International Journal of Applied Engineering Research ISSN 0973-4562 Volume 11, Number 1 (2016) pp 427-434

Numerical and Experimental Comparision of Heat Transfer Enhancement in A 2-Pass Double Pipe Heat Exchanger with and without Inserted Twisted Tapes

Dr.Bhramara Panitapu
Professor, Department of Mechanical Engineering
JNTU College of Engineering, Kukatpally, Ranga Reddy District, Telangana, India.
E-mail: bhramara74@yahoo.co.in

T. Kanthimathi
Assistant Professor, Department of Mechanical Engineering
K L University, Vaddeswaram, Guntur District, Andhra Pradesh, India.
E-mail: pkanthi1978@gmail.com

Krishna Chaitanya
P.G. Student, Department of Mechanical Engineering
JNTU College of engineering, Kukatpally, Ranga Reddy District,
Telangana, India.

Abstract
Enhancement of heat transfer using twisted tapes inserted in a 2-pass double pipe heat exchanger is studied experimentally and numerically using Ansys Fluent. A numerical model is developed to investigate the effects of cold water flow rates on the Nusselt number, overall heat transfer coefficient in both inner pipe and annulus. Results show that the numerical and experimental results are in good agreement with each other. The twist ratio for numerical heat transfer enhancement is taken as y=10.

Keywords: Heat transfer enhancement, Twisted tapes, Ansys Fluent, Nusselt number, Numerical investigation, twist ratio, double pipe heat exchanger.

Introduction
Forced convection heat transfer in a circular tube has been the subject of interest among researchers during the past few decades. Significant effort has been made to develop heat transfer enhancement techniques in order to improve overall performance of heat exchangers. The interest in these techniques is linked with the energy prices. With the present increase in the energy cost it is expected that the heat transfer enhancement field will go through a new growth phase. In order to achieve energy efficient heat exchangers there are two methods passive method and active method. Active method involves some external power input for the enhancement of heat transfer. Some examples of active method include pulsation by cams and reciprocating plungers. Passive methods generally use surface or geometrical modifications to the flow channel by incorporating inserts or additional devices. These inserts include louvered strips and twisted tapes, turbulent or swirl flow devices, coil wire and helical coil in a circular tube. Recently the concept of heat transfer enhancement using twisted tapes is developed and investigated numerically. Variously developed twisted tapes inserts are frequently used to strengthen the heat transfer efficiency for heat exchangers [1].

Wire coil inserts are currently used in applications as oil cooling devices, preheaters and fire boilers. Advantages of wire coil inserts are low cost, easy installation and removal, no need to change dimension of the tube etc Full length twisted tapes have length equal to the length of the test section. Varying length twisted tape do not have length equal to the length of the test section, but equal to ½ , ¾ and ¼ th length of the test section etc. Regularly spaced twisted tapes are short length tapes of different pitches spaced by connecting together. Some of the twisted tapes have have baffles attached at some intervals so as to achieve more augmentation. Slotted tapes with holes have slots and holes of suitable dimensions made in twisted tapes so as to create more turbulence. Tapes with different surface modifications are provided with insulating material so that fin effect can be avoided.

Related Work
Experimental investigation is carried out by Bodius Salam [2] for measuring tube side heat transfer coefficient of water for turbulent flow in a circular tube fitted with twisted tape insert of 5.3 twist ratio and they found that Nusselt number increase with increase in Reynolds number. Dr. Prasant Baredar., [3] investigated different heat transfer augmentation techniques and it is found that compared to conventional heat exchanger the augmented heat exchanger has shown a significant improvement in heat transfer coefficient by 61 % for twisted tape I and 78% for twisted tape II. Twisted tape of lower twist ratio (p/d =3.5) gives higher heat transfer coefficient (by 1.39 times) than higher twist ratio of p/d = 7.

Royds [4] carried out experimental work in a circular tube with full length tapes and with higher twist ratio. According to his observations higher twist ratio resulted in higher heat transfer and pressure drop. Smithberg & Landis [5] had done numerical analysis with full length tapes and prposed mathematical model with swirl mechanism.

Experimental work done by Cressell [6] with twisted tapes focussed on the friction factor. In his observations it is concluded that for all configurations of twisted tapes friction factor increases because the tapes will object the fluid flow in the pipe.

Effect of coil pitch and other corresponding parameters on heat transfer enhancement and friction factor of the horizontal concentric tubes with coiled wire inserts are proposed by Naphon [7] and it was found that coiled wire inserts are specially effective on laminar flow region in terms of heat transfer enhancement. A Garcia et al. [8] study effect of coil wire insert inside a horizontal tube in a laminar, transition and turbulent region with different pitch and wire diameter. At low Reynolds number coil wire found to be behaves as a smooth tube but accelerates critical Reynolds number down to 700. In turbulence region coil wire increase heat transfer rate up to four times compared to smooth tube.

Y Shoji [9] study effect of length and segmentation of coil wire insert on heat transfer enhancement. It was found that increase with increase of coil wire length.

In the present study a numerical and experimental investigation of heat exchanger with and without twisted tapes is carried out. Water is considered as working fluid for heat exchanger. Cold water is allowed to flow through the inner pipe whereas hot water is flowing through the annulus. Numerical analysis is carried out using ansys to study the enhancement of heat transfer for different flow rates of cold water. The twisted tape ratio considered is y=10.

Description of Numerical analysis

Governing equations

a) Continuity equation: It is derived by applying conservation of mass to a small differential volume of fluid.

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho v) = S_m$$

The above equation is valid for incompressible as well as compressible flows. Source S_m is the mass added to the dispersed second phase.

b) Momentum equation is derived using Newtons second law of motionto a differential volume of fluid. Resulting momentum equation in Cartesian coordinates is

$$\frac{\partial}{\partial t}(\rho\vec{v})+\nabla(\rho\vec{u}\vec{v}) = -\nabla P+\nabla\bar{\bar{\tau}}+\rho\vec{g}+\vec{F}$$

Where 'p' is the static pressure $\bar{\bar{\tau}}$ is the stress tensor, $\rho\vec{g}$ and $\vec{F}$ are gravitational force and external force on the body respectively.

$$\bar{\bar{\tau}} = \mu\,[(\nabla\vec{v}+\overrightarrow{\nabla v^{T}}) - \frac{2}{3}\nabla.\vec{v}I]$$

Where μ is molecular viscosity, I is the unit tensor and second term to the right side is the effect of volume dilation.

c) Energy equation is derived from the first law of thermodynamics. In general form it is given as

$$\frac{\partial}{\partial t}(\rho E) + \nabla.(\vec{v}(\rho E + P)) = \nabla.\,(k_{eff}\nabla T - \textstyle\sum_j h_j\vec{J_j} + (\bar{\bar{\tau}}_{eff}\,.\,\vec{v})) + S_h$$

Where effective thermal conductivity $k_{eff} = (k + ki)$, ki is the turbulent thermal conductivity. $\vec{J_j}$ is the diffusion flux of species j. First three terms of the equation on the right hand side represent energy transfer due to conduction, species dissipation and viscous dissipation. S_h includes heat of reaction and any other volumetric heat sources. Energy E per unit mass is defined as

$$E = h - \frac{p}{\rho} + \frac{u^2}{2}$$

Turbulence models

k- ϵ models are the simplest complete models for turbulent flows in which the solution of two separate transport equations allows the turbulent velocity and length scales to be determined independently. For the present study K-ϵ model and second order upwind discretization methods are selected.

Geometry modelling

The analysis performed on a double pipe heat exchanger with inner diameter of the inner pipe 0.019m and the outer diameter of inner pipe 0.025m. The annulus pipe inner diameter is 0.05m and the outer diameter is 0.056m and the total length of the heat exchanger is 2.36m (2-pass). Hot water is allowed to flow through the annulus whose mass flow rateis constant. Cold water is allowed to flow through the inner pipe whose mass flow rate is varied.
The outer wall of the annulus pipe is maintained in isolated condition. Geometry model is shown in fig.1

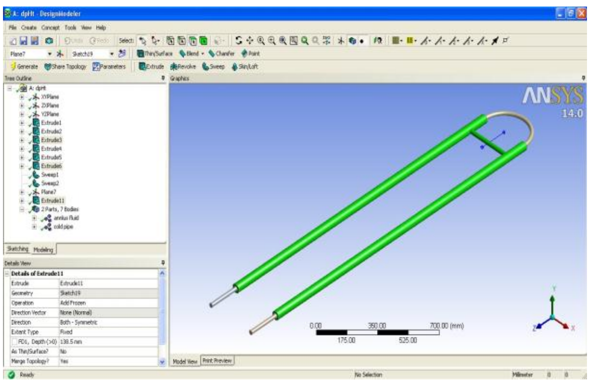


Figure 1: Geometrical model of double pipe heat exchanger

Twisted tape is inserted in cold pipe with 10 number of twists, pitch 220mm, diameter 17mm and thickness 1mm as shown in fig.2

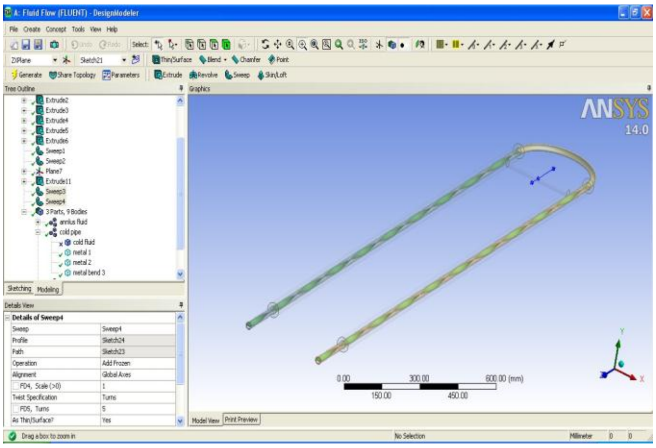


Figure 2: Twisted tape inserted in the heat exchanger

Meshing

Structured meshing method in Ansys work bench was used for meshing the geometry as shown in the fig.3

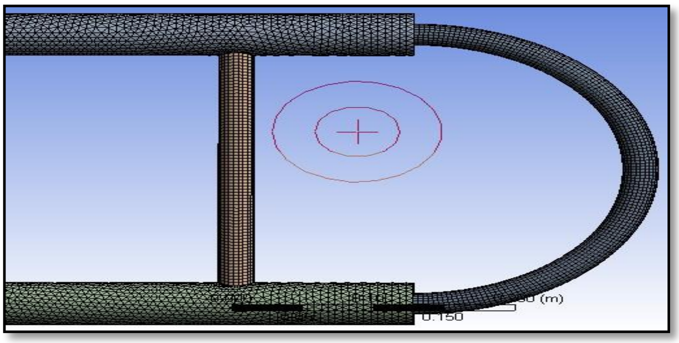
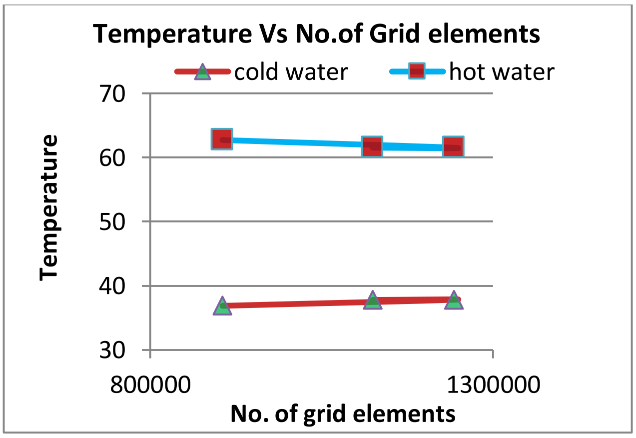


Figure 3: Structured meshing of the heat exchanger

The element considered is hexahedral shape with no. of elements 1124397 to 1420000. Grid independence test was conducted at 10 Lpm hot water and 8 Lpm cold water flow rates in CFD Fluent by increasing and decreasing the size of elements the results are as shown below for outlet temperatures of cold water and hot water in a 2-pass DPHE without inserts.



Figue 4: Variation of temperature with no. of grid elements

Physical models

The standard k- ϵ models are used to model single phase turbulent flow in a circular pipe channel. Based on the Reynolds number either viscous laminar model or standard k- ϵ model is used for laminar and turbulent flow respectively. The choice of the model is based on the Reynolds number. If Re< 2000, viscous laminar model is used, If Re> 2000 turbulent flow (k- ϵ) model is used

Properties

Properties of pure water which is used as working fluid at 25°C are as given below

Density ρ (Kg/m3)	997.5
Specific Heat Cp (j/kg-k)	4182
Thermal Conductivity K(w/m-k)	0.6
Viscosity (Kg/m-s)	0.001003

Pipes of heat exchangers are considered to be made up of steel. The steel properties are given below

Density ρ (Kg/m3)	8030
Specific Heat Cp (j/kg-k)	502.48
Thermal Conductivity K(w/m-k)	16.27

Boundary conditions

No slip boundary condition was assigned to non porous wall surfaces. A uniform mass flow inlet and a constant inlet temperature were assigned at the channel inlet.

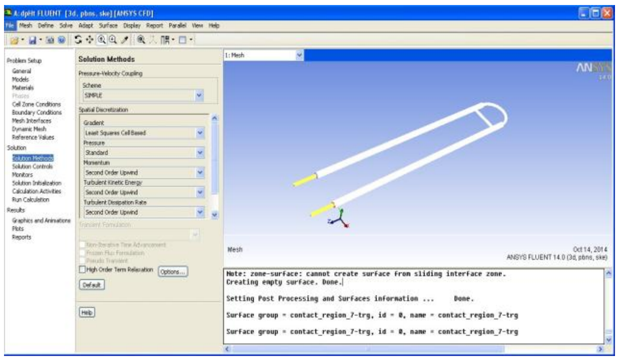


Figure 5: Specification of solution methods

S.no	Boundary type	Annulus Pipe inlet	Pipe inlet
1	Mass flow rate	0.1 *kg/s*	0.1 to 0.266 *Kg/s*
2	Temperatures	342K	302K
3	Constant heat flux at pipe wall (insulation)	0 W/m^2	-

Solution methods

The CFD method follows the use of commercial software ANSYS FLUENT 14.0 to solve the problem. The specified solver in FLLUENT uses a pressure correction based iterative SIMPLE algorithm with 2nd order upwind scheme for discretizing the convective transport terms. The convergence criteria for all the dependent variables are specified as 0.0001. The default values of under-relaxation factors are used in the simulation work.

In the present study, the analytical values of heat transfer coefficients are calculated. The heat transfer coefficients are also obtained using CFD methods and Experimental results both the values are compared.

After determining the important features of the problem following procedure is followed for solving the problem.

1. Initially the geometry model is created in the Project schematic of the ANSYS work bench by using the commands like Extrude, sweep in plane, creating bodies as per the sketch of the experimental setup.
2. Meshing was done to the geometry model by program controlled and sizing was done to get the required element size, nodes and smoothing. After getting the required size of element and meshing, naming section was done to the dominie as we are going to see the results.
3. After meshing is completed setup section is opened in the project schematic in fluent, where governing equations are selected like ENERGY, VISCOUS-Standard K-epsilon (2equ), Standard wall Function is given to necessary equations to simulate, material creating and boundary conditions are assigned to calculate the moment, pressure etc., by using standard finite element method equations.
4. Second order upwind scheme is selected for solution.

Experimental analysis

CFD analysis results validation is done by experimentally on the experimental setup of 2 pass double pipe heat exchanger (DPHE). The experimental setup of the 2 pass double pipe heat exchanger is shown in Fig.,4. used for the experimental study calculations of heat transfer enhancement

Figure 6: Experimental Setup

The Heat Exchanger has Inner pipe inner diameter 19mm, Inner pipe outer diameter 25mm, Outer pipe inner diameter 50mm,Outer pipe outer diameter 56mm.Pipe material is made up of mild steel, and the inserts are made up of aluminium. Heat transfer length is 4.72m. The outer pipe (annulus) is well insulated using 15mm dia asbestos rope to reduce heat losses to the atmosphere. Two pressure tapings, one just before the test section and the other just after the test section are attached to the U-tube manometer for pressure drop measurement. Water at room temperature was allowed to flow through the inner pipe while hot water (set point 70°C) flowed through the annulus side in the counter current direction.
Hot water is allowed to flow through the annulus at a temperature of 70°C with constant mass flow rates of 6,10,14LPM. Cold water is allowed to flow through the inner pipe at a temperature of 26°C with mass flow rates ranging from 6 to 16 LPM.

The heat transfer rate is calculated using the analytical equations as given below

$$Re_h = \frac{4mh}{\pi\ D1\mu}$$

$$Nu_h = 0.023*Re_h^{0.8}*Pr^{0.333}$$

$$Nu_h = \frac{h_o*D_e}{K}$$

Heat transfer rate $Q_h = m_h * C_{ph} * (T_{hi} - T_{ho})$

Where, $Q_h = UA(LMTD)_{counter}$

$$A = \pi*Davg*L$$

$$Q_c = m_c*C_{pc}*(t_o-t_i)$$

$$Q_{avg} = \frac{Qc+Qh}{2}$$

$(LMTD)_{counter} = [\frac{(\Delta T_1 - \Delta T_2)}{Ln(\frac{\Delta T_1}{\Delta T_2})}]$; Where $\Delta T1 = (T1\text{-}t2)$ and $\Delta T2 = (T2\text{-}t1)$

$U = \frac{Qavg}{A*LMTD}$ where U is the overall heat transfer coefficient

$$\frac{1}{hi} = \frac{1}{U} - \frac{1}{hi}$$

Results & Discussion

Validation of Numerical Results:

Numerical results are made credible by comparing them with Experimental work. Nusselt number for the base fluid (water) in the turbulent regime is compared with that of Dittus-Boelter correlation. The experimental results and CFD results are plotted and compared with analytical results. It can be seen from Figure. 7 the Numerical Nusselt number is in very good agreement with the correlation values. Friction factor comparison is in the acceptable limit, though not as good as the heat transfer coefficient comparison. This shows that the numerical model is validated.

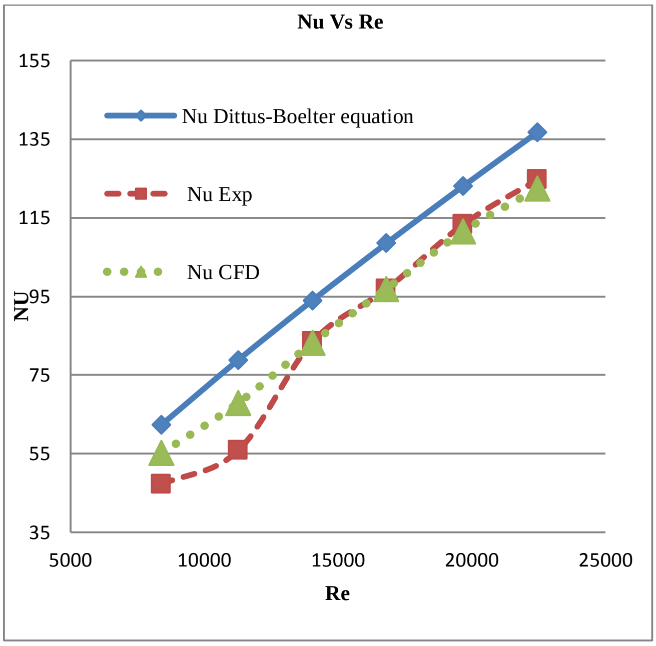


Figure 7: Reynolds number vs Nusselt number for Hot water 6LPM:

The results are compared with those of plain tube to estimate the enhancement of heat transfer rate in the absence of inserts both by CFD and experimental verification was done at different cold water flow rates from 6LPM to 16LPM by keeping hot water flow rate constant. The Nusselt number was increased by increasing Reynolds.

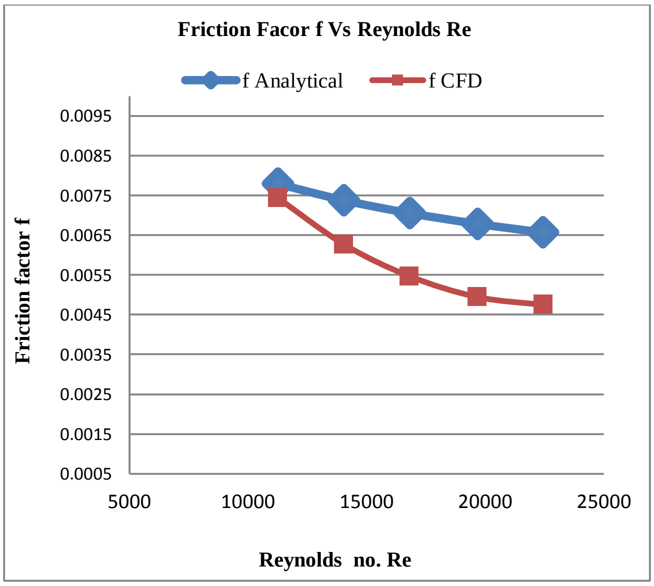


Figure 8: Reynolds Number Vs Friction Factor for Hot Water At 6LPM

The above figure 8 indicates that the friction factor is gradually decreasing by increasing the Reynolds number i.e for different (8LPM-16LPM) cold water flow rate. Graph is plotted for the friction factor f vs Reynolds number Re for the analytical and CFD f values at constant 6LPM Hot water flow rate.

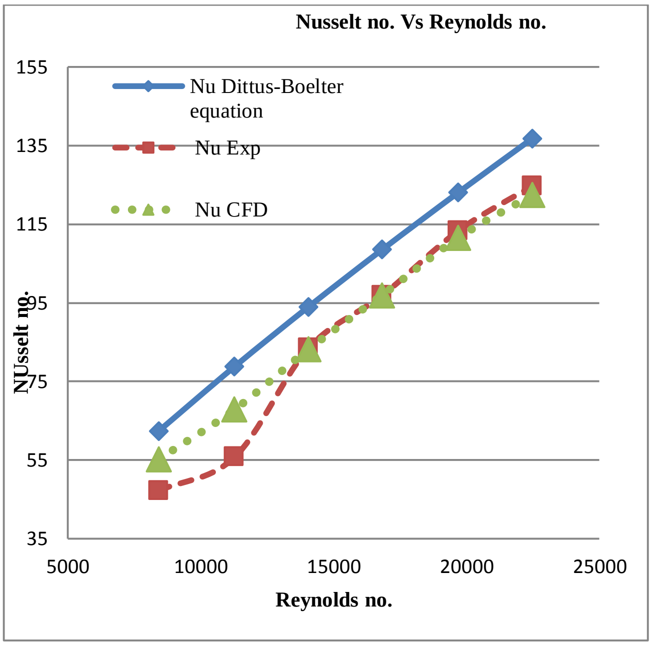


Figure 9: Nusselt number Vs Reynolds number

The above figure 9 shows the Nusselt number is gradually increasing with increase in Reynolds number Re from 8LPM to 16LPM of cold water flow rates for constant 10LPM hot water flow in annulus. The plotted results are compared with Analytical values, Experimental values and CFD results. The heat transfer effect is increased from 8LPM cold water flow rate

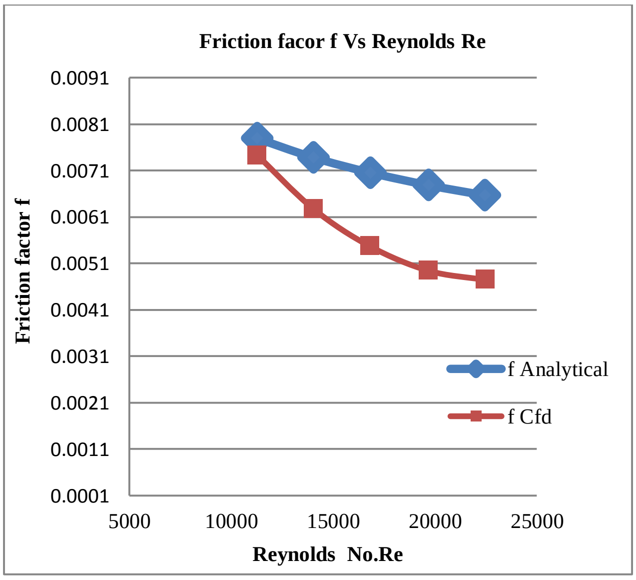


Figure 10: Reynolds Number Vs Friction Factor for Hot Water at 10LPM

The above figure 10 tells that the friction factor is gradually decreasing by increasing the Reynolds number i.e cold water flow rate. Graph is plotted for the friction factor f vs Reynolds number Re for the analytical and CFD friction factor f values at constant 10LPM Hot water flow rate

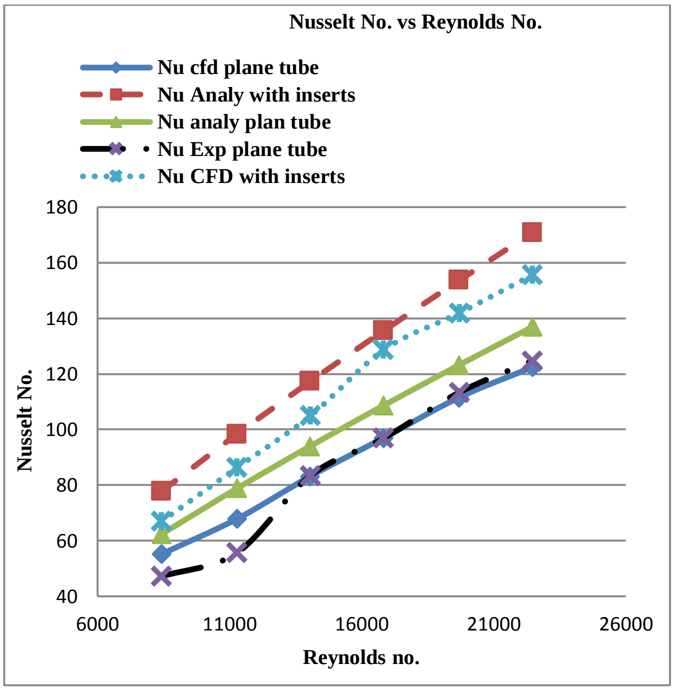


Figure 11: Reynolds Number vs Nusselt Number for Hot Water 6LPM using Inserts

The above figure 11 indicates that the Nusselt number calculated for twisted tape inserts is increasing when compared to plain tube. By using twisted tape inserts the maximum enhancement was 33% increase at Reynolds number of 17000 and 28% increase enhancement at Reynolds number 22000.

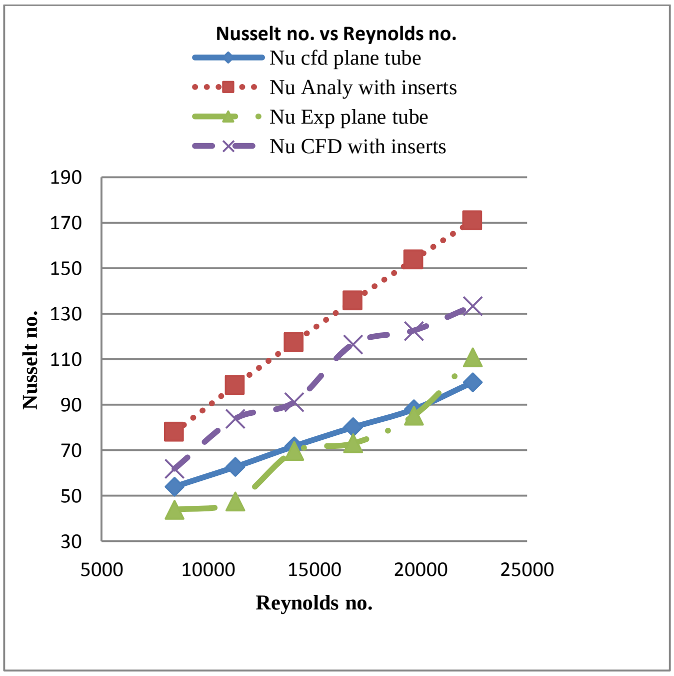


Figure 12: Reynolds Number vs Nusselt Number for Hot Water 10LPM using Inserts

The above figure 12 indicates that by using twisted tape inserts the maximum enhancement was 46% increase at Reynolds number of 17000 and 40% increase enhancement at Reynolds number 20000.

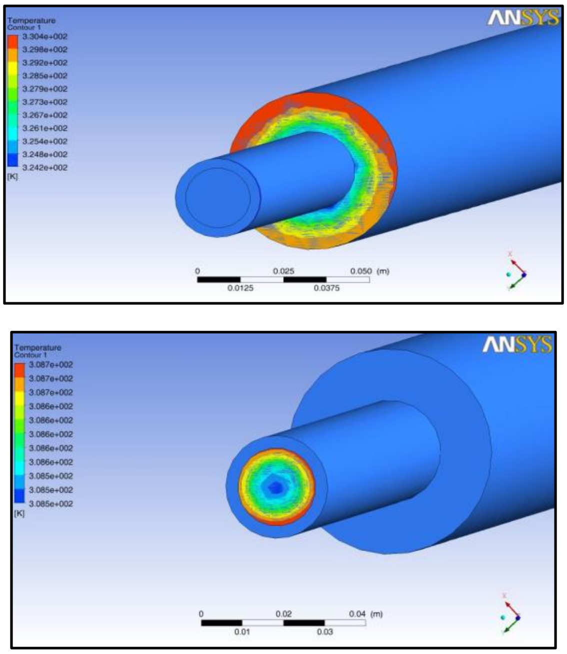


Figure 13: Temperature Counters at Outlets of the AnnulusPipe and Cold Pipe of DPHE

The above figure 13 shows that the temperature counters of the outlet cold water pipe side and outlet hot water pipe(annulus side) of the double pipe heat exchanger at 6LPM hot water flow in annulus side and 8LPM cold water flow rate in cold pipe side.

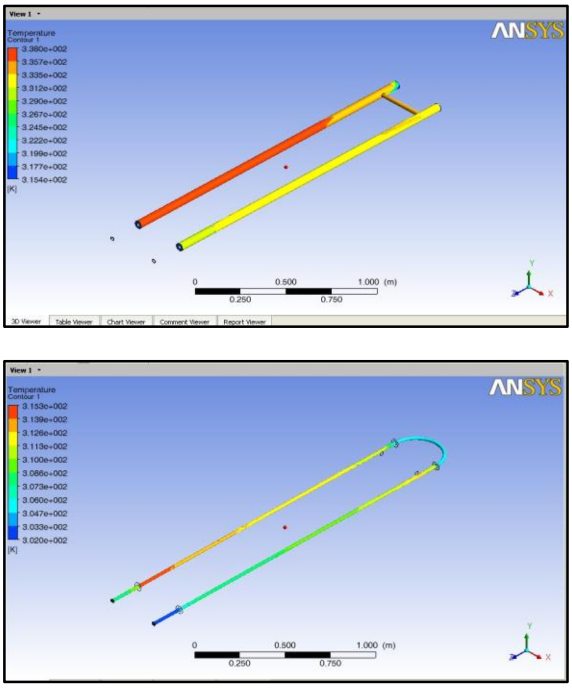


Figure 14: Temperature Distribution of Temperature Counters in Annulus Pipe and cold water pipe

The above figure 14 shows the Temperature counter of the annulus pipe side and inner pipe side temperature distribution along the length of the heat exchanger.Temperature variation at 6LPM hot water flow rate and 8LPM hot water flow rate.

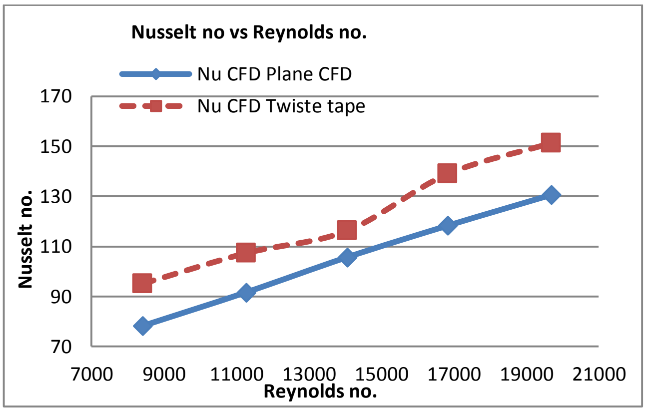


Figure 15: Nusselt no. vs. Reynolds no. for DPHE at 14LPM Hot Water, With and Without Inserts

The above graph shows the comparison between the heat transfer enhancement of plane tube vs twisted tape.The heat transfer enhancement is increased by 10% to 21% at constant 14LPM hot water in annulus side by varying cold water flow rates from 6LPM to 14LPM. The temperature enhancement is observed more in turbulent regime.

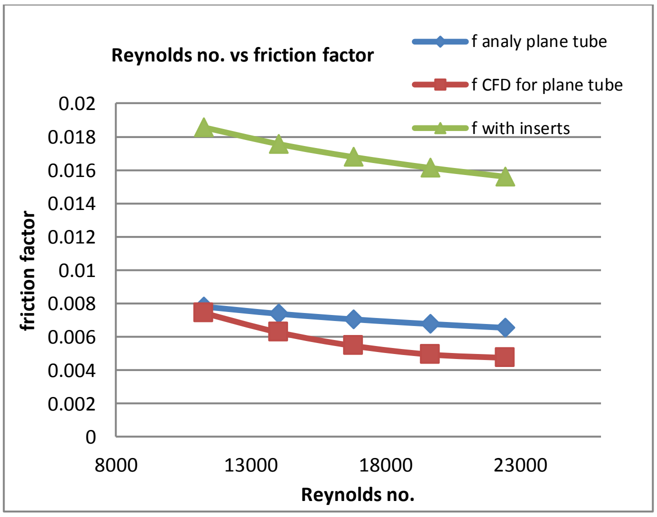


Figure16: Reynolds no. vs Friction Factor for Plane Tube vs With Inserts.

The above plot indicates that friction factor f increase with twisted tapes when compared to plane tube at 6LPM hot water condition. And friction factor decreases by increasing the Reynolds number.

Conclusion

In this work the hydrodynamics and thermal behavior of double pipe Heat Exchanger (DPHE) were studied.The purpose of this study was to investigate the heat transfer and fluid flow characteristics of a DPHE. The majority of the work was done numerically, using a commercial computational fluid dynamics package. Hot water and cold water were considered in DPHE in counter flow direction. A steady state computational fluid dynamics (CFD) models was simulated by ANSYS FLUENT 14.0. The effect of Reynolds number and Nusselt number on the flow behavior of the pipe was studied. Simulations were performed using different flow rates in the inner tube and in the annulus. Validation of the model was performed with the boundary conditions of constant wall temperature and constant heat flux. The results of the inner Nusselt number from these values were compared to work done by experimental work results for plane tube.

- The friction factor decreases with increasing Reynolds number and the heat transfer coefficient increases with increase in Reynolds number in plane tube.
- The friction factor increased with inserting twisted tape inserts in double pipe heat exchanger as compared with plane type.
- Heat transfer enhanced by 22% to 33% in temperature outlet at various Reynolds numbers with using inserts at 6LPM hot water.
- Heat transfer enhanced by 15% to 45% at various Reynolds number by using inserts at 10LPM hot water, the enhancement was maximum in turbulent regime.
- Increase of pressure drop is more by using twisted type inserts in cold pipe of 2 pass double pipe heat exchanger compared to plane tube.

References

[1] S.Liu and M.Sakri , A comprehensive review on heat transfer enhancement in pipe exchangers., Renewable Sustain energy Rev.,VOl.19 pp 64-81,2013
[2] Bodius Salam et al. experimental investigation was carried for measuring tube side heat transfer coefficient of water for turbulent flow in a circular tube fitted with twisted tape insert of 5.3 twist ratio.
[3] R. Sethumadhavan, M. Raja Rao, Turbulent flow heat transfer and fluid friction in helical-wire-coil-inserted tubes, Int. J. Heat Mass Transfer 26 (1983) 1833–1845.
[4] Royds did experimental in circular tube water fluid with different twist ratios they observed Tighter twist ratios results in greater heat transfer and pressure drop.
[5] Smithberg & Landis., Proposed mathematical model for tape generated swirl mechanism of full length twisted tape of DPHE.
[6] Cresswell., did Experimental in circular tube of twisted tape observed Ratio of maximum velocity to mean velocity is smaller in swirl flow compared with straight flow
[7] Paisarn Naphon, Effect of coil-wire insert on heat transfer enhancement and pressure drop of the horizontal

concentric tubes, International Communications in Heat and Mass Transfer 33 (2006) 753– 763

[8] A.García, P.G. Vicente, A. Viedma, Experimental study of heat transfer enhancement with wire coil inserts in laminar-transition-turbulent regimes at different Prandtl numbers, Int. J. Heat Mass Transfer 48 (2005) 4640–4651.

[9] Y. Shoji, K. Sato, D.R. Oliver, Heat transfer enhancement in round tube using wire coil: influence of length and segmentation, Heat Transfer – Asian Res. 32 (2003) 99–107.