International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2020): 7.803

Improvement in Performance of Heat Exchanger by using Promoters

Saurabh D Vighe[1], VG Gore[2]

[1]PG Student Department of Mechanical Engineering PR Potepatil College of Engineering and Technology, Amravati, Maharashtra, India
Email: *saurabhvighe7391[at]gmail.com*

[2]Assistant Professor, Department of Mechanical Engineering P R Pote Patil College of Engineering and Technology, Amravati, Maharashtra, India
Email: *gorevijay1973[at]gmail.com*

Abstract: ***Heat transfer enhancement has been always a significantly interesting topic in order to develop high efficient, low cost, light weight, and small heat exchangers. The energy cost and environmental issue are also encouraging researchers to achieve better performance than the existing designs. Two of the most effective ways to achieve higher heat transfer rate in heat exchangers are using different kinds of inserts and modifying the heat exchanger tubes. There are different kinds of inserts employed in the heat exchanger tubes such as helical/twisted tapes, coiled wires, ribs/fins/baffles, and winglets. This paper presents an overview about the early studies on the improvement of the performance of thermal systems by using different kinds of inserts. Louvered strip insert had better function in backward flow compared to forward one. Modifying the shape of twisted tapes led to a higher efficiency in most of the cases excpet for perforated twisted tape and notched twisted tape. Combination of various inserts and tube with artificial roughness provided promising results. In case of using various propeller types, heat transfer enhancement was dependent on higher number of blades and blade angle and lower pitch ratio.***

Keywords: Heat transfer enhancement, compact surface, turbulence parameter, pressure drop, thermodynamic performance

1. Introduction

Using passive techniques in order to enhance heat transfer characteristics in heat exchanger has been an interesting topic for scientists and researchers during recent decades. Numerical and experimental studies have been conducted in order to improve heat transferred by these techniques. The demand of reduction of the cost and dimensions of heat exchanger has motivated the searchers to investigate different ways of heat transfer enhancement. Passive heat transfer enhancement techniques are mostly preferred due to their simplicity and applicability in many applications. Furthermore, in passive techniques, there is no need of any external power input except to move the fluid. The devices in this category include surface coating, rough surfaces, extended surfaces, turbulent/swirl flow devices, convoluted (twisted) tube, and tube inserts. Various kinds of inserts have been employed in the heat exchangers such as helical/twisted tapes, coiled wires , ribs/fins/baffles , and **winglets,** refrigeration, air-conditioning, and commercial heat pump industries as well as in the chemical, petroleum, and numerous other industries. Using inserts in tubular heat exchangers not only reduced the heat exchanger size but also provided thermal, mechanical, and economic advantages in heat exchangers. The quantities of the two fluids resident in heat exchangers, as an important safety consideration, have been greatly decreased by compact enhanced designs.

1.1 Inserts

The heat transfer coefficient improvement capability beside a minimum loss in friction factor defines the thermohydraulic performance of an insert. Tube inserts have been utilized for heat transfer enhancement and fouling mitigation in different industrial fields such as petroleum refineries and chemical plants for several years. In this paper, the literature reviews are classified into louvered strip insert twisted tape, swirl flow devices insert, wirecoil insert, conical ring insert, winglet-type vortex generators, and brush and pin elements inserts.

1.2 Louvered Strip Insert

In case of using louvered strip insert, several parameters such as inclination angles, distance of wings, shape of wings, and direction of flow can be modified to improve the heat transfer enhancement. Schematic of forward and backward louvered strips are shown in Figure 1 [11]. Eiamsa-ard et al. [11] reported that, in case of using louvered strips, the general backward flow has better performance compared to forward one. Figure 2 also shows a better thermohydraulic performance was achieved with use of bigger inclination angles together with a smaller pitch (smaller distance between wings).

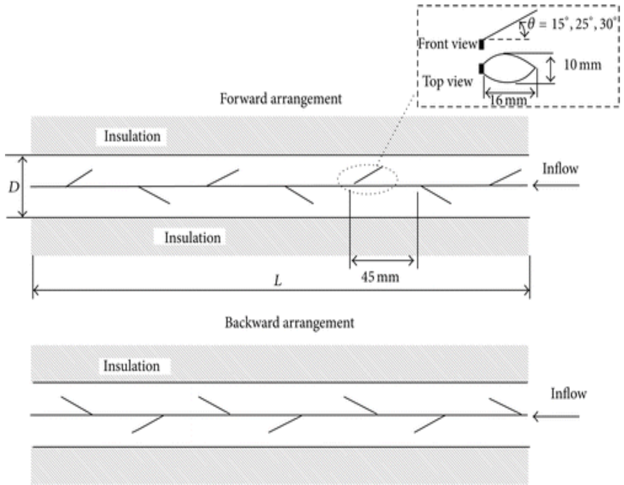

Figure 1: Forward and backward arrangements of louvered strips

Effect of different inclination angles (θ) and different distance between wings (*S*) of conical strip inserts on PEC at different Reynolds number.

2. Methodology

2.1 Computational Fluid Dynamics (CFD) Analysis

2.1.1 Introduction of CFD

Computational Fluid Dynamics or CFD can be described, as the use of computers to analyze systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions based on numerical approach. In this numerical approach the equations (usually partial differential in form) govern a process of interest are solved numerically. The technique is very powerful and spans over a wide range of industrial and non-industrial application areas. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid, and Nervier-Stokes equations for viscid flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Lastly, if the problem is highly dynamic and occupies a wide range of scales, the grid itself can be dynamically modified in time, as in adaptive mesh refinement methods. CFD is attractive to industry since it is more cost-effective than physical testing. However, one must note that complex flow simulations are challenging and error-prone and it takes a lot of engineering expertise to obtain validated solutions. CFD embraces a variety of technologies including mathematics, computer science, engineering and physics and these disciplines have to be brought together to provide the means of modeling fluid flows. Such modeling is used in many fields of science and engineering but, if it is to be useful, the results that it yields must be a realistic simulation of a fluid in motion. At present this depends on the problem being simulated, the software being used andthe skill of the user. Until recently the user of CFD has been a specialist, probably trained to doctoral level, working in a research and development department. Now, however, the technology is more widely available both in industry and academia and so it is being used to provide insights into many aspects of fluid motion. 'CFD' is becoming vital component in the design ofindustrial products and processes. The reasons for its popularity are:

1) Dramatic increase in computer power.
2) Commercial CFD codes at reasonable cost.
3) Easy to use front-ends in CFD codes for rapid set-up of problems and efficient analysis of the results.
4) Improvements in the physics and number of sub-models available to the user e.g. Thermal radiation, turbulence, soot, and pollution chemistry.
5) Increased validation studies by code vendors leading to increased confidence in the results.

2.1.2 Application Areas for CFD

1) Aerodynamics of aircraft and vehicles (Lift and Drag): CFD is used in conjunction with wind tunnel tests to determine the performance of various configurations.
2) Hydrodynamics of ships
3) Flow over missiles (Lift, Drag and side force data): Simulation performance can be obtained by using CFD tool.
4) Power plant, Combustion in IC engines and gas turbines (Air flow inside I.C. engines).
5) Turbo-machinery: Flows inside rotating passages, diffusers etc.
6) Electrical and electronic engineering: Thermal modeling of (cooling) equipment including microcircuits, chips, PCB for efficient working.
7) Jet flow inside nuclear reactor halls: Such problems involve the simulation of fault conditions (like failure of nuclear reactors), which is very difficult to perform in actual experiment. Hence, computation is the only way of trying to understand such flows.
8) Chemical process engineering: Mixing and separation, polymer moldings.
9) External and internal environment of buildings: wind loading and heating/ventilation.
10) Hydrology and oceanography: Flows in rivers, estuaries, oceans
11) Biomedical engineering: Blood flows through arteries and veins.

2.1.3 The Strategy of CFD

Broadly, the strategy of CFD is to replace the continuous problem domain with a discrete domain using a grid. In the continuous domain, each flow variable is defined at every point in the domain. For instance, the pressure p in the continuous 1D domain shown in the figure 3.8 below would be given as p = p(x), 0 < x < 1

In the discrete domain, each flow variable is defined only at the grid points. So, in the discrete domain shown below, the pressure would be defined only at the N grid points.

pi = p(xi), i = 1, 2, . . . ,N

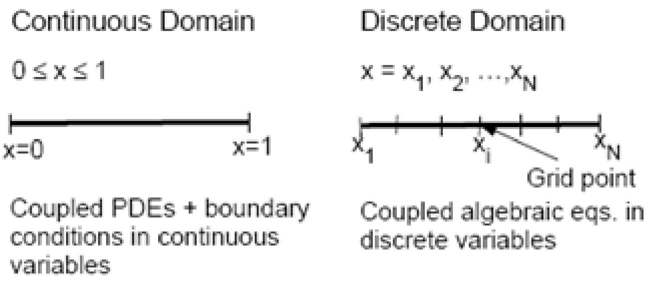

Figure 2: Difference between continuous and Discrete Domain

In a CFD solution, one would directly solve for the relevant flow variables only at the grid points. The values at other locations are determined by interpolating the values at the grid points. The governing partial differential equations and boundary conditions are defined in terms of the continuous variables p, V etc. One can approximate these in the discrete domain in terms of the discrete variables pi, Vi etc. The discrete system is a large set of coupled, algebraic equations in the discrete variables. Setting up the discrete system and solving it (which is a matrix inversion problem) involves a very large number of repetitive calculations, a task we humans palm over to the digital computer. This idea can be extended to any general problem domain. The following figure 3.9 shows the grid used for solving the flow over an airfoil.

3. CFD Analysis of Heat Exchanger with Twisted Pipes

For the analysis of solar air heater without baffled duct following steps have been performed

- 3D Modelling
- Meshing
- Application of boundary conditions
- Plotting of Results

i) Modeling

3D Model has been created in modeling tool of Ansys Fluent 14.5 by using following steps.

- A plane has been selected for sketching of heat exchanger
- A rectangle of proper dimension is created in the selected plane of sketcher tool.

2D sketch has been extruded in modelling tool with the proper thickness.

The dimensions of geometry are given below.
Diameter of cylinder =0.25 m
Length of cylinder=1m

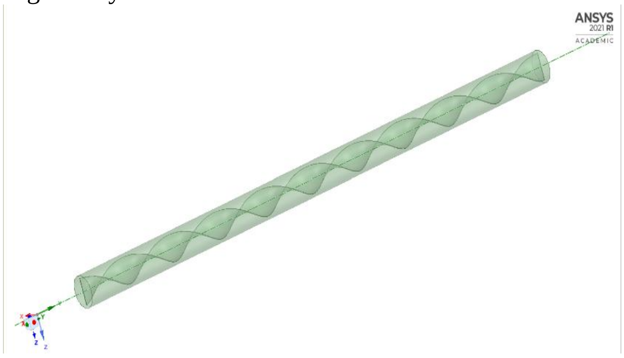

Image 1: Cylindrical heat exchanger without twisted pipe insetrs

ii) Meshing

Meshing is done in the same software Ansys fluent. After achieving satisfied nodes it has been sent to the processing by applying boundary conditions in processor

iii) Processing

For CFD simulation specific boundary conditions have been specified in fluent tool is given below:

- Heat flux= 1000 w/m^2
- Temperature at inlet= 300 k
- Pressure at outlet= 1 atm

iv) Post processing

a) For inlet velocity (V= 0.05 m/s)

For mass flow rate V= 0.05 m/s the temperature distribution is found as per follows:

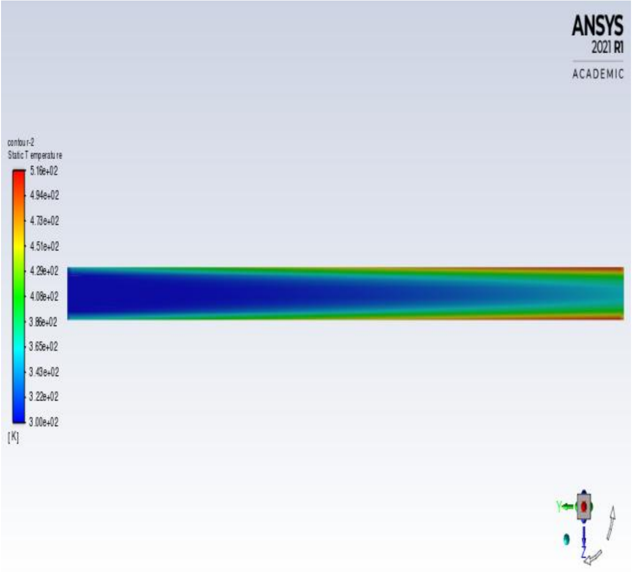

Image 2

For V= 0.05 m/s the temperature distribution is found as per follows:

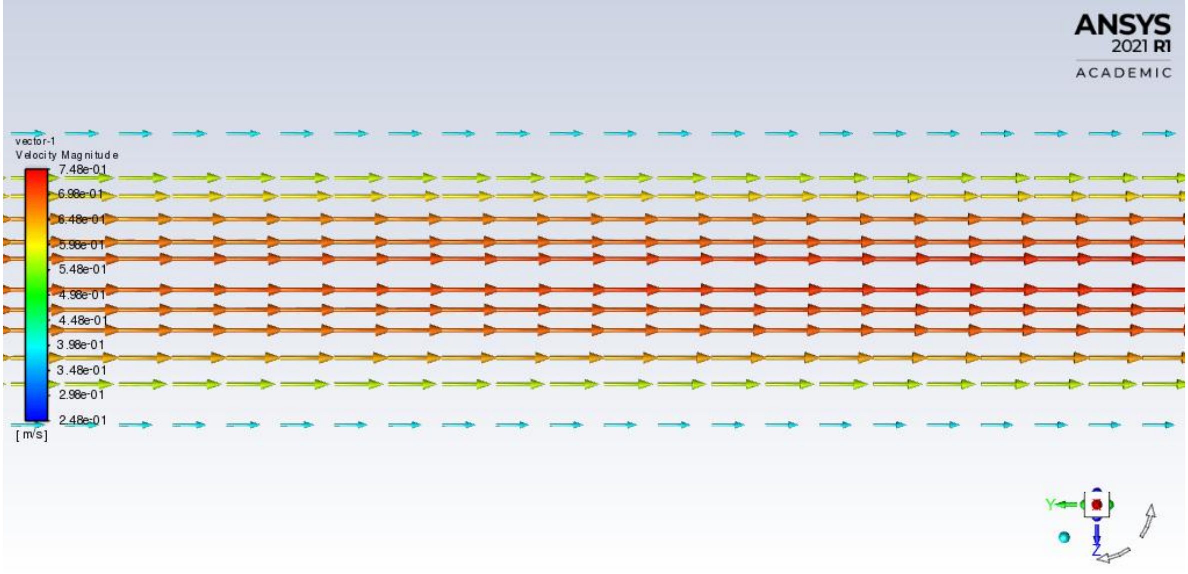

Image 3

For V= 0.05 m/s the Nusselt number is found as per follows:

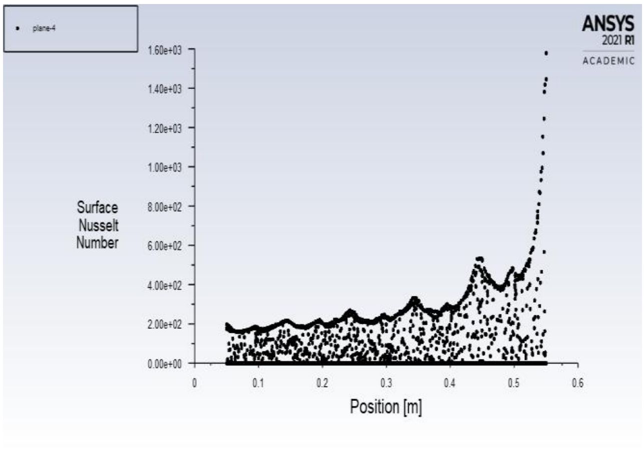

Image 4

- Results for the given inputs are as per follows:
- Inlet velocity V= 0.05
- Outlet temperature T=463.6878 K and
- Nusselt number at the middle plane Nu=20.1471.

b) For inlet velocity (V= 0.75 m/s)

For mass flow rate V= 0.75 m/s the temperature distribution is found as per follows:

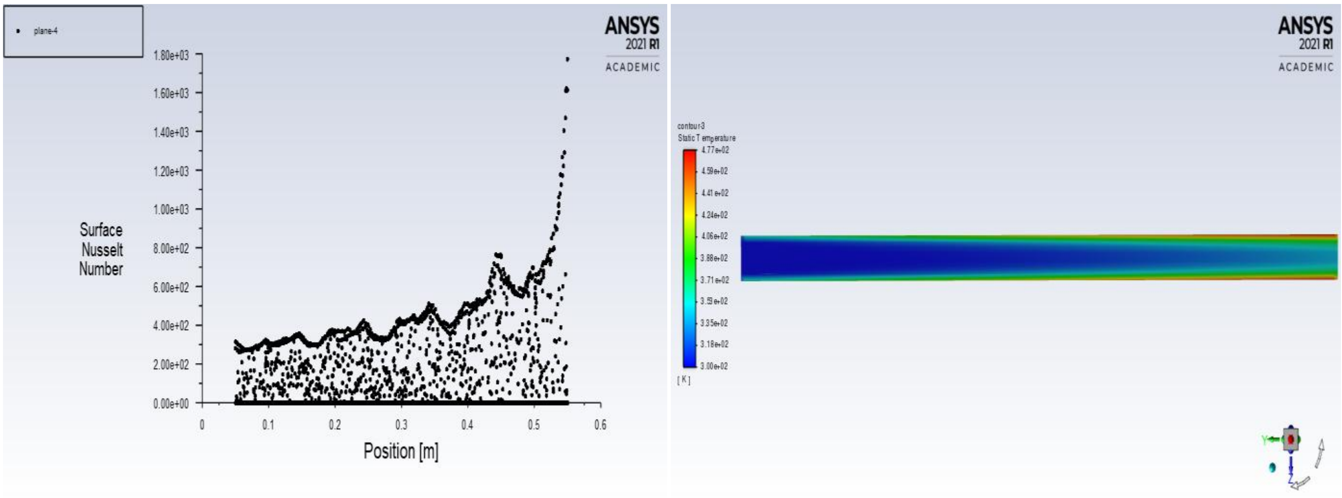

Image 5

For V= 075 m/s the Nusselt number is found as per follows:

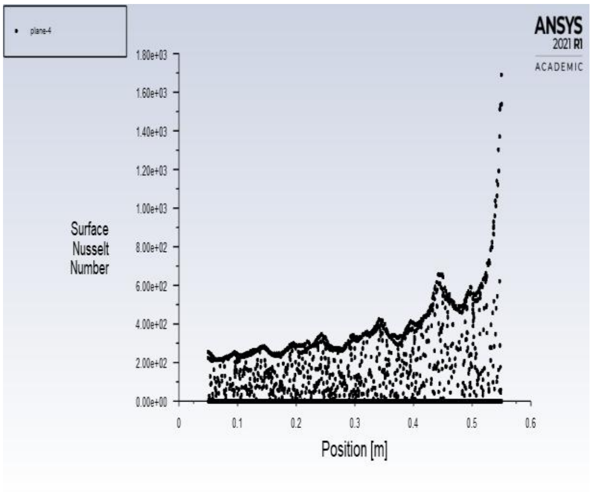

Image 6

V= 0.75, T=409.2177K and Nu=25.94149

c) For inlet velocity (V= 1 m/s)

For mass flow rate V= 1 m/s the temperature distribution is found as per follows:

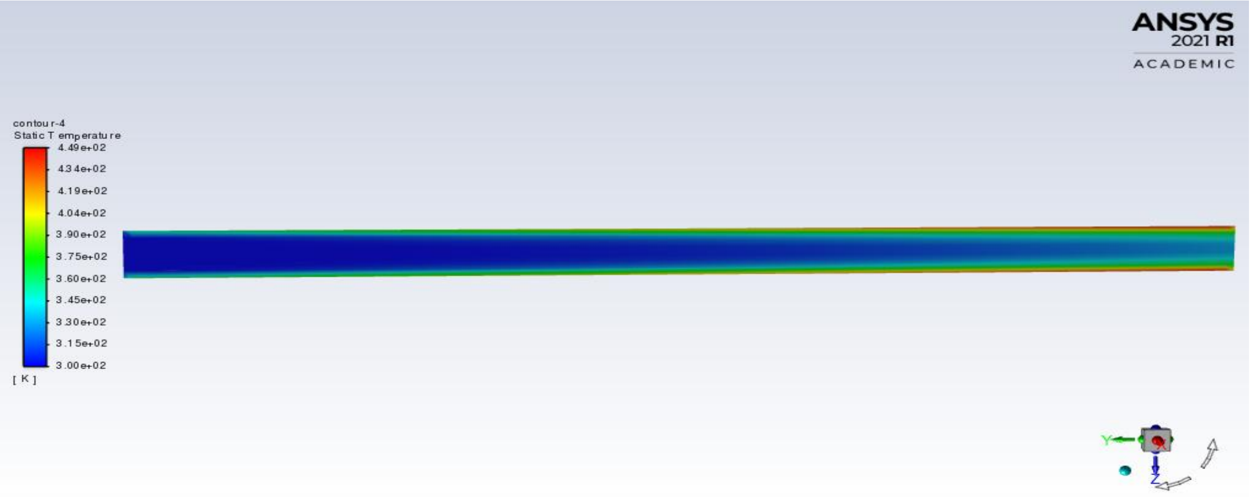

Image 7

For V= 1 m/s the Nusselt number is found as per follows

Results for the given inputs are as per follows:
Inlet velocity V= 0.75 m/s
Outlet temperature T=409.2177K K and
Nusselt number at the middle plane Nu=25.94149

c) For inlet velocity (V= 1.25 m/s)
For mass flow rate V= 1.25 m/s the temperature distribution is found as per follows

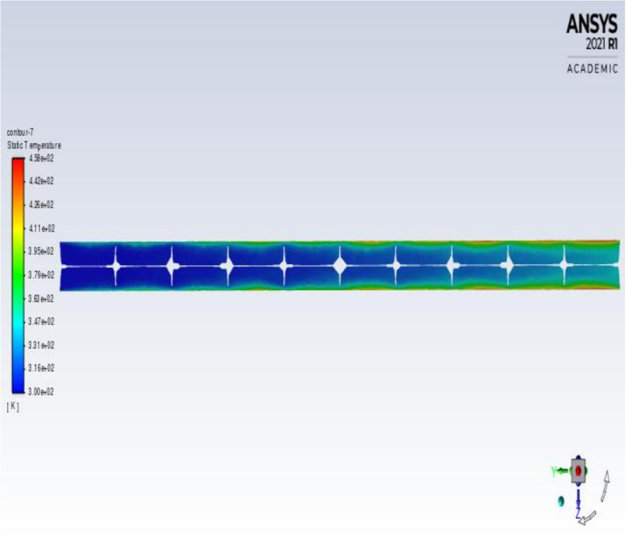

Image 8

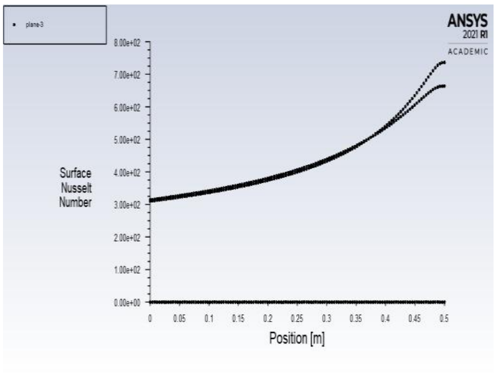

Image 9

1.25

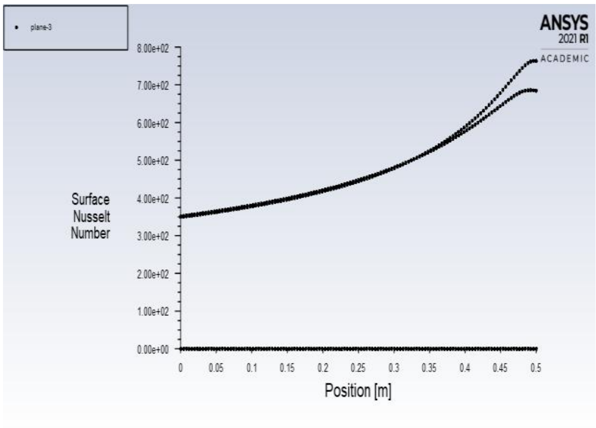

Image 10

Results for the given inputs are as per follows:
Inlet velocity V= 1.25 m/s
Outlet temperature T=365.1849 K and
Nusselt number at the middle plane Nu=34.4837

d) For inlet velocity (V= 1.5 m/s)
For mass flow rate V= 1.5 m/s the temperature distribution is found as per follows

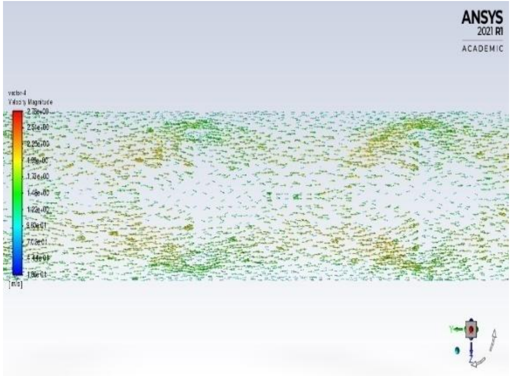

Image 11

Inlet velocity V= 1.25 m/s
Outlet temperature T=356.1461 Kand
Nusselt number at the middle plane Nu=147.0147

4. Result Analysis

Parameters	Without twist tape	With twist tape	Without twist tape	With twist tape
Inlet velocity (m/s)	Outlet temperature (K)	Outlet temperature (K)	Surface Nu no at mid plane	Surface Nu no at mid plane
0.5	463.6878	471.0154	20.1471	83.71223
0.75	409.2177	413.3624	25.94149	103.7774
1	381.6812	384.7363	31.36958	120.9104
1.25	365.1849	367.5039	34.4837	134.7815
1.5	354.2062	356.1461	39.8215	147.0147

5. Future Scope

Further studies are essential in future to discover and optimize geometries of various inserts and vortex generators in order to enhance the thermal characteristics of heat exchangers. Additionally, the recent development in nanotechnology has widened a promising field in heat transfer investigation. Combination of inserts and nanofluids might be a kind of propitious idea to enhance the thermal systems performance which has been rarely reported in the open literature. Numerous opportunities will be available when authors focus their efforts on using nanofluids combined with different kinds of inserts in large or even microscale heat exchangers and heat sinks. The comparison of nanofluids and conventional fluids in different shapes of compact heat exchangers and heat sinks in terms of heat transfer and pressure drops has shown huge improvement.

The authors declare that there is no conflict of interests regarding the publication of this paper

6. Conclusion

The main resistance to heat transfer in conventional heat exchangers is the thermal conduction through the laminar sub layer attached to the surface. Improvement of the heat transfer rate involves the removal of this layer at the expense of increased pressure drop. Heat transfer enhancement techniques can be applied at the design stage of new units or in the retrofit of existing units. In design, fluid velocity is a degree of freedom that can be manipulated by appropriate choice of the exchanger dimensions related to cross sectional area. Alternatively, mechanical devices such as inserts are available to promote local turbulence and increase the heat transfer rate. Such devices can also be used in retrofit for increased heat recovery or increased production. New exchanger technology has emerged to provide alternative solutions to accomplish the following goals: (1) to achieve the given heat load within the limitations imposed by pressure drop in the smaller heat transfer equipment, or (2) increase the heat load within the limitations of pressure drop for the same installed heat transfer area.

New exchanger technology is evolving in the direction of more compact surfaces. A compact surface is designed such that the thermohydraulic performance shows higher heat transfer rate and reduced pressure drop. One of the main problems still to overcome with compact surfaces is the limitations they have in terms of the operating conditions that can withstand, since they cannot operate at high temperatures and pressures. Research and development in this area are focused in the development of new geometries and materials of construction.

References

[1] Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, AhmerRais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan5, CFD Applications In Various Heat Exchangers Design: A Review, Department Of Mechanical Engineering, University Of Engineering & Technology, Applied Thermal Engineering, 2011.[1]

[2] Hamidou Benzenine, Rachid Saim , Said Abboudi and Omar Imine, “Numerical analysis of a turbulent flow in a channel provided with transversal waved baffles”,International Journal of thermal science, Vol. No. 17, pp. 1491-1499, 2013.]

[3] Apu Roy, D.H.Das, CFD Analysis Of A Shell And Finned Tube Heat Exchanger For Waste Heat Recovery Applications, National Institute Of Technology, 2011.

[4] Movassang SZ, Taher FN, Razmi K, Azar RT. Tube bundle replacement for segmental and helical shell and tube heat exchangers. Applied Thermal Engineering. 2013;51:1162-1169

[5] Zhang L, Xiaa Y, Jianga B, Xiaoa X, Yang X. Pilot experimental study on shell and tube heat exchangers with small-angles helical baffles. Chemical Engineering and Processing. 2013;69:112-118

[6] Chen GD, Zeng M, Wang QW, Qi SZ. Numerical studies on combined parallel multiple shell-pass shell-and-tube heat exchangers with continuous helical baffles. Chemical Engineering Transactions. 2010;21:229-234

[7] Gao B, Bi Q, Nie Z, Wu J. Experimental study of effects of baffle helix angle on shellside performance and shell-and-tube heat exchangers with discontinuous helical baffles. Experimental Thermal and Fluid Science. 2015;68:48-57

[8] Picón-Núñez M, García-Castillo JL, Alvarado-Briones B. Thermo-hydraulic design of single and multi-pass helical baffle heat exchangers. Applied Thermal Engineering. 2016;105(25):783-791

[9] Kays WM, London AL. Compact Heat Exchangers. 3rd ed. EUA: McGraw Hill; 1984

[10] Marriott J. Recover more heat with plate heat exchangers. Chemical Engineering. 1971; 78(8):127-134