41
Orthogonal	0.98	0.99	0.63	3.52
Skewness	0.12	0.62	0.02	9.18

3.5 Establish CFD simulation

All boundaries requires proper data input, proper selection of solver type, model selection, and other important steps. CFD simulation is performed using ANSYS FLUENT software.

3.6 Set Solution Controls

After selection of proper selection of data inputs time to select proper solution techniques to solve governing equations. This step is solved using ANSYS FLUENT software.

3.7 Monitor Simulation parameters (Residuals)

Various types of errors are present in CFD simulation like truncation error, residuals errors and many more. Residuals are required step to show the error percentage in CFD simulation.

3.8 Post Processing of Simulation

The required results are extract from solved files of Ansys Fluent software. The main results extracted from CFD simulation are contours for temperature.

4 FACTOR AND LEVELS

In this section the application of the Taguchi experimental design method and Signal to Noise ratio approach is discussed. The scheme of carrying out experiments was selected and the experiments were conducted to investigate the effect of process parameters on the output parameters e.g. Nusselt number, Dimensionless parameter.

For the present experimental work the three process parameters each at four levels have been decided for Taguchi method. It is desirable to have four minimum levels of process parameters to reflect the true behavior of output parameters of study. The process parameters are renamed as factors and they are given in the adjacent column. The levels of the individual process parameters/factors are given in Table 2.

Table 2 : Process Parameters and their Levels (Taguchi Method)

Factor	**Unit**	**Level**			
		L1	**L2**	**L3**	**L4**
A: Extended Surface	-	SC	RI	TI	WI
B: MFR (Mass Flow Rate)	m/s	3.0	4.0	5.0	6.0
C: HF (Heat Flux)	W/m^2	795	1195	1590	1990

Here Factor A represent design condition of circular tube, SC represent simple circular tube design, RI represent Rectangular Insert, TI represent Twisted Insert and WI level represent Wave Insert. L-16 orthogonal array is developed for this research study.

5 RESULT AND DISCUSSION

4.1 Taguchi Analysis

Taguchi technique has been used for two type of works, first to develop an experimental plan which is presented in table 3 and second is analysis of experimental data with response. There are two responses Nusselt number and dimensionless temperature difference parameter. The analysis is performed using “Signal to Noise ratio” curves. These curves are helpful to find most effective factors which are responsible for response improvement or decrements [11, 12].

Table 3 : S/N Ratio Analysis of NU

Sr. No.	**A (ES)**	**B (MFR)**	**C (HF)**	**Nu**	**S/N RATIO**
1	1	1	1	18.58	25.3809
2	1	2	2	25.41	28.1001
3	1	3	3	28.93	29.2270
4	1	4	4	32.96	30.3597
5	2	1	2	40.82	32.2175
6	2	2	1	47.20	33.4788
7	2	3	4	76.98	37.7276
8	2	4	3	57.07	35.1282
9	3	1	3	37.42	31.4621
10	3	2	4	45.27	33.1162
11	3	3	1	49.85	33.9533
12	3	4	2	54.06	34.6575
13	4	1	4	40.20	32.0845
14	4	2	3	48.32	33.6825
15	4	3	2	57.81	35.2401
16	4	4	1	67.47	36.5822

4.2 S/N Ratio Analysis

Signal to Noise Ratio analysis is performed for both responses Nu and θ and present in this section. S/N ratio analysis for Nu is present in table 3 for each experiment.

For Nu response, larger is better option is select for S/N ratio analysis, as seen in table 4, the best factor which improve Nu number is extended surface install in circular tube, whereas least factor which show low effect for Nu number is heat flux.

Table 4 : S/N ratio for Nu number (larger is better)

Level	**A (ES)**	**B (MFR)**	**C (HF)**
1	28.27	30.29	32.35
2	34.64	32.09	32.55
3	33.3	34.04	32.37
4	34.4	34.18	33.32
Delta	6.37	3.9	0.97
Rank	**1**	**2**	**3**

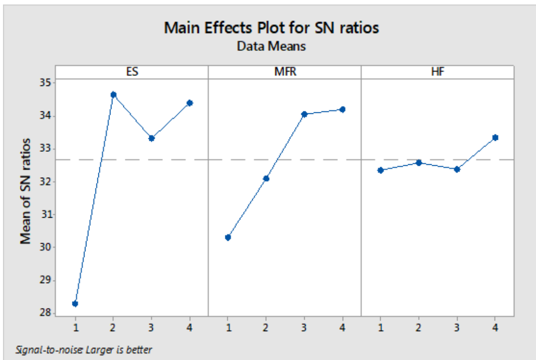


Figure 8 : S/N ratio analysis for Nu number

As seen in figure 8, the best case for optimal Nu number is following, extended surface must be design-2, mass flow rate must be level-IV and heating flux must be level-IV.

S/N ratio analysis for θ is present in table 5 for each experiments.

Table 5 S/N ratio analysis for θ response variable

Sr. No.	A (ES)	B (MFR)	C (HF)	θ	S/N RATIO
1	1	1	1	3.57	-11.0534
2	1	2	2	4.01	-12.0629
3	1	3	3	4.40	-12.8691
4	1	4	4	4.63	-13.3116
5	2	1	2	1.87	-5.4368
6	2	2	1	2.16	-6.6891
7	2	3	4	1.65	-4.3497
8	2	4	3	2.68	-8.5627
9	3	1	3	2.04	-6.1926
10	3	2	4	2.25	-7.0437
11	3	3	1	2.55	-8.1308
12	3	4	2	2.83	-9.0357
13	4	1	4	1.90	-5.5751
14	4	2	3	2.11	-6.4856
15	4	3	2	2.20	-6.8485
16	4	4	1	2.26	-7.0822

For θ response, smaller is better option is select for S/N ratio analysis, as seen in table 6, the best factor which improve θ number is extended surface install in circular tube, whereas least factor which show low effect for θ number is heat flux.

Table 6 : S/N ratio analysis for θ (smaller is better)

Level	A (ES)	B (MFR)	C (HF)
1	-12.324	-7.064	-8.239
2	-6.26	-8.07	-8.346
3	-7.601	-8.049	-8.528
4	-6.498	-9.498	-7.57
Delta	6.065	2.434	0.957
Rank	**1**	**2**	**3**

Like table 6 for S/N ratio, graphical representation is also present in figure 9 for S/N ratio.

S/N ratio figure help to predict the best case among all possible combinations of experiments and this is the best part of Taguchi method. The peak of hill is always present the best case of selective response and lowest peak of figure always present worse case of selective response.

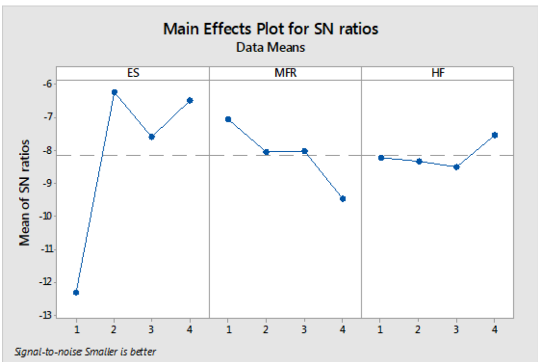


Figure 9 : S/N ratio analysis for θ number

As seen in figure 9, the best case for optimal θ number is following, extended surface must be Level-II, mass flow rate must be level-I and heating flux must be level-IV.

4.3 Optimal Solution Analysis

Taguchi method also develop a optimal solution for selective responses using S/N Ratio technique. Optimal solution for Nu and θ is present in table 7 and table 8 respectively. For both cases best and worse optimal solutions are present with CFD contour analysis for these optimal conditions.

Table 7 : Optimal solution for Nu number

Factors	Level- ES	Level- MFR	Level- HF
Best Case	RI	6.0	1990
Worse Case	SC	3.0	795

The best case for Nu have optimal level for extended surface is Rectangular insert, optimal mass flow rate is 6.0 m/s and optimal level for heat flux is 1990 W/m^2.

Table 8 : Optimal solution for θ number

Factors	Level- ES	Level- MFR	Level- HF
Best Case	RI	3.0	1990
Worse Case	SC	6.0	1590

The best case for θ have optimal level for extended surface is Rectangular insert, optimal mass flow rate is 3.0 m/s and optimal level for heat flux is 1990 W/m^2.

4.4 Visual Analysis of CFD Result

The CFD simulation has been done for all the 16 cases as per DOE given in table 3. CFD has one best advantage of visual analysis of results of Nu and θ. CFD contours for static temperature magnitude for cases 1-16 have been shown in fig. 10 to fig. 25.

It can be observed from these contours that Rectangular Insert (RI) having higher rate of heat transfer then other combinations. It can be visualize by static temperature contour in figure 10 - 25. This observation of CFD analysis verify the optimum results given by S/N ratio analysis and is shown in table 3 and table 5.

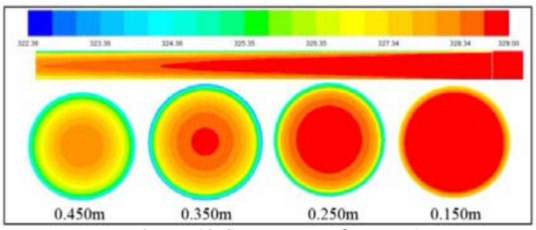


Figure 10 CFD contour for case 1

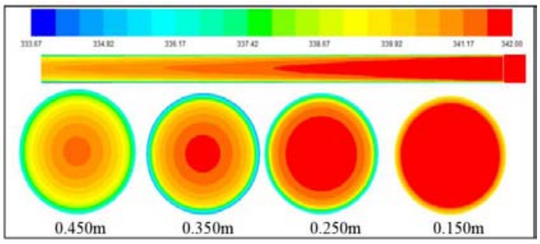


Figure 11 CFD contour for case 2

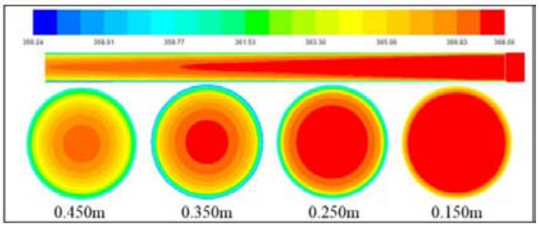


Figure 12 CFD contour for case 3

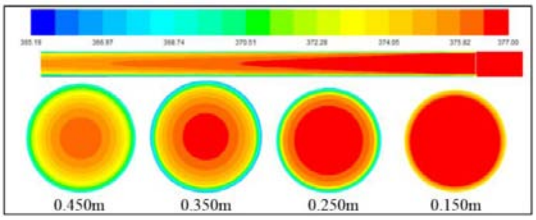


Figure 13 CFD contour for case 4

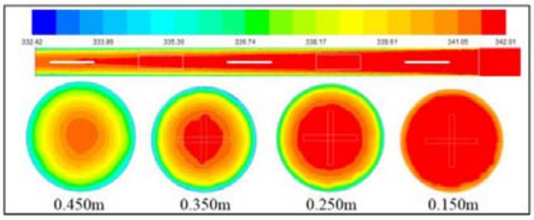


Figure 14 CFD contour for case 5

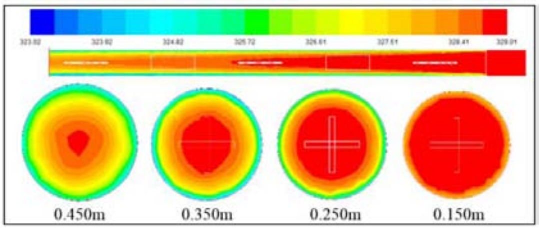


Figure 15 CFD contour for case 6

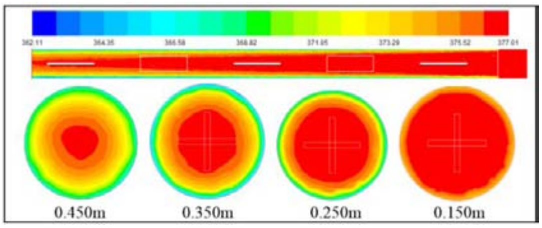


Figure 16 CFD contour for case 7

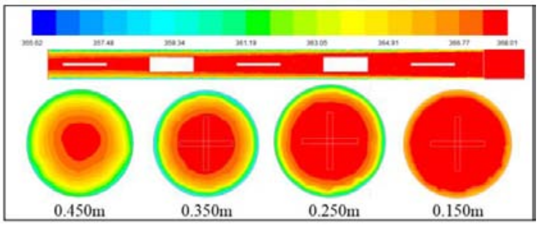


Figure 17 CFD contour for case 8

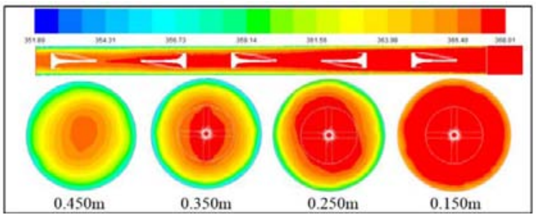


Figure 18 CFD contour for case 9

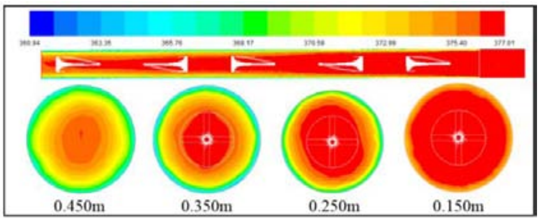


Figure 19 CFD contour for case 10

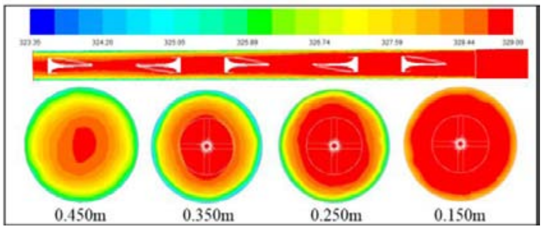


Figure 20 CFD contour for case 11

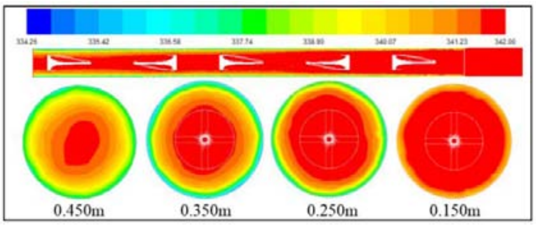


Figure 21 CFD contour for case 12

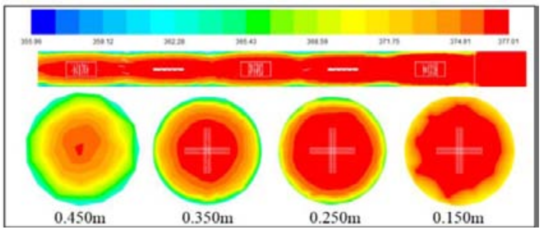


Figure 22 CFD contour for case 13

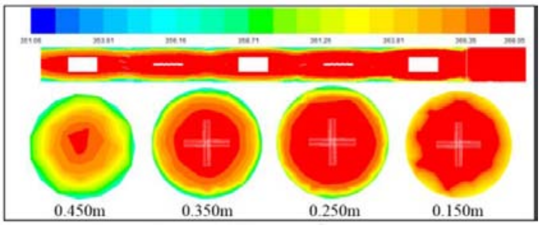


Figure 23 CFD contour for case 14

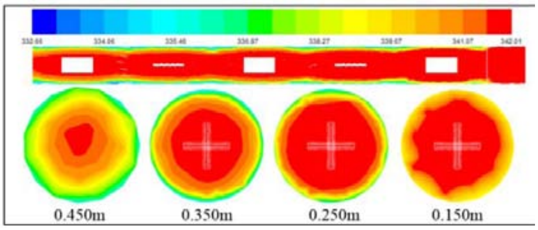


Figure 24 CFD contour for case 15

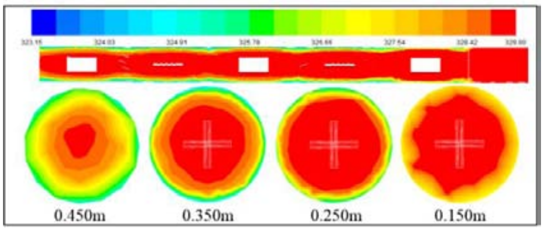


Figure 25 CFD contour for case 16

6 CONCLUSIONS

1. In the present paper, force convective heat transfer analysis for a circular tube is studied experimentally and numerically.
2. Experimental results are validated by CFD simulation analysis.
3. In this research work, experiments were design by using DOE software. Taguchi technique is applied for analysis of the observed data. Total three factors and two responses are selected for four levels each. Total 16 experiments were performed as per L16 design layout.
4. For all 16 cases CFD contours are generated and discussed.
5. Signal to Noise ratio analysis is performed for both responses (i.e. Nu & θ). This analysis gives the rank of factors on selective response. For both responses extended surface (ES) is ranked number 1.
6. Another result given by the signal to noise ratio is the optimal level for both responses. The second level (2^{nd}) i.e. Rectangular insert the optimal level of the factors for optimum heat transfer.

REFERENCES

[1] A. Tohidi, H. Ghaffari, H. Nasibi and A. S. Mujumdar, "Heat transfer enhancement by combination of chaotic advection and nanofluids flow in helically coiled tube" Applied Thermal Engineering (2015), 86, 91-105.

[2] C. X. Lin and M. A. Ebadian, "The effects of inlet turbulence on the development of fluid flow and heat transfer in a helically coiled pipe" International Journal of Heat and Mass Transfer (1999), 42, 739-751.

[3] A. M. Husseina and H. K. Dawood, "Numerical study on turbulent forced convective heat transfer using nanofluids TiO2 in an automotive cooling system" Case Studies in Thermal Engineering (2017), 9, 72–78.

[4] F. Akbaridoust and M. Rakhsha, "Experimental and numerical investigation of nanofluid heat transfer in helically coiled tubes at constant wall temperature using dispersion model" International Journal of Heat and Mass Transfer (2013), 58, 480–491.

[5] Jalaluddin and A. Miyara, "Thermal performance and pressure drop of spiral-tube ground heat exchangers for ground-source heat pump" Applied Thermal Engineering (2015), 90, 630-637.

[6] P. Naphon, "Heat transfer and pressure drop in the horizontal double pipes with and without twisted tape insert" International Communications in Heat and Mass Transfer (2006), 33, 166-175.

[7] V. Singh, S. Chamoli, M. Kumar and A. Kumar, "Heat transfer and fluid flow characteristics of heat exchanger tube with multiple twisted tapes and solid rings inserts" Chemical Engineering and Processing: Process Intensification (2016), 102, 156-168

[8] Y. A. Cengel "Heat Transfer: A Practical Approach" 2nd ed., McGraw-Hill, 2003.

[9] J. S. Jayakumara, S. M. Mahajani and J. C. Mandal, "CFD analysis of single-phase flows inside helically coiled tubes" Computers and Chemical Engineering (2010), 34, 430–446.

[10] L. S. Sundar and M. K. Singh, "Convective heat transfer and friction factor correlations of nanofluid in a tube and with inserts: A review" Renewable and Sustainable Energy Reviews (2013), 20, 23–35.

[11] Zhenyu Wang and Yuchen Wang, "Improvement of the aerodynamic performance of vertical axis wind turbines with leading-edge serrations and helical blades using CFD and Taguchi method" Energy Conversion and Management (2018), 177, 107–121

[12] M. T. J. Abad, A. Zamzamian and M. Dehghan, "Experimental studies on the heat transfer and pressure drop characteristics of Cu–water and Al–water nanofluids in a spiral coil" Experimental Thermal and Fluid Science (2013), 47, 206–212.