Experimental and CFD Analysis of Twisted Tube Heat Exchanger under Forced Convection

Vivek P. Thawkar[1], Hemant S. Farkade[2]

[1]PG Scholar, Mechanical Engineering Department, Government College of Engineering, Amravati, MH, India

[2]Assistant Professor, Mechanical Engineering Department, Government College of Engineering, Amravati, MH, India

Abstract: ***Experimental investigation has been carried out to study the effect of overall heat transfer coefficient in twisted elliptical tube heat exchanger. The main aim of this experimentation is to determine the overall heat transfer coefficient and friction factor of twisted elliptical tubes in multipass arrangement, with water as a working fluid. The experimental model was validated with the computational model. Twisted elliptical tubes with major diameter 18 mm and minor diameter 12 mm with twist pitch 60 mm are used. The material used is commercially pure copper. Reynolds numbers were varied in turbulent zone in the range of 50000 to 350000. Experimental data obtained from test section for different flow rates of water, 0.2 kg/s, 0.147 kg/s, 0.095 kg/s and 0.055 kg/s. The purpose of this study is to determine the feasibility of twisted elliptical tubes for use in applications like automobile radiators, air conditioners or similar type of multipass applications.***

Keywords: Twisted tube heat exchanger, CFD, Reynolds Number, Friction factor.

1. Introduction

Heat Exchangers are the devices that are used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. In heat exchangers, there are usually no external heat and work interactions. They control system or substance temperature by adding or removing thermal energy. There are many different sizes, levels of sophistication, types of heat exchangers, but all of them use same conducting element in the form of plate or tube, to separate two fluids which exchange thermal energy. In most of the heat exchangers, heat exchange between two fluids takes place through a separating wall. Typical applications involve heating and cooling of fluid or condensation and evaporation of fluid streams. Some of the common heat exchangers are shell and tube, Radiators, Condensers, Air-preheaters and Evaporators. Heat exchangers are extensively used in many engineering applications like space craft, automobiles. Due to increase in demand for heat exchange equipment, it is required to have more efficient heat exchanger for higher heat transfer rates with lower pressure drop. In designing the space craft applications it is necessary to design a compact and light weight heat exchanger. Thus such applications led to development of heat transfer enhancement techniques. The purpose of Heat transfer enhancement techniques is to maximize heat transfer rate and minimize pressure drop.

There are three types of heat transfer enhancement techniques namely active, passive and compound techniques.

Active method: It requires external power input to enhance heat transfer. Some of the active techniques are electric field, surface vibration. These techniques are more complex from design point of use. It finds limited applications since it requires power supply.

Passive method: This technique does not need any external power to enhance heat transfer. It takes available power in the system for enhancement of heat transfer. It includes special additives or special surface geometries. Curved tubes geometries can be classified as torus, coiled tubes, serpentine tubes.

Compound method: The compound method is hybrid method which uses both active and passive methods.

Twisted tube technology is one of the passive heat transfer enhancement technology, which has been able to overcome some of the limitations of the conventional shell and tube technology, and in addition, provide superior overall heat transfer coefficient through tube side enhancement. Twisted tube heat exchanger offers the solution to most of the shell and tube heat exchanger problems. Twist produces the swirling flow, increases the turbulence thus even at low Reynolds numbers the heat transfer is more as the heat transfer coefficient is improved. Baffles are eliminated thus no pressure drop and dead spots in the flow fields. Increased turbulence flushes away the fouling thus prohibits the fouling buildups. Twisted tubes are formed into an elliptical or oblate cross section with a superimposed twist. Twisted tubes are mostly chosen in the design of industrial shell and tube heat exchangers, which can considerably reduce the size of the heat exchangers. Here the tubes are such that they are self supported by each other at many points therefore there is no problem of tube

2. Literature Review

Number of investigations has been carried out by using various inserts and tube geometries for passive heat transfer. Investigations of various researchers and their findings are as follows:

Li Zhang, Sheng Yang et al. [2] conducted an experiment on condensation heat transfer characteristics of steam on horizontal twisted elliptical tubes (TETs) with different geometrical parameters. A smooth circular tube, an elliptical tube (ET) and five TETs with different structure parameters were tested at the steam saturation temperature of around 100.5 ºC with the wall subcooling from around 2 ºC to 14 ºC.

A smooth circular tube has higher condensation heat transfer coefficients than all the tested TETs. The enhancement factors of the five TETs from TET No. 1 to TET No. 5 range from 0.87 to 1.34. The condensation heat transfer coefficients of steam on the TETs increase with the rise in the tube ellipticity. The condensation enhancement of around 34% was the highest for the TET No. 3 with largest ellipticity of 0.86. The condensation heat transfer coefficient was lower for a smaller twist pitch. Sheng Yang, Hong Xu, et al. [3] experimentally investigated the heat transfer and flow resistance characteristics of water flow inside the twisted elliptical tubes (TETs) with different structural parameters. Effects of tube structural parameters (aspect ratio and twist pitch) on the performance of TETs were analyzed. Experimental results indicate that the heat transfer performance is better for both a larger tube aspect ratio (A/B) and a smaller twist pitch (S). The effect of twist pitch on heat transfer performance is more notable than that of tube aspect ratio. Xiang-hui Tan, Dong-sheng Zhu, et al. [4] studied that the tube side and shell side heat transfer and pressure drop performances of a twisted oval tube heat exchanger experimentally. Experimental study shows that the tube side heat transfer coefficient and pressure drop in a twisted oval tube are both higher than in a smooth round tube. The analyzing result shows that the twisted oval tube heat exchangers is preferred to work at low tube side flow rate an high shell side flow rate. Guo-yan Zhou, Liu Yang, et al. [5] studied the fluid flow and heat transfer characteristics in the shell side of twisted oval tube type of heat exchanger are studied numerically with Realized k-ε model. Influence of the geometrical parameters including twisted pitch length P and aspect ratio A/B on the performance of the shell side are analyzed. In this numerically study, a twisted oval tube heat exchanger with 7 tubes in the spell is applied. In order to avoid leakage from the flow channel between the shell wall and tube bundle, profile of the shell is modeled hexagonal in the present work. During the simulation of the fluid flow and heat transfer in the shell side flow channel of the twisted oval tube heat exchanger, the Realized k-ε model is chosen. SIMPLEC algorithm is employed to solve the flow and pressure equations. The second order upwind algorithm is employed in simulating the boundary layer motion of the fluid. All the governing equations are solved with pressure based coupled algorithm. By comparing the numerical results with the experimental results from former researchers, the Realized k-ε model and grid size 0.3-0.5 are verified to be employed in the numerical study. Numerical results shows that Nusselt number and friction factor both increase with the increasing of P and A/B on the overall heat transfer performance h/ΔP of the heat exchanger are also analyzed. The result shows that the overall heat transfer performance of these cases increase with the increasing of A/B. Analysis of the secondary shows that the magnitude of the secondary flow increases with the increasing of A/B and decreases with the increasing of P. A.R. Sajadi *et al.* [6] discussed that heat transfer and flow resistance of alternating elliptical axis tubes, both experimentally and numerically. The working fluid is heat transfer oil. The flow's Reynolds number ranges from 300 to 2000. The grid and numerical models are generated using Gambit 2.4.6 and Fluent 6.3. The experimental results were verified with the numerical results. The numerical results show that decreasing the aspect ratio and pitch length, increases heat transfer and flow resistance.

In order to compare the heat transfer and flow resistance simultaneously, the non-dimensional heat transfer enhancement ratio is defined. The comparison of this ratio shows that alternating elliptical axis tubes perform better than the flattened or circular ones. It is observed that this ratio increases with the increase in Reynolds number.

3. Experimental Model

3.1 Experimental set-up description

The schematic diagram of experimental setup is shown in figure 1. The test section consists of elliptical copper tube (major diameter=18 mm, minor diameter=12 mm, thickness= 2 mm) in multipass arrangement with length of 350 mm of each tube pass. The test section was fitted in the centre of the duct of size 350 mmx400 mm with a length of 1000 mm. Two k-type thermocouples were placed at inlet and outlet of the test section. Water is used as working fluid and it was heated in the hot water tank with coiled heater of 1000 Watts capacity. For maintaining the forced convection condition blower is used. Mass flow rate was controlled with the help of ball-valve. A U-tube manometer was used to measure the pressure drop across the tube; water was used as manometric fluid. All these components were required to assemble for carrying out experimentation.

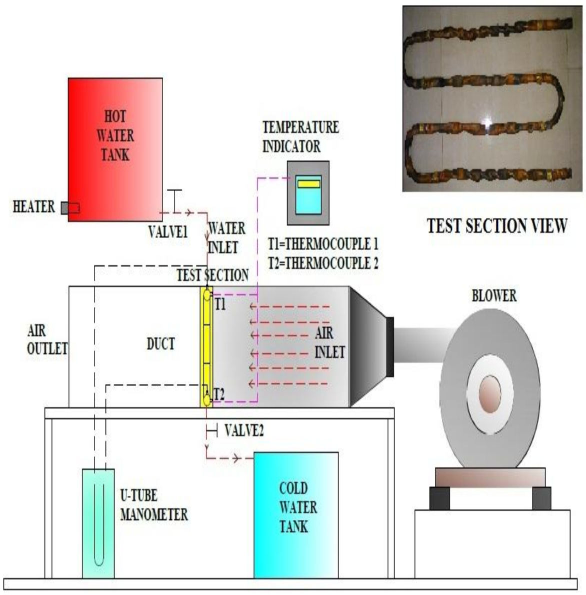

Figure 1: Schematic diagram of experimental setup.

3.2 Experimental Procedure

In present setup natural circulation of water through the tube is employed. The tank is kept at a height of 230 cm and the inlet of tube is at 122 cm, both measured from the floor in positive direction. Therefore there is sufficient head difference to cause the hot water to flow from the heater tank to the tubes. After the tubes, cooled water is collected in an outlet tank.

At first, water is blown through the whole system so that the system is free from air. Water is heated in the tank up to the temperature of 363 K. Then the ball valve is opened and the water is allowed to flow through the tubes. The flow is regulated using second ball valve only it ensures that the tubes are always full of water. Air is blown over tubes using a fan which results in forced convection type of heat transfer process. After the tubes cooled water is collected into a tank. The collected water is reused for next reading, this completes the process. Temperature is recorded at two points, one at the inlet of twisted tube and other at the outlet of the tube. Anemometer is used to measure the velocity of air. Four flow rates of water 0.2 kg/s, 0.147 kg/s, 0.095 kg/s, 0.055 kg/s are used. For each flow rate four temperatures 363 K, 353 K, 343 K and 333K are used. Air side flow rate is fixed at 0.34 kg/s and water side flow rate is varied. Water flow rate is varied with the help of second ball valve. Particular mass flow rate is set with the help of valve opening. First for full opening of valve, readings are taken. With the help of measuring jar and stopwatch water mass flow rate is measured. First water mass flow rate is set, then readings are taken for above given four temperatures and the same procedure is repeated for the remaining mass flow rate.

4. Computational Model

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions and related phenomena by solving the mathematical equations which govern these processes using numerical methods in the computer. CFD provides a qualitative and quantitative prediction of fluid flows by means of mathematical modeling, numerical methods, and software tools. It gives an insight into flow patterns that are difficult, expensive or impossible to study using experimental techniques.CFD simulation is a three step process namely Pre-processing, Solver, Post- processing.

4.1 Pre-processing

The CAD model was developed in CATIA V5 R21 in Generative shape design module. CAD model is exactly as experimental model. The meshing of geometry is done in ICEM CFD 14.5 module of software ANSYS 14.5. In meshing, both surface meshing and volume meshing were used. In surface meshing, mesh type was all tri used and mesh method patch independent was used. In volume meshing, mesh type tetra/mixed was used and mesh method robust Octree was used. The meshing model consists of 2174430 elements and 359108 nodes. The quality of mesh was 0.39122.

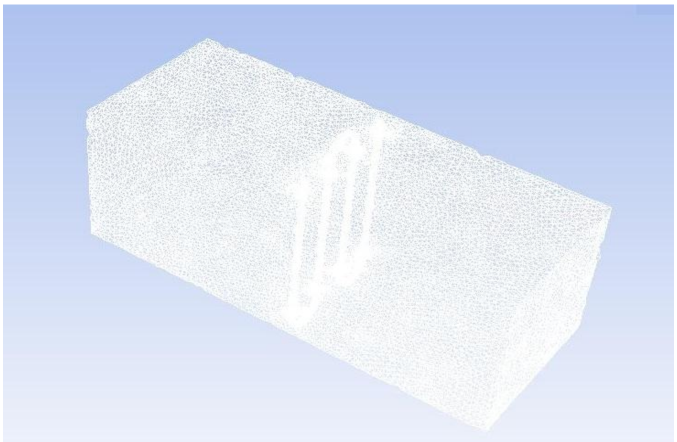

Figure 2: Meshing Model.

4.2 Solver and Post-processor

In present computational process, calculations were made by using ANSYS FLUENT 14.5 software. Pressure-based solver was used. Standard k-ε turbulence model with enhanced wall treatment was used. SIMPLE algorithm was employed to solve the flow and pressure equations. The standard method was employed in the pressure term. The second order upwind algorithm was employed in simulating the boundary layer motion of the fluid. All the governing equations were solved with pressure-based coupled algorithm. Firstly, it solved a coupled system of equations comprising the momentum equations and pressure based continuity equations and then solved the energy, turbulence and other scalar equations. The post-processor provides for visualization of the results, and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. FLUENT 14.5 is the post-processor used for present work.

5. Data Reduction

As we know that, for cross-flow and multipass heat exchangers, the heat transfer equation is given as:

$$Q = FUA(\Delta T)_{LMTD} \quad (5.1)$$

Where, ΔT_{LMTD} = Logarithmic mean temperature difference
It is given by:

$$\Delta T_{LMTD} = \frac{\theta_1 - \theta_2}{\ln\frac{\theta_1}{\theta_2}} \quad (5.2)$$

Where, $\theta_1 = T_{hi} - T_{co}$
$\theta_2 = T_{ho} - T_{ci}$
F = correction factor, for cross flow F=1 (from charts)
U = overall heat transfer coefficient in W/m^2-k
A= n Π D_h L = surface area of twisted tube in m^2
Where, n = Number of tube passes
D_h = Hydraulic diameter of twisted tube in meter
L= Length of each tube pass in meter

Here all the temperatures are known from experiment. Heat transfer from hot and cold fluid is given by:

$$Q = m_h.C_p.(Th_i - Th_o) = m_c.C_p.(Tc_o - Tc_i) \quad (5.3)$$

From equation (5.1) and equation (5.3), the value of U can be found experimentally.

Friction factor is also calculated by:

$$\Delta P = f\left(\frac{L}{D_h}\right)\left(\frac{\rho u^2}{2}\right) \quad (5.4)$$

Where, ΔP = Pressure drop in twisted tube in Pascal
L = Length of test section in meter
D_h = hydraulic diameter of twisted tube in meter
f = friction factor
u = velocity of water in m/sec

Experimental pressure drop (ΔP) is calculated by formula:

$$\Delta P = \rho g \Delta h \quad (5.5)$$

Where, Δ h = Manometric height in meter
ρ = density of fluid in kg/m^3
g = Acceleration due to gravity m/sec^2

Theoretical friction factor for twisted tube is determined by a unified equation given by "Sheng Yang *et. al.*" This equation can be applied for the Reynolds number in laminar as well as for turbulent range as:

$$f = 1.529 Re^{-0.350} (A/B)^{1.686} \left(\frac{S}{d_h}\right)^{-0.366} \quad (5.6)$$

The Reynolds Number and hydraulic diameter can be determined by using following equations:

$$Re = \frac{\rho U D_h}{\mu} \quad (5.7)$$

$$D_h = \frac{4 A_c}{P} \quad (5.8)$$

Where, A_c = Cross sectional area in m^2
P = Perimeter in meter

6. Results and Discussion

Experimental study is done to determine the behavior of twisted tube with circular end connections in order to analyze twisted tube's applicability in multipass applications like radiators, air conditioners and similar kind of multipass application. Experimentally the flow rate used is in high Reynolds number range of Re 50000-350000. Experimental results were validated with the CFD results.

Figure 3 shows the comparison of mass flow rate of water with temperature difference (ΔT) at temperature 363k in twisted elliptical tubes and validated with the computational results. From the graph it is clear that temperature difference is decreases with the increase in mass flow rate

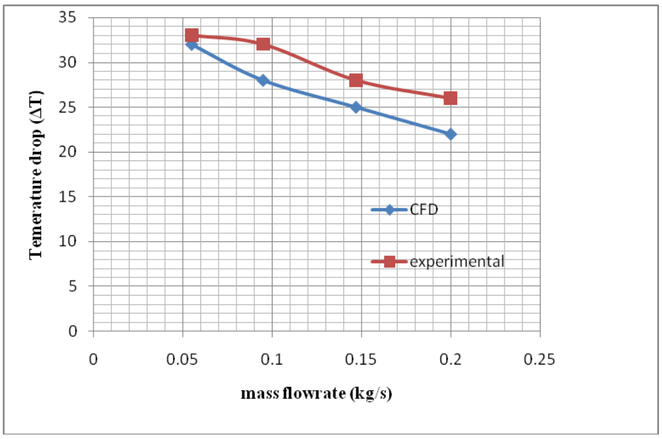

Figure 3: Comparison between temperature drop (ΔT) with the mass flow rate at temperature 363k, experimentally and computationally

Figure 4 shows the comparison between overall heat transfer coefficient with the Reynolds number experimentally and computationally. Overall heat transfer coefficient of twisted elliptical tube is increases with the increase in the Reynolds number. A logarithmic trend pattern is also shown.

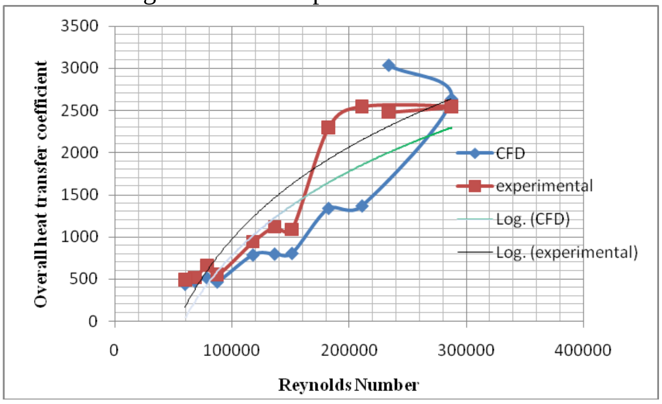

Figure 4: Comparison of overall heat transfer coefficient with the Reynolds Number, experimentally and computationally

Figure 5 shows the comparison between the pressure drop (ΔP) with the Reynolds number, experimentally and computationally. Pressure drop was measured by U-tube manometer in test section tube. From the graph it is cleared that the pressure drop is increased with the increase in the Reynolds number.

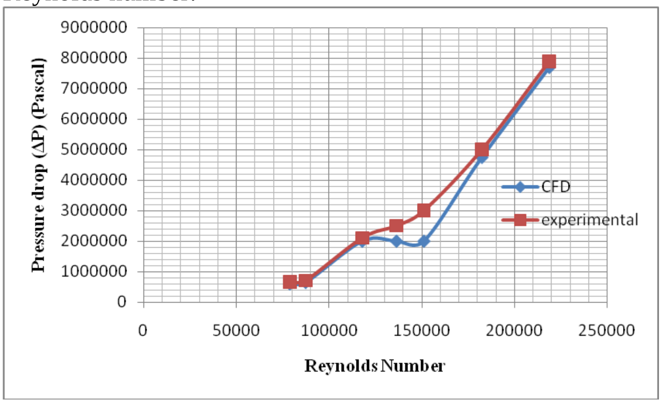

Figure 5: Comparison of pressure drop (ΔP) with the Reynolds number, experimentally and computationally

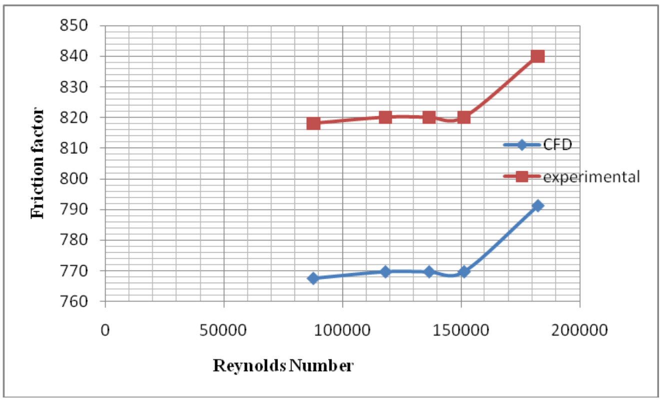

Figure 6: Comparison of friction factor with Reynolds number, experimentally and computationally

Figure 6. shows the comparison between friction factor with the Reynolds number. The experimental friction factor and computational friction factor were compared. The

computational results were computed for different mass flow rates. The contours of temperature were shown in the figure 6 and figure 7. From the computational results, it was cleared that temperature drop was more for low mass flow rates.

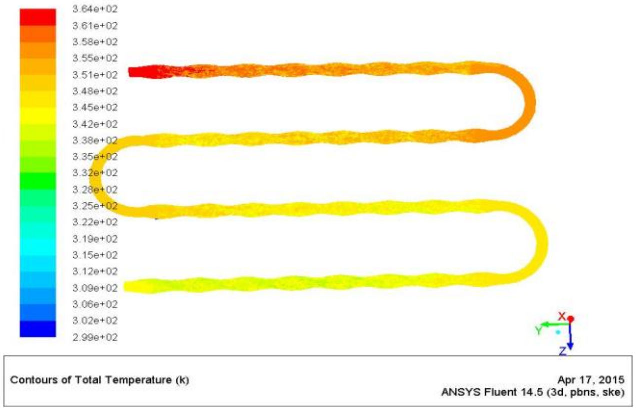

Figure 6: Temperature distribution in tube side for mass flow rate of 0.2 kg/s at 363k.

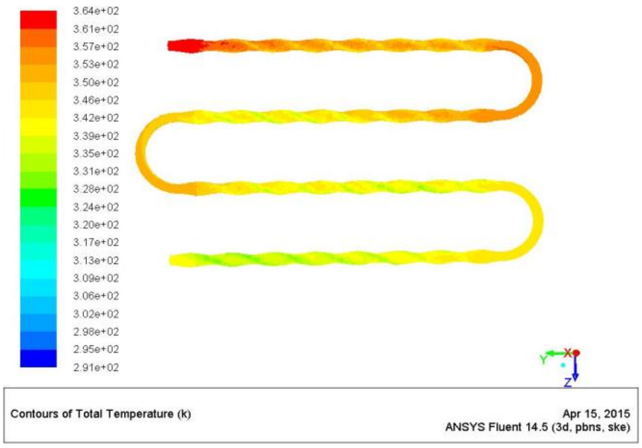

Figure 7: Temperature distribution in tube side for mass flow rate of 0.055kg/s at 363k

7. Conclusions

Heat transfer enhancement of twisted elliptical tube in multipass arrangement has been studied experimentally and computationally. The work has been conducted in the turbulent flow regime (Reynolds number in between 50000 to 350000) using water as working fluid. The findings of the work can be drawn as follows:

- At low flow rates turbulence is more in twisted tube further the fluid follows the tube's profile and thus there is a good velocity profile. This result in a good synergy between temperature and velocity profile and thus heat transfer is more. For the applications like radiators and air conditioners where the flow rate is high twisted tubes should be used. As in these applications reducing the heat exchanger size is very important. In case of automobile the space in passenger compartment increases and in refrigerating units the cost and floor space can be saved which results from compact construction of the heat exchangers.
- With the circular connecting ends in twisted tubes the flow tries to generate swirl in the twisted part. As the length is very short, fully developed velocity profile cannot be produced. Further with the plain circular ends the little developed velocity profile gets again disturbed. The result is that everywhere in the domain the velocity and temperature profiles are not fully developed.
- With undeveloped velocity and temperature profiles in the twisted tube synergy between them is not established. Thus the heat transfer is not good. In the present study, performance of twisted tube is better in high Reynolds number range. The reason may be that the flow under such condition is highly turbulent and the swirls are developed, results in good heat transfer.

Nomenclature

Symbol	*Meaning*	*Unit*
A	Surface area of twisted elliptical tube	m^2
A_c	Cross-sectional area of twisted tube	m^2
A	Major diameter of twisted tube	m
B	Minor diameter of twisted tube	m
C_p	Specific heat of water at constant pressure	J/kg-k
D_h	Hydraulic diameter of twisted tube	m
F	Correction factor	-
f	Friction factor	-
g	Acceleration due to gravity	Kg/s^2
Δh	Manometric height	m
L	Length of test section	m
m_h	Mass of hot water	Kg/s
m_c	Mass of cold air	Kg/s
n	Number of tube passes	-
p	Perimeter of twisted tube	m
ΔP	Pressure drop in test section	Pascal
Q	Heat transfer rate	W
R_e	Reynolds Number	-
S	Twist pitch	m
Th_i	Inlet temperature of water	k
Th_o	Outlet temperature of water	k
Tc_i	Inlet temperature of air	k
Tc_o	Outlet temperature of air	k
ΔT_{LMTD}	Logarithmic mean temperature difference	k
U	Overall heat transfer coefficient	W/m^2-k
u	Velocity of water	m/s
ρ	Density of water	Kg/m^3

References

[1] Y. A. Cengel, "Heat Transfer- a Practical Approach", SI units 2nd Edition, Tata McGraw Hill, 2004.

[2] Li Zhang, Sheng Yang, Hong Xu, "Experimental study on condensation heat transfer characteristics of steam on horizontal twisted elliptical tubes", Applied Energy, vol. 97(2012), p.p.881-887.

[3] Sheng Yang, Li Zhang, Hong Xu, "Experimental study on connective heat transfer and flow resistance characteristics of water flow in twisted elliptical tubes", Applied Thermal Engineering, vol. 31(2011), p.p. 2981-2991.

[4] Xiang-hui Tan, Dong-Sheng Zhu, Guo-van Zhou, Li-ding Zeng, "Heat transfer and pressure drop performance of twisted oval tube heat exchanger", Applied Thermal Engineering, vol. 50(2013), p.p. 374-383.

[5] Xiang-hui Tan, Dong-sheng Zhu, Guo-van Zhou, "3D numerical simulation on the shell side heat transfer and pressure drop performance of twisted and oval tube heat

exchanger", International Journal of Heat and Mass Transfer, vol. 65(2013), p.p. 244-253.
[6] Gh.S.Jahanmir, F. Farhadi, "Twisted bundle heat exchanger performance evaluation by CFD", International Communications in Heat and Mass Transfer, vol. 39(2012), p.p. 1654-1660.
[7] Smith Eiamsa-ard, Pongjet Provonge, "Performance assessment in a heat exchanger tube with alternate clockwise and counter-clockwise twisted-tape inserts", International Journal of Heat and Mass transfer, vol. 53(2010), p.p. 1364-1372.
[8] V. Hejazi, M. A. Akhavan-Behabadi, "Experimental investigation of twisted tape inserts performance on condensation heat transfer enhancement and pressure drop", International Communications in Heat and Mass transfer, vol. 37(2010), p.p. 1376-1387.
[9] C. Thianpong, K. Yongsiri, K. Nanan, "Thermal performance evaluation of heat exchangers filled with twisted-ring turbulators", International Communications in Heat and Mass Transfer, vol. 37(2012), p.p. 861-868.
[10] Ramesh K. Shah, Dusan P. Sekulic, Fundanemtals of Heat Exchanger Design, (Publication John Wiley & Sons, Inc. 2003, ISBN0-471-32171-0, p.p. 13-64.
[11] Suvanjan Bhattacharyya, Subhankar Saha, Sujoy Kumar Saha "Laminar flow heat transfer enhancement in a circular tube having integral transverse rib roughness and fitted with centre-cleared twisted-tape." Experimental Thermal and Fluid Science 44 (2013) p.p.727–735.
[12] S. Eiamsa-ard, P. Seemawute, "Decaying swirl flow in round tubes with short-length twisted tapes", International Communications in Heat and Mass Transfer, Vol. 39, (2012), pp. 649–656.
[13] Si-hong Song, Qiang Liao, Wei-dong Shen "Laminar heat transfer and friction characteristics of microencapsulated phase change material slurry in a circular tube with twisted tape inserts." Applied Thermal Engineering 50 (2013) pp.791-798.

Author Profile

Vivek P. Thawkar completed B.E. (Mechanical Engineering) in year 2011 from R.T. M. Nagpur University. Currently perusing M. Tech (Thermal Engineering) from Government College of Engineering Amravati, Maharashtra State, 444604, India. An autonomous institute of Government of Maharashtra.

Hemant S. Farkade is an assistant professor of Mechanical Engineering at Government College of Engineering, Amravati, Maharashtra State, 444604, India. An autonomous institute of Government of Maharashtra. His research area is Heat transfer and IC Engines.