Paper ID HP 10

COMPUTATIONAL ANALYSIS OF HEAT TRANSFER ENHANCEMENT IN A CIRCULAR TUBE USING SEMICIRCULAR NOTCHED ROTARY TWISTED TAPE INSERTS

Kiran Gund
Department of mechanical engineering, SPPU, Pune, India.
kiransgund@gmail.com

Vinayak Kale
Department of mechanical engineering, SPPU, Pune, India.
vmkale@gmail.com

Hemant Darokar
Department of mechanical engineering, SPPU, Pune, India.
hemantdarokar@rediffmail.com

Abstract

Heat transfer enhancement of air flow using twisted tape inserts in a circular copper tube have been investigated under turbulent flow regime. A simulation technique is employed to simulate the swirling flow of twisted tape insert with and without semi-circular notch with three different pitch ratios (P/w = 3,4 and 5). The Reynolds number is varied in the range of 25000 to 35000. The heat transfer enhancement represented by a Nusselt number (Nu) in turbulent flow regime is investigated by using CFD (Ansys 16.0 FLUENT) to investigate the physical behaviour of the thermal and fluid flows under uniform heat-flux condition. Results shows that the semi-circular notch twisted tape inserts develops a strong turbulence flow within fluid flow which increases heat transfer rate. The use of semi-circular notch twisted tape lead to a considerable increase in heat transfer and pressure drop in comparison with the plane tube. The heat transfer rate increases with increase in Reynolds number and decrease in pitch ratio.

Introduction

Most of the industries use the heat exchanger for their operations. Numbers of researchers have performed experimentation by using different heat transfer enhancement techniques to enhance performance of the heat exchanger to increase the heat transfer efficiency of existing heat exchanger, to reduce the floor size requirement of the heat exchanger, to reduce the installation cost by reducing material cost. These techniques are termed as heat augmentation techniques. These are classified as; Active technique, Passive technique, Compound technique. Active techniques use external power source for heat transfer enhancement while in passive techniques don't required any external power source for its operation. [1]

Thereafter and due to the advancement in computer hardware and software and consequent increase in computation speed, the CFD modelling technique has been recognized as a powerful and effective tool for better understanding of the complex hydrodynamics in many industrial processes. [2]

Literature Review

Marner and Bergles firstly investigated the importance of uniform wall temperature (UWT) boundary condition to a major group of heat exchangers used in chemical industry. They studied UWT heating and cooling of ethylene glycol at (Pr = 24-85, Re =3803470) using single twisted tape insert of y=5.4 in a tube and internally finned tubes. Observations show that both heat transfer and friction factor increased substantially beyond particular range of Reynolds number, at which secondary swirl flow and turbulence were induced in the flowing fluid. [3]

P. Murugesan, K. Mayilsamy, S. Suresh Experimentally investigated heat transfer and friction factor characteristics of circular tube fitted with plain twisted tapes (PTT) and U–cut twisted tapes (UTT) with twist ratios y = 2.0, 4.4 and 6.0. The Nusselt number and friction factor for the tube with UTT are higher than that of plain tube and also tube equipped with PTT. Over the range of Reynolds number considered average thermal enhancement factors in the tube equipped with PTT are found 1.15, 1.06, and 1.02 and tube equipped with UTT are 1.22, 1.10 and 1.06 respectively for twist ratios y =2.0, 4.4 and 6.0. The thermal enhancement factors for all the cases are more than unity. This indicates the effect of heat transfer enhancement due to the enhancing tool is more dominant than the effect of rising friction factor and vice versa. [4]

Shabanian et al. conducted an experiment and (CFD)modelling on heat transfer, friction factor and thermal performance of an air cooled heat exchanger equipped with three types of tube insert including butterfly,

classic, and jagged twisted tape. They found that the predicted results in terms of turbulence intensity are in good agreement with measured values of Nusselt number and friction factor. [5]
Naveen et al. carried out study on a computational fluid dynamics modeling on heat transfer, friction factor and thermal performance of water in concentric tube heat exchanger using twisted tapes (Plain, V-cut, double V-cut, Jagged V-cut) with different twist ratios (y=2.0, 4.0). The maximum thermal performance factor was obtained by the Jagged V-cut twisted tape insert compare to other twisted tapes [6]

CFD Modelling

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problem that involves fluid flow. CFD modelling is based on fundamental governing equations of fluid dynamics as the conservation of mass, momentum, and energy. CFD helps to predict the fluid flow behaviour based on the mathematical modelling using software tools. It is now widely used and is acceptable as a valid engineering tool.
Steps involved in CFD:

Pre-Processing: This is the first step of CFD simulation process which helps in describing the geometry in the best possible manner. One needs to identify the fluid domain of interest. For the present work the twisted tape geometry with and without semi-circular notch at 3deiiferent pitch ratios 3,4 and 5 has been created by using catia. Cu tube is considered as enclosure. By using the Boolean operation we subtract the twisted tape geometry from the solid Pipe which is our flow domain. The domain of interest is then further divided into smaller segments known as mesh generation step. ANSYS is used as Pre-Processing software for Meshing process.

Solver: Once the problem physics has been identified, fluid material properties, flow physics model, and boundary conditions are set to solve using a computer. I have used air as fluid and copper as solid pipe. The boundary conditions are then specified. air Inlet velocity is 12 m/s. temperature of the incoming air is 314 K. Heat flux of 1555.21 w/m^2 is applied to the external surface area of the copper pipe of 800 mm length. ANSYS FLENT is used as solver. Using this software, the governing equations related to flow physics problem are solved.

Post-Processing: The next step after getting the results is to analyse the results with different methods like contour plots, vector plot, streamlines, data curve etc. for appropriate graphical representations and report. ANSYS CFD-Post is used for the post processing to plot contour, streamlines, eddy viscosity gradient, turbulence kinetic energy, etc. [7]

Results and discussion

In the post processing stage some of plots obtained are:

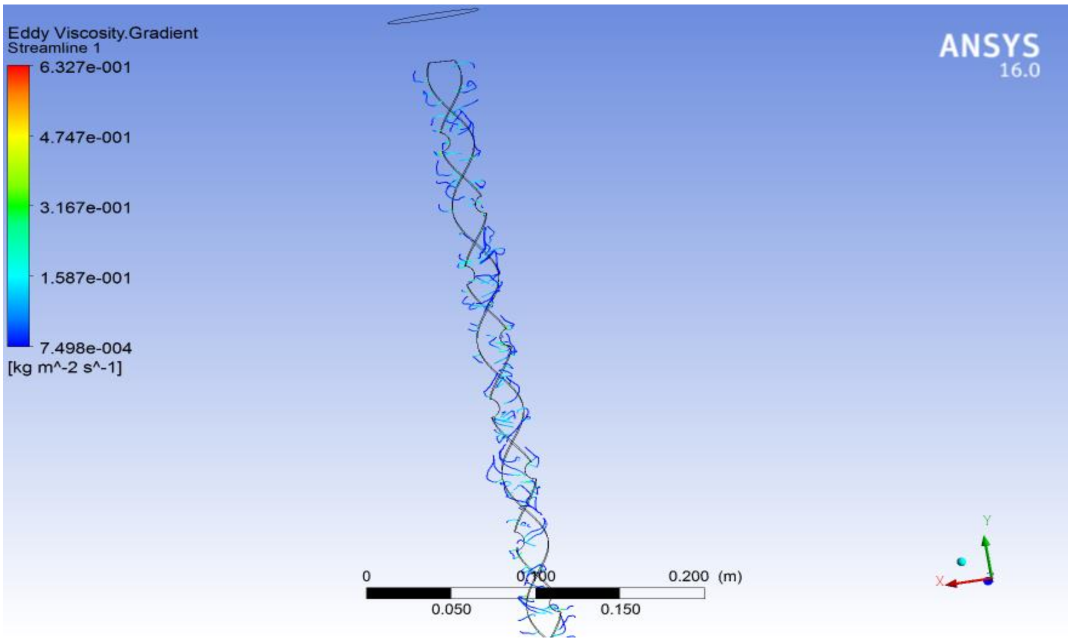

Figure 1 Plot Eddy viscosity Gradient.

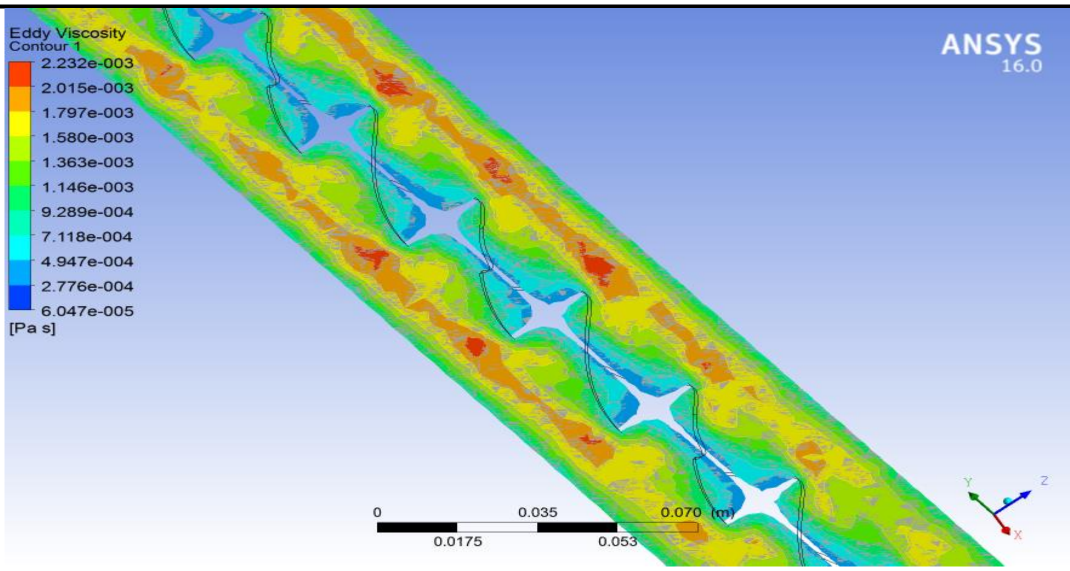

Figure 2 Plot of Eddy viscosity.

From the plot it is observed that Eddy viscosity is larger near the notch zone of the twisted tape. The turbulent transfer of momentum by eddies giving rise to an internal fluid friction, in a manner analogous to the action of molecular viscosity in laminar flow, but taking place on a much larger scale. The increasing turbulence intensity increases heat transfer rate.

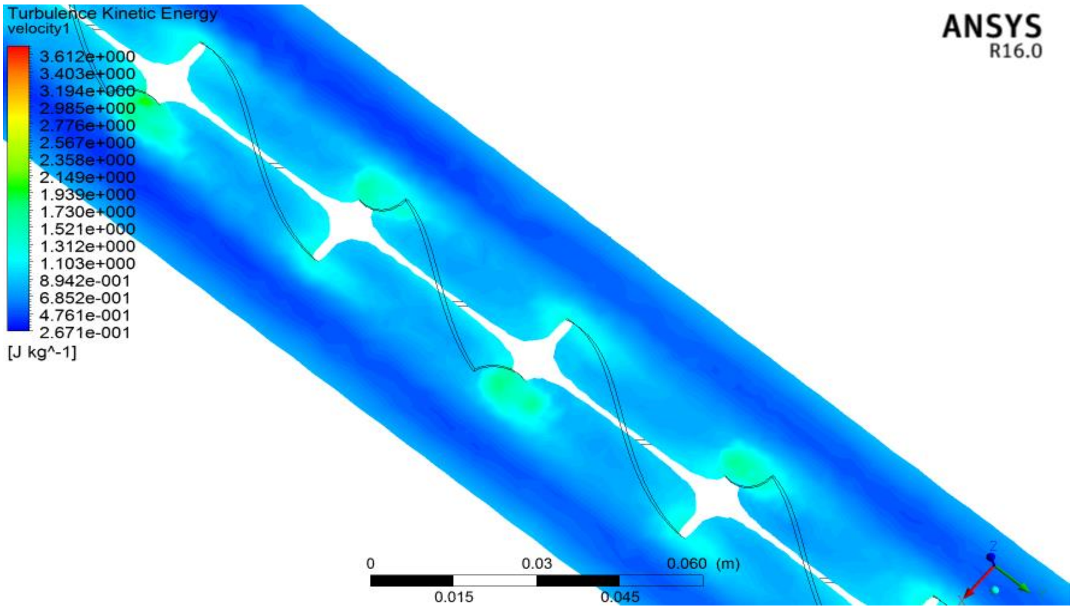

Figure 3 Plot of Turbulence Kinetic Energy.

In fluid dynamics, turbulence kinetic energy is the mean kinetic energy per unit mass associated with eddies in turbulent flow. Physically, the turbulence kinetic energy is characterized by measured root-mean-square velocity fluctuations. Plot shows the higher Kinetic energy is observed near the notch zone which helps in better mixing of the fluid and turbulence in flow field.

To observe the variation in temperature within the test section one line is located within the copper tube. The temperature at 10 steps is plotted against the step count for all three twist ratios.

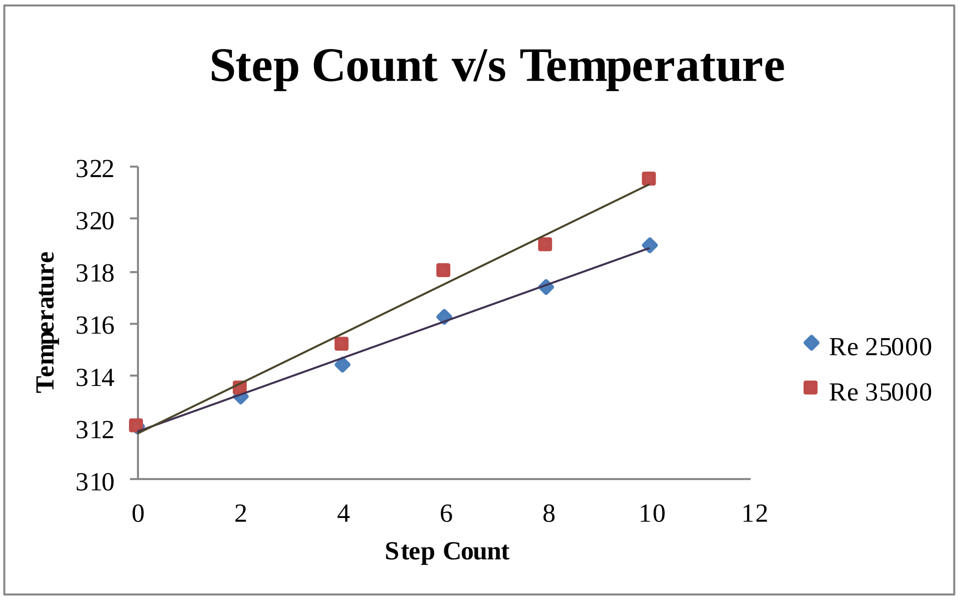

Figure 4. Step count V/S Temperature along the line For SFTT at y=3.0 at different Re

In Figure 4 the temperature variation within the pipe with semi-circular notch has been recorded at two different Reynolds number. It shows that as Reynolds number goes on increasing the heat carried away by the air goes on increasing i.e. heat transfer rate goes on increasing.

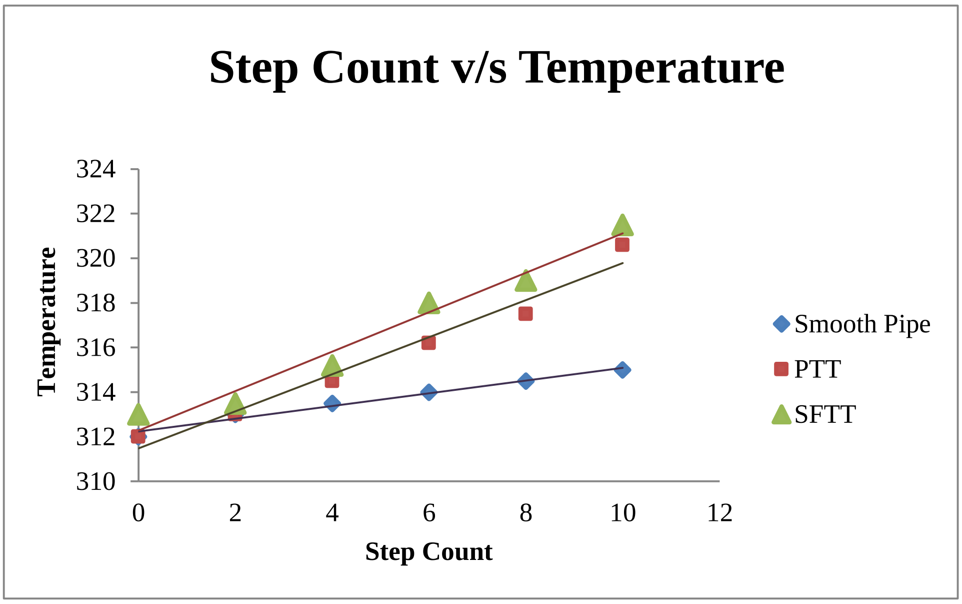

Figure 5. Step count V/S Temperature along the line For SFTT at y=3.0

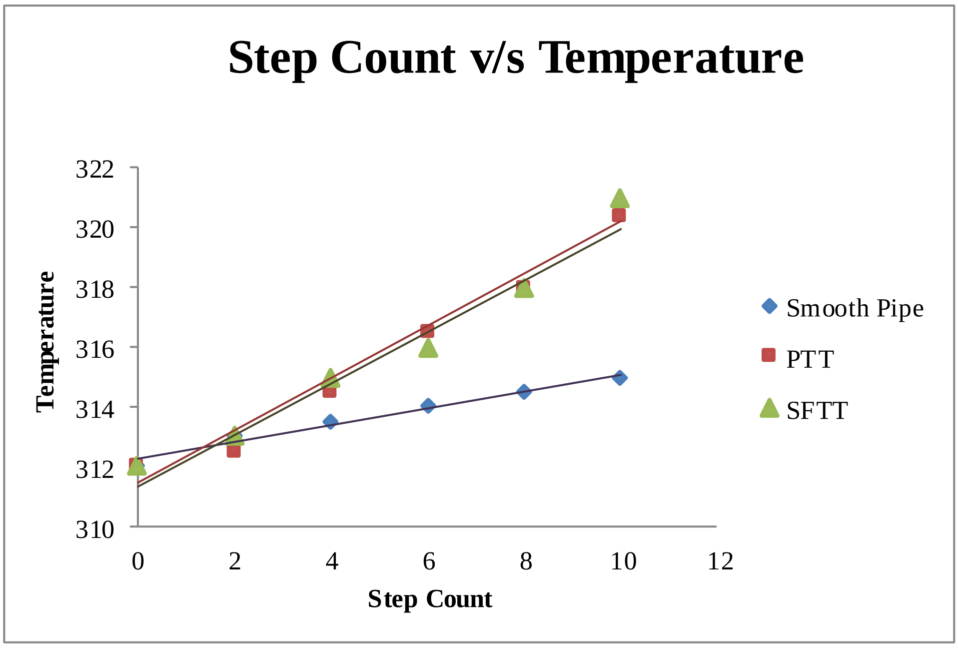

Figure 6 Step count V/S Temperature along the line For SFTT at y=4.0

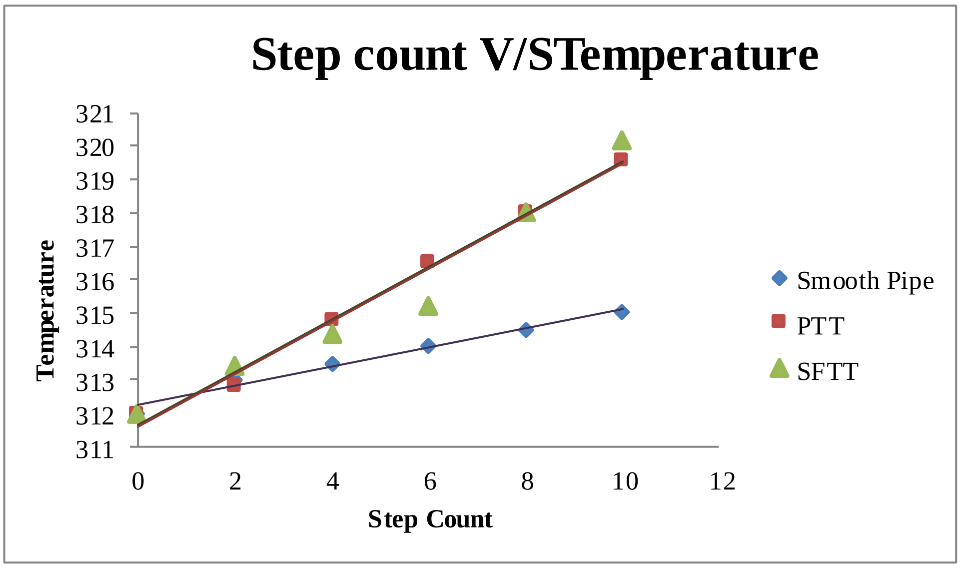

Figure 7 Step count V/S Temperature along the line For SFTT at y=5.0

From the above figures it is cleared that as twist ratio goes on decreasing the difference in temperature between inlet and outlet of test section goes on increasing i.e. heat transfer rate increases as the twist ratio decreases. In comparison of SFTT,PTT and Smooth pipe the heat transfer with SFTT is maximum. Maximum Heat transfer occurs when a circular tube is inserted with semi-circular notched twisted tape insert of twist ratio 3 at Re = 35000

Conclusion:

- Twisted tape inserts increases the heat transfer rate in the heat exchanger by increasing turbulence in the fluid flow.
- Swirl flow increases the thermal contact by reducing boundary layer thickness.
- The turbulent flow ensures the better mixing of the fluid particles which leads to increase in heat transfer efficiency.
- The heat transfer rate increases as Reynolds number goes on increasing.
- The heat transfer rate increases as twist ratio goes on decreasing.

- The maximum heat transfer obtained in this experimentation is at pipe with semicircular notch twisted tape insert at smaller twist ratio y=3.0 and Re = 35000.

References

[1]Varun, M.O.Garg, Himanshu Nautiyal , Sourabh Khurana, M.K.Shukla, ' Heat transfer augmentation using twisted tape inserts: A review',Renewable and Sustainable Energy Reviews 63(2016)193–225.
[2] V. Kumar, S. Saini, M. Sharma, and K. D. P. Nigam, "Pressure drop and heat transfer study in tube-in-tube helical heat exchanger," Chemical Engineering Science, vol. 61, no. 13, pp. 4403–4416,2006.
[3] W.J.Mrner and A.E.Bergles, 'Augmentaion of highly viscous laminar heat transfer inside the tube with constant wall temperature', Exp. Thermal fluid science,2,1989,252-267.
[4] P. Murugesan, K. Mayilsamy, S. Suresh, 'Heat Transfer and Friction Factor in a Tube Equipped with U-cut Twisted Tape Insert', Jordan Journal of Mechanical and Industrial Engineering, Volume 5, Number 6, Dec. 2011 ISSN 1995-6665 Pages 559 - 565
[5] S. R. Shabanian, M. Rahimi, M. Shahhosseini, and A. A. Alsairafi, "CFD and experimental studies on heat transfer enhancement in an air cooler equipped with different tube inserts,"InternationalCommunicationsinHeatandMassTransfer,vol.38,no.3,pp.383–390,2011.
[6] Naveen, S., and S. Bhuvaneshwaran. "CFD Analysis of Concentric Tube Heat Exchanger Using Twisted Tapes." International Journal of Advance Research, Ideas and Innovations in Technology 3, no. 1 (2017).
[7] Ahmed. L. Abdullah and Fuat Yilmaz, "Computational Analysis of Heat Transfer Enhancement in a Circular Tube Fitted with Different Inserts" Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, ISSN: 2289-7879.