

# Numerical and Experimental Investigation of Heat Transfer Enhancement in Slot Groove Circular Tube with Internal Twisted Tape

Humam Kareem Jalghaf

Mechanical Engineering Dept. - University of Technology, Baghdad – Iraq  
E-mail: ([Humam.karem@yahoo.com](mailto:Humam.karem@yahoo.com)) or ([20310@uotechnology.edu.iq](mailto:20310@uotechnology.edu.iq))

## Abstract:

Numerical and experimental investigation on the turbulent enhancement convective heat transfer inside slot and plain dimples tubes with internal twisted tape were performed in this study. An experimental rig was constructed and instrumented to evaluate the heat transfer enhancement and pressure drop at this surface. Air was used as working fluid, and steam was used as a heating source where constant wall temperature condition of ( $135^{\circ}\text{C}$ ) was achieved. Heat transfer and pressure drop data were obtained from four configuration tube. The test facility was capable for providing turbulent flow with Reynolds number varied from 4000 to 15000. Thermal and hydrodynamic flow pattern was numerically studied using commercial code FLUENT15. The average heat transfer of the experimental results was in good agreement with the numerical ones. The result depict that the slot dimple tube with twist tape and plain tube with twist tape give high enhancement in heat transfer relative to plain tube due to increase in area of heat transfer. The average enhancement ratio for slot dimple tube with (TR =4 and 8) are (1.204 and 1.202) respectively. This indicates that 15.5-20.4% of heat transfer area can be saved at the same pumping power for present cases configuration compared with the plain tube heat exchanger.

## 1- Introduction

The study of heat transfer growth in heat exchanger has important benefit in the design of economical and effective heat exchanger. Accretion technicality raise convective heat transfer by decrease thermal resistance in a heat exchanger. A reduce in heat transfer surface size, area, and hence weight of heat exchanger for a given pressure drop and heat duty. The heat transfer can be raised by various techniques that have been suggested in last years and generally referred to Enhancement.

Heat transfer enhancement can be categorized in two main type: efficient, inefficient and hybrid technique. Hybrid technique includes two or more from each of inefficient and efficient technique [1].

Efficient techniques: In these techniques, an external energy is utilized to simplify the required

flow adjustment and the concomitant refinement in the heat transfer rate [2], Inefficient techniques mostly uses geometrical or surface modifications to the flow channel by additional devices or combine inserts, they not need any external energy input. The additional energy that needed to achieve the required heat transfer enhancement is taken from the available power in the system which in the end leads to amplify pressure drop of a fluid, and Hybrid technique there compound enhancement includes two (or more) of the technicality used together to product an enhancement that is greater than the single techniques utilized separately.

Watcharin et al. [3], studied experimentally the effect of a twisted tape intercalation on the heat transfer and flow friction characteristics in a concentric double pipe heat exchanger for the range of Reynolds number of 2000-12000. The experimental results showed that the enhancement in heat transfer rate of the twisted-tape inserts is found to be robustly affected by tape- produce swirl motion.

Eiamsa-ard, et. al. [4] studied the heat transfer characteristics and pressure drop of flow through twisted tape insert in a circular tube. In the experimental work, a twisted-tape was inserted into the internal tube with different free spacing twisted-tapes:  $s = 2P$ ,  $3P$ , and  $4P$ , respectively ( $P$ =pitch length of twisted tape). All of the experiments were carried out at Reynolds number of the internal tube,  $Re=2300$  to  $7500$ . The study revealed that the free spacing twisted-tapes,  $s = 2P$  gives the heat transfer lower than full length twisted tape around 5-15%, while it can decrease the pressure drop around 90%.

Chinaruk Thianpong et al [5] studied an empirical investigation of fully developed turbulent flow in a dimpled tube in conjunction with a twisted tape, the effect of the twist ratio and pitch ratio on the friction factor and heat transfer rate characteristics was also studied. Their results showed that the dimpled tube in common with a twisted tape has significant influence on the friction factor and heat transfer enhancement. The friction factor and heat transfer raise with decreasing both of twist ratio ( $y/w$ ) and pitch ratio (PR). Depending on the twist ratio and pitch ratio, the friction factor and heat transfer rate in the dimpled tube with twisted tape are

respectively 5 to 6.31 and 1.66 to 3.03 times of those in the plain tube. The experimental correlations for the friction factor and the Nusselt number were developed.

In the current study the enhancement in heat transfer is tested in six cases: three in a plain tube and three in a plain tube with dimpled slot. Each tube was studied three times: once without inserted twisted tape, then the case was tested with inserted twisted tapes of twist ratio  $TR=4$  and  $TR=8$  respectively.

The effect of the twisting ratio and Reynolds number on Nusselts number with a uniform heat flux were studied and compared with that of plain tube and calculating the rate of friction factor and Nusselts number based on experimental data. The theoretical analysis of the current study will be completed using ANSYS FLUENT to simulate the influence of Reynolds number on average Nusselts number, heat transfer enhancement and velocity profile.

## 2- Experimental Setup:

In the present work, a constant wall temperature technique was used by applying saturated steam on the external wall of test tube. The wall temperature will be more likely equal to the condensation temperature. This means that no set of thermocouples is required to be placed on the tube wall. The photo of the experimental apparatus used in the current study is shown in Fig. 1, consists of an arrangement of force convection open loop flow system. This system contains air blower, piping, instrumentation and test section.

The test rig contain: 1-Bordan gauge. 2- Steam delivers. 3-Steam condensate drian. 4- Steam shell with insulation. 5-Flange of steam the shell. 6-Flange of the ring. 7-Outer air tube. 8-Inlet air tube. 9-Union joint. 10-Water manometer. 11-Pressure tapping.

The blower is connected to a carbon steel pipe of 50 mm diameter and 2 m length. Orifice plate intermediate this pipe is to measurement the air flow rate through the system. Two pressure taps were placed upstream and downstream the orifice flange to measurement the pressure difference, using water manometer. Temperature tap located upstream of orifice plate is for measuring the air blowing temperature. To achieve a fully developed flow at the test section entrance, a tube with 1m length has the same test tube diameter was connected with the test tube by a special coupling. The test section is one of the main components in an empirical test facility which consisted of a tube in tube heat exchanger in cross flow configuration. The test section contains a tube of 35 mm diameter surrounded by a jacket pipe of 75mm inner diameter with 1m long, and is insulated by 5 cm rock wool. Steam was used as source of heat and air as working

fluid. The steam is supplied from a boiler available at the laboratory, and pressure regulator valve was used to obtain constant saturated steam pressure and temperature at annulus of test tube. The jacket is instrumented with pressure, temperature gauge, drain and vent valve to get rid of the non-condensable gases and to drain the condensate steam during the test period. Temperature and pressure taps are placed at the entrance and the exit of the test tube to measurement the temperature and pressure of air across the tested section. The following subsections show the experimental apparatus in details.

## A- Flow System

In the present work, a turbulent flow regime was considered, therefore the system was designed for obtaining Reynolds number in the range of 4000 to 15000. The air flow system includes the following parts:

- 1- Blower.
- 2- Air flow tube
- 3- Inlet air tube.
- 4- Test section.
- 5- Test tubes.

The following notes classify technical description for each part:

1-Blower: Centrifugal fan with impeller diameter (410 mm), was used for supplying air to the test section. The fan is capable to provide volume flow rate of 40 m<sup>3</sup>/hr and the maximum discharge pressure of 4000Pa. The fan is driven by an electric motor of 500W and 2800 r.p.m. The air flow rate is controlled by a slide vane located at the fan's inlet section.

2-Air Flow Tube: Air flow was measured by orifice plate manufactured in according to British Standard 1042. The pressure tap is connected with water manometer by flexible polyethylene tube [6].

3- Test Section: The test section contained of two parts:

- I. Shell side: The test section is a shell and tube configuration, where steam flows in shell side, while air flows inner the tube. The shell side is made from carbon steel tube of 75 mm inner diameter, outer diameter 82 mm and 1200 mm length. The shell ends are welded with the special flange that machined to produce sealing housing, where sealing cap can be pushed through by screw bolts. Teflon rope of 10 mm is packed in the flange housing and warping round the test tube while sealing cup is forced by the four bolts to compress the teflon rope on the test tube and on the inner ring of the flange housing to produce tight sealing and to maintain the saturation steam pressure and temperature.
- II. Test Tubes: Tube made from copper of (1600 mm) length and inner diameter of 35 mm, as plain tube and then form a slot groove with (40mm)

center to center distance, 5mm radius of slot and 5 mm groove depth. See Fig. 2, were tested in the present work, CNC machine was used in the manufacturing process.

III. Twisted Tape: The twisted tapes were manufactured from a metal strip of finite length twisted with different pitches and twist ratios (the pitch is the distance required for the strip to rotate

180°), and the twist ratio (TR) is the ratio of pitch to the width of the tape. The twisted tape was fabricated from a copper strip of length (160cm), width (17 mm) and thickness (0.70 mm), both ends of every tape were clamped by metallic clamps.

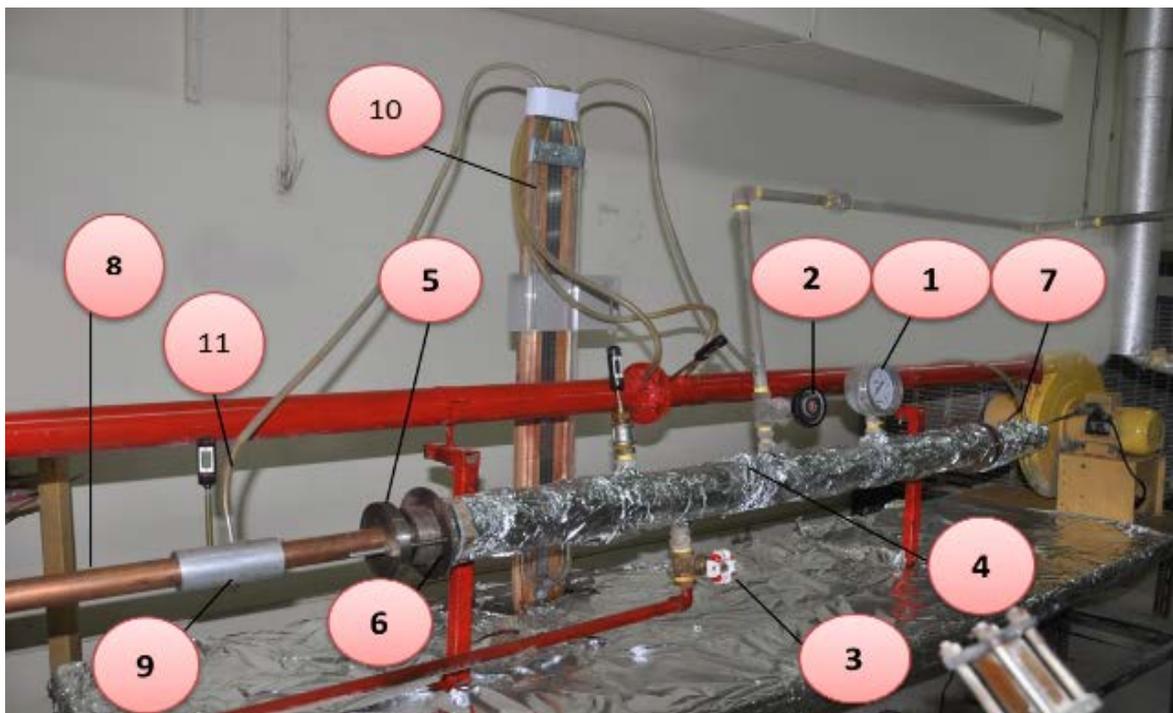
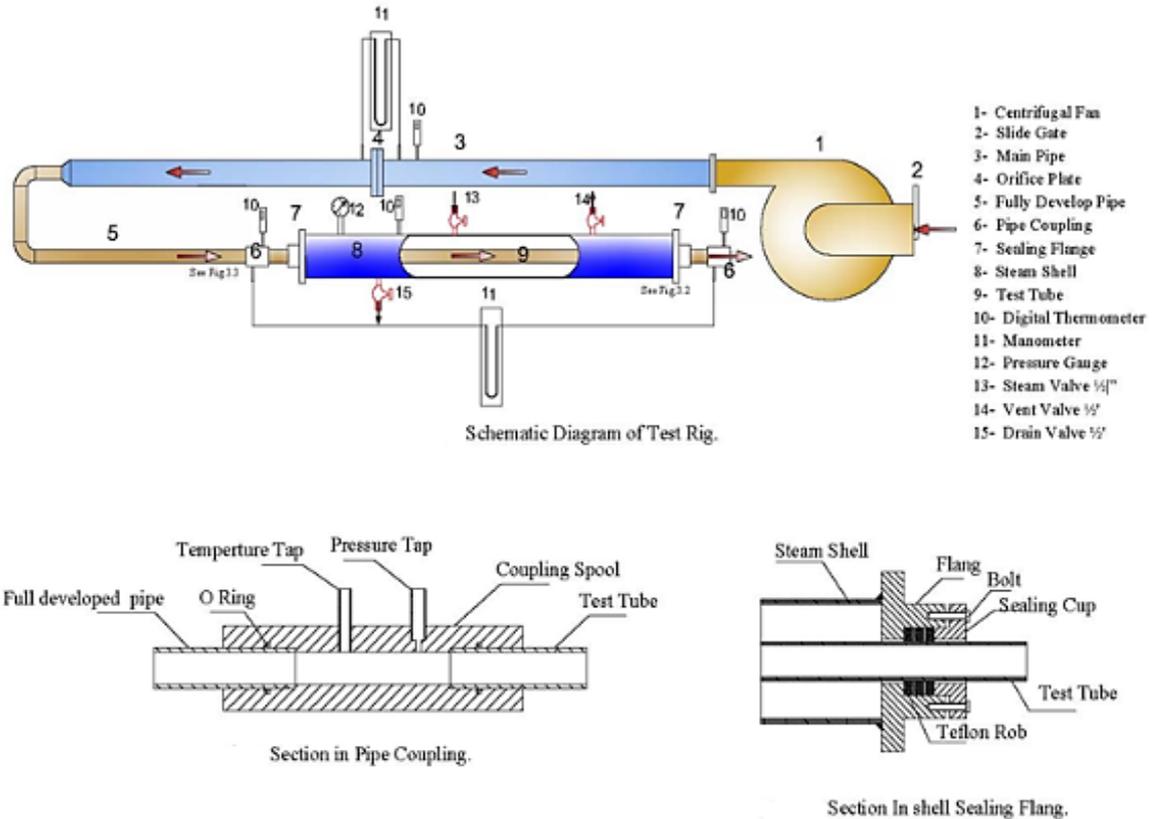


Figure 1: schematic and photo of test rig

**B- Steam Generation System**

In the present work, a fire tube boiler was used to produce a complete saturated steam. The boiler is supplied with safety controls, fully automatic burner, other tube fittings and measuring tools. Steam is supplied at pressure up to 12 bar and temperature 191.7°C. The saturated steam was provided to the test section from the boiler by flow regulation valve and throttling valve. The steam status at the inlet to the test section can be controlled for the desired temperature and pressure.

**C- Instrumentations**

The essential measurements required are volume flow rate, temperature of air and steam, and pressure drop through test tube. The following sections describe the instrumentation that had been used in the experimental test.

- 1- Air flow rate instrument: as explain above.
- 2- Temperature thermometers: Four (K-type) were used to measure the temperature. These thermocouples were fixed at the following positions: One thermocouple put upstream of the orifice plate. Two thermocouples were placed across the test tube to measure the inlet and the outlet air temperature. One thermocouple placed in the steam shell to measure the saturated steam temperature which represents the wall temperature of the test tube.
- 3- Pressure Instrument: Two types of pressure instruments were used:  
 Borden gauge produced by (Amet) range of (0-10 bar) with increments of 0.2bar was used to measure the saturated steam pressure in steam shell.  
 Two water manometers of range (1000 mm H2O) were used to measure the pressure difference across the orifice plate and the pressure drop through test tubes.

**3-Calculations**

The main objective of the present study is to evaluate the pressure drop and convective heat transfer coefficient across the test tubes. According to the first law of thermodynamic for incompressible flow with constant specific heat, the heat transfer to the air can be calculated.

$$Q_a = m^{\circ}_a C p_a (T_{a out} - T_{a in}) \dots \dots (1)$$

Where,  $m^{\circ}_a$  is the air mass flow,  $T_a$  is the inlet and outlet of air temperature.

To express the heat transfer of convection from the tube wall to the air, one can apply the newton's law for cooling. [7]

$$Q_w = h_a A_w (T_w - T_b) \dots \dots \dots (2)$$

The system is thermally balanced ( $Q_w = Q_a$ ), i.e, the heat gained by the air is the heat released by the tube wall. By combining equations 2 and 1, the heat transfer coefficient can be evaluated.

$$h_a = \frac{m^{\circ}_a C p_a (T_{out} - T_{in})}{A_w (T_{wi} - T_b)} \dots \dots \dots (3)$$

Where,  $T_b$  is the bulk air temperature and can be evaluated as follow:

$$T_b = \frac{T_{in} + T_{out}}{2} \dots \dots \dots (4)$$

To evaluate the inner wall temperature  $T_{wi}$  it can be evaluated from the measured heat flow through the test tube and the saturation steam temperature using the following equation [8]:

$$Q = \frac{2 \pi L (T_s - T_{wi})}{\frac{\ln(r_o/r_i)}{k} + \frac{1}{r_o h_c}} \dots \dots \dots (5)$$

Where  $h_c$ , is the condensation heat transfer coefficient on the horizontal tube and can be calculated from the following equation [8].

$$h_c = 0.725 \left[ \frac{\rho_l (\rho_l - \rho_v) g h_{fg} k^3}{\mu_l D_o (T_s - T_{wo})} \right]^{0.25} \dots (6)$$

**3.1- Data Reduction**

Heat transfer represented by dimensionless parameter Nusselt number is computed by:

$$Nu = \frac{h_a D}{k_a} \dots \dots \dots (7)$$

To validate the present experimental test apparatus the test was performed on a plain tube, and the results are compared with well-known equations of Dittus-Boelter ,Gueblinki and Blasins equation for heat transfer inside a tube for a turbulent flow at constant wall temperature [9].

$$Nu_u = 0.023 Re^{0.8} Pr^{0.3} \dots \dots \dots (8)$$

$$Nu_u = 0.012 (Re^{0.87} - 280) Pr^{0.4} \dots (9)$$

$$f = \frac{0.316}{Re^{0.25}} \dots (10)$$

The friction factor can be defined on the principle of an equivalent shear force in the flow direction per heat transfer unit (or friction area), whether this equivalent shear force is a primarily a pressure force. For the surface considered, the pressure drop across the tube is due to the viscous shear, turbulence and eddies induced by dimples. The friction factor (f) can be calculated by measuring the pressure downstream and upstream of the tested tube and according to the equation:

$$f = \frac{2 \Delta P D}{L \rho U^2} \dots \dots \dots (11)$$

And,  $\Delta p$  represents the pressure drop across the test section and is evaluated from the following equation:

$$\Delta P = \rho g h \dots \dots \dots (12)$$

The data obtained from the current configuration tubes were compared with plain tube and the results is presented in the result section.

**3.2- Overall performance enhancement ratio:**

Overall enhancement ratio is defined as the heat transfer enhancement ratio to the friction factor ratio based on the equal pumping power. This parameter is used to differentiate the negative technique and a comparison of various configurations for the technique itself. The overall enhancement ratio is defined as: [10]

$$\eta = \left(\frac{Nue}{Nup}\right) / \left(\frac{fe}{fp}\right)^{0.3} \geq 1 \dots (13)$$

This relation was based on the pressure drop and heat transfer of plain tube, and according to above relation, the overall enhancement ratio should be greater than unit. The greater value indicates better performance for that geometry, while value less than one indicate bad or worthless approach.

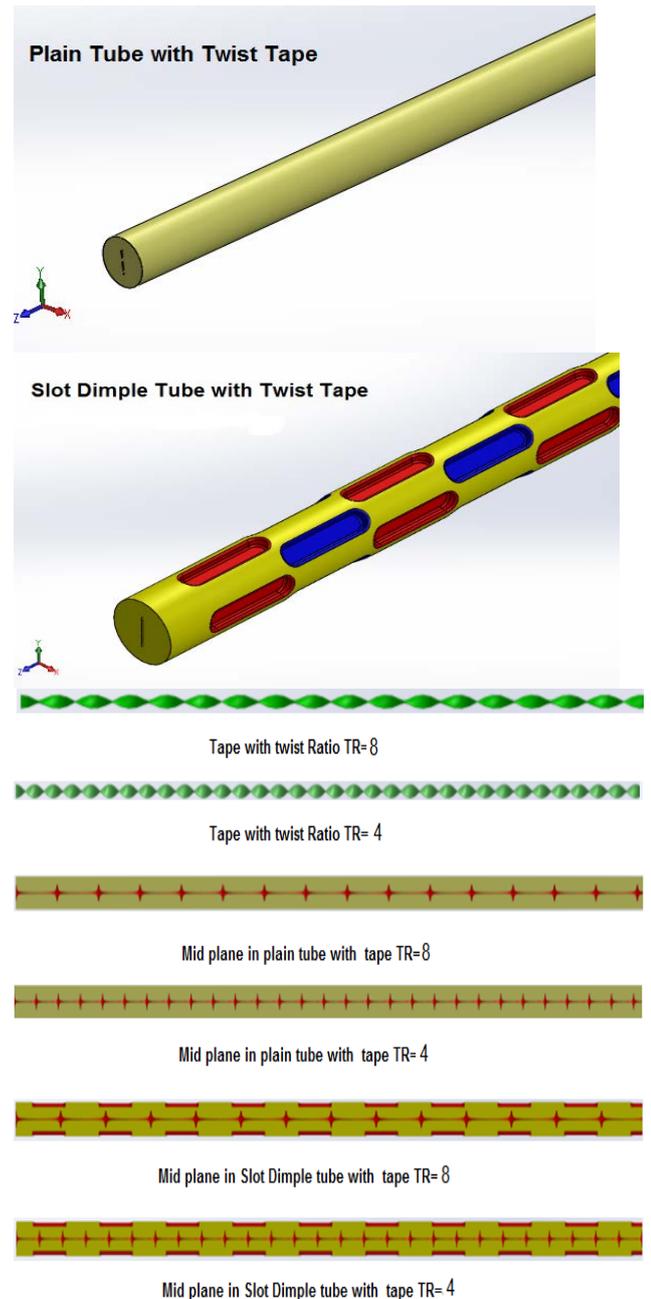
**4- Numerical Solution:**

The numerical simulations or computational fluid dynamic CFD allow analysis of a complicated phenomenon without resort to an costly prototype for flow visualization with sophisticated experimental measurements in this section, geometry, mesh, setup (assumption, governing equations, turbulence model and boundary condition), solution and post processor results are discussed.

**A- System Geometry:** There are four configurations of geometry the first two are plain tube in dimensions (35mm diameter and 1600mm length) with insert a twist tape with two twisted ratio TR (4, 8) and the second two are tube contained a slot dimple with dimensions (40 mm center to center distance, 5mm slot end radius and 5 mm depth) with insert a twist tape with two twisted ratio (4, 8) too. The system geometry is drawn by using (Solid Works) as shown in fig. 2.

**B- Mesh:** Unstructured grids are in general successful for complicated geometries, so for this reason the unstructured tetrahedron grids was used in the current study.

In the present work, a higher order element type, as shown in Figure 3 is used for mesh generation to approximate precisely the boundaries of high curvature. The final point in a good mesh is the total number of cells generated. It is vital to have enough number of cells for a good resolution but memory requirements increase as the number of cells increases. For the present cases, an average of (1.2) million cells is used.



**Figure 2:** The photos of the test tube

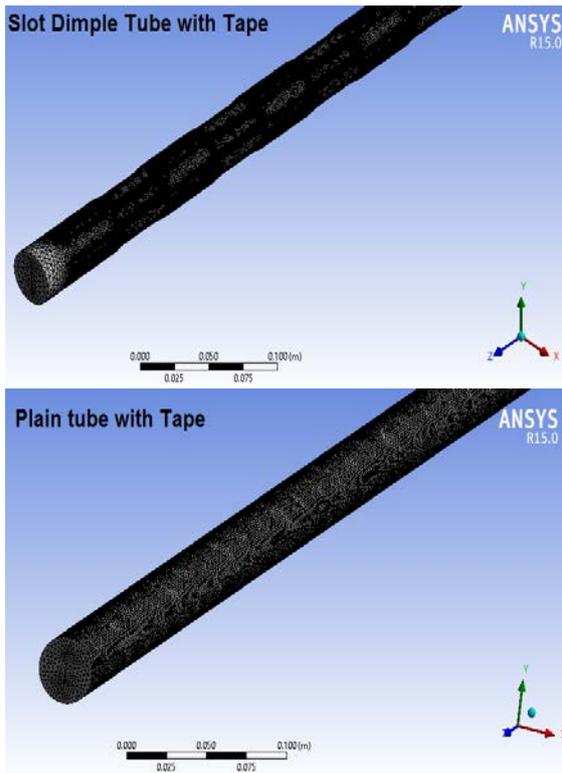


Figure 3: Test section mesh

**C- Assumption and governing equations:**

- **Assumption:** In the present case simulation, the flow characteristics are assumed to be as (Three dimensional, steady flow, Incompressible fluid, Newtonian fluid and Turbulent flow) and air is taken as the working fluid.
- **Governing Equations:** [11] The equations used to solve are the continuity, momentum and energy equations. The mass of a fluid is conserved. The change rate in the momentum is equal to the sum of forces on a fluid particle. The change rate of energy is equal to the sum of the rate of the work done and the rate of heat added to on a fluid particle.

**A. Continuity equation:**

$$\frac{\partial}{\partial x} \rho u + \frac{\partial}{\partial y} \rho v + \frac{\partial}{\partial z} \rho w = 0 \quad \dots\dots(14)$$

**B. Conservation of momentum:**

$$\frac{\partial}{\partial x} \rho u^2 + \frac{\partial}{\partial y} \rho uv + \frac{\partial}{\partial z} \rho uw = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \mu_e \frac{\partial u}{\partial x} + \frac{\partial}{\partial y} \mu_e \frac{\partial u}{\partial y} + \frac{\partial}{\partial z} \mu_e \frac{\partial u}{\partial z} + S_x \dots (15)$$

$$\frac{\partial}{\partial x} \rho vu + \frac{\partial}{\partial y} \rho v^2 + \frac{\partial}{\partial z} \rho vw = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \mu_e \frac{\partial v}{\partial x} + \frac{\partial}{\partial y} \mu_e \frac{\partial v}{\partial y} + \frac{\partial}{\partial z} \mu_e \frac{\partial v}{\partial z} + S_y \dots (16)$$

$$\frac{\partial}{\partial x} \rho wu + \frac{\partial}{\partial y} \rho wv + \frac{\partial}{\partial z} \rho w^2 = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \mu_e \frac{\partial w}{\partial x} + \frac{\partial}{\partial y} \mu_e \frac{\partial w}{\partial y} + \frac{\partial}{\partial z} \mu_e \frac{\partial w}{\partial z} + S_z \dots (17)$$

**C. Conservation of energy:**

$$\frac{\partial}{\partial x} \rho u T + \frac{\partial}{\partial y} \rho v T + \frac{\partial}{\partial z} \rho w T = \frac{\partial}{\partial x} \left( \rho \Gamma_e \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \rho \Gamma_e \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \rho \Gamma_e \frac{\partial T}{\partial z} \right) \dots\dots(18)$$

Where,  $\Gamma_e = \Gamma_1 + \Gamma_t$   
and

$$\Gamma_1 = \frac{\mu_1}{Pr_1}; \Gamma_t = \frac{\mu_t}{Pr_t}$$

- **Turbulence Model:** The selected of turbulent model will be based on consideration such as the physics included in the flow, the scale of accuracy required, the ready computational resources, and the time available for the simulation. The standard( $k - \epsilon$ ) model is economical with sensible accuracy for a wide range of turbulent flows and it is widely used in heat transfer simulation.
- **Boundary Conditions:** The boundaries applied are similar to that in the experimental condition and are prescribed at all boundary surfaces of the computation domain. At solid walls, constant wall temperature boundary conditions are used with range of experimental measured temperature, and no-slip boundary condition is set for the geometry, inlet velocity condition for the mainstream air inlet which was set to have a uniform velocity profile, Four inlet velocity and temperature are used in the same experimental measured, and at outlet set the boundary as pressure outlet with 1 atm.

**D- Solution**

The segregated solver, ANSYS is the solution algorithm used by FLUENT and is adopted in the present work. Using segregated solver way, the governing equations are solved sequentially (i.e., segregated from one another). Since the governing equations are non-linear, many iterations must be done before a converged solution is obtained. The SIMPLE algorithm is used in the present work.

- **Initial Condition:** The flow field is known after iteration is done. The initialization of the model is important for convergence. If the initial conditions are poor, then it takes longer time to converge or it may even solution in divergence. For the present work, all variables are initiated from the inlet boundary conditions.
- **Number of Iterations:** This is the number of iterations done before the solution is converge. In the present case, number of iteration was 2500.
- **Convergence criteria:** The iterations are stopped when the solution remains the same

within the accuracy of the selected convergence criteria. The most widely used method to check the solution convergence is the error residuals, which is the difference between the values of a variable in two consecutive iterations normalized by the largest absolute residual for the first five iterations. The solution is said to be converged when the residuals are below a set tolerance limit.

- This limit was set to be of  $10^{-4}$  for all variables expect the energy limit was set to be  $10^{-6}$ .

**5. Results and discussion:**

In this section, the experimental and numerical results of cases are discussed. Pressure drop and heat transfer for test tube were evaluated and presented as dimensionless value by friction factor and Nusselt number for the range of Reynolds number from 4000 to 15000. Overall enhancement ratios of dimpled tube are discussed. Thermal and hydrodynamic results of CFD study are presented in form velocity vector, contour, and Nusselt number.

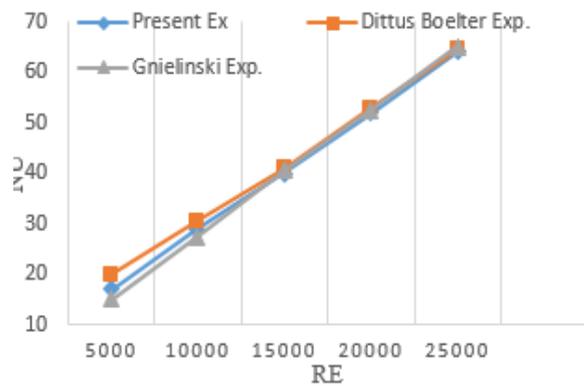
In order to verify the data obtained from the present test facility for heat transfer and pressure drop, experimental tests were performed on a standard plain tube (smooth) of 1.6 m length. The experimental data were then compared with the results obtained by the well-known correlations under a similar condition to evaluate the validity of the plain tube. Comparisons of Nusselt number and friction factor are shown in Figure 4 and 5, respectively. Obviously, the experimental results of heat transfer are in good agreement with empirical correlations developed by Dittus–Boelter and Gnielinski. It is noted that the average deviation in Nusselt number was approximately of  $\pm 2\%$ , and the maximum deviation of 10% with Dittus correlation for  $Re < 10000$ , whereas this correlation was developed for  $Re > 10000$ .

The friction factor of present plain tube and the results obtained by Blasins correlation are shown in Figure 5. The deviation was approximately of +30% for the range of tested Re. This was attributed to the surface pipe roughness, since Blasins equation was developed for smooth pipe while in present work, the tube surface has certain wall roughness, and it was not considered in Blasins equation. In the present work, the experimental data obtained from the plain tube will be used as a reference for comparison with other configuration, since the tested tubes are made from the same materials and manufacturing process.

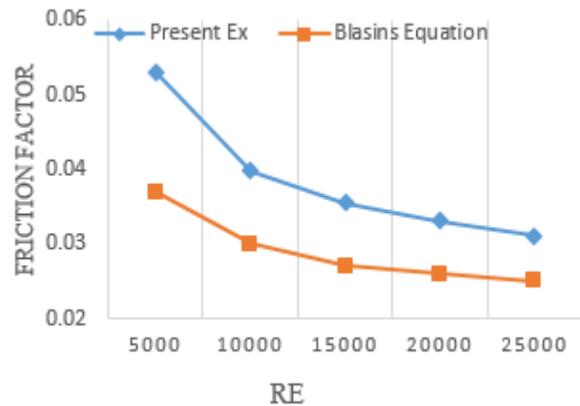
**A. The local heat transfer coefficient on slot dimples and plain surface:**

It is clear that high heat transfer can be observed on the dimples surfaces and at the surface downstream of the dimpled body due to

the flow impingent and reattachments at these regions where high turbulence are existing. The heat transfer in dimpled tube can be classified into two surfaces which are the dimples and the plain surface, and can be note that the maximum heat transfer occur at dimple area which represent stagnation zone as shown in figure 6 and 7.



**Figure 4:** Comparison of heat transfer of the present experimental test of plain tube with published empirical equation



**Figure 5:** Comparison of friction factor of present test with published equation of Blasins equation for smooth tube

The experimental Nusselt number for the present dimple tube are shown in figure 8. It can note that the Nusselt number increase with increase Reynold number which is natural behavior due to increase the turbulent flow which leads to destroy the flow pattern and the flow become more irregular and that allow to cool air in the core of flow to reach into the wall and this temperature difference allow to heat transfer between the flow and wall. While the friction factor decrease with increase in the Reynold number for dimple tube and increase with increase in Reynold Number for plain tube as shown in figure 9.

In general, for a laminar flow, the pressure various across the protrusion is small in the absence of separation. Across the dimple dome, the pressure various between the upward and downward side of the dimple is also small. Hence, at low Reynolds numbers, the shear or

surface friction dominates the total losses. For turbulent or high Reynolds number, flow separation and wake formation on the protrusion with separation and reattachment within the dimple. This is attributing to the high pressure associated with slot dimpled tube compared with the plain tube.

The mean friction factor for the slot dimple tube with (TR =4 and 8) are (6.89 and 6.82) times higher than that for the plain tube and for plain tube with twist tape (TR =4 and 8) are (22.46 and 18.51) times higher than that for the plain tube as shown in figure 9.

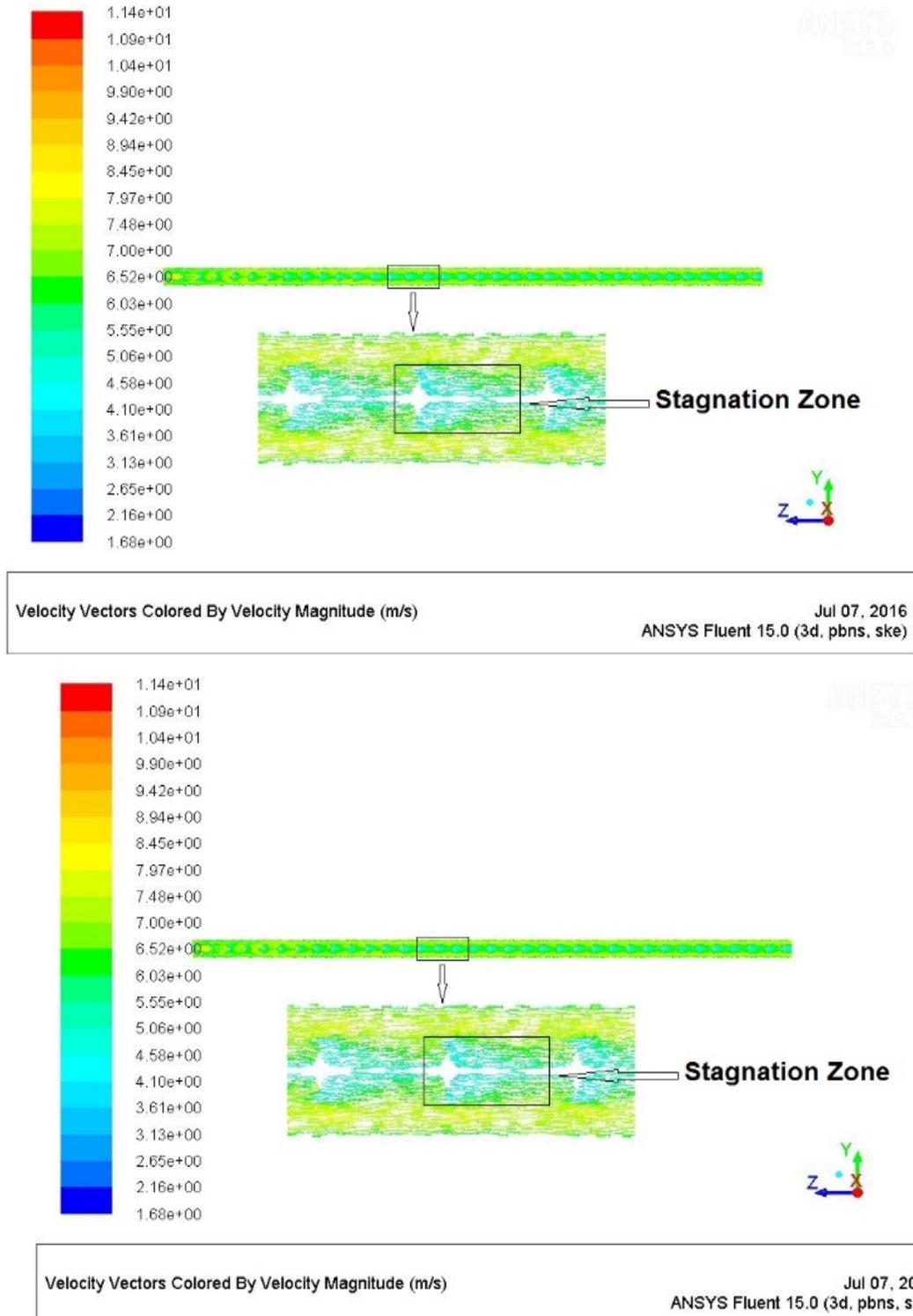


Figure 6: Velocity vector in tube with internal twisted tape for a) plain tube, b) Slot Dimple tube.

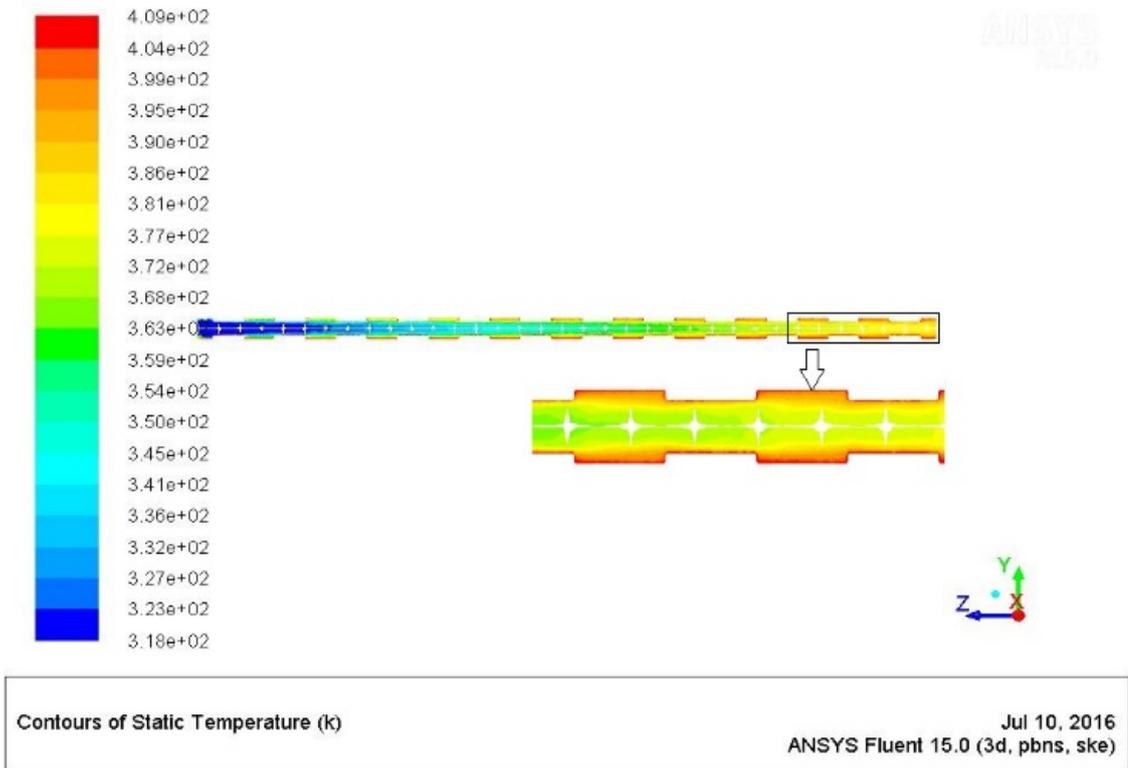
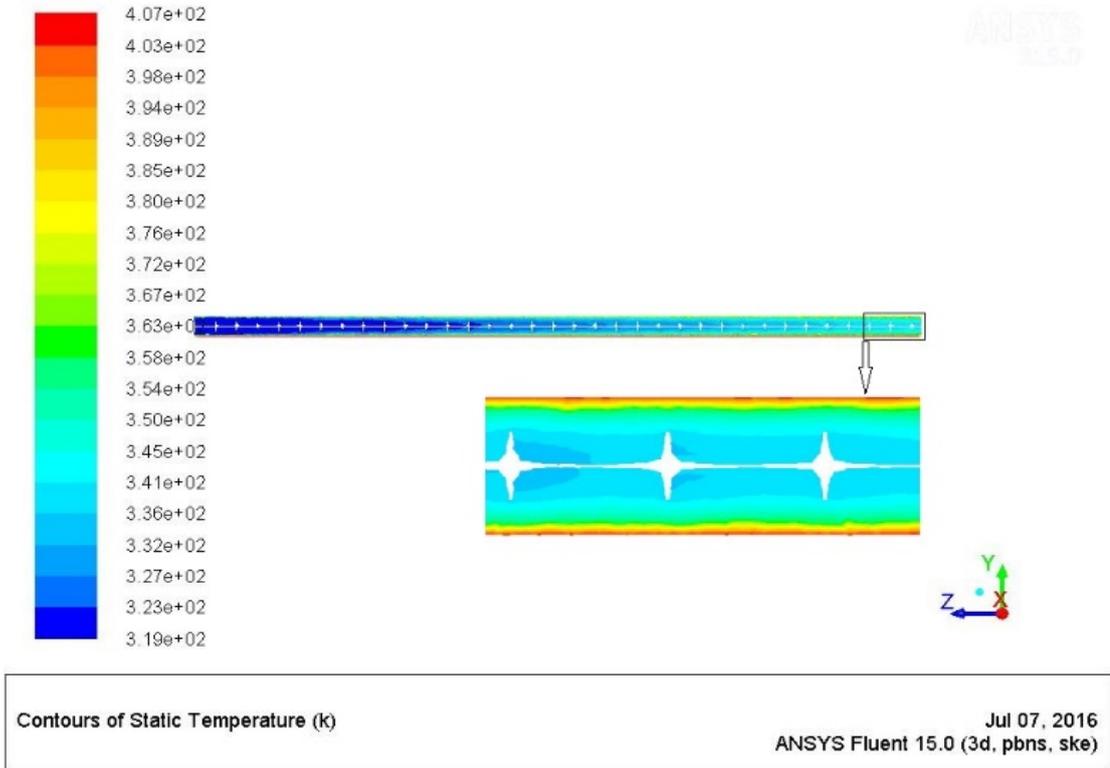
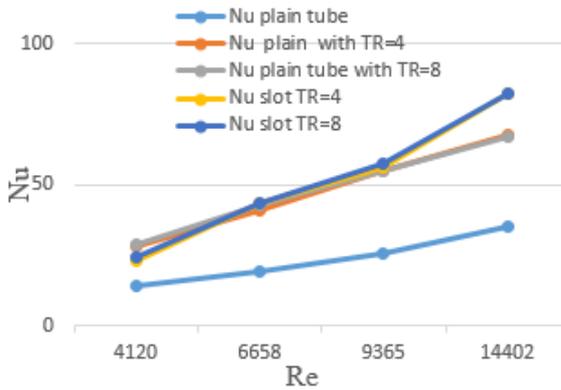
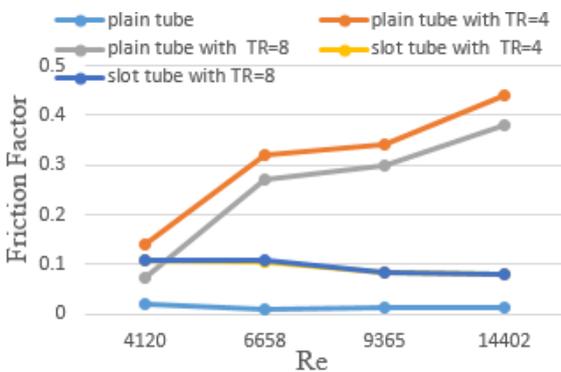


Figure 7: contour of temperature in tube with internal twisted tape for a) plain tube, b) Slot Dimple tube.



**Figure 8:** Comparison between Nusselt number for all present cases and plain tube for the present experimental results.



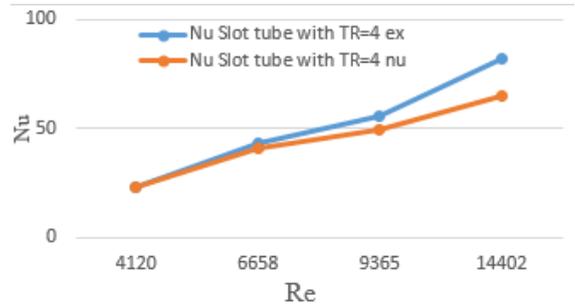
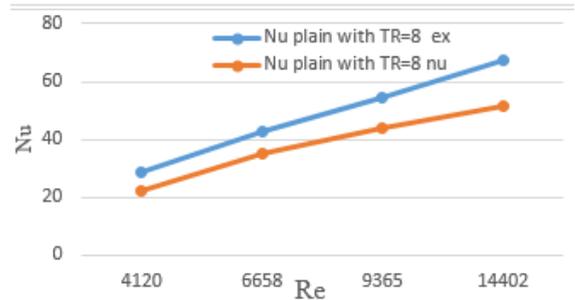
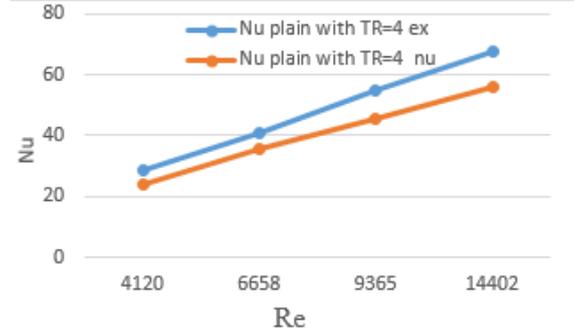
**Figure 9:** Comparison experimental friction factor between present cases and plain tube

**A- Comparison of enhancement technique:**

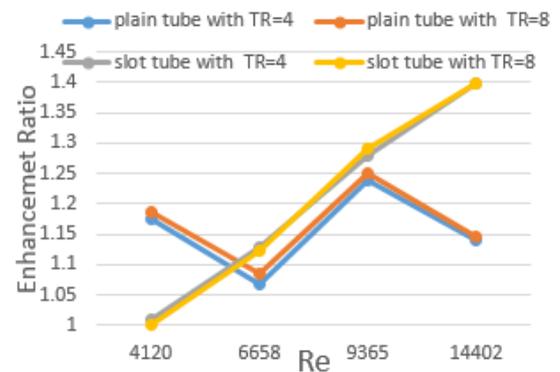
**1- Heat transfer and pressure drop:**

Figure 8 illustrates the Nusselt number of plain tube in comparison with the present cases, the result shows that slot dimples tube with (TR =4 and 8) can enhance the average heat transfer by 2.05- 2.07 times the plain tube, and for plain tube with (TR =4 and 8) can enhance the average heat transfer by 2.18- 2.20 times the plain tube. In case of twisted tape with present tube geometry, more turbulence is created during the swirl of fluid and gives higher heat transfer rate compared to plain tube. The numerical results were in good agreement with the present experimental results.

Figure 10 reveals the comparison between the numerical and experimental results of Nusselt number for present cases. The deviation is within 15- 19% higher for numerical as average value. Figure 7 shows the variation of heat transfer along the tube, where high heat transfer can be noticed at the inlet section where a laminar boundary layer exists.



**Figure 10** comparison between numerical and experimental Nusselt number for present Plain and slot dimple tube for TR=4 and TR=8



**Figure 11:** the overall enhancement ratio for present cases dependent on plain tube

**2-overall enhancement ratio:** Figure 11 shows the overall enhancement ratio for present cases dependent on plain tube. The result depict that the slot dimple tube with twist tape and plain tube with twist tape give high enhancement in heat transfer relative to plain tube due to increase in heat transfer area. The average enhancement ratio for slot dimple tube with (TR =4 and 8)are (1.204 and 1.202) respectively .This indicates that 15.5-20.4% of heat transfer area can be saved at the same pumping power for present cases configuration compared with the plain tube heat exchanger.

**6. Conclusion:**

The present work reaches to the following conclusion:

- 1) Nusselt number increase with increase Reynold number due to increase the turbulent flow which leads to destroyed the flow pattern and the flow become more irregular
- 2) Friction factor decrease with increase in the Reynold number because the dimple create pressure difference between the downstream and upstream of dimple.
- 3) The experimental results were in good agreement with the present numerical results.
- 4) In case of twisted tape with present tube geometry, more turbulence is created during the swirl of fluid and gives higher heat transfer rate compared to plain tube.
- 5) Use slot dimple tube with twist tape and plain tube with twist tape give high enhancement in heat transfer relative to plain tube due to increase in heat transfer area.
- 6) Heat transfer area can be saved at the same pumping power for present cases configuration compared with the plain tube heat exchanger.

**7. Nomenclatures:**

A	Orifice cross-sectional area	(m <sup>2</sup> )
As	Tube surface area	(m <sup>2</sup> )
C	Discharge coefficient	
Cp	Specific heat at constant pressure	(J/kg.K)
D	Tube diameter	(m)
L	Test section length	(m)
f	friction factor	
h	Average heat transfer coefficient	(W/m <sup>2</sup> .K)
m	Mass flow rate	(kg/s)
Nu	Average Nusselt number	
P	Pressure	(pa)
Pr	Prandtl number	
Q	Volumetric discharge	(m <sup>3</sup> /s)
q	Heat energy	(W)
T	Temperature	(C, K)

t	time	sec
u	Mean velocity	(m/s)
ρ	Fluid Density	kg/m <sup>3</sup>

**8. References:**

- 1- Bergles, A. E, and Troy, NY, “High-Flux Processes Through Enhanced Heat Transfer”, Keynote at the 5th International Conference on Boiling Heat Transfer, Montego Bay, Jamaica, May 4-8, 2003.
- 2- Ohadi, M. M., Dessiatoun, S. V., Darabi, J.,and Salehi, M.," Active Augmentation of Single-Phase and Phase-Change Heat Transfer – an overview, in Process, Enhanced, and Multiphase Heat Transfer." Manglik, R. M. and Kraus, A. D., eds., Begell House, New York, 277-286,1996.
- 3- Watcharin Noothong, Smith Eiamsa-ard, and Pongjet Promvonge," Effect of Twisted-tape Inserts on Heat Transfer in a Tube", The 2<sup>nd</sup> Joint International Conference on “Sustainable Energy and Environment (SEE 2006)”, 21-23 November 2006, Bangkok, Thailand.
- 4- Eiamsa-ard S.et.al, Thianpong C., Eiamsa-ard P. and Promvonge P., "Convective heat transfer in a circular tube with short-length twisted tape insert",International Communications in Heat and Mass Transfer, 36, (2009), 365–371.
- 5- Chinaruk Thianpong,et.al, and Smith Eiamsa-ard,]" Compound heat transfer enhancement of a dimpled tube with a twisted tape swirl generator” International Communications in Heat and Mass Transfer 36 ,PP 698–704,2009.
- 6- British standard 1042, Measurement of flow through orifice plat.
- 7- R. L. Webb,and N. Hyun Kim, “Principles of Enhanced Heat transfer”, Second edition, Taylor Francis Group, 2005.
- 8- M .R. Jafari Nasr, G. T. Polley, and A. T. Zoghi, "Performance Evaluation of Heat Transfer Enhancement", CHISA, Prague, 2002.
- 9- M .R. Jafari Nasr, G. T. Polley, and A. T. Zoghi, "Performance Evaluation of Heat Transfer Enhancement", CHISA, Prague, 2002.
- 10- S. Naga Sarada.et.al,]” Enhancement of heat transfer using varying width twisted tape inserts” International Journal of Engineering, Science and Technology Vol. 2, No. 6, pp. 107-118, 2010
- 11- Ashish Kulkarni and Shane Moeykens, “Introduction to FLUENT Note”, Sixth edition, ANSYS Inc.,mber 2006..

## دراسة عملية ونظرية لانتقال الحرارة في الانبوب ذي النذب الشقي مع شريط ملتوي داخلي

همام كريم جلفف  
قسم الهندسة الميكانيكية  
الجامعة التكنولوجية

### الخلاصة:

تم في هذا البحث التحقق نظريا وعمليا لتحسين انتقال الحرارة بالحمل للجريان المضطرب داخل انبوب ذي نذب الشقي والانبوب العادي مع وجود شريط محلزن داخل هذه الانابيب. تم بناء الجهاز لاجراء التجارب العملية لإيجاد مقدار التحسن في انتقال الحرارة والانخفاض في الضغط بين مقدمة ومؤخرة هذه النذب الشقية. المانع المستخدم لاجراء التجارب العملية هو الهواء, بينما يستخدم بخار الماء كمصدر الحرارة حيث تصل درجة الحرارة عند الجدار (135 درجة سليزيه). التجارب العملية رتب بحيث ممكن الحصول على جريان مضطرب برقم رينولد يتراوح من 4000 الى 15000. نمط الجريان الهيدروليكي والحراري لهذة التصاميم تم دراستها نظريا باستخدام برنامج الفلونت 15. نتائج معدل انتقال الحرارة العملية كانت بتوافق جيد مع النتائج النظرية. النتائج بينت ان الانبوب ذي النذب الشقي مع شريط ملتوي داخلي والانبوب الاعتيادي مع شريط ملتوي داخلي اعطى تحسن عالي بانتقال الحرارة نسبة الى الانبوب الاعتيادي من دون شريط داخلي وذلك نتيجة لزيادة في مساحة انتقال الحرارة. معدل نسبة التحسين للانبوب ذي النذب الشقي مع شريط داخلي ( ذي نسبة التواء 4 و 8 ) كان (1.204 و 1.202) على التوالي. هذا النتائج تبين بان 15.5-20.4% من مساحة انتقال الحرارة ممكن الاحتفاظ بها لنفس قدرة الضخ لتصاميم الحالية مقارنة بالمبادلات الحرارية ذات الانبوب الاعتيادي.